

Linear one and two axis moves are accomplished by giving the axis and the value to move. The result of a command will depend on which mode is active: G90 or G91, and G73 (radius) or G72 (diameter).

X moves default to RADIUS mode (G73), not diameters. If you want your moves to be in diameter you must put a G72 in the begining of your program.

See the example at the end of this chapter

Feed and Rapid moves are accomplished as follows:

Movement in X:

Xn

Single axis (X) move

example: X.25

This will move to a diameter of .5

The programming in X can be done as radius movements (G73) or diameter (G72).

Movement in Z:

Zn

Single axis (Z) move

example: Z-.1

This will move the tool -.1 into the material

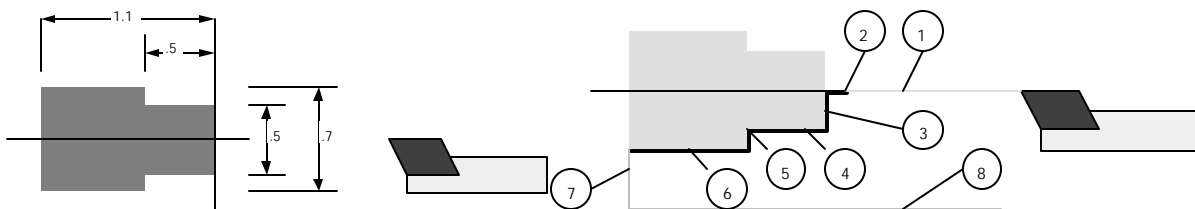
Z moves can be programmed directly off the print.

Movement in X and Z simultaneously

XnZn

Two axis linear move (X & Z)

Example for linear moves



The moves to generate this tool path could be as shown below. The tool path given does not show any of the codes needed to determin the feedrates or spindle speeds etc. The code is shown in the diameter mode:

radius mode:
 G90G94F300G72
 M03S2000
 M08
 T1
 X0 Z1
 (1) Z1
 (2) Z0 G95f.003
 (3) X.5
 (4) Z-.5
 (5) X.7

radius mode:
 G90G94F300G73
 M03S2000
 M08
 T1
 X0 Z1
 Z1
 Z0G95f.003
 X.25
 Z-.5
 X.35

Linear moves X n Z n XnZn

(6)
(7)
(8)

Z-1.1
X1
Z1 *G00*
M30

Z-1.1
X.5
Z1 *G00*
M30

A note about feed rate modes:

The feed rate commands are modal. Once they are set they stay in effect until they are changed. So once you set the mode you do not have to change it again until you change the mode. Once the feed rate mode is selected you can change the feed rate by adding the new feed rate to the line of code where you need it.

G94 - Inch per minute mode

The G94 command will make all feed rate commands effective in inch/minute. If you have a feed rate command of F4 in this mode it will move the slide at 4"/minute. F.001 would be .001"/minute, Very Very Slow!!! This is a good way to rapid around, but it is not a good way to program feed (cutting) moves. If the spindle is not turned on the slide will still have the tool make the cut.

G95 - Inch per revolution mode

This mode has the tool move in inches/revolution. If the spindle is not turning the slide will not move.

The feedrate used is the last one that was given. If a new feedrate is needed it can be added to the movement command. The type of feed rate mode can be determined at or before the line. The mode established stays in effect until it is changed.

Example formats are:

G94XnF4	Single axis (X) move with new feedrate, at 4"/minute
G95XnF.003	Single axis (X) move with new feedrate, at .003"/rev

Rapid travel (G00):

Rapid travel is accomplished by either

Using G94 for rapid travels:

Setting the feedrate in the Inches per minute mode (G94) with a value of F300 (check your system to see what the fastest IPM rate is, systems shipped before 7/93 are usually 200IPM. This will move the slide at its fastest rate. When the rapid moves are completed the feedrate is reset to the desired feed rate. An example format is:

G94F300	Set the feed to IPM @ 300"/minute
Xn	The move in X
Zn	Move in Z

Note: The advantage of using this way to rapid vs. the G00 command is that you can adjust the speed of the rapid travel by changing the Fnnn number. With G00 you are fixed to the rapid travel feed rate set in the first line of code. With the G94Fnnn method you can adjust the rapid travel for each move if you need to.

Linear moves X n Z n XnZn

Note good tip:

When you are running a program for the first time and proving it out you could make all of your rapid moves F100. Then when the program is known to work you could change all the F100 to F300 by using the editor with “find and replace”. See the notes in chapter 5, F3 on using this feature.

Using G00 for rapid travels

The rapid travel mode can be established by using the G00 command. This will set the mode to IPM at a feed rate specified in the first line of the program. If you do not set the feed rate in the first line of the program it will use the last rapid move it did in either the manual mode (100”/min) or from the last program that was run. So it is best to establish a rapid rate in your program. In the following example we set the speed to 300”/min in the first line of the program. The next time you use a G00 comand the rapid travel will be 300”/min.:

```
G90G94F300
```

In the following example the G00 is used to change to the rapid mode:

G00	Set the feed to what is set in the first line of the prog.
Xn	The move in X
Zn	Move in Z

Feed rate mode (G01):

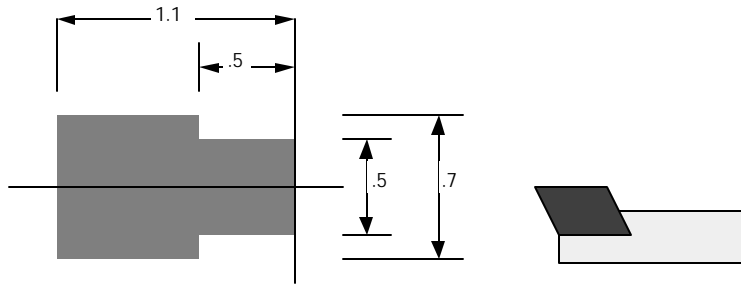
You can go from rapid travel mode to feed rate mode by either specifying a new feed rate and mode, IE: G94F300 for rapid travel, and G95F.00*n* for feed rate. or using G01 to revert to the last described feed rate mode in the program. This eliminates having to use the G95F.00*n* over and over. The other advantage of using G01 vs specifying the feed rate every time is that if you want to globally change the feed rate you only have to change one line.

To use the G01 command you must specify the first feed rate move in the program with the G95F.00*n* format. This way the program knows what feedrate to use. Here is an example showing establishing the rapid and feed rate modes and making some moves, then changing to rapid, and then back to the feed rate with G01:

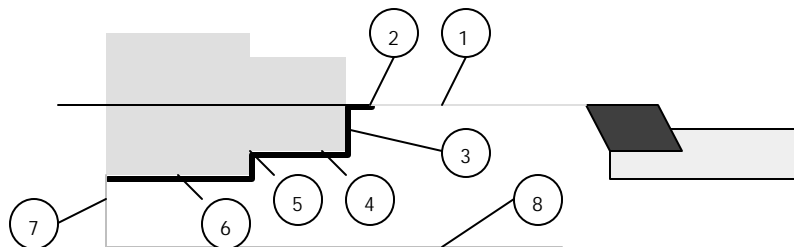
```
G94F300X0Z1
Z.1
G95F.003Z.005
X.5
G00Z.05
X0
G01Z0
X.5
G00Z1
```

Example

In the example below we will be finish turning a sample part with a left hand tool. The part dimensions are given in diameters. When the part is programmed these will have to be converted to radius moves. Z zero is set at the face of the part, X zero is set at the center.



The next diagram shows the rapid moves as a lighter line, feed moves are shown as the heavier lines.



The same example in radius mode

	G90 G 73 G94 F300	Establish Abs. positioning, Feed IPM @ 300"/minute, Radius mode
	T1 (turn tool)	Call Tool #1 offset
	X0 Z1	moves the tool to a safe start location
(1)	Z.1	Rapid to .1" from the face
(2)	G95 F.003 Z0	Set feed rate to IPR @ .003"/ rev, Feed to the face
(3)	X.25	Feed out to .25" radius (.5" diameter)
(4)	Z-.5	Feed to Z -.5
(5)	X.35	Feed to .7" diameter
(6)	Z-1.1	Feed to -1.1 in Z
(7)	G94 F300 X.5	Set feed rate to IPM @ 300"/minute, Pull the tool away in X
(8)	Z1	Withdraw the tool in Z
	M30	End of program command

The same example with G00 and G01

	G90 G94F300 G73	Establish Abs. positioning, rapid IPM @ 300"/minute, diameter mode
	T1	Call Tool #1 offset
	X0 Z1	moves the tool to a safe start location
(1)	Z1	Rapid to .1" from the face
(2)	G95 F.003 Z0	Set feed rate to IPR @ .003"/ rev, Feed to the face
(3)	X.25	Feed out to .25" radius (.5" diameter)
(4)	Z-.5	Feed to Z -.5

Linear moves X n Z n XnZ

(5)	X.35	Feed to .7" diameter
(6)	Z-1.1	Feed to -1.1 in Z
(7)	G00 X.5	Rapid feed rate, Pull the tool away in X
(8)	Z1	Withdraw the tool in Z
	M30	End of program command

The same example written in diameter mode (G72)

	G90 G72 G94 F300	Establish Abs. positioning, Feed IPM @ 300"/minute, Radius mode
	T1 (turn tool)	Call Tool #1 offset
	X0 Z1	moves the tool to a safe start location
(1)	Z1	Rapid to .1" from the face
(2)	G95 F.003 Z0	Set feed rate to IPR @ .003"/ rev, Feed to the face
(3)	X.5	Feed out to .25" radius (.5" diameter)
(4)	Z-.5	Feed to Z -.5
(5)	X.7	Feed to .7" diameter
(6)	Z-1.1	Feed to -1.1 in Z
(7)	G94 F300 X1	Set feed rate to IPM @ 300"/minute, Pull the tool away in X
(8)	Z1	Withdraw the tool in Z
	M30	End of program command