

Objectives

The focus of the previous chapter (*Fixed Cycles*) had been on the concept of programming holes and hole operations in an efficient way. The purpose of fixed cycles is to simplify programming and make changes easier, if necessary. The objective of projects presented in this section is to apply the fixed cycles in practical shop applications.

Applications

Machining holes is quite likely the most common operation in majority of machine shops. Unlike milling or turning, where the toolpath is unique to a particular job, machining holes is a point-to-point (positioning) operation, where the action of a particular tool is always the same. What changes between operations or even between jobs, are the hole and machining parameters.

The applications for machining holes are numerous. Typical operations will include spot drilling, center drilling, drilling, boring, counterboring, countersinking, reaming, and tapping. Good knowledge of these tools is essential for successful program development.

Overview

Fixed cycles provide the programming template, but the CNC programmer still has to do a lot of work. In addition to the general technological decisions, the main emphasis in programming holes is on calculating the R-level and the Z-depth. As the R-level is only a clearance (albeit an important one), it usually presents no difficulties. On the other hand, the Z-depth calculation depends on many factors, such as the drawing requirements, material thickness, tool type, tool diameter and cutting angle, and many others. The most common formula for calculating the drill point length P for 118° drill point angle is:


$$P = D \times K$$

 ... where D is the effective cutting diameter and K is the constant (*see chart on page 196*).

This is just one of many programming tools available. Calculating a spot drill depth or a center drill depth presents other challenges. Countersinking, for example, may require trigonometry, to calculate the Z-depth. Tapping requires not only a proper tool selection but properly calculated feedrate. Some hole operations require care in setup, for example, boring with a shift or back boring.

The following projects have been designed to address many important aspects of machining holes and provide a starting point for gaining experience.

References

-  *CNC Programming Handbook* - Chapter 25 *Fixed Cycles*
Chapter 26 *Machining Holes*
Chapter 27 *Pattern of Holes*