Soft Servo
SYSTEMS, INC

ServoWorks S-100T
Part Programming Manual

Revision 1.9

Warning

The product described herein has the potential – through misuse, inattention, or lack of understanding – to create conditions that could result in personal injury, damage to equipment, or damage to the product(s) described herein. Machinery in motion and high-power, high-current servo drives can be dangerous; potentially hazardous situations such as runaway motors could result in death; burning or other serious personal injury to personnel; damage to equipment or machinery; or economic loss if procedures aren't followed properly. Soft Servo Systems, Inc. assumes no liability for any personal injury, property damage, losses or claims arising from misapplication of its products. In no event shall Soft Servo Systems, Inc. or its suppliers be liable to you or any other person for any incidental collateral, special or consequential damages to machines or products, including without limitation, property damage, damages for loss of profits, loss of customers, loss of goodwill, work stoppage, data loss, computer failure or malfunction claims by any party other than you, or any and all similar damages or loss even if Soft Servo Systems, Inc., its suppliers, or its agent has been advised of the possibility of such damages.

It is therefore necessary for any and all personnel involved in the installation, maintenance, or use of these products to thoroughly read this manual and related manuals and understand their contents. Soft Servo Systems, Inc. stands ready to answer any questions or clarify any confusion related to these products in as timely a manner as possible.

The selection and application of Soft Servo Systems, Inc.'s products remain the responsibility of the equipment designer or end user. Soft Servo Systems, Inc. accepts no responsibility for the way its controls are incorporated into a machine tool or factory automation setting. Any documentation and warnings provided by Soft Servo Systems, Inc. must be promptly provided to any end users.

This document is based on information that was available at the time of publication. All efforts have been made to ensure that this document is accurate and complete. However, due to the widely varying uses of this product, and the variety of software and hardware configurations possible in connection with these uses, the information contained in this manual does not purport to cover every possible situation, contingency or variation in hardware or software configuration that could possibly arise in connection with the installation, maintenance, and use of the products described herein. Soft Servo Systems, Inc. assumes no obligations of notice to holders of this document with respect to changes subsequently made. Under no circumstances will Soft Servo Systems, Inc. be liable for any damages or injuries resulting from any defect or omission in this manual.

Soft Servo Systems, Inc. makes no representation or warranty, expressed, implied, or statutory with respect to, and assumes no responsibility for the accuracy, completeness, sufficiency, or usefulness of the information contained herein. NO IMPLIED WARRANTIES OF MERCHANTABILITY OR FITNESS OF PURPOSE SHALL APPLY.

Important Notice

The information contained in this manual is intended to be used only for the purposes agreed upon in the related contract with Soft Servo Systems, Inc. All material contained herein is subject to restricted rights and restrictions set forth in the contract between the parties.

These manuals contain confidential and proprietary information that is not to be shared with, nor distributed to, third parties by any means without the prior express, written permission of Soft Servo Systems, Inc. No materials contained herein are to be duplicated or reproduced in whole or in part without the express, written permission of Soft Servo Systems, Inc.

Although every effort and precaution has been taken in preparing this manual, the information contained herein is subject to change without notice. This is because Soft Servo Systems, Inc. is constantly striving to improve its products. Soft Servo Systems, Inc. assumes no responsibility for errors or omissions.

Table of Contents

List of Tables

List of Figures

Chapter 1: Overview of the ServoWorks S-100T Part Programming Manual

1.1 Description of the ServoWorks S-100T Part Programming Language

The ServoWorks S-100T part programming language consists of blocks of code made up of B codes, F codes, G codes, M codes, N codes, S codes, T codes, etc.

The most commonly used code is the G code, which is why this coding system is also sometimes referred to as "G code" or "G-code part programming." The G Codes are a wide range of preparatory codes that prepare the machine to treat the information that follows in a distinct manner and to execute it. In essence, G codes determine the mode of the system. G Codes are described in detail in *Chapter 6: Codes (Preparatory Functions)*.

M and B codes define miscellaneous functions related to CNC machine control. These codes work like on/off switches for the functions they control. The M codes are described in detail in *Chapter 7: M Codes and B Codes (Miscellaneous Codes)*.

S codes define the spindle speed command. S codes are described in *Chapter 8: Spindle Functions and S Codes*.

T codes define the tool selection command. T codes are described in *Chapter 9: Tool Functions and T Codes*.

1.2 Components of the ServoWorks S-100T Part Programming Language

The G-code coding system used by ServoWorks S-100T is made up of ***words***. These words are strung together to form ***blocks of code***. A block of code (or "instruction") is one or more words together that are grouped together on one line.

A word is an ***address*** followed by numbers. The address defines the meaning of the number that follows the address. In other words, ServoWorks S-100T uses the address to determine what the number stands for. A list of addresses recognized by ServoWorks S-100T is included in *Chapter 2: Address Descriptions*.

A typical block of code starts with the sequence number of the block, and looks like this:

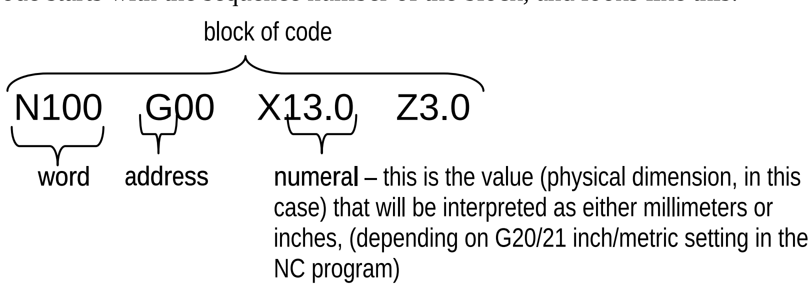


Figure 1-1: Typical Block of Code

1.3 Numbers in the ServoWorks S-100T Part Programming Language

The numbers that accompany the parameters X and Z for G codes are absolute values defining the distance or position for that parameter. These values are physical dimensions that you can set to be interpreted as either millimeters or inches, by setting the "unit of measurement" parameter specified in the "General" screen of Configuration Mode of ServoWorks S-100T. This program unit can be overridden by G20/21 inch/metric settings in a part program. (See *Section 3.3.6: Inch / Metric Data Input (G20, G21).*)

1.4 Blocks of Code and Duplicate Parameters

It's very important for each block to be on a separate line. If you try to put two blocks on the same line, they get combined together. This could have serious consequences: let's say you wanted to move from Point A to Point B in the first block, and from Point B to Point C in the second block. If you put both blocks on the same line, the parameters from the second block are ignored, and you will get movement from Point A to Point B only – there will be no movement to Point C.

Any time a parameter is defined more than once on the same line, the last instance of that parameter is used for the execution of the block. Any other instances of that parameter are ignored. For instance, in the block "N100 G00 X13.0 Z3.0 X2.0 X9.5," only the X9.5 would be considered. The other two instances of the X parameter (X13.0 and X2.0) would be ignored.

The maximum length of a block of code is 85 characters per line.

Chapter 2: Address Descriptions

The address defines the meaning of the number that follows the address. An address may have more than one meaning.

If an address has more than one meaning, the multiple meanings are listed here.

ADDRESS	DESCRIPTION
A	angle of tool tip (G76)
B	miscellaneous functions (See *Chapter 7: M Codes and B Codes (Miscellaneous Functions)*)
C	1. C axis absolute coordinate value designation 2. incremental data for tool radius offset (G10) 3. C component of hole position (G83, G84, G85, G87, G88, G89)
D	1. depth of cut (G71, G72) 2. number of repetitions (G73) 3. finishing allowance (G76)
E	retract amount (G71, G72, G74, G75)
F	1. feedrate [NOTE: feedrate is modal – the feedrate specified with this code is effective until a new value is specified; it doesn't need to be specified for each block.] 2. thread lead (G32, G76, G92) 3. drilling feedrate (G83, G87) 4. tapping feedrate (G84, G88) 5. boring feedrate (G85, G89)
G	preparatory function (see Chapter 6: G Codes (Preparatory Functions))
I	1. arc center modifier for the X axis (G02, G03) 2. cutting relief in X (G73) 3. movement amount in X (G74) 4. depth of cut in X (G75) 5. taper height component (G76)
H	incremental dimensioning for the C axis
K	1. arc center modifier for the Z axis (G02, G03) 2. cutting relief in Z (G73) 3. depth of cut in Z (G74) 4. movement amount in Z (G75) 5. thread height (G76)

Table 2-1: Summary of Address Descriptions (1 of 3)

ADDRESS	DESCRIPTION
L	1. number of times to repeat the execution of a custom macro (G65) 2. thread lead (G76, G92) 3. data category (G10)
M	1. miscellaneous functions (See *Chapter 7: M Codes and B Codes (Miscellaneous Functions)*) 2. number of finish-cutting passes (G76)
N	block/sequence number
O	minimum cutting depth (G76)
P	1. subprogram name call (M98) 2. dwell time (G04) 3. axis number (G10, for L108 or L10909 data categories) 4. dwell time to the start of spindle-speed checking (G26) 5. specifies which reference point (P2, P3 or P4) (G30) 6. macro number (G65) 7. sequence number of first block of finished shape (G70, G71, G72, G73) 8. movement amount in X (G74) 9. depth of cut in X (G75) 10. number of finish-cutting passes, chamfering amount, angle of tool tip (G76) 11. thread height (G76) 12. dwell time at the bottom of the hole (G83, G84, G85, G87, G88, G89)
Q	1. velocity feedforward gain (G02, G03) 2. speed-reach variation ratio (G26) 3. sequence number of last block of finished shape (G70, G71, G72, G73) 4. depth of cut in Z (G74) 5. movement amount in Z (G75) 6. minimum cutting depth (G76) 7. first cut amount (G76) 8. depth of each cut (G83, G87) 9. chamfering amount (G92)

Table 2-2: Summary of Address Descriptions (2 of 3)

ADDRESS	DESCRIPTION
R	1. subprogram repetition (M98) 2. arc radius designation (G02, G03) 3. smoothing mode (G10, for L106 or L107 data categories) 4. smoothing time (G10, for L108 data category) 5. position loop gain value (G10, for L10909 data category) 6. speed-alarm variation ratio (G26) 7. retract amount (G71, G72, G74, G75) 8. number of repetitions (G73) 9. finishing allowance (chamfering amount) (G76) 10. taper height component (G76, G90, G92, G94) 11. distance from initial level to point R level (G83, G84, G85, G87, G88, G89) 12. radius of the cylinder (G107)
S	spindle speed functions (See *Chapter 8: S Codes (Spindle Functions)*)
T	tool functions (“T0105” indicates Tool #01, Offset #05) (See *Chapter 9: T Codes (Tool Functions)*)
U	1. incremental dimensioning for the X axis 2. depth of cut (G71) 3. finishing allowance in X (G71, G72, G73) 4. cutting relief in X (G73)
W	1. incremental dimensioning for the Z axis 2. depth of cut (G72) 3. finishing allowance in Z (G71, G72, G73) 4. cutting relief in Z (G73)
X	1. X axis absolute coordinate value designation 2. data value for X axis (G10) 3. X coordinate component (G50, G52, G90) 4. X component of point B (G74, G75) 5. X-axis end point coordinate of thread (G76) 6. X component of hole position (G83, G84, G85) 7. X component from point R to the bottom of the hole (G87, G88, G89)
Z	1. Z axis absolute coordinate value designation 2. data value for Z axis (G10) 3. Z coordinate component (G50, G52, G90) 4. Z component of point C (G74, G75) 5. Z-axis end point coordinate of thread (G76) 6. Z component from point R to the bottom of the hole (G83, G84, G85) 7. Z component of hole position (G87, G88, G89)

Table 2-3: Summary of Address Descriptions (3 of 3)

NOTE: B, F, M, S and T are all independent codes, with no parameters (i.e. “F1500” is a valid, single line of code).

Chapter 3: Format for Part Programs

3.1 General Format for Part Programs

A program is made up of blocks of code. A program typically starts with the program number, followed by blocks of code, and ending with either M02 or M30 (explained in Chapter 7).

A typical program looks like this:

blocks of code

program end code

```
%Program Title
N100 G00 X13.0 Z3.0
.
.
.
N500 G01 X4.0 Z5.0
M02
%
```

Figure 3-1: Typical Format for Part Programs

Either the "%" symbol or a carriage return is required at the end of all part program files.

CAUTION

No lower-case letters are recognized except for a subprogram name. If a lower case letter is accidentally included other than for a subprogram name, it will be ignored.

3.2 Format for Negative Numbers

To enter negative numbers, use a minus sign (e.g. X-5.0).

3.3 Comment Code

Comments can be included in part programs as an aid to the programmer and to anyone who may want to use or edit the code at some time in the future. Code is specified as comment code by placing a "%" or "(" at the beginning of a line. Comment code is not executed.

Comments must be on separate lines from blocks of code. A "%"or "(" placed anywhere on a line except the beginning of the line will be ignored, and the words following the "%"or "(" will be executed, or will cause an error.

Examples of comment code:

```
% This is a comment.

( This is also a comment.
```

3.4 Optional Skip Code

Optional Skip Code, also known as Block Delete Code, refers to code that is skipped (not executed) when the Optional Skip switch is turned on. You can specify a portion of a block as Optional Skip Code by beginning such code with a "/" (forward slash,) or with "\\" (two backward slashes).

3.5 How ServoWorks S-100T Handles Unused Data

Unused data is data that is not recognized by ServoWorks S-100T as conforming to CNC Programming Code (G Code). Such code will be ignored.

Chapter 4: Order of Operations for ServoWorks S-100T Part Programming

The order of operations for CNC programming determines the order in which ServoWorks S-100T processes a block of code.

The following rules determine the order of operations for processing a block of code:

1) When M00, M01, M02 or M30 is with any motion in the same block, then motion will be executed first. After motion is done, then the above M code is processed.

2) When general M code (and/or B code) is with any motion in the same block, then M code (and/or B code) and motion are parallel processed.

3) When S code is with any motion in the same block, then motion will hold until spindle speed reaches the range that is defined in Spindle Parameters, at which time motion begins.

4) When S code is with general M code (and/or B code) in the same block, then S and general M (and/or B) are parallel processed.

5) When T code is with any motion in the same block, then T will be executed first, after which the associated tool offset compensation will become effective. Then motion is executed after.

6) When any of G53, G54, G55, G56, G57, G58 or G59 is with any motion in the same block, then the associated workpiece coordinate system is effective first. Then motion is executed later.

7) When G20/G21, G96/G97, G98/G99 are with any motion in the same block, then the associated modes (types) are effective first. Then motion is executed later.

For simplicity, you can separate motion and (T or S or M or B) in different blocks. The execution order then just follows the block order.

Chapter 5: Subprogram Functions

Description

If there are repeated blocks or sequences, then they can be stored as a subprogram. A subprogram can be called from the main program as needed. M98 is used to call the subprogram and M99 is used to return operation from the subprogram back to main program.

The subprogram is the same as a normal program, except that the end of subprogram is M99. The name of subprogram itself always begins with “O” as the first character.

Required Format

M98 Pp Rr
M99

Possible Parameters That Can Be Used With Subprogram Functions

p – subprogram name (without the first character “O”)
r – number of repetitions (maximum number of repetitions: 999,999)

Example

```
%
(Main Program)
(Subprogram: OTOOTH)

N00130 G53 G99 G97 G20 T0000 M5
N00140 G0 Z.1
N00150 G28 U0. W0. M11
N00160 S1000 M04 T0101
N00170 G0 X.75 Z.1
N00180 G10 L2 P0 X0. Z0.
N00190 G54
N00240 M98 PTOOTH R6
N00250 G53 G1 X1. W.05
N00260 G00 X1.05 M05
N00270 G28 U0. W.6
N00280 M30
%
```

Figure 5-1: Main Program (Example)

```
%
OTOOTH

N00040 G1 X.75 Z 0. F0.01
N00050 Z-.05
N00060 X.9 Z-.1
N00070 Z-.4
N00080 X.75 Z-.45
N00090 Z-.5
N00110 M99

%
```

Figure 5-2: OTOOTH.dat (Subprogram Example)

Variations

The subprograms can also call other subprograms (which can in turn call other subprograms). The nesting depth of subprogram calls is up to eight levels.

Notes

- When R is omitted, then the subprogram is just called once.
- The maximum number of repeated callings is 999,999.
- The main program and its associated subprograms should be stored in same file folder.
- The subprogram call (M98) cannot be with any motion in the same block. If it is, the motion in that block will be ignored.

Chapter 6: G Codes (Preparatory Functions)

6.1 Modes and Modal Commands

G codes are categorized into groups. Most groups are modal groups, as explained in the next paragraph. Only Group 00 is nonmodal.

A ***mode*** is a particular functioning condition. A ***modal group*** is a group of two or more related modes. Only one mode in a modal group can be active at a time. The ***default mode*** is the mode in a modal group that is automatically activated if no other mode from that modal group is programmed in the code.

Modal codes are codes that set a mode. That mode stays in effect until it is cancelled by another code from the same modal group. Many G codes are modal. F codes, which specify feedrate, are also modal.

Nonmodal codes stay in effect only for the blocks that they are programmed. They are one-shot codes. Afterwards, their function is turned off automatically. All G codes in Group 00 are nonmodal.

More than one G code may be programmed in a block, such as "G20 G53 G97 G99" – but they must be from different groups. If more than one G code from the same group is programmed in one block of code, the last G-code value will be executed, and will cancel out the other G codes from that group.

6.2 Summary of G Codes

G-CODE	GROUP	DESCRIPTION
G00	01	rapid positioning (rapid traverse)
G01	01	linear interpolation
G02	01	circular interpolation (clockwise)
G03	01	circular interpolation (counterclockwise)
G04	00	dwell
G09	00	exact stop check mode
G10	00	programmable data input
G20	06	inch data input
G21	06	metric data input
G22	09	barrier check on
G23	09	barrier check off

Table 6-1: Summary of G Codes (1 of 4)

G-CODE	GROUP	DESCRIPTION
G25	08	spindle speed fluctuation detection off
G26	08	spindle speed fluctuation detection on
G28	00	automatic zero return to the reference point
G29	00	automatic return from the reference point
G30	00	automatic return to 2^{nd}, 3^{rd} and 4^{th} reference points
G32	01	thread cutting with a constant lead
G40	07	tool nose radius compensation cancel
G41	07	tool nose radius compensation right
G42	07	tool nose radius compensation left
G50	00	coordinate system preset and maximum spindle RPM
G52	00	local coordinate preset
G53	14	machine coordinate selection
G54	14	workpiece coordinate 1 selection
G55	14	workpiece coordinate 2 selection
G56	14	workpiece coordinate 3 selection
G57	14	workpiece coordinate 4 selection
G58	14	workpiece coordinate 5 selection
G59	14	workpiece coordinate 6 selection
G61	15	exact stop check mode
G64	15	continuous cutting mode

Table 6-2: Summary of G Codes (2 of 4)

G-CODE	GROUP	DESCRIPTION
G65	00	custom macro simple call
G70	00	finishing cycle
G71	00	stock removal in turning
G72	00	stock removal in facing
G73	00	pattern repeat cycle
G74	00	end face peck drilling/grooving
G75	00	outer diameter/inner diameter grooving
G76	00	multiple-pass threading cycle
G80	10	hole machining canned cycle cancel
G83	10	face drilling cycle
G84	10	face tapping cycle
G85	10	face boring cycle
G87	10	side drilling cycle
G88	10	side tapping cycle
G89	10	side boring cycle
G90	01	outer diameter / inner diameter cutting cycle
G92	01	thread cutting cycle
G94	01	end face cutting cycle
G96	02	constant surface speed control
G97	02	constant revolution per minute (constant surface speed control cancel)

Table 6-3: Summary of G Codes (3 of 4)

G-CODE	GROUP	DESCRIPTION
G98	05	per minute feed
G99	05	per revolution feed
G107	00	cylindrical interpolation
G112	21	polar coordinate interpolation mode
G113	21	polar coordinate interpolation cancel
G164	15	continuous cutting mode with block rollover

Table 6-4: Summary of G Codes (4 of 4)

In addition to these preset G codes, you can create customized G codes using custom G code macro calls. See *Section 6.5: Custom Macro Calls Using G Codes, M Codes, S Codes or T Codes* in the *ServoWorks CNC Macro Programming Manual*.

6.3 Detailed Explanations of G Codes

Note that the following explanations assume that the tool is moving and that the part is stationary.

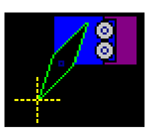
Red	Part Outline
Green	Rapid Traverse (G00)
Yellow	Cutting (G01)
	Tool (showing tool geometry and location of tool nose)

Figure 6-1: Key to Plots

6.3.1 Rapid Positioning (G00)

Description

The G00 command is used for rapid positioning of a tool. This command will move a tool to a specified position in the workpiece system. The tool will move along each axis at its rapid traverse rate (See *Section 7.4: Rapid Traverse Feedrate* in the *ServoWorks S-100T Parameters Manual*) for that axis until it reaches the specified position on that axis. Acceleration and deceleration are based on the "Smoothing Time" parameter (see *Section 5.3: Smoothing Time* in the *ServoWorks S-100T Parameters Manual*).

The rapid positioning movement can be done with or without linear interpolation, as shown in the following figure:

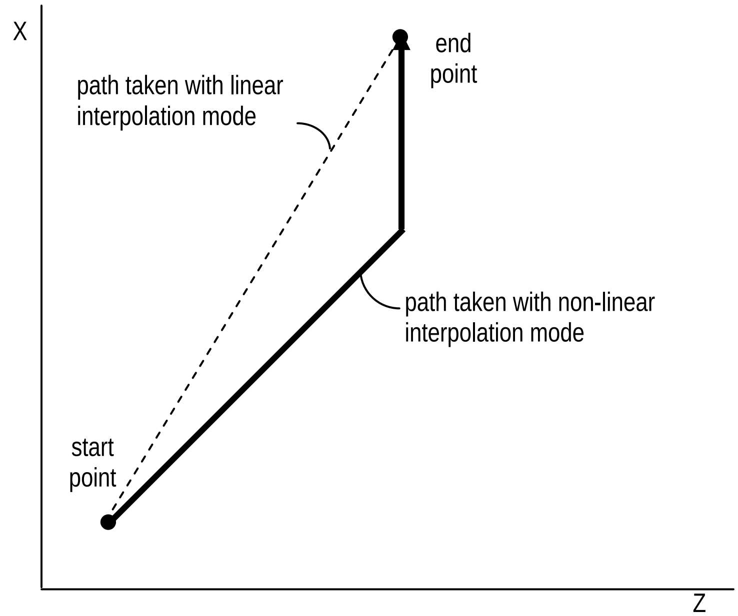


Figure 6-2: Rapid Traverse

Whether rapid positioning is done with or without linear interpolation is determined by the "Rapid Traverse Type" parameter set within the Feedrate Parameters in Configuration Mode of ServoWorks S-100T (see *Section 7.4: Rapid Traverse Feedrate* in the *ServoWorks S-100T Parameters Manual*.)

⚠ CAUTION
Do not use this code for cutting, as the path from point to point is uncertain, and depends upon the rapid traverse rate for the machine tool set by the machine tool builder.

Required Format

G00 X/U Z/W

Parameters Typically Used With G00

X – absolute coordinate value for the X axis (diameter or radius programming)
Z – absolute coordinate value for the Z axis
U – incremental coordinate value (relative to the current tool position) for the X axis (diameter or radius programming)
W – incremental coordinate value (relative to the current tool position) for the Z axis

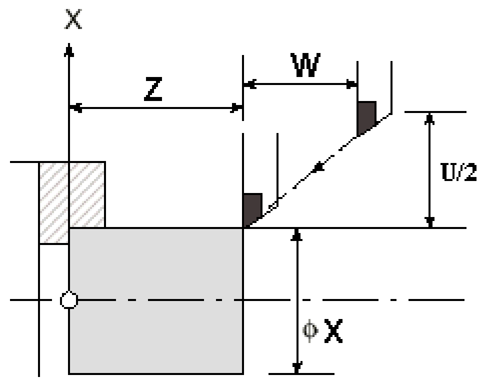


Figure 6-3: G00 Parameters

Example

```
G00 X1.05 Z-1.0
```

Notes

G00 is always executed with the feedrate per minute defined by the Feedrate Parameters in Configuration Mode of ServoWorks S-100T (see *Section 4.9: Setting Parameters for Feedrates* in the *ServoWorks S-100T Operator's Manual*), regardless of whether G98 (per minute feed) or G99 (per revolution feed) is in effect. [See also *Chapter 7: Feedrate Parameters* in the *ServoWorks S-100T Parameters Manual*.]

6.3.2 Linear Interpolation (G01)

Description

The G01 command is used for moving a tool along two or more axes to provide a linear (straight line) path. This command will move a tool to a specified position in the workpiece system. The movement of the multiple axes is at a specified feedrate, which is the feedrate in the direction of movement, not the feedrate of an individual axis. The CNC calculates the linear trajectory, and synchronizes multiple motors to maintain a straight path between two points.

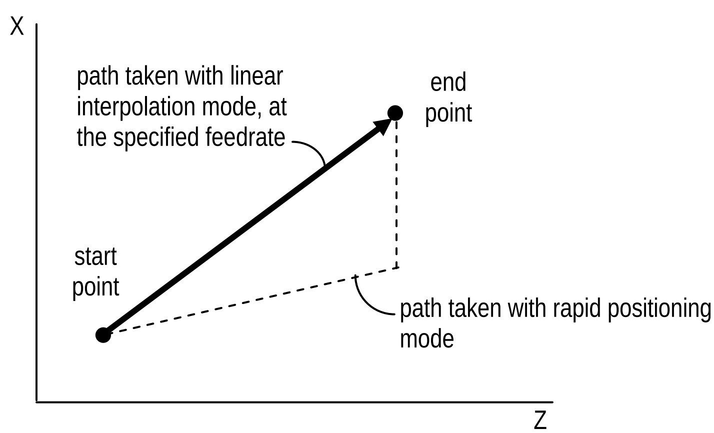


Figure 6-4: Linear Interpolation

Required Format

G01 X/U Z/W F

Parameters Typically Used With G01

X – absolute coordinate value for the X axis (diameter or radius programming)
Z – absolute coordinate value for the Z axis
U – incremental coordinate value (relative to the current tool position) for the X axis (diameter or radius programming)
W – incremental coordinate value (relative to the current tool position) for the Z axis
F – feedrate (of the movement along the linear tool path); [NOTE: the feedrates for the individual axes are calculated so as to produce the specified feedrate along the linear path]

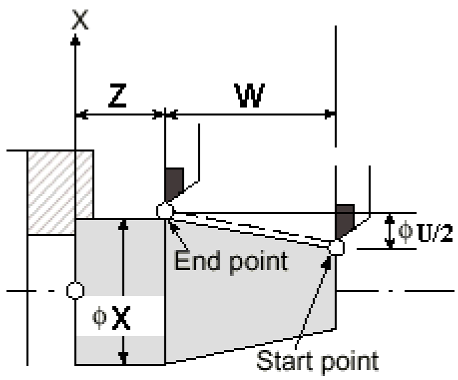


Figure 6-5: G01 Parameters

Example

```
G01 Z-1.0 F.01
```

Notes

G01 is always executed with the F code determined by G98 (per minute feed) or G99 (per revolution feed).

6.3.3 Circular Interpolation (G02, G03)

Description

The G02 and G03 commands are used for moving a tool along two axes to provide a circular path (arc). G02 specifies clockwise circular interpolation, and G03 specifies counterclockwise interpolation. This command will move a tool to a specified position in the workpiece system. The movement of the multiple axes is at a specified feedrate.

You must specify either the center of the arc in incremental values relative to the tool start point (with the correct positive or negative sign), *OR* you must specify the arc radius.

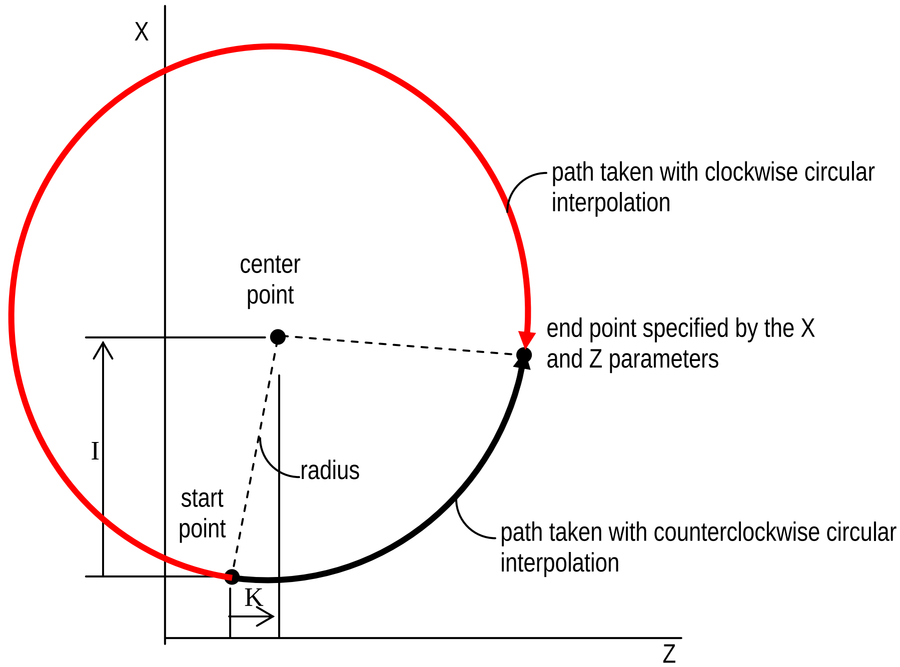


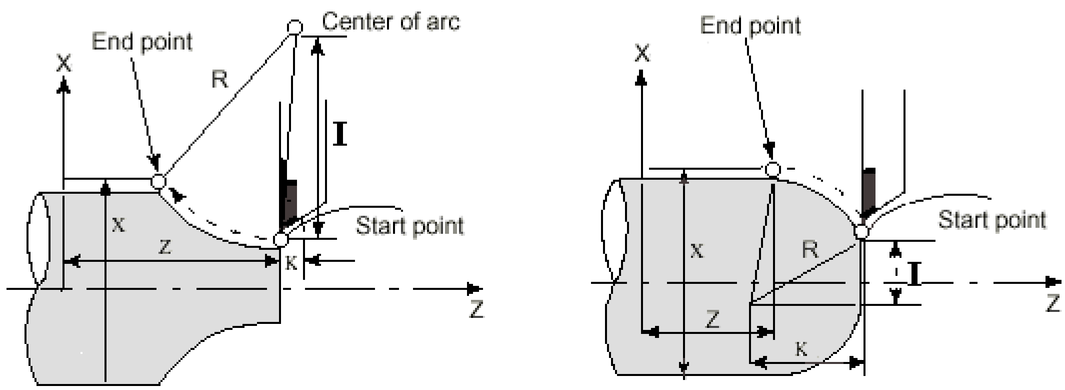
Figure 6-6: Circular Interpolation

Required Format

G02/G03 X/U Z/W I K F Q
G02/G03 X/U Z/W R F Q

Parameters Typically Used With G02 and G03

X – absolute coordinate value (relative to the part origin) for the X axis (diameter or radius programming)
Z – absolute coordinate value (relative to the part origin) for the Z axis
U – incremental coordinate value (relative to the current tool position) for the X axis (diameter or radius programming)
W – incremental coordinate value (relative to the current tool position) for the Z axis
I – arc center modifier for the X axis (radius value)
K – arc center modifier for the Z axis
R – arc radius designation (with sign)
F – feedrate (of the movement along the circular tool path); [NOTE: the feedrates for the individual axes are calculated so as to produce the specified feedrate along the circular path]
Q – velocity feedforward gain (0 ~ 99)



G02 **G03**

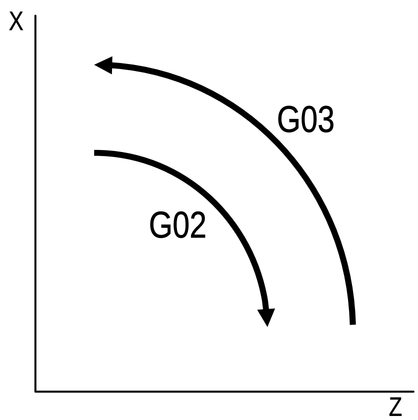
Figure 6-7: G02/G03 Parameters

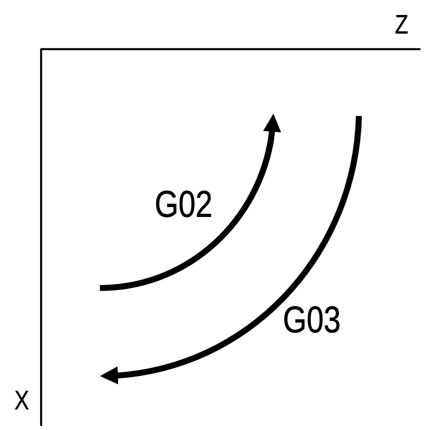
NOTE: If you specify the R parameter (arc radius designation), then you don't need to specify the I and K parameters (arc center modifiers). If you do specify all three parameters (R, I and K), only the R parameter will be effective.

Examples

```
G3 X.625 Z-.3437 R.3437 F.01
G3 U.6874 W-.3437 R.3437 F.01
```

G02 and G03 in the ZX and XZ Planes





ZX Plane **XZ Plane**

Figure 6-8: G02/G03 in the ZX and YZ Planes

Limitations

Circular interpolation can only be performed in a two-axis plane (a Z-X plane).

Variations

To program a full circle, omit coordinates for the end point (or specify an end point which is the same as the start point), and specify the center points of the circle using the I and K arc center modifiers. The end point will then be considered the same as the start point, and a 360° arc (a circle) will be specified.

The arc radius may be specified as either positive or negative. A positive arc radius will produce an arc of less than 180°, while a negative arc radius will produce an arc of greater than 180°, as shown in the following figure:

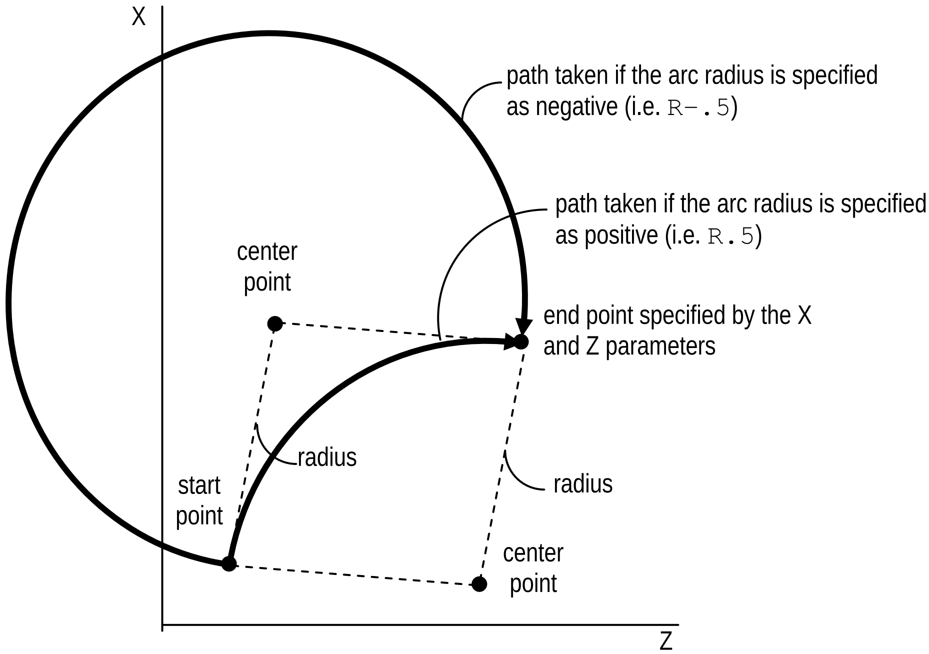


Figure 6-9: Positive and Negative Arc Radii for Circular Interpolation

Notes

- G02 and G03 are always executed with the F code determined by G98 (per minute feed) or G99 (per revolution feed).
- Q is used to reduce the servo lag or the following error such that the finished radius is close to the commanded radius. However, the Q setting is related to the machine's critically damped response.

CAUTION

If an arc is very close to 180°, you must specify that arc by using the I and K parameters to specify the center of the arc. Otherwise, for arcs very near 180°, we cannot guarantee consistent and perfect results because the calculation of the center coordinates may produce an error.

CAUTION

If you specify the center points of the circle using the I and K parameters (arc center modifiers), and the end point specified by the X and Z parameters is not on the arc, an arc movement will still be performed, but I and K will be recalculated, and the recalculated I and K will be used in lieu of the given I and K, in order to use the current start point (the existing position of the axes), and the end point specified by the user.

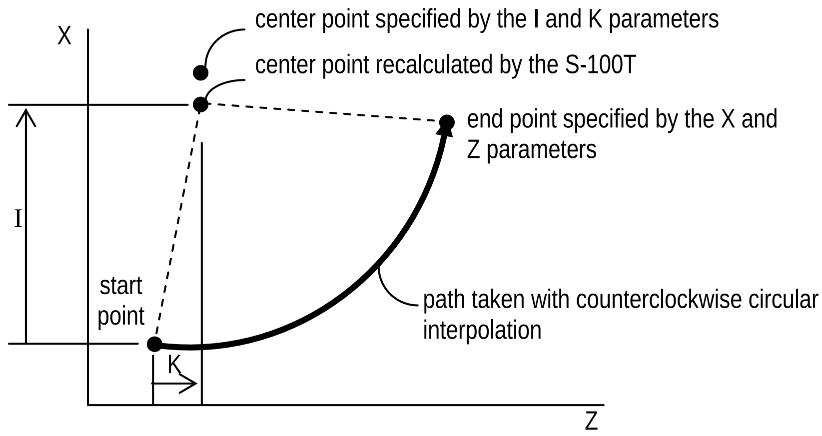


Figure 6-10: Arc Movement Produced by Incorrect Parameters for Circular Interpolation

Similarly, if you specify the arc radius with the R parameter, and the end point specified by the X and Z parameters is not on the arc, an arc movement will still be performed, but the I and K will be calculated and used in lieu of the given R parameter.

6.3.4 Dwell (G04)

Description

This command specifies a delay in program execution. Shifting to the next block is delayed for the number of minutes specified by the parameters.

Required Format

G04 P

Possible Parameters That Can Be Used With G04

P – dwell time (without sign, floating data, units = seconds)

Example

`G04 P5.0` (to dwell five seconds)

6.3.5 Exact Stop Check (G09)

Description

When G09 is programmed in a block, an in-position check (exact stop check) is performed at the end of the block.

Possible Parameters That Can Be Used With G09

None.

Example

```
G1 G9 U.02 Z-1.5 F.01
```

Limitations

Only the current block will perform an in-position check at the end of the movement.

Variations

Whereas the G61 command is modal, the G09 command checks the in-position status only for the block for which it is assigned.

6.3.6 Programmable Data Input (G10)

Required Format

G10 L__ P__ R__

OR

G10 L__ X__ Z__ ……

Possible Parameters That Can Be Used With G10

L – specifies the data category
P – specifies the data index
R – specifies the data value
X – specifies the data value for the X axis
Z – specifies the data value for the Z axis

Data Input Categories

The currently supported data input categories (specified by the L parameter) are listed as follows:

- **L106 – Smoothing (Acc/Dec) Mode Settings For Rapid Motion (G00))**
  - Description: Specifies the smoothing (acceleration/deceleration) mode setting of all axes in rapid motion (G00)
  - Parameters:

    P parameter: Unused

    R parameter: Smoothing (acceleration/deceleration) mode

    Valid Values for R Parameter:

NO_SMOOTHING	0	// No velocity smoothing
SMOOTH_LINEAR	1	// Trapezoidal velocity profile
SMOOTH_BELLSHAPE	2	// Bell-Shaped velocity profile
SMOOTH_EXPONENTIAL	3	// Exponential velocity profile

CAUTION

It is invalid to set smoothing mode before acceleration/deceleration motion has finished. We recommend using G04 exact stop before every G10 L106 command.

  - Reference
    It is strongly recommended that you review *Chapter 5: Smoothing Parameters* in the *ServoWorks S-100T Parameters Manual* for additional information.

- **L107 – Smoothing (Acc/Dec) Mode Settings For Cutting Motion (G01)**
  - Description*:* Specifies the smoothing (acceleration/deceleration) mode setting of all axes in cutting motion (G01)
  - Parameters:

    P parameter: Unused

    R parameter: Smoothing (acceleration/deceleration) mode

    Valid Values for R Parameter:

NO_SMOOTHING	0	// No velocity smoothing
SMOOTH_LINEAR	1	// Trapezoidal velocity profile
SMOOTH_BELLSHAPE	2	// Bell-Shaped velocity profile
SMOOTH_EXPONENTIAL	3	// Exponential velocity profile

CAUTION

It is invalid to set smoothing mode before acceleration/deceleration motion has finished. We recommend using G04 exact stop before every G10 L107 command.

  - Reference
    It is strongly recommended that you review *Chapter 5: Smoothing Parameters* in the *ServoWorks S-100T Parameters Manual* for additional information.

- **L108 – Smoothing (Acc/Dec) Time Settings**
  - Description: Specifies the smoothing (acceleration/deceleration) time setting for each axis
  - Unit: milliseconds
  - Parameters:

    P parameter: Axis number (Range: 1 ~ 16)

    R parameter: Smoothing (acceleration/deceleration) time value.

    Valid Values for R Parameter: vary depending on servo loop update rate, as follows:

1 KHz or lower:	0 ~ 5000 (ms)
2 KHz:	0 ~ 2500 (ms)
4 KHz:	0 ~ 1250 (ms)

  - Examples:

    G04 (Exact stop)

    G10 L108 X50 Y50 (Sets X and Y axes smoothing time to 50 ms)

    G10 L108 P1 R1000 (Sets axis X smoothing time to 1000 ms)

⚠ CAUTION

It is invalid to set smoothing time before acceleration/deceleration motion has finished. We recommend using G04 exact stop before every G10 L108 command.

- **L10909 – Position Loop Gain Settings**
  - Description: Specifies the position loop gain setting of each axis.
  - Unit: Hz
  - Parameters:

    P parameter: Axis number (Range: 1 ~ 16)

    R parameter: Position loop gain value (Range: 0 ~ 999999.9 Hz)

  - Notes: Position loop gain can be set at any time in the program.
  - Examples:

    G10 L10909 X10 Y5 (Sets X axis position loop gain to 10 Hz, and Y axis to 5 Hz)

    G10 L10909 P1 R15 (Sets X axis position loop gain to 15 Hz)

⚠ CAUTION

A high position loop gain value could lead to system vibration and instability, which might cause damage to the machine. Please be very careful in setting values. Refer to *Section 3.2: Overall Position Loop Gain* in the *ServoWorks S-100T Parameters Manual*.

6.3.7 Inch / Metric Data Input (G20, G21)

Description

The G20 and G21 commands are used to specify which measurement system you will use to specify coordinates, distances, radii, etc.

G20: Input distances will be in inches
G21: Input distances will be in millimeters

Possible Parameters That Can Be Used With G20 and G21

None.

Limitations

The G20 and G21 commands are typically used at the beginning of a program. All commands that follow that include parameters specifying distance are affected by the G20 and G21 commands.

Notes

- Put the G20 and G21 commands at the beginning of the program.
- You should stick with one measurement system for an entire program.

Default

Whether the measurement system is in inches or millimeters when neither G20 nor G21 has been commanded depends upon a parameter set by the machine builder. These values can be set using the “program unit” parameter specified in the “General” screen of Configuration Mode of ServoWorks S-100T.

6.3.8 Barrier Check On / Off (G22, G23)

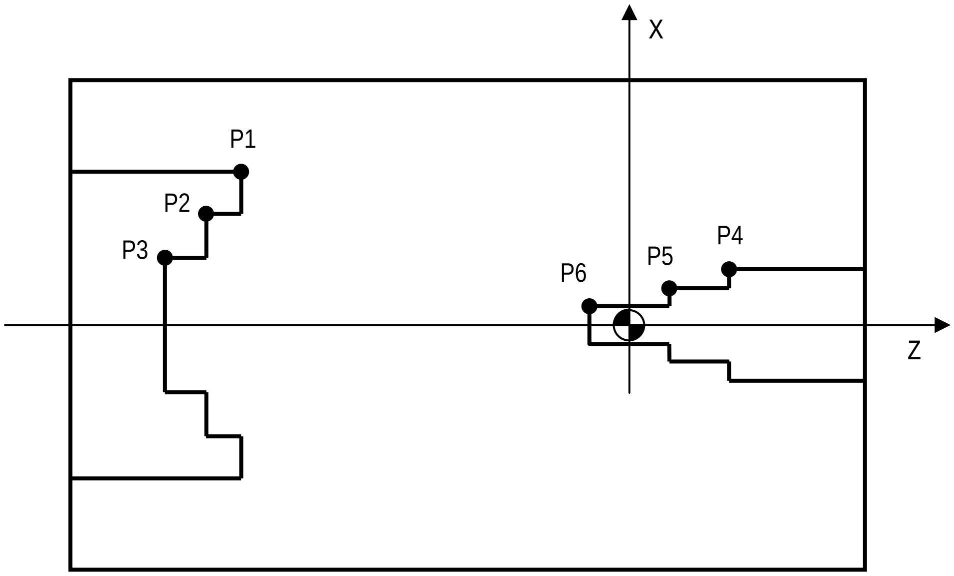
Description

This command sets barrier check on or off. Both chuck barriers and tailstock barriers are valid. By limiting the tool nose movement range, the chuck barrier and tailstock barrier prevent collision with the chuck and the tailstock due to programming errors. If movement is commanded which exceeds the region set in the parameters, the tool will automatically stop at the barrier boundary.

G22: Barrier check on
G23: Barrier check off

Possible Parameters That Can Be Used With G22, G23

None.



P1, P2, P3: Chuck Barrier
P4, P5, P6: Tailstock Barrier

Figure 6-11: Top View of a Turning Machine Showing Chuck Barrier and Tailstock Barrier Points

Default

G23 is the default mode when neither G22 nor G23 has been programmed.

6.3.9 Spindle Speed Fluctuation Detection Off / On (G25, G26)

Description

With these commands, you can enable or disable spindle speed fluctuation detection.

G25: Spindle speed fluctuation detection off
G26: Spindle speed fluctuation detection on

Possible Parameters That Can Be Used With G25, G26

None.

Default

G25 is the default mode when neither G25 nor G26 has been programmed.

6.3.10 Automatic Zero Return To / From Reference Points (G28, G29)

Description

The G28 and G29 commands are used to return to and to return from the reference point. The reference point (home position) is set by the machine builder. [See *Section 8.9: Extra Home Positions* in the *ServoWorks S-100T Parameters Manual*.]

G28: Return to the reference position
G29: Return from the reference position

The G28 command is used to return the tool from the start point (the current tool position) to the reference point, via an intermediate point, the coordinates of which are specified by the parameters.

The G29 command is used to return the tool from the reference point (the current tool position) to a destination point specified by the parameters, via an intermediate point specified by the G28 command.

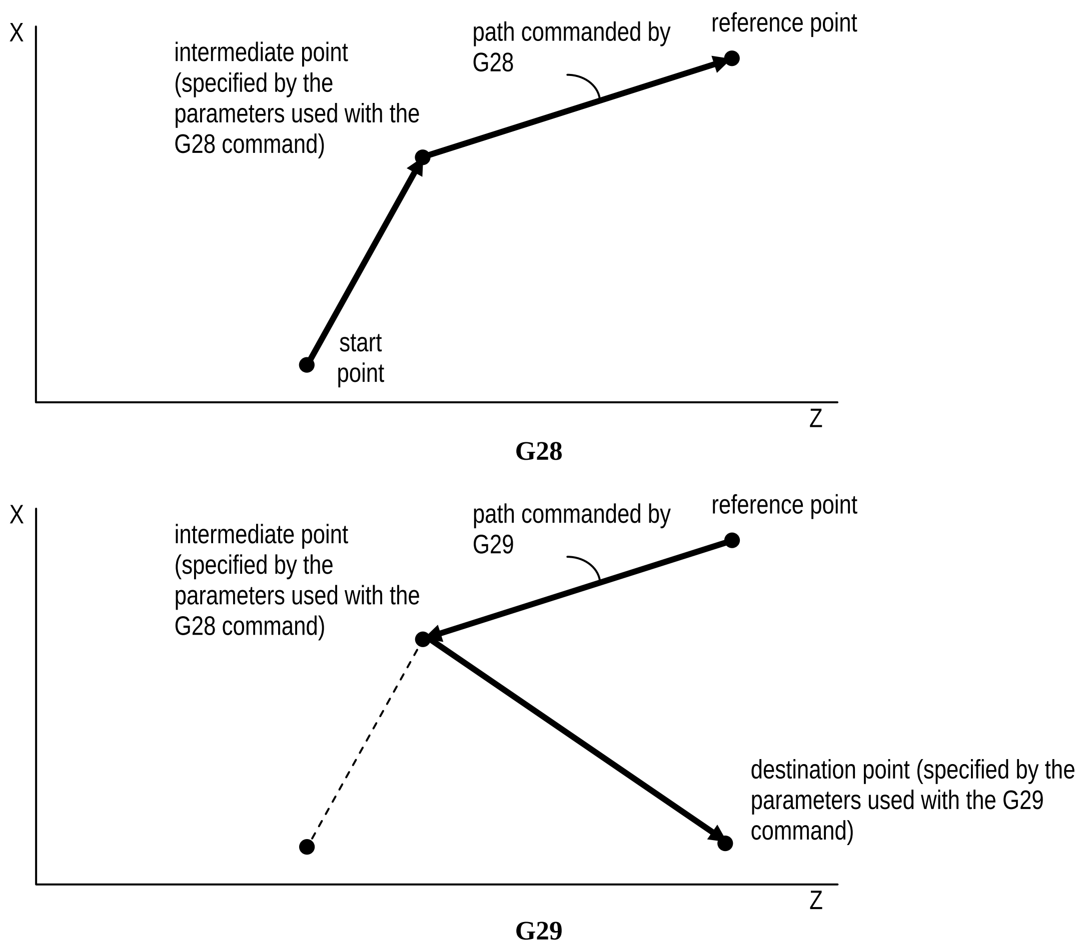


Figure 6-12: Automatic Return To/From Reference Points (G28, G29)

Required Format

G28 X/U Z/W
G29 X/U Z/W

Possible Parameters That Can Be Used With G28 and G29

X – absolute coordinate value for the X axis (diameter or radius programming)
Z – absolute coordinate value for the Z axis
U – incremental coordinate value (relative to the current tool position) for the X axis (diameter or radius programming)
W – incremental coordinate value (relative to the current tool position) for the Z axis

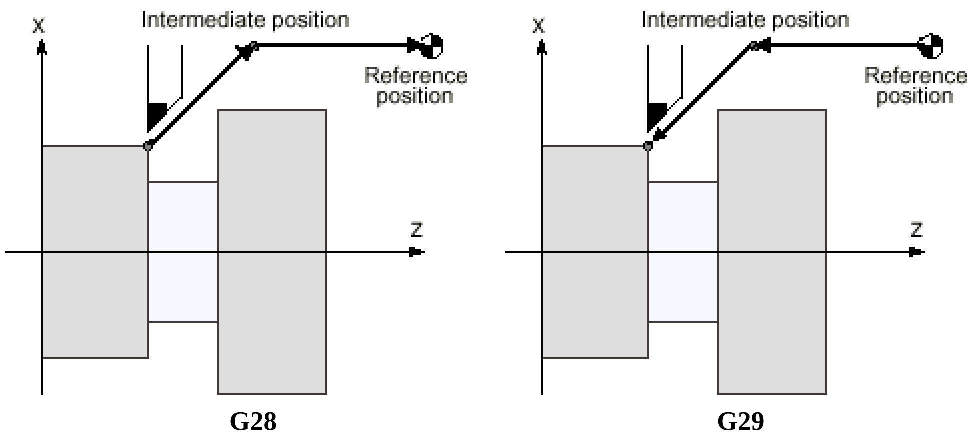


Figure 6-13: G28/G29 Parameters

Example

```
G28 X0.5 Z7.8
G29 X7.5 Z2.1
```

Notes

G28 and G29 are effective only after a home operation has been completed since ServoWorks S-100T was powered up.

6.3.11 Automatic Zero Return To 2nd, 3rd and 4th Reference Points (G30)

Description

The G30 command is used to return the tool from the start point (the current tool position) to an additional specified reference point, via an intermediate point, the coordinates of which are specified by the parameters. The additional reference points are set in ServoWorks S-100T (see *Section 8.9: Extra Home Positions* in the *ServoWorks S-100T Parameters Manual* for additional information).

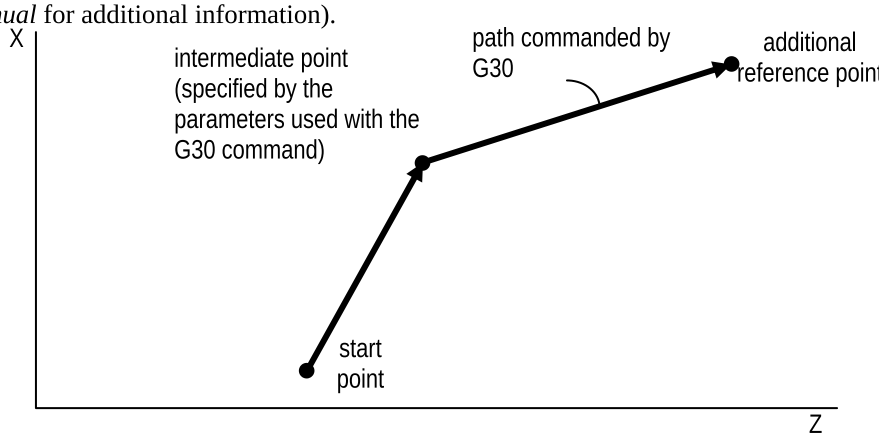


Figure 6-14: Automatic Return to Additional Reference Points (G30)

Required Format

G30 P X/U Z/W

Possible Parameters That Can Be Used With G30

X – absolute coordinate value for the X axis (diameter or radius programming)
Z – absolute coordinate value for the Z axis
U – incremental coordinate value (relative to the current tool position) for the X axis (diameter or radius programming)
W – incremental coordinate value (relative to the current tool position) for the Z axis
P – specifies which reference point (P2, P3, or P4) – if P is not specified, P2 is the default.

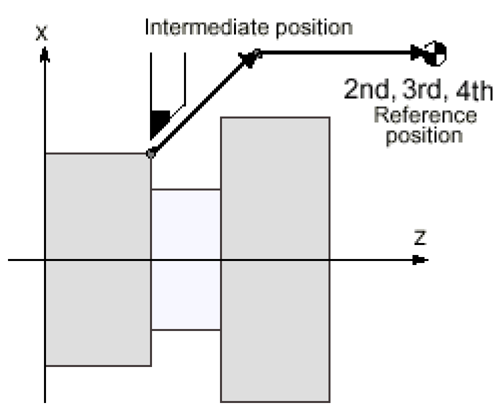


Figure 6-15: G30 Example

Example

```
G30 P3 X0.5 Z7.8
```

6.3.12 Thread Cutting With a Constant Lead (G32)

Description

This command is used to cut equal lead straight threads, threads in tapered screws, and scroll threads.

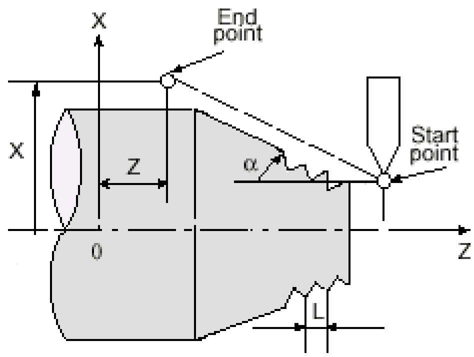


Figure 6-16: G32 Parameters

Required Format

G32 X/U Z/W F

Possible Parameters That Can Be Used With G32

X – absolute coordinate value for the X axis (diameter or radius programming)
Z – absolute coordinate value for the Z axis
U – incremental coordinate value (relative to the current tool position) for the X axis (diameter or radius programming)
W – incremental coordinate value (relative to the current tool position) for the Z axis
F – specifies thread lead (L)

Notes

- Manual Feedrate Override (MFO) is ineffective (fixed at 100 %)
- Spindle Speed Override (SSO) is ineffective (fixed at 100 %)
- When thread cutting is executed in SINGLE BLOCK mode, the tool will stop after execution of the 1st block not specifying thread cutting.
- Use G97 instead of G96 (Constant Surface Speed) for thread cutting.
- In G99 mode, after G32 block has been executed, the modal feedrate is the same as the one defined in the G32 block.
- In G98 mode, while the G32 block is being executed, the feed mode becomes G99 automatically. After the G32 block has been executed, the feed mode returns to G98 and the modal feedrate is the same as the modal feedrate in effect before the G32 block was executed.

6.3.13 Tool Nose Radius Compensations (G40, G41 and G42)

Description

G40: Tool nose radius compensation (TNRC) cancel
G41: Tool nose radius compensation (TNRC) right
G42: Tool nose radius compensation (TNRC) left

TNRC enables the contour of the part to be programmed directly without taking the dimensions of the tool into account.

Possible Parameters That Can Be Used With G40 – G42

None.

Example of G42 and G40

```
%
N00040 G53 G99 G97 G20 T0000 M5
N00050 G0 Z0.1
N00060 G28 U0. W0. M11
N00070 G50 S2000
N00080 S1000 M4 T0101
N00090 X1.05 Z1.
N00100 Z.1
N00110 G1 Z.005 F.01
N00120 G96 S500 M8
N00130 X-.05
N00140 Z.0738
N00150 G0 X1. Z.1

N00180 G42 G0 X-.0624
N00190 G1 Z0
N00200 G3 X.625 Z-.3437 R.3437
N00210 G1 Z-.8125
N00220 X.7226
N00230 G3 X.7667 Z-.8216 R.0312
N00240 G1 X.8567 Z-.8666
N00250 G3 X.875 Z-.8887 R.0312
N00260 G1 Z-1.5437
N00270 G40 G0 X1.

N00290 Z1. M9
N00300 T0100 M5
N00310 G28 U0. W0.
N00320 M30
%
```

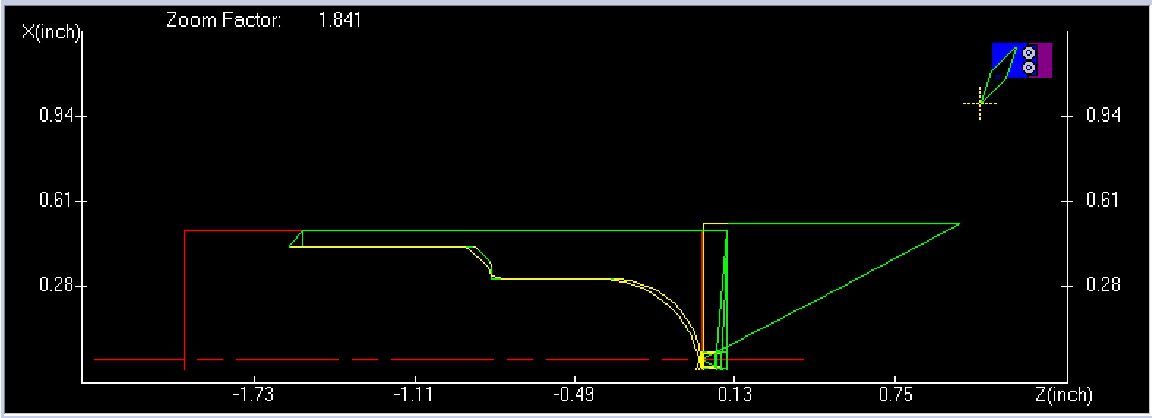


Figure 6-17: Plot of G40/G42 Example

Limitations

- Look-ahead buffer: 4 blocks
- Only motion (G00, G01, G02, G03) and non-motion (G04, M, S, B, and F codes) are eligible in TNRC mode.
- The 1st and the last motions of TRNC mode cannot be G02 or G03.
- G41 and G42 cannot be active at the same time.

Notes

- Tool location code (imaginary tool nose location) is combined into TRNC calculation.
- Tool movement around inside corner is the intersection.
- Tool movement around outside corner is the circular transition.
- The compensation of tool nose center is parallel to programmed path with radius offset.
- The tool-path shown on PLOT is the compensated path of imaginary tool nose.

Default

G40 is the default mode when none of the G40, G41 or G42 codes has been programmed.

6.3.14 Coordinate System Preset and Maximum Spindle RPM (G50)

Description

A workpiece coordinate system (selected with a code from G54 to G59) is shifted to set a new workpiece coordinate system so that the current tool position matches the specified coordinates given in the G50 block of code.

Required Format

G50 Xx Zz Ss

Possible Parameters That Can Be Used With G50

x – X coordinate component (abs value, diameter or radius programming)
z – Z coordinate component (abs value)
s – maximum spindle RPM for constant surface speed mode (G96)

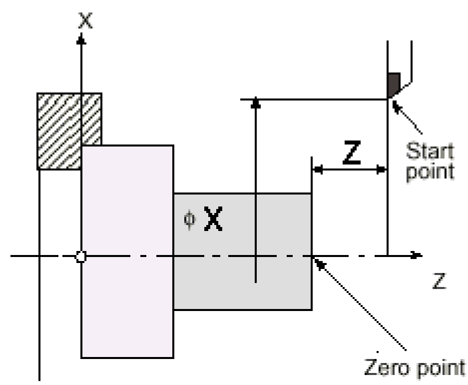


Figure 6-18: G50 Parameters

Limitations

G50 will only shift the current workpiece coordinate system (not all coordinate systems).

Variations

- G10 L2 P0 could be used to change the External Workpiece Zero Point Offset Value (to shift all coordinate systems).
- G10 L2 P1 ~ P6 could be used to change the corresponding Workpiece Zero Point Offset Value.

6.3.15 Local Coordinate System Preset (G52)

Description

This command establishes a local coordinate system for the current workpiece coordinate system, *within* that workpiece coordinate system. The coordinates set by the parameters become the new zero position for the current workpiece coordinate system.

Required Format

G52 Xx Zz

Possible Parameters That Can Be Used With G52

x – X Coordinate Component (incremental value, diameter or radius programming)
z – Z Coordinate Component (incremental value)

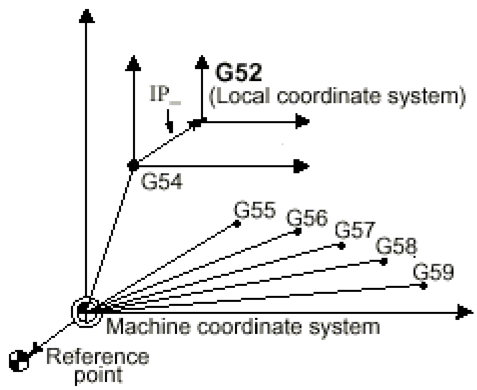


Figure 6-19: G52 Explanation

Example

```
G52 X5.6 Z8.6
```

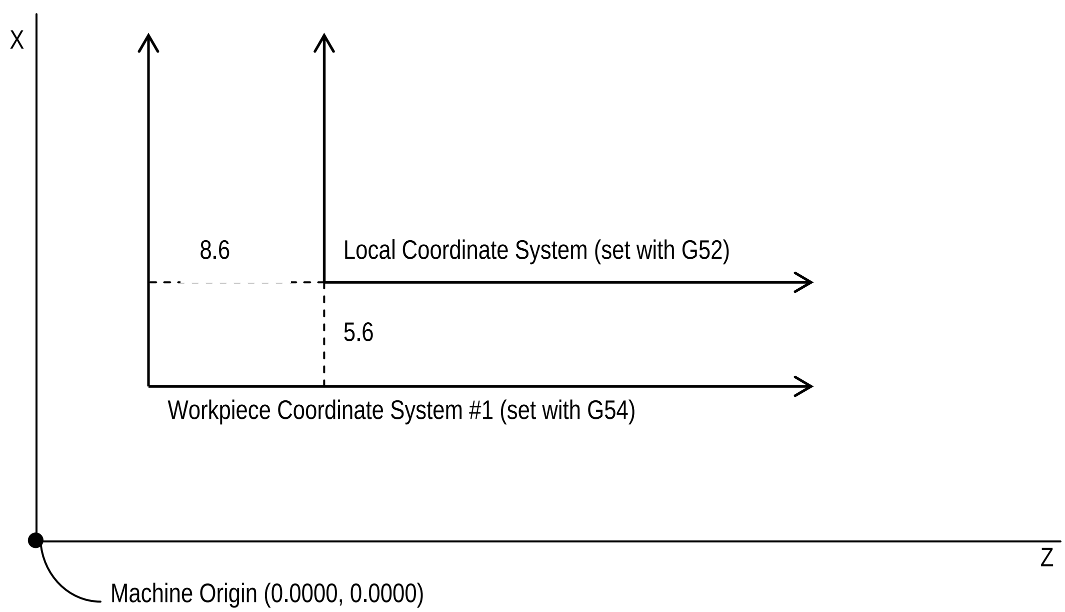


Figure 6-20: G52 Example

Limitations

- G52 will only shift the current workpiece coordinate system (not all coordinate systems).
- You can set different local coordinate systems for each workpiece coordinate system.

Variations

- G10 L2 P0 could be used to change the External Workpiece Zero Point Offset Value (to shift all coordinate systems).
- G10 L2 P1 ~ P6 could be used to change the corresponding Workpiece Zero Point Offset Value.

Notes

To cancel the local coordinate system, and return to the workpiece coordinate system, you must use the G52 command with each parameter set to zero. In other words, if you set up your local coordinate system using "`G52 X5.6 Z8.6`," you must use the block "`G52 X0 Z0`" to cancel the local coordinate system.

CAUTION
This command only works with absolute coordinates. You must not try to specify incremental coordinates, or this G52 code will be ignored.

6.3.16 Machine Coordinate Selection (Modal) (G53)

Description

When this command is executed:

- Any active tool offset will be cancelled.
- Any active workpiece coordinate system (G54 ~ G59) or/and coordinate shift (G50 or G52) will be cancelled.
- Program positions will be identical to machine positions.

Possible Parameters That Can Be Used With G53

None.

Example #1 of G53

```
%
N00130 G53 G99 G97 G20 T0000 M5
N00140 G0 Z.1
N00150 G28 U0. W0. M11
N00160 S1000 M04 T0101
N00170 G0 X.75 Z.1
N00180 G10 L2 P0 X0. Z0.
N00190 G54
N00240 M98 PTOOTH
N00250 G53 G1 X1. W.05
N00260 G00 X1.05 M05
N00270 G28 U0. W.6
N00280 M30
%
```

Example #2 of G53

```
N00100 G53 G00 X1.0 Z-.05
```

is equivalent to:

```
N00100 G53
N00110 G00 X1.0 Z-.05
```

6.3.17 Workpiece Coordinate Selection (Modal) (G54-G59)

Description

These codes are used to select different coordinate systems that correspond to different workpieces. Up to six coordinate systems can be selected using G54-G59:

G54	Workpiece Coordinate System #1
G55	Workpiece Coordinate System #2
G56	Workpiece Coordinate System #3
G57	Workpiece Coordinate System #4
G58	Workpiece Coordinate System #5
G59	Workpiece Coordinate System #6

By setting the coordinate system in such a way that the zero point is set to a fixed piece on the workpiece, you make programming that workpiece very simple. You can use the same blocks of G code to cut six different workpieces, simply by moving the reference point.

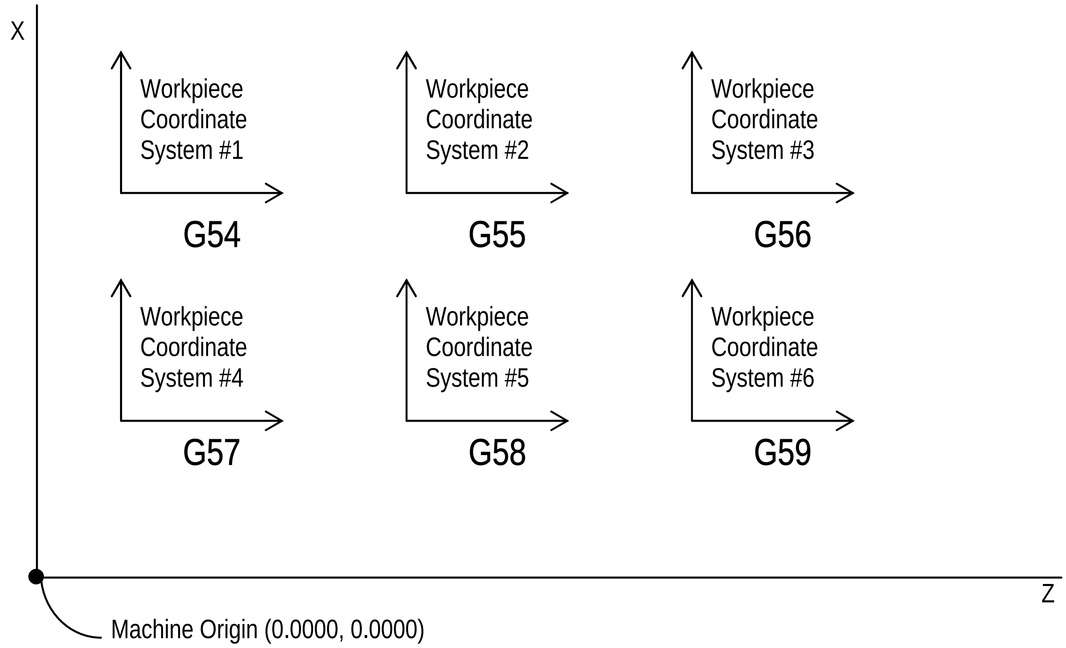


Figure 6-21: G54-G59 Explanation

Possible Parameters That Can Be Used With G54 – G59

None.

Example #1 of G54 – G56

```
%
N00130 G53 G99 G97 G20 T0000 M5
N00140 G0 Z.1
N00150 G28 U0. W0. M11
N00160 S1000 M04 T0101
N00170 G0 X.75 Z.1
N00180 G10 L2 P0 X0. Z0.
N00190 G54
N00200 M98 PTOOTH
N00210 G55
N00220 M98 PTOOTH
N00230 G56
N00240 M98 PTOOTH
N00250 G53
N00260 G00 X1.05 M05
N00270 G28 U0. W.6
N00280 M30
%

%
OTOOTH

N00040 G1 X.75 Z 0. F0.01
N00050 Z-.05
N00060 X.9 Z-.1
N00070 Z-.4
N00080 X.75 Z-.45
N00090 Z-.5
N00110 M99
%
```

Example #2 of G54 – G56

```
N00100 G54 G00 X1.0 Z-.05
```

is equivalent to:

```
N00100 G54
N00110 G00 X1.0 Z-.05
```

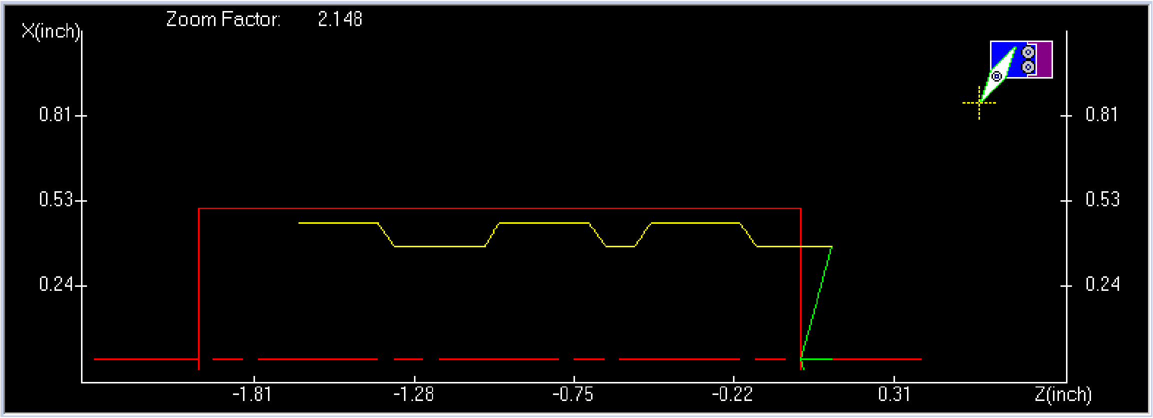


Figure 6-22: Plot of Example for G54 – G56

Limitations

- Both G54 ~ G59 and Subprogram Call (M98) should not be in the same block.
- G50 or G52 will only shift the current workpiece coordinate system (not all coordinate systems).

Variations

- G10 L2 P0 could be used to change the external workpiece zero point offset value (to shift all coordinate systems).
- G10 L2 P1 ~ P6 could be used to change the corresponding workpiece zero point offset value.

Notes

- The G54 – G59 codes are used to *select* (not *set up*) different coordinate systems. These six coordinate systems need to be set up in advance of using these codes to select the coordinate systems.

CAUTION

You should check the workpiece coordinate system setup on your machine tool prior to running a part program that uses the G54 – G59 codes.

CAUTION

If you should call a G54, G55, G56, G57, G58 or G59 code with any parameters, they will be ignored.

CAUTION

G54 – G59 blocks of code should appear on separate lines, with no parameters.

Reference

You can set up the workpiece coordinate systems using Offset Mode in ServoWorks S-100T. See *Section 14.8: Setting the External Zero Offset and Workpiece Zero Point Offsets* in the *ServoWorks S-100T Operator's Manual*.

6.3.18 Exact Stop Check Mode, Continuous Cutting Mode and Continuous Cutting Mode with Block Rollover (G61, G64, G164)

Description

In Exact Stop Check Mode (G61), all move commands (such as G00, G01, G02 and G03) are executed with the tool decelerating at the end point, and with an in-position check after each move command. The next block is not executed until the in-position check is performed.

Continuous Cutting Mode (G64) is used mainly to cancel G61, Exact Stop Check Mode. In Continuous Cutting Mode, no in-position check is made at the end of each block (except G00 blocks). Instead, program execution continues automatically with the next block.

In Continuous Cutting Mode with Block Rollover, no tool deceleration occurs and no in-position check is made at the end of each block. Instead, program execution continues automatically with the next block (similar to G64 Continuous Cutting Mode).

Continuous Cutting Mode with Block Rollover uses “block rollover” technology to provide more accurate feedrate control and smoother motion, especially for high-speed, small segment contour control.

G61: Exact Stop Check Mode
G64: Continuous Cutting Mode
G164: Continuous Cutting Mode with Block Rollover

Detailed Description of the Block Rollover Function (G164)

When a single block of code cannot be completed in one servo loop/cycle time (as determined by the interpolation rate of the servo interface system), the remainder of that block of code is completed in subsequent servo loop(s). When the last servo loop for a single block of code only needs to move a small portion of the distance moved in the previous servo loop(s), but in the same cycle time, the velocity is adjusted to slow axis movement for this lesser distance in the same time frame. The block rollover function prevents this change in velocity by moving the remainder of one block to the next block and recalculating the interpolation for the next block.

For example, suppose you have a part program consisting of a series of 0.25 mm segments, and you run the machine at 6000 mm/min. 6000 mm/min is the equivalent of 0.1 mm/ms. Let’s assume the servo interface platform has an interpolation rate of 1 ms, so the ServoWorks CNC Engine interpolates the trajectory every 1 ms, meaning that our CNC moves the axes 0.1 mm every interpolation cycle. If the CNC happens to execute a block of 0.25 mm, it takes three interpolation cycles (= 3 ms) to execute the block. But, as you can see, the motion behaves like this: in the first millisecond, it moves 0.1 mm; in the second millisecond, it also moves 0.1 mm; and in the third millisecond (the third cycle) it moves 0.05 mm. As a result, the velocity is not very smooth, but the trajectory can be kept accurate.

If the smoothing time (T_s) is big, then the irregular velocities (0.1 mm/ms, 0.1 mm/ms and 0.05 mm/ms) will be smoothed out, but at the cost of losing the trajectory accuracy. But, if you reduce T_s for more accuracy, then the ServoWorks CNC Engine strictly follows this unsmooth velocity (0.1 mm/ms, 0.1 mm/ms and 0.05 mm/ms), causing an unsmooth, "jerky" motion. Obviously, if the velocity commands are so unsmooth, the servos cannot catch up with these frequent changes of the velocity commands, resulting in huge servo errors, causing a huge trajectory error, which is obviously undesirable. For milling, this would create bad traces on the milling surface.

In the above case, with block rollover, the ServoWorks CNC Engine moves 0.1 mm only for two cycles, and adds the remainder of 0.05 mm to the next block (let's assume it is 0.25 mm), making a 0.3 mm segment. In this way, the ServoWorks CNC Engine can follow the trajectory more accurately while making the velocity very smooth.

Possible Parameters That Can Be Used With G61, G64, G164

None.

Example

```
G61 G1 X1.0 Z-1.25 F.01
```

Variations

Whereas the G61 command is modal, the G09 command checks the in position status only for the block for which it is assigned.

Notes

Use Exact Stop Check Mode when sharp edges are required for the corners of a workpiece. Use Continuous Cutting Mode or Continuous Cutting Mode with Block Rollover at all other times.

Default

G64 is the default mode when none of the G61, G64 or G164 codes has been programmed.

6.3.19 Simple Macro Call (G65)

Description

Using customized macros is very similar to calling a subprogram from a main motion program, except with the added advantage that you can use variables in macros. This means that macros are more reusable than subprograms. For instance, you can have one macro for a bolt hole, and just change the variables (arguments) you assign when you call the macro (with customized macro instructions). If you were using a subprogram for programming a bolt hole, you would need a different subprogram for each size bolt hole.

The format of a macro is the same format as a subprogram:

```
%
O Macro number;
 .
 .
 .    body of macro
 .
 .
M99;
%
```

A simple macro call is a one-shot command in which a macro is called once using G65 (although it may be executed more than once, depending upon the L parameter, the number of times the macro is to be repeated).

Required Format

G65 P L <argument assignment>

Possible Parameters That Can Be Used With G65

P – macro number
L – number of times to repeat the execution of the macro (1 by default)

In addition to these two parameters, macros are called with argument assignments, which are not considered to be parameters.

Example

G65 P100 Z20.0 R2.5 F500

Limitations

- G65 must be specified before any argument.
- The macro program (the subroutine) must be in the same file as the part program file that calls the macro program.

Reference

See the *ServoWorks CNC Macro Programming Manual* for additional information on writing and using macros. This section deals only with calling and canceling macros.

6.3.20 Finishing Cycle (G70)

Required Format

G70 Pp Qq

Possible Parameters That Can Be Used With G70

p – sequence number of 1st block of finished shape
q – sequence number of last block of finished shape

Example of G70

```
%

N00040 G53 G99 G97 G20 T0000 M5
N00050 G0 Z0.1
N00060 G28 U0. W0. M11
N00070 G50 S2000
N00080 S1000 M4 T0101
N00090 X1.05 Z1.
N00100 Z.1
N00110 G1 Z.005 F.01
N00120 G96 S500 M8
N00130 X-.05
N00140 Z.0738
N00150 G0 X1. Z.1
N00160 G71 U.05 R.1
N00170 G71 P00180 Q00270 U.02 W.005 F.01
N00180 G0 X-.0624
N00190 G1 Z0
N00200 G3 X.625 Z-.3437 R.3437
N00210 G1 Z-.8125
N00220 X.7226
N00230 G3 X.7667 Z-.8216 R.0312
N00240 G1 X.8567 Z-.8666
N00250 G3 X.875 Z-.8887 R.0312
N00260 G1 Z-1.5437
N00270 G0 X1.
N00280 G70 P00180 Q00270
N00290 Z1. M9
N00300 T0100
N00310 G28 U0. W0. M5
N00320 M30

%
```

Limitations

- Any T code between p and q will be ignored.
- Every M, S, or B code between p and q are valid.

Notes

- q should be equal to or larger than p.
- All block numbers between p and q should be both larger than p and less than q.

6.3.21 Stock Removal in Turning (G71)

Required Format

G71 Ud Re
G71 Pp Qq Uu Ww Ss Ff Mm Bb

Possible Parameters That Can Be Used With G71

d – depth of cut (radius value w/o sign)
e – retract amount (w/o sign)
p – sequence number of 1st block of finished shape
q – sequence number of last block of finished shape
u – finishing allowance in X (with sign, diameter or radius programming)
w – finishing allowance in Z (with sign)
s – spindle speed
f – cutting feedrate
m – M code
b – B code

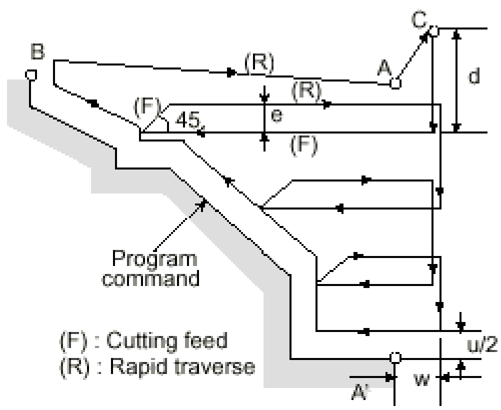


Figure 6-23: G71 Parameters

Example of G71

```
%

N00040 G53 G99 G97 G20 T0000 M5
N00050 G0 Z0.1
N00060 G28 U0. W0. M11
N00070 G50 S2000
N00080 S1000 M4 T0101
N00090 X1.05 Z1.
N00100 Z.1
N00110 G1 Z.005 F.01
N00120 G96 S500 M8
N00130 X-.05
N00140 Z.0738
N00150 G0 X1. Z.1
N00160 G71 U.05 R.1
N00170 G71 P00180 Q00270 U.02 W.005 F.01
N00180 G0 X-.0624
N00190 G1 Z0
```

```
N00200 G3 X.625 Z-.3437 R.3437
N00210 G1 Z-.8125
N00220 X.7226
N00230 G3 X.7667 Z-.8216 R.0312
N00240 G1 X.8567 Z-.8666
N00250 G3 X.875 Z-.8887 R.0312
N00260 G1 Z-1.5437
N00270 G0 X1.
N00280 G70 P00180 Q00270
N00290 Z1. M9
N00300 T0100
N00310 G28 U0. W0. M5
N00320 M30

%
```

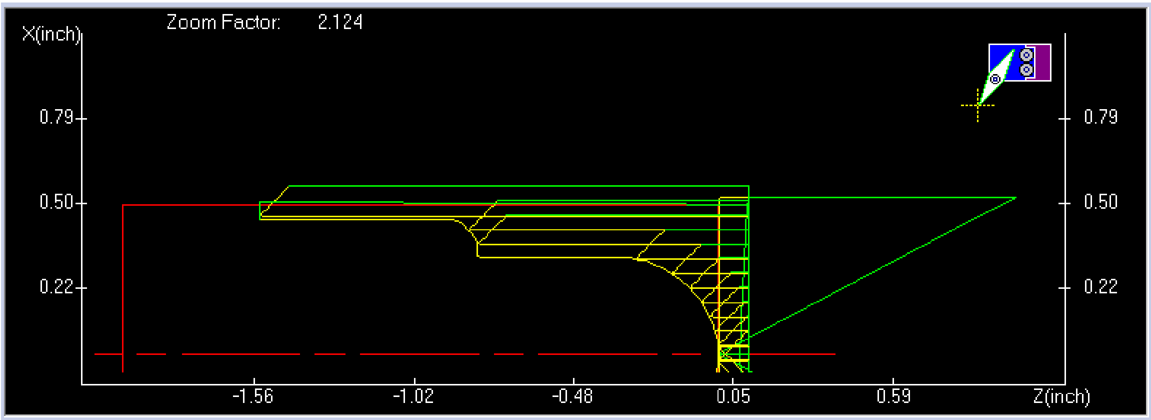


Figure 6-24: Plot of Example for G71

Limitations

- Any T code in a G71 block will be ignored.
- Any M, S, T, or B codes between p and q will be ignored.

Notes

- The finished shape changes monotonically in both X and Z directions.
- q should be equal to or larger than p.
- All block numbers between p and q should be both larger than p and less than q.

6.3.22 Stock Removal in Facing (G72)

Required Format

G72 Wd Re
G72 Pp Qq Uu Ww Ss Ff Mm Bb

Possible Parameters That Can Be Used With G72

d – depth of cut (radius value w/o sign)
e – retract amount (w/o sign)
p – sequence number of 1st block of finished shape
q – sequence number of last block of finished shape
u – finishing allowance in X (with sign, diameter or radius programming)
w – finishing allowance in Z (with sign)
s – spindle speed
f – cutting feedrate
m – M code
b – B code

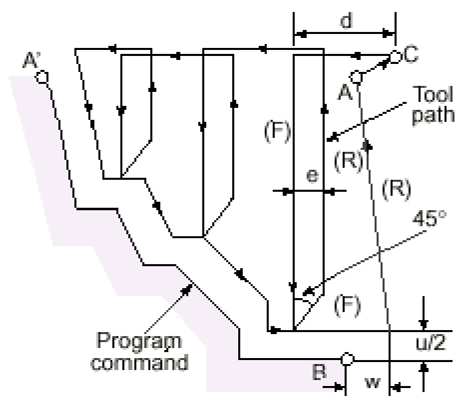


Figure 6-25: G72 Parameters

Example of G72

```
%

N00040 G53 G99 G97 G20 T0000 M5
N00050 G0 Z0.1
N00060 G28 U0. W0. M11
N00070 G50 S2000
N00080 S1000 M4 T0101
N00090 X1.05 Z1.
N00100 Z.1
N00110 G1 Z.005 F.01
N00120 G96 S500 M8
N00130 X-.05
N00140 Z.0738
N00150 G0 X1. Z.1
N00160 G72 W.08 R.05
N00170 G72 P00180 Q00280 U.02 W.005 F.01
N00180 G0 Z-1.5437
```

```
N00190 G1 X.875
N00200 Z-.8887
N00210 G2 X.8567 Z-.8666 R.0312
N00220 G1 X.7667 Z-.8216
N00230 G2 X.7226 Z-.8125 R.0312
N00240 G1 X.6250
N00250 Z-.3437
N00260 G2 X-.0624 Z0 R.3437
N00270 G1 Z0.1
N00280 G0 X1.
N00290 G70 P00180 Q00270
N00300 Z1. M9
N00310 T0100 M5
N00320 G28 U0. W0.
N00320 M30

%
```

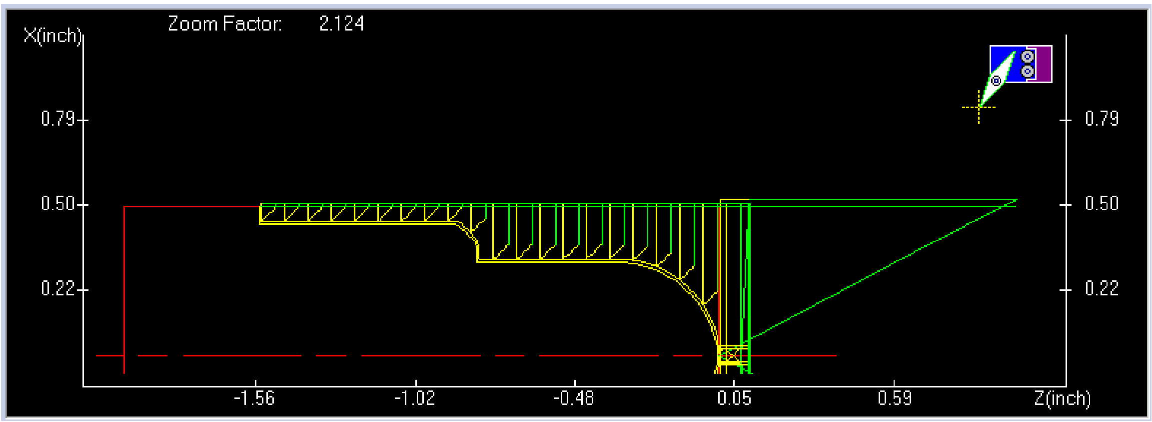


Figure 6-26: Plot of Example for G72

Limitations

- Any T code in a G72 block will be ignored.
- Any M, S, T, or B codes between p and q will be ignored.

Notes

- The finished shape changes monotonically in both the X and Z directions.
- q should be equal to or larger than p.
- All block numbers between p and q should be both larger than p and less than q.

6.3.23 Pattern Repeat Cycle (G73)

Required Format

G73 Ui Wk Rd
G73 Pp Qq Uu Ww Ss Ff Mm Bb

Possible Parameters That Can Be Used With G73

i – cutting relief in X (with sign, radius value)
k – cutting relief in Z (with sign)
d – number of repetitions
p – sequence number of 1st block of finished shape
q – sequence number of last block of finished shape
u – finishing allowance in X (with sign, diameter or radius programming)
w – finishing allowance in Z (with sign)
s – spindle speed
f – cutting feedrate
m – M code
b – B code

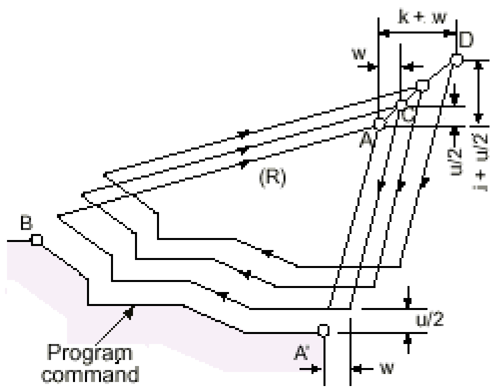


Figure 6-27: G73 Parameters

Example of G73

```
%

N00040 G53 G99 G97 G20 T0000 M5
N00050 G0 Z0.1
N00060 G28 U0. W0. M11
N00070 G50 S2000
N00080 S1000 M4 T0101
N00090 X1.05 Z1.
N00100 Z.1
N00110 G1 Z.005 F.01
N00120 G96 S500 M8
N00130 X-.05
N00140 Z.0738
N00150 G0 X1. Z.1
N00160 G73 U.06 W.03 R3
```

```
N00170 G73 P00180 Q00270 U.02 W.005 F.01
N00180 G0 X-.0624
N00190 G1 Z0
N00200 G3 X.625 Z-.3437 R.3437
N00210 G1 Z-.8125
N00220 X.7226
N00230 G3 X.7667 Z-.8216 R.0312
N00240 G1 X.8567 Z-.8666
N00250 G3 X.875 Z-.8887 R.0312
N00260 G1 Z-1.5437
N00270 G0 X1.
N00280 G70 P00180 Q00270
N00290 Z1. M9
N00300 T0100 M5
N00310 G28 U0. W0.
N00320 M30
%
```

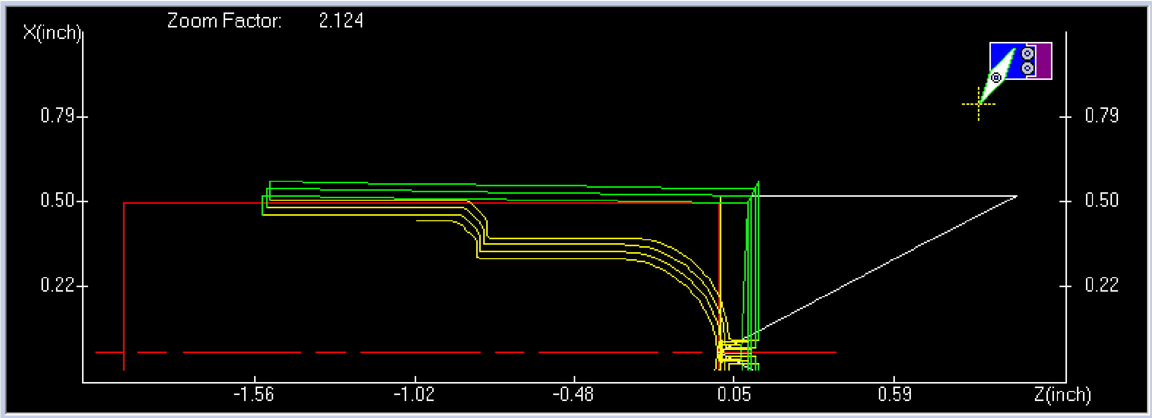


Figure 6-28: Plot of Example for G73

Limitations

- Any T code in a G73 block will be ignored.
- Any M, S, T, or B codes between p and q will be ignored.

Notes

- q should be equal to or larger than p.
- All block numbers between p and q should be both larger than p and less than q.

6.3.24 End Face Peck Drilling/Grooving (G74)

Required Format

G74 Re
G74 X/Ux Z/Wz Pi Qk Ss Ff Mm Bb

Possible Parameters That Can Be Used With G74

e – retract amount (w/o sign)
x – X component of point B (abs/inc value, diameter or radius programming)
z – Z component of point C (abs/inc value)
i – movement amount in X (without sign, radius value)
k – depth of cut in Z (without sign)
s – spindle speed
f – cutting feedrate
m – M code
b – B code

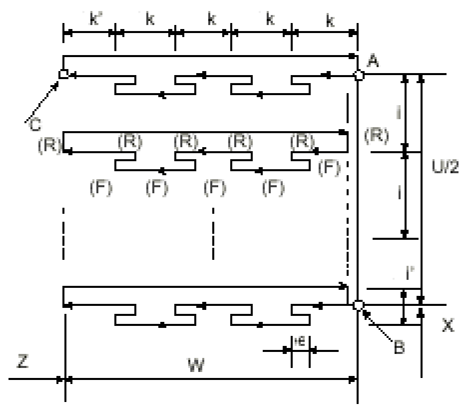


Figure 6-29: G74 Parameters

Example of G74

```
%

N00040 G53 G99 G97 G20 T0000 M5
N00050 G0 Z0.1 M11
N00060 G28 U0. W0.
N00070 S2000 M3 T0202
N00080 X1.05 Z1.
N00090 X0 M8
N00100 Z0.1
N00110 G74 R0.05
N00120 G74 Z-1. Q0.1 F0.002
N00130 T0200 M9
N00140 G28 U0. W0. M5
N00150 M30

%
```

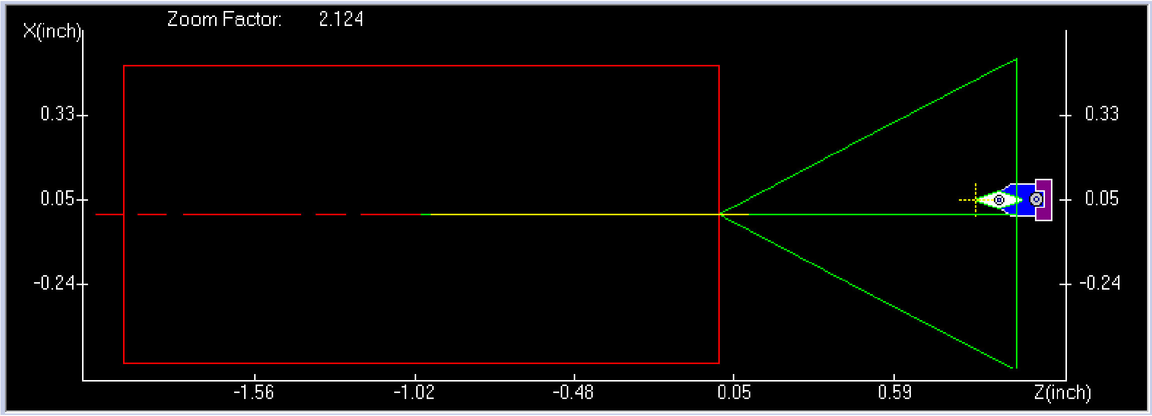


Figure 6-30: Plot of Example for G74

Limitations

Any T code in a G74 block will be ignored.

6.3.25 Outer Diameter / Inner Diameter Grooving (G75)

Required Format

G75 Re
G75 X/Ux Z/Wz Pi Qk Ss Ff Mm Bb

Possible Parameters That Can Be Used With G75

e – retract amount (w/o sign)
x – X component of point B (abs/inc value, diameter or radius programming)
z – Z component of point C (abs/inc value)
i – depth of cut in X (w/o sign, radius value
k – movement amount in Z (w/o sign)
s – spindle speed
f – cutting feedrate
m – M code
b – B code

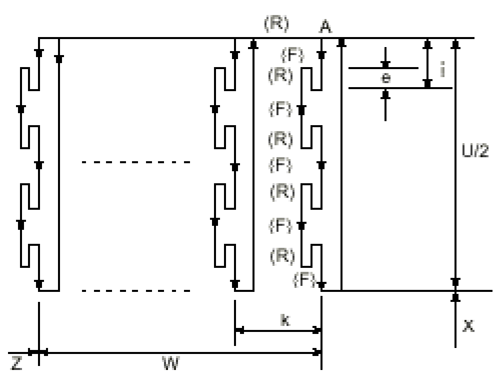


Figure 6-31: G75 Parameters

Example of G75

```
%

N00040 G53 G99 G97 G20 T0000 M5
N00050 G0 Z0.1 M11
N00060 G28 U0. W0.
N00070 S2000 M4 T0303
N00080 X1.05 Z1.
N00090 Z-0.6415 M8
N00110 G75 R0.05
N00120 G75 U-0.8 Z-1. P0.1 Q0.1 F0.002
N00130 T0300 M9
N00140 G28 U0. W0. M5
N00150 M30

%
```

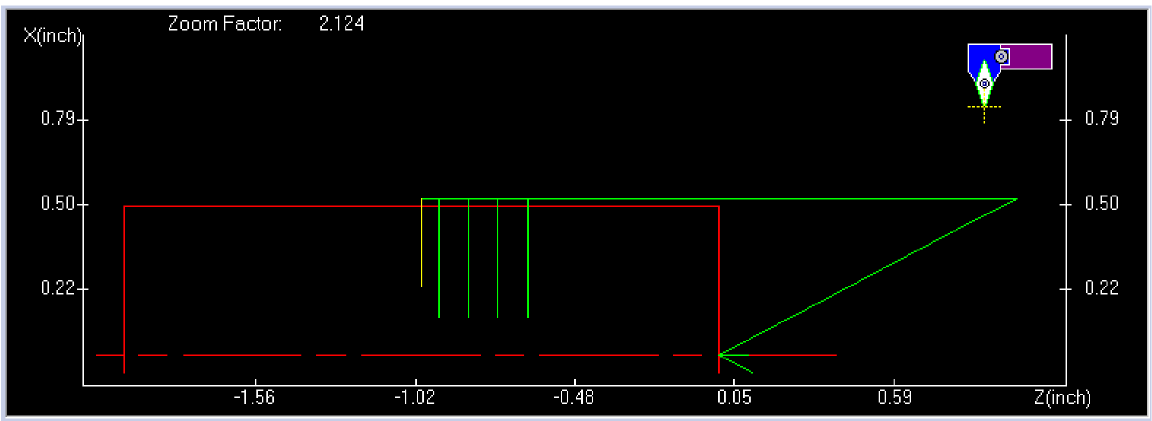


Figure 6-32: Plot of Example for G75

Limitations

Any T code in a G75 block will be ignored.

6.3.26 Multiple-Pass Threading Cycle (G76)

Required Format

G76 Pmra Qo Rd
G76 X/Ux Z/Wz Ri Pk Qq Ss Fl Mm Bb

Possible Parameters That Can Be Used With G76

m – number of finish-cutting passes: 01 ~99 (2-digit)
r – chamfering amount: 00 ~ 99 (2 digit, unit: 0.1 thread lead)
a – angle of tool tip: 00 ~ 99 (2 digit, unit: 1 degree)
o – minimum cutting depth (floating value w/o sign, radius value)
d – finishing allowance (floating value w/o sign, radius value)
x – X-axis end point coordinate of thread (abs/inc value, diameter or radius programming)
z – Z-axis end point coordinate of thread (abs/inc value)
i – taper height component (floating value with sign, radius value)
k – thread height (floating value w/o sign, radius value)
q – first cut amount (floating value w/o sign, radius value)
s – spindle speed
l – thread lead (floating value w/o sign)
m – M code
b – B code

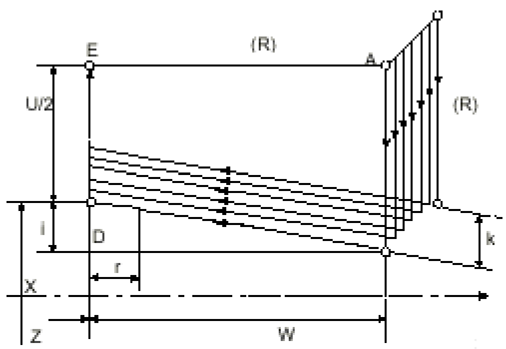


Figure 6-33: G76 Parameters

Example of G76

```
%
N00040 G53 G99 G97 G20 T0000 M5
N00050 G0 Z0.1
N00060 G28 U0. W0. M11
N00080 S1000 M4 T0404
N00090 X1.05 Z1.
N00090 Z-0.6415 M8
N0730 G76 P010060 Q0.005 R0
N0760 G76 X.814 Z-1.0 R0 P0.0307 Q0.005 F.05
N0770 G0 Z1.
N00300 T0400 M9
N00310 G28 U0. W0. M5
N00320 M30
%
```

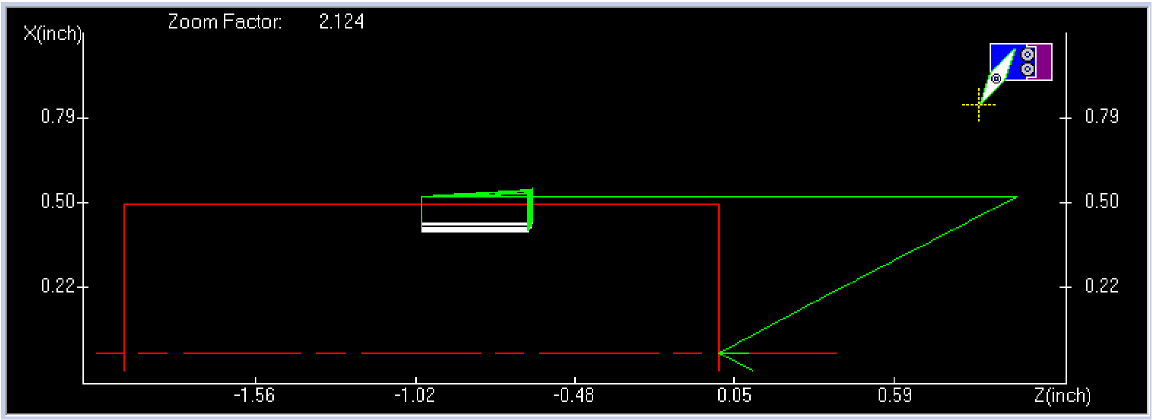


Figure 6-34: Plot of Example for G76

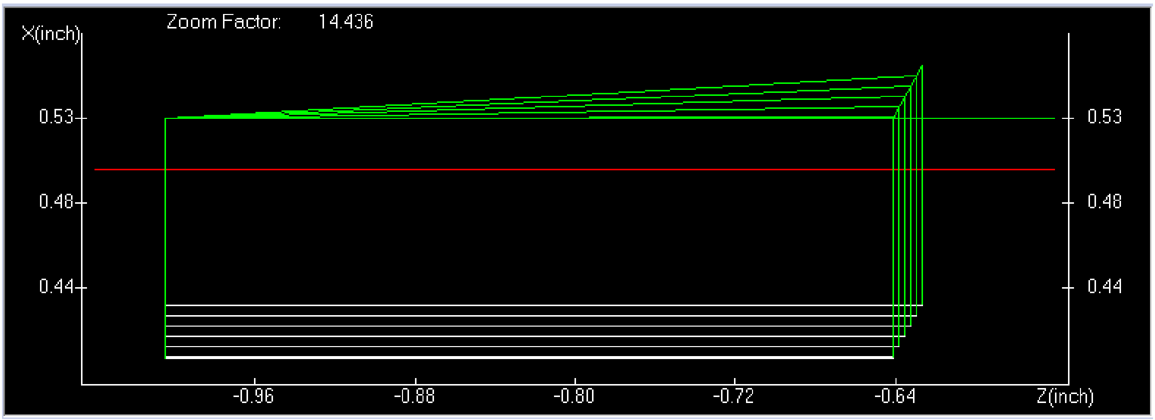


Figure 6-35: Enlarged Plot of Cutting Passes for Example for G76

Limitations

Any T code in a G76 block will be ignored.

Notes

- The thread cutting of G76 is the same as the thread cutting of G32 and G92.
- Manual Feedrate Override (MFO) is ineffective (fixed at 100 %).
- Spindle Speed Override (SSO) is ineffective (fixed at 100 %).
- Use G97 instead of G96 (Constant Surface Speed) for a G76 cycle.
- When FEEDHOLD is applied during threading in a G76 cycle, the tool will immediately retract and return to the start point on the X axis and then the Z axis. When CYCLE START is applied, the multiple thread cutting cycles resumes.
- G76 should be programmed under G99 mode (per revolution feed); otherwise, the “G76 Feed Mode Error” alarm will be triggered.
- The feedrate load factor, chamfer angle and chamfer amount to be used for retraction at the end of the G76 cycle (thread cutting retreat) are specified in Configuration Mode of ServoWorks S-100T, in the General Parameters Display Area.

6.3.27 Hole Machining Canned Cycle Cancel (G80)

Example of G80

```
%

N00160 G53 G99 G97 G20 T0000 M20
N00170 G98 M5
N00180 G0 Z.1
N00190 G28 U0. W0. M11
N00200 M19
N00210 T0808 M54
N00220 G0 X1.05 Z1.
N00230 Z.15

N00250 G0 X.7 C0 M8
N00260 G84 Z-1.25 R.1 F20
N00270 C90
N00280 C180
N00290 C270
N00300 G80

N00320 G0 X1.05 M9
N00330 Z1.
N00340 T0800 M55
N00350 M20
N00360 G28
N00370 M30

%
```

Notes

Any Group-01 G-code will also cancel the hole machining canned cycle.

6.3.28 Face Drilling Cycle (G83)

Required Format

G83 X/Ux C/Hc Z/Wz Rr Qq Pp Ff

Possible Parameters That Can Be Used With G83

x – X component of hole position (abs/inc value, diameter or radius programming)
c – C component of hole position (abs/inc value)
z – Z component from point R to the bottom of the hole (with sign, abs/inc value)
r – distance from initial level to point R level (w/o sign)
q – depth of each drilling (floating value w/o sign)
p – dwell time at the bottom of the hole
f – drilling feedrate

Example of G83

```
%

N00160 G53 G99 G97 G20 T0000 M20
N00170 G98 M5
N00180 G0 Z.1
N00190 G28 U0. W0. M11
N00200 M19
N00210 T0808 M54
N00220 G0 X1.05 Z1.
N00230 Z.15

N00250 G0 X.7 C0 M8
N00260 G83 Z-1.25 R.1 Q.1 F20
N00270 C90
N00280 C180
N00290 C270
N00300 G80

N00320 G0 X1.05 M9
N00330 Z1.
N00340 T0800 M55
N00350 M20
N00360 G28
N00370 M30

%
```

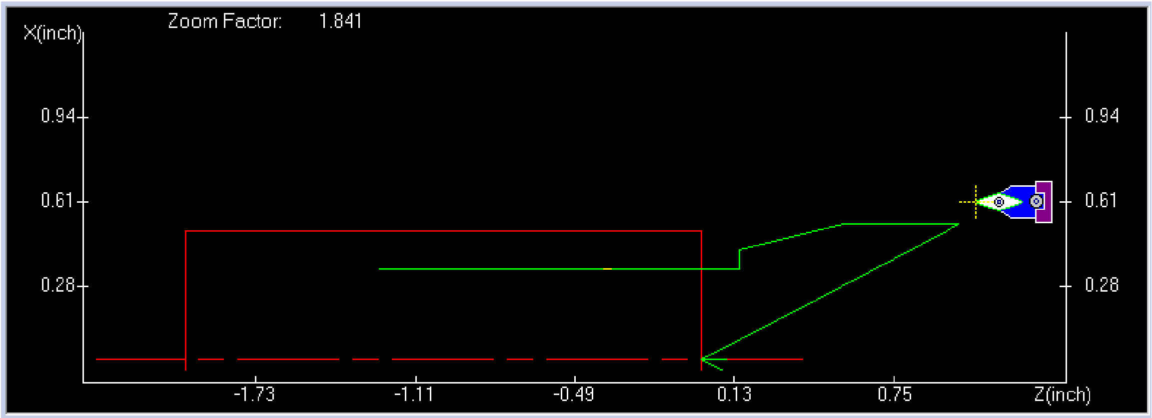


Figure 6-36: Plot of Example for G83

Notes

The high-speed peck drilling cycle is enabled/disabled with Spindle parameters in Configuration Mode.

6.3.29 Face Tapping Cycle (G84)

Required Format

G84 X/Ux C/Hc Z/Wz Rr Pp Ff

Possible Parameters That Can Be Used With G84

x – X component of hole position (abs/inc value, diameter or radius programming)
c – C component of hole position (abs/inc value)
z – Z component from point R to the bottom of the hole (with sign, abs/inc value)
r – distance from initial level to point R level (w/o sign)
p – dwell time at the bottom of the hole
f – tapping feedrate

Example of G84

```
%

N00160 G53 G99 G97 G20 T0000 M20
N00170 G98 M5
N00180 G0 Z.1
N00190 G28 U0. W0. M11
N00200 M19
N00210 T0808 M54
N00220 G0 X1.05 Z1.
N00230 Z.15

N00250 G0 X.7 C0 M8
N00260 G84 Z-1.25 R.1 F20
N00270 C90
N00280 C180
N00290 C270
N00300 G80

N00320 G0 X1.05 M9
N00330 Z1.
N00340 T0800 M55
N00350 M20
N00360 G28
N00370 M30

%
```

6.3.30 Face Boring Cycle (G85)

Description

After positioning at the hole position, the tool moves with rapid traverse to point R. Boring is performed from point R to the bottom of the hole. After the tool reaches the bottom, it returns to point R at the same boring feedrate. Then the tool is retracted with rapid traverse to initial level.

Required Format

G85 X/Ux C/Hc Z/Wz Rr Pp Ff)

Possible Parameters That Can Be Used With G85

x – X component of hole position (abs/inc value, diameter or radius programming)
c – C component of hole position (abs/inc value)
z – Z component from point R to the bottom of the hole (with sign, abs/inc value)
r – distance from initial level to point R level (w/o sign)
p – dwell time at the bottom of the hole
f – boring feedrate

Example of G85

```
%

N00160 G53 G99 G97 G20 T0000 M20
N00170 G98 M5
N00180 G0 Z.1
N00190 G28 U0. W0. M11
N00200 M19
N00210 T0808 M54
N00220 G0 X1.05 Z1.
N00230 Z.15

N00250 G0 X.7 C0 M8
N00260 G85 Z-1.25 R.1 F20
N00270 C90
N00280 C180
N00290 C270
N00300 G80

N00320 G0 X1.05 M9
N00330 Z1.
N00340 T0800 M55
N00350 M20
N00360 G28
N00370 M30

%
```

6.3.31 Side Drilling Cycle (G87)

Required Format

G87 Z/Wz C/Hc X/Ux Rr Qq Pp Ff)

Possible Parameters That Can Be Used With G87

z – Z component of hole position (abs/inc value)
c – C component of hole position (abs/inc value)
x – X component from point R to the bottom of the hole (with sign, abs/inc value, diameter or radius programming)
r – distance from initial level to point R level (w/o sign)
q – depth of each drilling (floating value w/o sign)
p – dwell time at the bottom of the hole
f – drilling feedrate

Example of G87

```
%

N00160 G53 G99 G97 G20 T0000 M20
N00170 G98 M5
N00180 G0 Z.1
N00190 G28 U0. W0. M11
N00200 M19
N00210 T0909 M56
N00220 G0 X1.25 Z1.
N00230 Z-0.495

N00250 G0 Z-.5 C0 M8
N00260 G87 X0.4 R.1 Q.1 F10
N00270 C90
N00280 C180
N00290 C270
N00300 G80

N00320 G0 X1. M9
N00330 Z.3
N00340 T0900 M57
N00350 M20
N00360 G28
N00370 M30

%
```

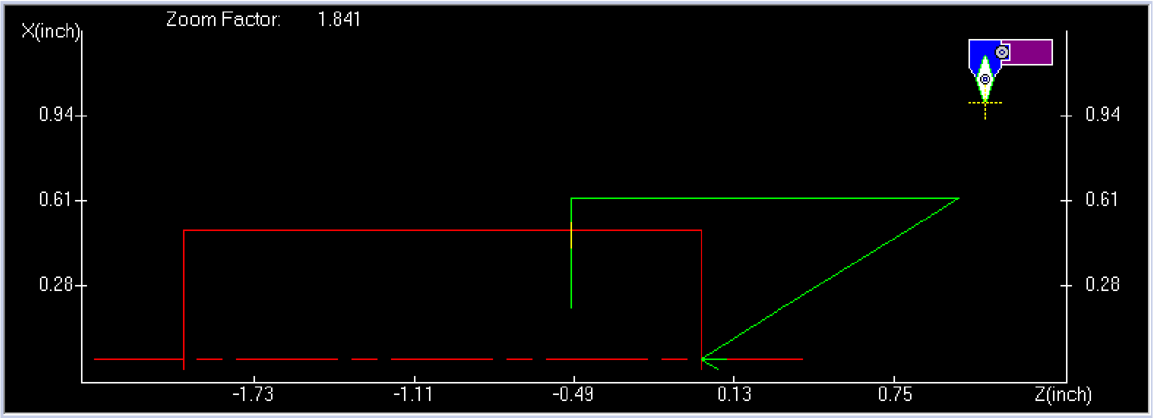


Figure 6-37: Plot of Example for G87

Notes

The high-speed peck drilling cycle is enabled/disabled with Spindle parameters in Configuration Mode.

6.3.32 Side Tapping Cycle (G88)

Required Format

G88 Z/Wz C/Hc X/Ux Rr Pp Ff

Possible Parameters That Can Be Used With G88

z – Z component of hole position (abs/inc value)
c – C component of hole position (abs/inc value)
x – X component from point R to the bottom of the hole (with sign, abs/Inc value, diameter or radius programming)
r – distance from initial level to point R level (w/o sign)
p – dwell time at the bottom of the hole
f – tapping feedrate

Example of G88

```
%

N00160 G53 G99 G97 G20 T0000 M20
N00170 G98 M5
N00180 G0 Z.1
N00190 G28 U0. W0. M11
N00200 M19
N00210 T0909 M56
N00220 G0 X1.25 Z1.
N00230 Z-0.495

N00250 G0 Z-.5 C0 M8
N00260 G88 X0.4 R.1 F10
N00270 C90
N00280 C180
N00290 C270
N00300 G80

N00320 G0 X1. M9
N00330 Z.3
N00340 T0900 M57
N00350 M20
N00360 G28
N00370 M30

%
```

6.3.33 Side Boring Cycle (G89)

Description

After positioning at the hole position, the tool moves with rapid traverse to point R. Boring is performed from point R to bottom of the hole. After the tool reaches the bottom, it returns to point R at the same boring feedrate. Then the tool is retracted with rapid traverse to initial level.

Required Format

G89 Z/Wz C/Hc X/Ux Rr Pp Ff

Possible Parameters That Can Be Used With G89

z – Z component of hole position (abs/inc value)
c – C component of hole position (abs/inc value)
x – X component from point R to bottom of the hole (with sign, abs/inc value, diameter or radius programming)
r – distance from initial level to point R level (w/o sign)
p – dwell time at the bottom of the hole
f – boring feedrate

Example of G89

```
%
N00160 G53 G99 G97 G20 T0000 M20
N00170 G98 M5
N00180 G0 Z.1
N00190 G28 U0. W0. M11
N00200 M19
N00210 T0909 M56
N00220 G0 X1.25 Z1.
N00230 Z-0.495

N00250 G0 Z-.5 C0 M8
N00260 G89 X0.4 R.1 F10
N00270 C90
N00280 C180
N00290 C270
N00300 G80

N00320 G0 X1. M9
N00330 Z.3
N00340 T0900 M57
N00350 M20
N00360 G28
N00370 M30
%
```

6.3.34 Outer Diameter / Inner Diameter Cutting Cycle (G90)

Description

A canned cycle for outer diameter / inner diameter cutting.

Required Format

G90 X/Ux Z/Wz Rr Ff

Possible Parameters That Can Be Used With G90

x – X component (abs/inc value, diameter or radius programming)
z – Z component (abs/inc value)
r – taper height component (with sign, radius value
f – cutting feedrate

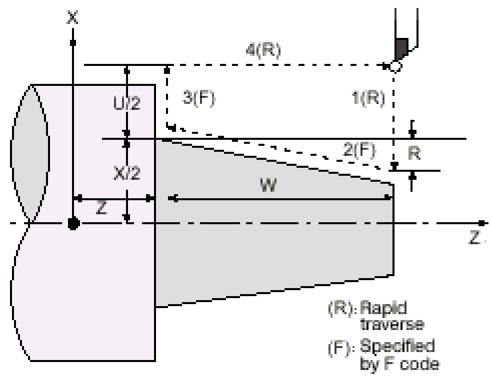


Figure 6-38: G90 Parameters

G90 Example #1: U < 0, W < 0, R = 0

```
%

N00040 G53 G99 G97 G20 T0000 M5
N00050 G0 Z0.1
N00060 G28 U0. W0. M11
N00070 S2000 M4 T0101
N00080 X1.05 Z1.
N00090 Z.1
N00110 G90 X.90 Z-1.4 F.01
N00130 G1 Z1. M9
N00140 T0100
N00150 G28 U0. W0. M5
N00160 M30

%
```

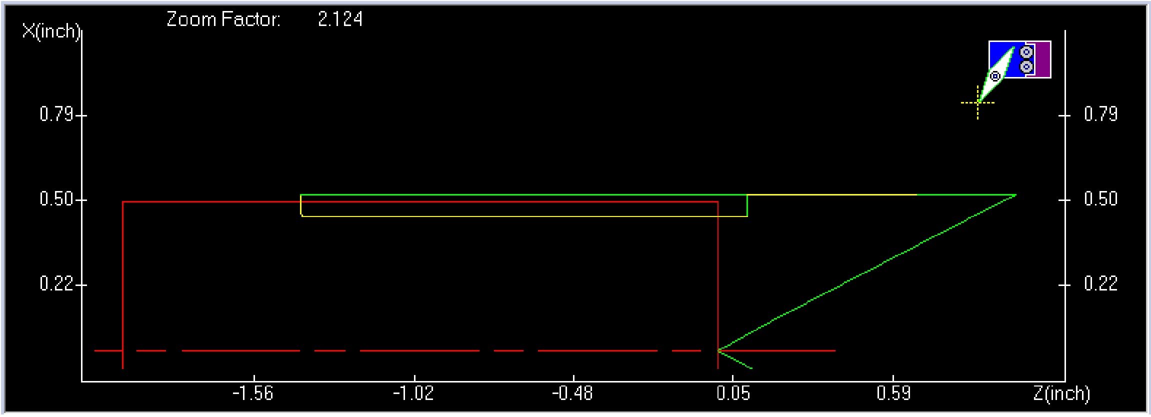


Figure 6-39: Plot of G90 Example #1

G90 Example #2: U < 0, W < 0, R < 0

```
%

N00040 G53 G99 G97 G20 T0000 M5
N00050 G0 Z0.1
N00060 G28 U0. W0. M11
N00070 S2000 M4 T0101
N00080 X1.05 Z1.
N00090 Z.1
N00110 G90 X.9 Z-1.4 R-.1 F.01
N00130 G1 Z1. M9
N00140 T0100
N00150 G28 U0. W0. M5
N00160 M30

%
```

Figure 6-40: Plot of G90 Example #2

G90 Example #3: U < 0, W < 0, R > 0

```
%

N00040 G53 G99 G97 G20 T0000 M5
N00050 G0 Z0.1
N00060 G28 U0. W0. M11
N00070 S2000 M4 T0101
N00080 X1.05 Z1.
N00090 Z.1
N00110 G90 X.7 Z-1.4 R.1 F.01
N00130 G1 Z1. M9
N00140 T0100
N00150 G28 U0. W0. M5
N00160 M30

%
```

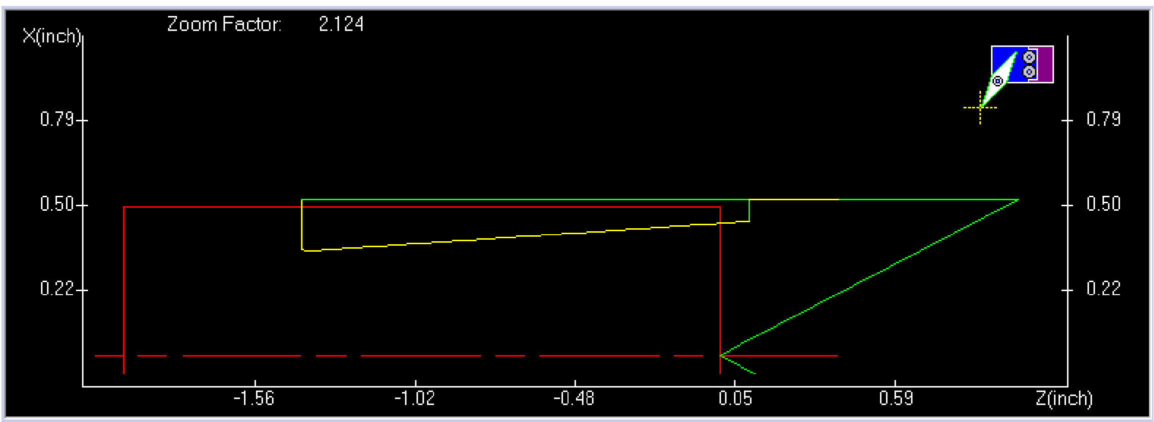


Figure 6-41: Plot of G90 Example #3

G90 Example #4: U > 0, W < 0, R > 0

```
%

N00040 G53 G99 G97 G20 T0000 M5
N00050 G0 Z0.1
N00060 G28 U0. W0. M11
N00070 S2000 M4 T0505
N00080 X-1.05 Z1.
N00090 Z.1
N00110 G90 X-.9 Z-1.4 R.1 F.01
N00130 G1 Z1. M9
N00140 T0500
N00150 G28 U0. W0. M5
N00160 M30

%
```

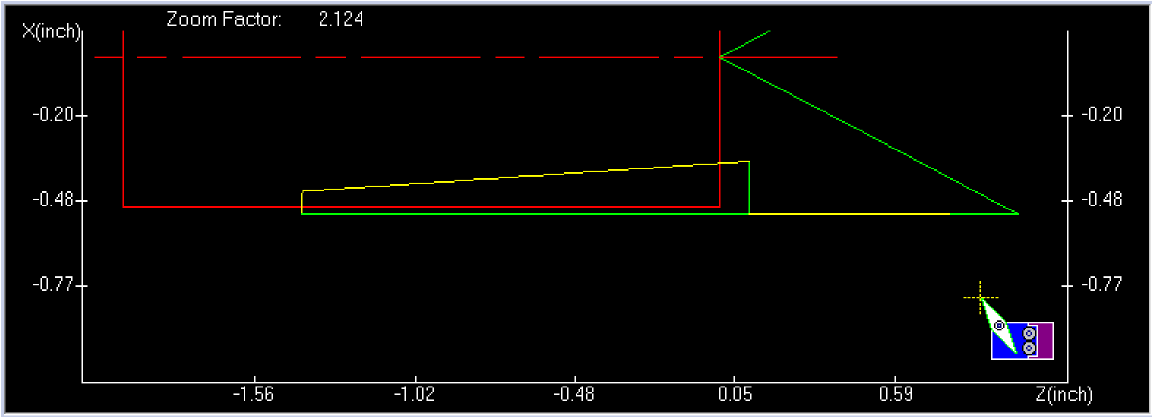


Figure 6-42: Plot of G90 Example #4

G90 Example #5: U > 0, W < 0, R < 0

```
%

N00040 G53 G99 G97 G20 T0000 M5
N00050 G0 Z0.1
N00060 G28 U0. W0. M11
N00070 S2000 M4 T0505
N00080 X-1.05 Z1.
N00090 Z.1
N00110 G90 X-.7 Z-1.4 R-.1 F.01
N00130 G1 Z1. M9
N00140 T0500
N00150 G28 U0. W0. M5
N00160 M30

%
```

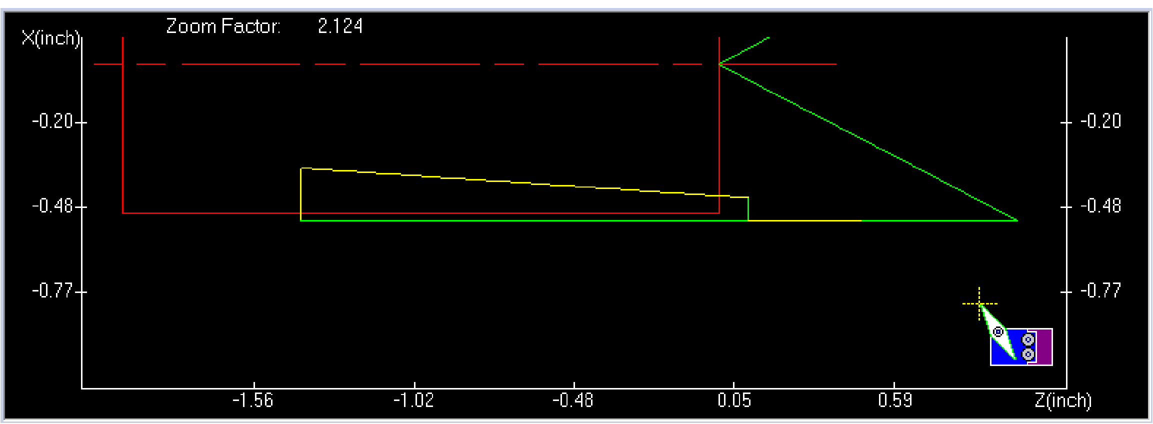


Figure 6-43: Plot of G90 Example #5

6.3.35 Thread Cutting Cycle (G92)

Description

A canned cycle for thread cutting.

Required Format

G92 X/Ux Z/Wz Rr Qq Fl

Possible Parameters That Can Be Used With G92

x – X component (abs/inc value, diameter or radius programming)
z – Z component (abs/inc value)
r – taper height component (with sign, radius value)
q – chamfering amount: 00 ~ 99 (2 digit, unit: 0.1 thread lead)
l – thread lead (same as G32)

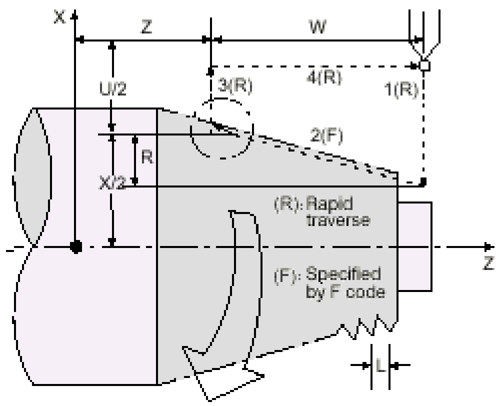


Figure 6-44: G92 Parameters

G92 Example #1: U < 0, W < 0, R = 0, Q = 0

```
%

N00040 G53 G99 G97 G20 T0000 M5
N00050 G0 Z0.1
N00060 G28 U0. W0. M11
N00070 S1000 M4 T0404
N00080 X1.05 Z1.
N00090 Z.1
N00110 G92 X.9 Z-1.4 Q00 F.05
N00130 G1 Z1. M9
N00140 T0400
N00150 G28 U0. W0. M5
N00160 M30

%
```

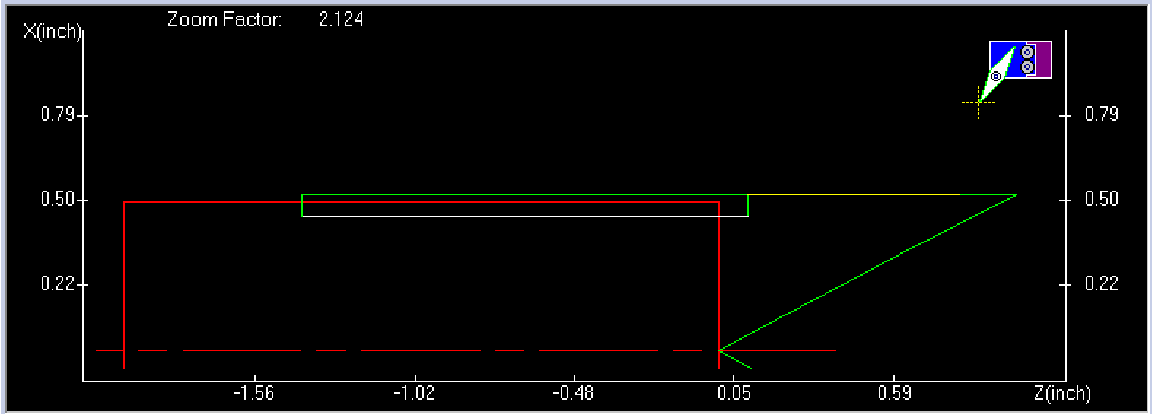


Figure 6-45: Plot of G92 Example #1

G92 Example #2: U < 0, W < 0, R < 0, Q = 06

```
%

N00040 G53 G99 G97 G20 T0000 M5
N00050 G0 Z0.1
N00060 G28 U0. W0. M11
N00070 S1000 M4 T0404
N00080 X1.05 Z1.
N00090 Z.1
N00110 G92 X.9 Z-1.4 R-.1 Q06 F.05
N00130 G1 Z1. M9
N00140 T0400
N00150 G28 U0. W0. M5
N00160 M30

%
```

Figure 6-46: Plot of G92 Example #2

Notes

- Manual Feedrate Override (MFO) is ineffective (fixed at 100 %).
- Spindle Speed Override (SSO) is ineffective (fixed at 100 %).
- Use G97 instead of G96 (Constant Surface Speed) for a G92 cycle.
- When FEEDHOLD is applied during threading in a G92 cycle, the tool will immediately retract and return to the start point on the X axis and then the Z axis.
- G92 should be programmed under G99 mode (per revolution feed); otherwise, the “G92 Feed Mode Error” alarm will be triggered.
- The feedrate load factor, chamfer angle and chamfer amount to be used for retraction at the end of the G92 cycle (thread cutting retreat) are specified in Configuration Mode of ServoWorks S-100T, in the General Parameters Display Area.

6.3.36 End Face Cutting Cycle (G94)

Description

A canned cycle for end face cutting.

Required Format

G94 X/Ux Z/Wz Rr Ff

Possible Parameters That Can Be Used With G94

x – X component (abs/inc value, diameter or radius programming)
z – Z component (abs/inc value)
r – taper height component (with sign, radius value)
f – cutting feedrate

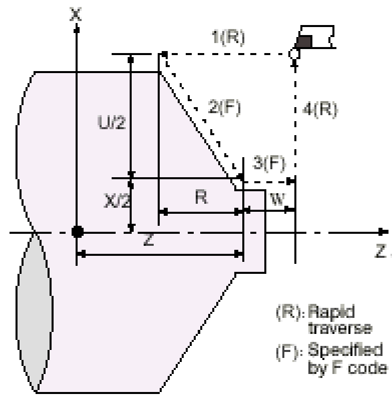


Figure 6-47: G94 Parameters

G94 Example #1: U < 0, W < 0, R = 0

```
%

N00040 G53 G99 G97 G20 T0000 M5
N00050 G0 Z0.1
N00060 G28 U0. W0. M11
N00070 S2000 M4 T0101
N00080 X1.05 Z1.
N00090 Z.1
N00110 G94 X.25 Z-.15 F.01
N00130 G1 Z1. M9
N00140 T0100
N00150 G28 U0. W0. M5
N00160 M30

%
```

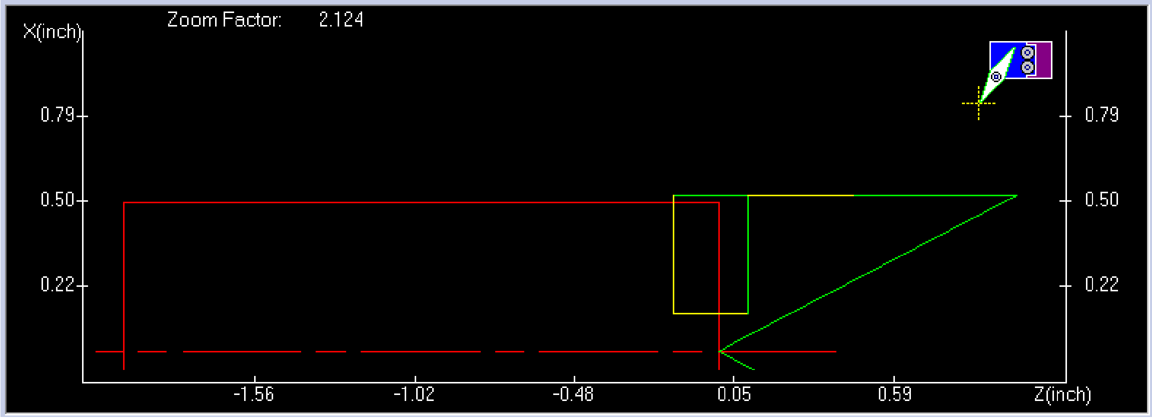


Figure 6-48: Plot of G94 Example #1

G94 Example #2: U < 0, W < 0, R < 0

```
%

N00040 G53 G99 G97 G20 T0000 M5
N00050 G0 Z0.1
N00060 G28 U0. W0. M11
N00070 S2000 M4 T0101
N00080 X1.05 Z1.
N00090 Z.1
N00110 G94 X.25 Z-.15 R-.1 F.01
N00130 G1 Z1. M9
N00140 T0100
N00150 G28 U0. W0. M5
N00160 M30

%
```

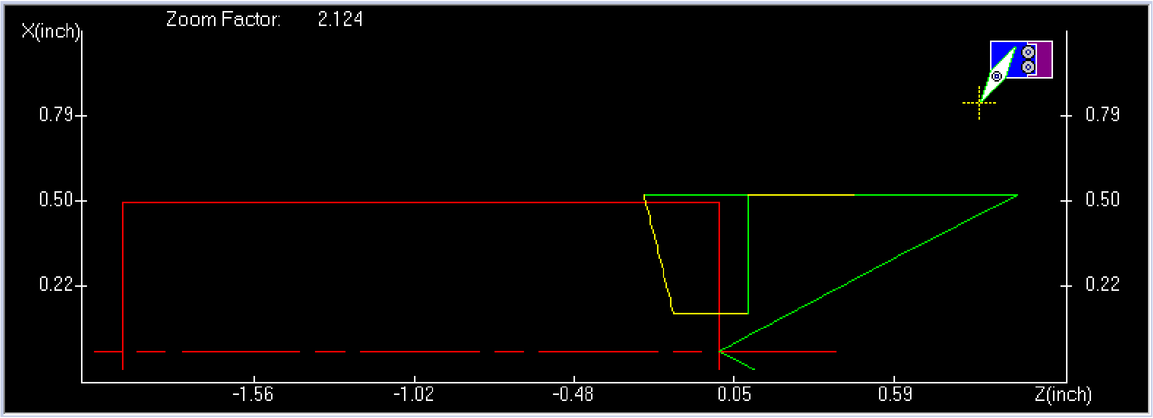


Figure 6-49: Plot of G94 Example #2

G94 Example #3: U < 0, W < 0, R > 0

```
%

N00040 G53 G99 G97 G20 T0000 M5
N00050 G0 Z0.1
N00060 G28 U0. W0. M11
N00070 S2000 M4 T0101
N00080 X1.05 Z1.
N00090 Z.1
N00110 G94 X.25 Z-.25 R.1 F.01
N00130 G1 Z1. M9
N00140 T0100
N00150 G28 U0. W0. M5
N00160 M30

%
```

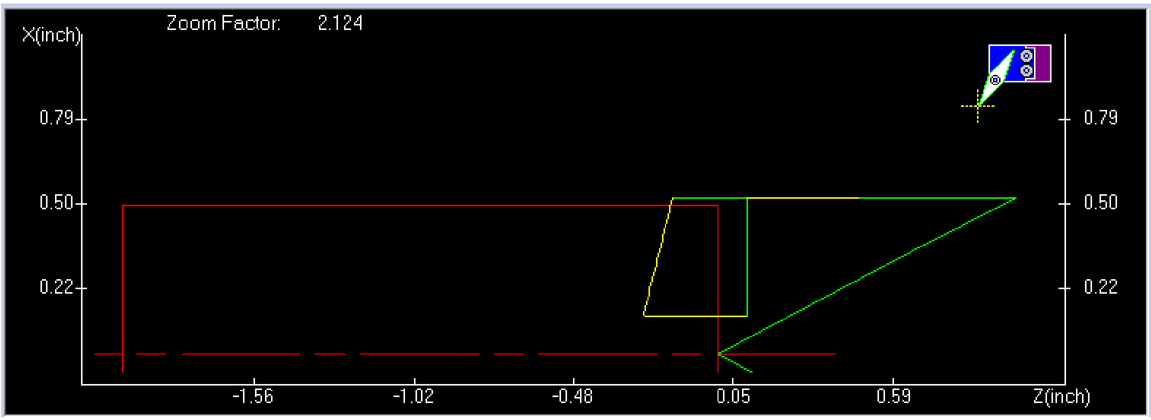


Figure 6-50: Plot of G94 Example #3

G94 Example #4: U > 0, W < 0, R < 0

```
%

N00040 G53 G99 G97 G20 T0000 M5
N00050 G0 Z0.1
N00060 G28 U0. W0. M11
N00070 S2000 M4 T0505
N00080 X-1.05 Z1.
N00090 Z.1
N00110 G94 X-.25 Z-.15 R-.1 F.01
N00130 G1 Z1. M9
N00140 T0500
N00150 G28 U0. W0. M5
N00160 M30

%
```

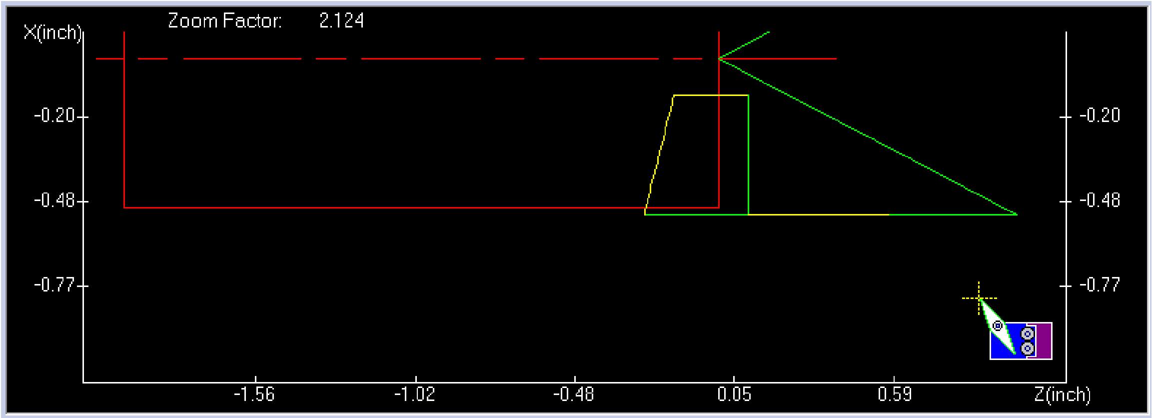


Figure 6-51: Plot of G94 Example #4

G94 Example #5: U > 0, W < 0, R > 0

```
%

N00040 G53 G99 G97 G20 T0000 M5
N00050 G0 Z0.1
N00060 G28 U0. W0. M11
N00070 S2000 M4 T0505
N00080 X-1.05 Z1.
N00090 Z.1
N00110 G94 X-.25 Z-.25 R.1 F.01
N00130 G1 Z1. M9
N00140 T0500
N00150 G28 U0. W0. M5
N00160 M30

%
```

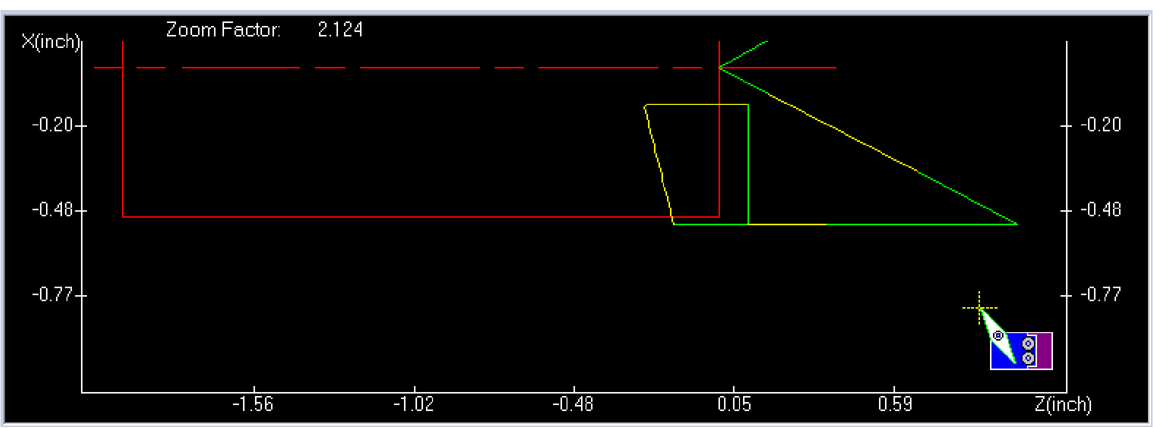


Figure 6-52: Plot of G94 Example #5

6.3.37 Constant Surface Speed Control / Constant Surface Speed Cancel (G96, G97)

Description

The G96 and G97 commands are used to specify either constant surface speed control or constant surface speed cancel, which refer to two different ways to set the rotation speed of a spindle.

G96: Constant surface speed control (spindle speed is given in meters per minute, or feet per minute).
G97: Constant surface speed cancel (spindle speed is given in RPM).

Constant surface speed is dependent upon the diameter of the spindle. In fact, the speed of the spindle in revolutions per minute is calculated based upon the desired surface speed and the diameter of the spindle. For a given constant surface speed, the greater the diameter of the spindle, the slower the spindle speed (as measured in revolutions per minute).

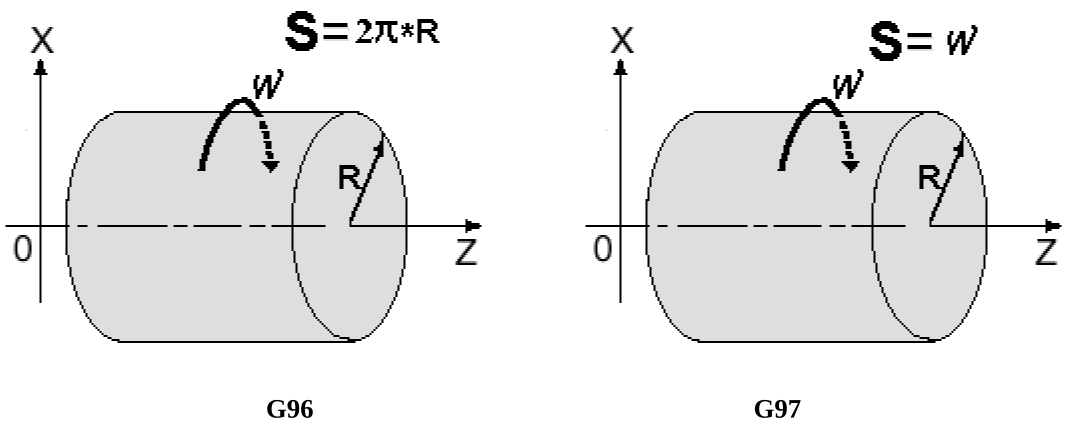


Figure 6-53: G96/G97 Parameters

Example

```
G96 S100.9
G97 S1500 M4
```

Default

G97 is the default mode when neither G96 nor G97 has been programmed.

6.3.38 Feedrate in Inch Per Minute / Inch Per Revolution (G98, G99)

Description

The G98 and G99 commands are used to set the feedrate for the spindle in inches per minute or inches per revolution.

G98: Feedrates will be in inches per minute or millimeters per minute
G99: Feedrates will be in inches per revolution or millimeters per revolution

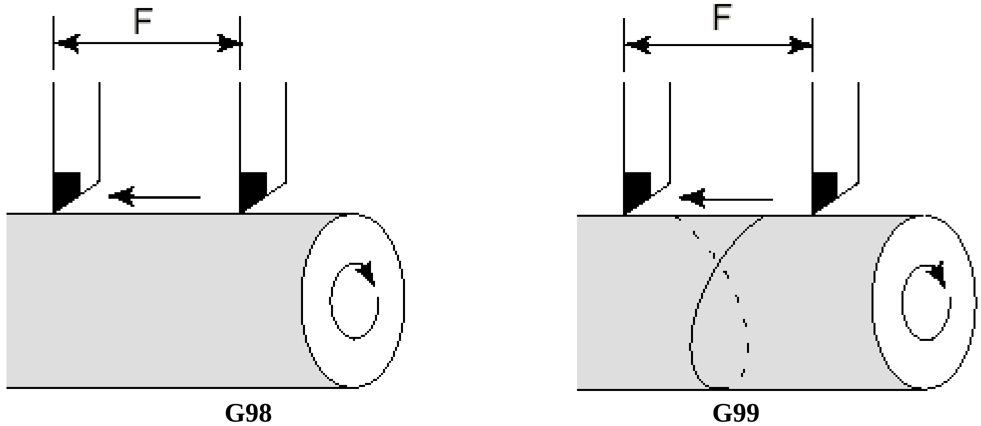


Figure 6-54: G98/G99 Parameters

Default

G98 is the default mode when neither G98 nor G99 has been programmed.

6.3.39 Cylindrical Interpolation (G107)

Required Format

G107 Rr

Parameters Typically Used With G107

r – radius of the cylinder (floating value w/o sign, rotation axis: C)

Example of G107

```
%
N00140 G53 G99 G97 G20 T0000 M20
N00150 G98 M5
N00160 G0 Z.1
N00170 G28 U0. W0. M11
N00180 M19
N00190 T0707 M52
N00200 G0 X-1.
N00210 Z-.4625
N00220 X-.7
N00230 G107 R.3125

( S )
```

```
N00260 G1 X-.61 F14
N00270 G4 P.1
N00280 C50
N00290 G4 P.1
N00300 Z-.5625
N00310 G4 P.1
N00320 C0
N00330 G4 P.1
N00340 Z-.6625
N00350 G4 P.1
N00360 C50
N00370 G4 P.1
N00380 X-.7
N00390 G4 P.1
N00400 G0 C65 Z-.4625

( N )
N00430 G1 X-.61 F14
N00440 G4 P.1
N00450 Z-.6625
N00460 G4 P.1
N00470 C115 Z-.4625
N00480 G4 P.1
N00490 Z-.6625
N00500 G4 P.1
N00510 X-.7
N00520 G4 P.1
N00530 G0 C130 Z-.4625


( K )
N00560 G1 X-.61 F14
N00570 G4 P.1
N00580 Z-.6625
N00590 G4 P.1
N00600 X-.7
N00610 G4 P.1
N00620 G0 C180 Z-.6625
N00630 G1 X-.61 F20
N00640 G4 P.1
N00650 C130 Z-.5625
N00660 G4 P.1
N00670 C180 Z-.4625
N00680 G4 P.1
N00690 X-.7
N00700 G4 P.1
N00710 G0 C0 Z-.35
N00720 G107 R0

N00740 G0 X-.7
N00750 Z.3
N00760 T0700 M53
N00770 M20
N00780 G28
N00790 M30
%
```

Limitations

Only G00, G01 and G04 are eligible in Cylindrical Interpolation Mode.

Notes

- r > 0: Cylindrical Interpolation Mode is enabled.
- r = 0: Cylindrical Interpolation Mode is cancelled.

Other Codes Which Affect G107

- M19: Spindle orientation & positioning mode
- M20: Spindle rotation mode
- G98 (feed per minute) should be declared before M19

6.3.40 Polar Coordinate Interpolation Mode / Polar Coordinate Interpolation Mode Cancel (G112, G113)

Description

With polar coordinate interpolation mode, commands programmed in a Cartesian coordinate system are converted to separate movements of a linear axis and a rotary axis (for a tool and a workpiece, for example).

G112: Polar coordinate interpolation mode start
G113: Polar coordinate interpolation mode cancel

Default

None

Chapter 7: M Codes and B Codes (Miscellaneous Codes)

7.1 Summary of M Codes and B Codes

Both M codes and B codes are miscellaneous codes used for sending commands from CNC to PLC. Default M codes are used on most lathes and turning machines. In addition to default M codes, there are user customizable M codes and B codes that will change based on application and user definition. There are 100 sets of M codes (M00 to M99) and 100 sets of B codes (B00 to B99).

NOTES:

- There are no default B codes; all B codes are user-defined.
- M code is sent to PLC through BCD format (F10.0 ~ F10.7).
- For each new M code, a one-shot M strobe signal (F07.0) is also sent to PLC.
- B code is sent to PLC through BCD format (F30.0 ~ F30.7).
- For each new B code, a one-shot B strobe signal (F07.4) is also sent to PLC.

7.2 Default M Codes

Default M codes are listed in the following table. (If there is more than one possible meaning, both meanings are listed.)

M-Code	Description
M00	program stop
M01	Program optional stop
M02	program end
M03	spindle start – clockwise
M04	spindle start – counterclockwise
M05	spindle stop
M08	coolant on
M09	coolant off
M10	chuck unclamp / collet open
M11	chuck clamp / collet close

Table 7-1: Summary of M Codes (1 of 2)

M-Code	Description
M19	spindle orientation
M20	spindle rotation mode
M30	program end and rewind
M50	live tool #1 on
M51	live tool #1 off
M52	live tool #2 on
M53	live tool #2 off
M54	live tool #3 on
M55	live tool #3 off
M98	subprogram call from a main program (ex.: M98 PTest_1 R3 calls subprogram OTest_1.dat in the same folder as the parent program, and repeats it three times)
M99	return to main program from a subprogram OR returns the flow of execution to the beginning of the main program, if used in the main program

Table 7-2: Summary of M Codes (2 of 2)

Except for M98 and M99, all M codes listed here are programmed in the default LadderWorks PLC sequence program provided with ServoWorks S-100T. In order to use these M codes (i.e. M00, M01, etc.), you must put the relevant PLC programming code for each M code in the customized PLC sequence program for your machine. We recommend you use the default LadderWorks PLC sequence program as the starting point for your customized PLC sequence program.

In addition to these preset M codes, you can create customized M codes using custom M code macro calls. See *Section 6.5: Custom Macro Calls Using G Codes, M Codes, S Codes or T Codes* in the *ServoWorks CNC Macro Programming Manual*.

NOTE: M00, M01, M02 and M30 codes are processed after any motion specified in the same block is completed. Therefore, if you want one of these M codes to be executed first, specify it in a separate line of code prior to the block of code specifying the motion you want it to precede.

All other M codes and all B codes are processed simultaneously with any motion specified in the same block.

Chapter 8: Spindle Functions and S Codes

8.1 Overview of S Codes

Description

S codes are 5-digit codes used to specify either the spindle speed (RPM) directly or the constant surface speed (the relative speed between the tool tip and the workpiece).

Required Format

Sxxxxx

Parameters

xxxxx – spindle speed (integer)

Examples

`G97 Sxxxxx` (for turning speed control, where the unit is in RPMs)
`G96 Sxxxxx` (for surface speed control, where the unit is in either meter/min. or feet/min.)

Limitations

None of the M03, M04, M05 or S codes is eligible in spindle positioning mode.

Notes

- The S code is sent to PLC through BCD format (F22.0 ~ F22.3).
- For each new S code, a one-shot S strobe signal (F07.2) is also sent to PLC.

⚠ CAUTION

Before M19 is activated, the feed mode should be changed to G98 (feed per minute). Otherwise an Alarm Message will appear.

⚠ CAUTION

Before M19 is activated, the spindle should be stopped. Otherwise, the ServoWorks S-100T will stop the spindle first, then execute the M19 code.

Other Codes Which Affect S Codes

- M03 rotates the spindle in the CW direction.
- M04 rotates the spindle in the CCW direction.
- M05 stops the spindle.
- G50 Sxxxxx is used to clamp the maximum spindle speed (RPM) while in G96 (constant surface speed mode).
- G25 is to disable spindle speed fluctuation check.
- G26 is to enable spindle speed fluctuation check.
- G107 (cylindrical interpolation) and G112, G113 (polar coordinate interpolation) are the contour control in M19 mode.

Reference

For information on spindle parameters, see *Chapter 10: Spindle Parameters* in the *ServoWorks S-100T Parameters Manual*.

8.2 Spindle Functions and Parameters

If the machine has a servo type spindle drive, then M19 is used to specify the spindle orientation (positioning mode/C-axis Homing), and M20 is used to release spindle positioning mode and change to spindle rotating mode. Both M19 and M20 are default values; they can be redefined in Spindle Parameters in Configuration Mode of ServoWorks S-100T.

In spindle positioning mode, C is the absolute dimensioning for the C axis (floating value, the unit is degree). H is the incremental dimensioning for the C axis (floating value, the unit is degree.).

Depending on the setting of Spindle Parameters, ServoWorks S-100T provides one of the following speed voltage outputs:

a) Analog Voltage (0 ~ 10 V) for a unipolar driver such as an inverter.
b) Analog Voltage (+/- 10 V) for a bipolar driver such as a servo amplifier.

Notes

- When an S code is with any motion in the same block, then motion will hold until spindle speed reaches the range that is defined in Spindle Parameters, at which time motion begins.
- When an S code is with a general M code (and/or B code) in the same block, then S codes and general M (and/or B) codes are parallel processed.

Variations

In addition to these preset S codes, you can create customized S codes using custom S code macro calls. See *Section 6.5: Custom Macro Calls Using G Codes, M Codes, S Codes or T Codes* in the *ServoWorks CNC Macro Programming Manual*.

Chapter 9: Tool Functions and T Codes

9.1 Overview of T Codes

Description

T codes are 4-digit codes used to select the tool and associated offset number.

Required Format

Txxyy

Parameters

xx – tool number 01 ~ 99 (2-digit)
yy – tool offset number 01 ~ 99 (2-digit)

Notes

- Txx00 or T0000 is used to cancel any active tool offset value.
- Only tool number xx is sent to PLC through BCD format (F26.0 ~ F26.7).
- For each new tool number, a one-shot T strobe signal (F07.3) is also sent to PLC.
- If the machine has a tool turret, then during tool selection, the command signal of PLC (G05.3) should be low until the selection is done.
- If the machine has gang tools, then G05.3 can always be kept high.

⚠ CAUTION

If a T code is programmed with any canned cycle G code (G70, G71, G72, G73, G74, G75, G76, G90, G92, G94) in the same block, then this T code will be ignored.

Other Codes Which Affect T Codes

- G10 L10 is used to update Tool Geometry Offsets.
- G10 L11 is used to update Tool Wear Offsets.
- G53, which is the machine coordinate selection code, will cancel any active tool offset compensation.

Variations

In addition to this preset T code, you can create customized T codes using custom T code macro calls. See *Section 6.5: Custom Macro Calls Using G Codes, M Codes, S Codes or T Codes* in the *ServoWorks CNC Macro Programming Manual*.

9.2 Tool Offsets

In the Tool Offset Table (in Offset Mode of ServoWorks S-100T), there are 99 sets of offsets. Each set of offsets consists of the following:

- X Geometry Offset (diameter or radius programming)
- X Wear Offset (diameter or radius programming)
- Z Geometry Offset
- Z Wear Offset
- Radius Geometry Offset
- Radius Wear Offset
- Tool Location Code: 0 ~ 9

When the specified tool offset number is selected, the offset values for X, Z and Radius are the sum of the corresponding Geometry Offset and the corresponding Wear Offset, respectively:

Offset Data = Geometry Offset + Wear Offset

Both the X and Z tool offsets are used as Tool Length Compensation. The Tool Location Code and Radius Offset are used as Tool Nose Radius Compensation (TNRC) with G40, G41 or G42.

When the specified tool offset number is selected, there is no actual movement. The current workpiece coordinate is automatically shifted for the associated tool offset data.

Notes

When a T code is with any motion in the same block, then the T code will be executed first, and the associated tool offset compensation will become effective. After this, motion is executed.

Example 1

```
N00100 T0101 G00 X1.0 Z.05
```

Tool #1 will be selected first. When the tool selection is done, the coordinate system will be shifted with respect to the amount of tool offset #1 (i.e., a new coordinate is set). Then the tool will move to the corresponding point (X1.0 and Z.05) of this new coordinate system.

The above is equivalent to:

```
N00100 T0101
N00110 G00 X1.0 Z.05
```

Example 2

```
N00100 T0100 G00 X.0 Z.0
```

The tool compensation of offset #1 is cancelled. The coordinate system is set back to the one that was in effect before the tool offset was activated. Then the tool will move to corresponding point (X1.0 and Z.05) of this coordinate system.

The above is equivalent to:

```
N00100 T0100
N00110 G00 X.0 Z.0
```

Index