Program for CNC Keyway Broaching

For OKUMA CNC MILL

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

```
%
BROACH MAIN .MIN

( 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)

G00 Z20.
T__ M06 (ENTER YOUR BROACH TOOL # HERE)

G15 H59
G00 G90 X0 Y0
M05
M19
G56 H__ Z1.0  (ENTER HEIGHT OFF-SET NUMBER HERE)
Z0.5
M08

(THE Q NUMBER  # 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 )
( Q = TOTAL DEPTH / D.O.C. )

N001 PSELECT SUB.MIN Q130 ( *** ENTER Q DIGITS HERE NO SPACE
NO DECIMAL)

G00 G90  Z20. M09
G28 Y20.
M30
%


%
SUB.MIN

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 - )

GOTO N001
%
```