Program for CNC Keyway Broaching

For MAZAK CNC MILL

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

THE MACHINE MUST BE IN EIA MODE
& THE SUB PROGRAM IS A SEPERATE
FILE BY IT SELF.

(You must enter the values for your tool and part information correctly)

```
%
O0001 (MAZAK CNC LATHE BROACH SAMPLE)
G28 U0 W0

T0101 (ENTER BROACH TOOL NUMBER HERE, IE.. T5 =T0505)

M05
M19

G00 X1.0 Z.5 (X= BORE SIZE, Z=STARTING POSITION OF TOOL)

M08

(NO DECIMAL ON L)
M98 P100 L130 (L= NUMBER OF PASSES MUST BE DIVISIBLE BY D.O.C. )

G00 G99 Z1.0 M09
G28 U0 W0
M30
%


%
O100 (SUB PROGRAM)
G00 U.001 ( U IS D.O.C.)

G01 G98 Z-1.0 F100. ( Z = END OF TOOL POSITION, F = FEED IMP )
G00 U-.250
Z.5
U.250
M99
%
```