

## **Objectives**

Any CNC program depends on the correct input of dimensional data, found in the part drawing. The purpose of the projects and exercises for subjects described in this chapter, is to enable the reader to understand and apply these basic elements of programming.

## **Overview and Applications**

Inputting dimensions in a CNC program means the correct way of writing - inputting - dimensional data, generally found in the part drawing. This does not mean just the linear or radial dimensions, it also means the use of proper units (metric or English) and the proper reference of dimensions (absolute or incremental). Many addresses in the program are affected by the proper selection, such as cutting feedrates, offset input values, and various screen position displays.

In terms of units, metric units are selected by the G21 command, English units are selected by the G20 command. Absolute dimensions are set by the G90 command, incremental dimensions by the G91 command. Selecting one from each option and using it in the program will establish the mode of operation. Each command in the pair cancels the other command. Although it is quite common to switch from absolute to incremental in the same program, switching between metric and English units in the same program is definitely *not* recommended.

As a rule, the absolute mode of dimensional input is preferred, unless there is a need to use the incremental mode, for example, in subprograms. Absolute mode indicates the location from part origin, incremental mode indicates the actual distance and direction from the current location. The XYZ data must correspond to the mode selected.

There are differences between milling and turning programs, mainly in two features. One is that on most CNC lathes, the X axis represents the diameter, and the feedrate is measured per revolution, rather than per time, which is typical for the milling systems.

Another important aspect of a dimensional input is the decimal point. Early machine controls did not use decimal point at all, and all modern controls *convert* any dimensional input without a decimal point. How each control interprets the dimension without a decimal point depends on the units used and the method of zero suppression. Knowledge of these features is very important in programming.

When the decimal point is used in the program, which is the common method, the number of decimal places are determined by the units selection, as well as the minimum and maximum amount of increment.

Correct using of dimensional data in a part program is one of the most elementary requirements of a CNC programmer and operator.

## **Recommended Resources**



CNC Programming Handbook - Chapter 11 *Input of Dimensions*  
Chapter 13 *Feedrate Function*  
Chapter 7 *Format Notation*  
Chapter 8 *Preparatory Commands*