

### Objectives

In this chapter, the main objective is to provide the opportunities to exercise linear interpolation in the most typical programming forms. In the exercises, the main emphasis is on learning the correct program syntax, correct using of the control modal features, and programming of a linear motion in both absolute and incremental modes.

### Overview and Applications

Linear interpolation is defined as the shortest motion between two points, programmed at a specified feedrate. It is programmed with the G01 command, in one, two, or three axes. The minimum and maximum programmable feedrate for each axis is determined by the control manufacturer and cannot be changed. Linear interpolation is used for cutting straight (linear) motions.

The programming format for linear interpolation is:

G01 X.. F..	for a single X axis motion
G01 Y.. F..	for a single Y axis motion
G01 Z.. F..	for a single Z axis motion
G01 X.. Y.. F..	for a two axis motion in X and Y
G01 X.. Z.. F..	for a two axis motion in X and Z
G01 Y.. Z.. F..	for a two axis motion in Y and Z

Typically, a linear interpolation is programmed after a rapid motion to the location where the straight cut is to take place. It is also programmed when a contour cutting motion changes from a circular motion to a linear motion. Both, the G01 command and the programmed feedrate command *F* are modal.

Linear interpolation can be canceled by any other motion command (G00, G02, G03), but the last active feedrate will remain active, in memory. If the feedrate specification is missing in the program, the last active feedrate will be used. If there were no previously active feedrate, the control system issues an alarm (error) and the program has to be corrected at the control unit.

Two feedrate modes are normally used for linear interpolation - one for milling, the other for turning:

❑ Milling feedrate	... units per time	mm/min or in/min
❑ Turning feedrate	... units per revolution	mm/rev or in/rev

In a simultaneous linear motion in two or three axes, the control system automatically calculates actual feedrate for each axis, in order to produce a straight motion.

A linear motion G01 can be programmed in absolute or incremental mode, using the G90 and G91 commands respectively. The CNC operator can control the actual cutting feedrate at the machine, using the *feedrate override* switch. This switch is located on the machine operation panel and can be used to increase or decrease the programmed feedrate, usually from 0% to 200%. When the program is optimized, the best feedrate should be reflected in the master program, which usually requires some minor changes to the program. To temporarily stop the linear motion during a machining operation, the *feedhold* button is used.

The two simple examples on the next page show two versions of programs using linear interpolation. The first example - *Example 1* - shows the most rudimentary form, while the *Example 2* is an actual program, albeit a simple one. Note the format and the modal entries.

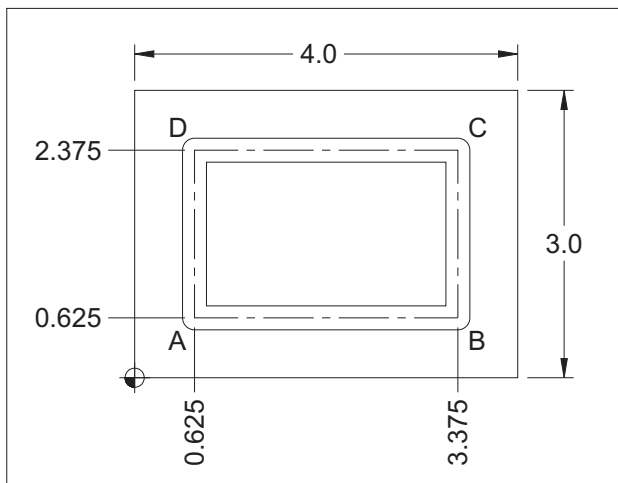
### ➡ Example - 1

The following four blocks use linear interpolation in incremental mode, to cut a square 50 x 50 mm, based on the program zero at the lower left corner. Only critical motions are shown in this example:

```
G21
G91 G01 Y50.0 F100.0
X50.0
Y0
X0
```

### ➡ Example - 2

The second example is a simple, yet complete program for one tool ( $\varnothing 0.25$  end mill). The groove is programmed at 0.125 depth. In the program, note the use of linear interpolation along the Z axis for the plunge-in motion.



```
N1 G20
N2 G17 G40 G80
  (IN BLOCK N3 MOVE TO LOCATION A ...>>)
N3 G90 G54 G00 X0.625 Y0.625 S2000 M03
N4 G43 Z0.1 H01 M08
N5 G01 Z-0.125 F3.0
N6 X3.375 F5.0                (LOCATION B)
N7 Y2.375                    (LOCATION C)
N8 X0.625                    (LOCATION D)
N9 Y0.625                    (LOCATION A)
N10 G00 Z0.1 M09
N11 G28 Z0.1
N12 M30
%
```

In the example above, the tool moves first to location A (block N3), then cuts the groove in the direction B (N6), C (N7), D (N8), and back to location A (N9). Specified cutting depth is listed in block N5 (Z-0.125).

The main application of linear interpolation is in contour based tool motions. Machining operations such as contouring, face milling, slot cutting and pocketing use this type of motion either exclusively, or combined with another type of motions - such as circular interpolation and rapid motion.

Straight tool motions are also used internally by the system, within various fixed cycles for hole operations. The fixed cycle does not use the linear interpolation command G01 directly, as it is built in the cycle internally.

In a true three dimensional applications (known as 3D), when machining a surface, a circular interpolation cannot be used, and all motions consist of very small linear motions.

In summary, the linear interpolation mode G01 is used in a CNC program for the sole purpose of connecting any two points with a straight line (the shortest distance between points).

## **References**

- 📖 *CNC Programming Handbook - Chapter 11 Absolute and Incremental Modes*
- Chapter 13 *Feedrate Control*
- Chapter 13 *Feedhold and Override*
- Chapter 22 *Linear Interpolation*