Program for CNC Keyway Broaching

For TORMACH CNC MILL

Try using this format to reduce programming time.
Just enter your broach information in the fields shown.

```
%
O1234 ( USE A WORK OFFSET FOR BROACH TOOL BY IT SELF)
(YOU CAN ADJUST THE LOCATION IN X OR Y DEPENDING ON KEYWAY
LOCATION)
(ALSO THE CODE IS WRITTEN FROM CENTER OF HOLE)
(THIS SAMPLE IS WRITTEN TO CUT KEY WAY AT 12 O ' CLOCK)
(THIS IS 2  FILES ONE IS MAIN PROGRAM ONE IS SUB PROGRAM)
G91 G28 Z0

T__ M06 (ENTER YOUR BROACH TOOL # HERE)

G00 G90 G59 X0 Y0
M05
M19
G43 H__ Z1.0  (ENTER HEIGHT OFF-SET NUMBER HERE)
Z0.5
M08

( *** 1ST SET OF DIGITS # MUST BE THE TOTAL DEPTH OF KEY WAY DIVIDED
BY THE D.O.C.)
(IE... IF KEYWAY IS .130 DEEP AND D.O.C. IS .001 THEN = 130 )
(.130 / .001 = 130 )
( 1ST SET OF DIGITS = TOTAL DEPTH / D.O.C. )

M98 P***7777  ( *** ENTER 1ST SET OF DIGITS HERE NO SPACE NO DECIMAL)
(M98 P1307777)

G00 G90  Z1.0 M09
G91 G28 Z0
G28 Y0
M30
%


%
O7777 (BROACH SUB)

G00 G91 Y.001 (*** YOU CAN CHANGE TO X IF KEYWAY IS IN THAT DIRECTION)

G01 G90 Z_____ F_____ (END POINT -Z OF BROACH & FEED RATE IPM)

G00 G91 Y -.300 (*** THIS MUST BE SAME LETTER AS PREVIOUS MOVE X OR Y )
(*** MUST BE OPPOSITE SIGN TOO + OR -)
(VALUE MUST BE LARGER THAN TOTAL DEPTH OF KEY WAY )

G90 Z.5

G91 Y.300 (*** THIS VALUE MUST MATCH PREVIOUS LETTER AND # )
(BUT IT WILL BE OPPOSITE SIGN + OR - )

M99
%
```