8040/55M CANNED CYCLE PARAMETERS

Complex deep hole drilling	format:\	**G69** G98/G99 X Y Z I B C D H J K L R
Drilling canned Cycle		**G81** G98/G99 X Y Z I K
Drilling canned Cycle with dwell		**G82** G98/G99 X Y Z I K
Simple deep hole drilling		**G83** G98/G99 X Y Z I J
Tapping canned cycle		**G84** G98/G99 X Y Z I K R
Reaming canned cycle		**G85** G98/G99 X Y Z I K
Boring cycle with withdrawal in rapid		**G86** G98/G99 X Y Z I K
Rectangular pocket canned cycle		**G87** G98/G99 X Y Z I J K B C D H L V
Circular pocket canned cycle		**G88** G98/G99 X Y Z I J B C D H L V
Boring cycle with withdrawal at feedrate		**G89** G98/G99 X Y Z I K

G8? or G69 G98/G99 X Y Z I K have the standard meaning for all the cycles.
I= total depth ***exception G83, I = drilling step
K= dwell time * exception G87 K= distance from center to edge of pocket**

Lets go over the other parameters for each cycle:

B: G69,G87,G88---- drilling step in axis longitudinal to the main plane
C: G69---------- distance from the previous drilling step
G87, G88--------- milling pass along the main plane
D: G69,G87, G88--- distance between the reference and surface of the part
H: G69 ---------- distance the axis will withdraw after each drilling step
G87,G88--------- the feedrate for the finishing pass
J: G69---------- how many drilling steps the tool withdraws to reference plane
G83 ---------- number of steps which the drill is to make
G87 ---------- distance from center to the edge of the pocket
G88---------- radius of the pocket
L: G69---------- minimum value which the drilling step can acquire
G87, G88 -------- finishing pass along the main plane
R: G69 ---------- factor which reduces the drilling step "B"
G84 ---------- defines the type of tapping: normal or rigid
V: G87,G88----------defines tool penetrating feedrate

Canned Cycle Area of influence:
Once a canned cycle has been defined it remains active until cancelled. In other words every time a block has some axis movement programmed, the machining operation of the canned cycle is active also. If you program N at the end of the block, the CNC repeats the programmed move and the machining operation.

G98:withdrawal of tool as far as the initial plane
G99:withdrawal of tool as far as the reference plane
G80: cancels the canned cycle
G79: allows modifications of parameters within the canned cycle. programmed alone in a block

General Considerations:

a) a canned cycle can be defined at any point in a program.
b) calls to subroutines can be made while the canned cycle is active
c) execution of a canned cycle will not alter the history of previous G functions
d) a canned cycle can be entered with M3 or M4. if no direction is entered, CNC assumes M3
e) execution of a canned cycle cancels radius compensation. equivalent to G40
f) tool length compensation must be programmed in the before the canned cycle or same block

MULTIPLE MACHINING

Multiple machining in a straight line pattern	**G60** A (XI)or(XK)or(IK) P Q R S T U V
Multiple machining in a rectangular pattern	**G61** A B (XIor XK or IK) (YJ orYD or JD) P Q R S T U V
Multiple machining in a grid pattern	**G62** A B (XIorXK or IK) (YJorYD orJD) P Q R S T U V
Multiple machining in a circular pattern	**G63** X Y (I or K) C F P Q R S T U V
Multiple machining in a arc pattern	**G64** X Y B (I or K) C F P Q R S T U V
Multiple machining by means of an arc chord	**G65** X Y (A or I) C F

A:---------- defines the angle which forms the machining pass with the abscissa axis
B:------- G61----- defines the angle formed by two machining paths.
G64 ---- defines the angular stroke of the machining path
C:------- indicates how the movement is made between machining points 0=G0,1=G1,2=G2, 3=G3
D:---------- number of operations in the ordinate axis
F:---------- feedrate used between points
I:-------- G60,G61,G62 -- pitch between machining operations
G63,G64--------- pitch angle between machining operations
G65---------- defines the chord length
J:-------- pitch between machining operations according to the ordinate axis
K:------- number of total machining operations
PQRSTUV:----------points where it is not required to machine
X:------- G61,G62 -------- length of the machining path
G63,G64,G65 -- distance from the starting point to center along the abscissa axis
Y:------- G61,G62--------- pitch between machining operations according to the ordinate axis
G63,G64,G65--- distance from the starting point to center along the ordinate axis

General considerations:

Multiple functions are defined as a series of functions which allow a machining operation to be repeated along a given path.

The programmer will select the type of machining which can be a canned cycle or a modal subroutine.

These functions must be defined every time they are used. These functions will only make sense if they are under the influence of a canned cycle or modal subroutine

The following operations will be done under the same working conditions defined by the canned cycle

To perform multiple machining, follow these steps:

1) move the tool to the first point of the multiple machining operation
2) define the canned cycle or modal subroutine to be repeated at all the points
3) defined the multiple operation to be performed