

### Objectives

In this section, the projects and exercises focus at one main subject - subprograms. Subprograms are considered an intermediate subject in many programming courses, but can be both very easy or quite sophisticated and very advanced. In order to understand the concept of subprograms, it is necessary to have a prior knowledge of some key elements of CNC programming. Knowledge of the program structure is essential. Common commands, such as the G90 and G91 (absolute / incremental program modes), play an important role in subprograms. Understanding the fixed cycles is also important, as many subprograms involve programming various operations on holes.

The exercises included in this section offer a variety of programming approaches and problem solutions.

### Overview and Applications

Manual programming can be very time consuming. Yet, there are many programming methods that can shorten the overall program length. One such method is using subprograms.

Subprograms are separate programs, stored under their own program number, that contain any point locations and/or tool motions that repeat more than once in the program. In short, subprograms contain *repetitive* data. Instead of writing the same data over and over again, we write it only once, and call it in the program as many times as necessary. The main application of subprograms is to program any repetitive toolpath pattern once, and use it many times.

The subprogram number uses the same format as any regular program number - four or five digits, preceded with the letter *O*, depending on the control system. The repetitive toolpath - the subprogram - is called by using the M98 function. In order to distinguish the end of the subprogram from the end of the main program, two miscellaneous function are used:

- M30      Main program end
- M99      Subprogram end

A typical example of a subprogram call is:

**M98 Pxxxx    or    M98 Pxxxx Lxx**

 where ...

M98 Pxxxx    ... calls the previously stored subprogram  
Lxx (or Kxx)    ... is the number of repetitions

Use letter *O* to store all programs! Some CNC units use a format without the *L/K* repeat command, for example:

**M98 P1234 L5**

may be programmed as

**M98 P051234**


It is always advisable to check the control unit manual for the exact program specifications. Any subprogram can be called from any main program, or even from another subprogram, within certain limits. Many subprogram applications benefit from either absolute or incremental mode of programming. Typically, use the absolute mode for subprograms if there are several operations at the same location. Use the incremental mode for subprograms, if the stored toolpath is repeated at different locations.

One common characteristic of any subprogram call is that any called subprogram will always return to the program it was called from. Most control systems allow subprogram nesting up to four levels deep. That means one subprogram can call another subprogram, which can call another subprogram, and so on.

Although subprograms are used much more in milling applications than in turning, there are some excellent possibilities for subprograms applied to CNC lathe. For example, a groove can be programmed only once, then repeated at different location, using a common subprogram. Many subprograms have been designed to do no machining at all - for example, they can be used for tool positioning, tool change location, part removal position and many other applications.

Overall, subprograms provide an excellent way not only to shorten the length of a part program, but also provide an easier way to modify programs at the CNC machine, if necessary.

## **References**

-  *CNC Programming Handbook* - Chapter 39 *Subprograms*
  - Chapter 7 *Program Structure*
  - Chapter 11 *Absolute and Incremental Modes*
  - Chapter 25 *Fixed Cycles*
  - Chapter 25 *Using L0/K0 in a fixed cycle*
  - Chapter 27 *Bolt Hole Circle Pattern*
  - Chapter 33 *Circular Pockets*
  - Chapter 40 *Local Coordinate System - G52*