

Program for CNC Keyway Broaching

For HAAS CNC 2 spindle 4 axis lathe

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

```
%
O1234 (BROACH IN Y AXIS HAAS 2 SPINDLE 4 AXIS LATHE)

(TOOL WAS SET AS CENTERLINE X0 LIKE A DRILL)
(AND CUTTING EDGE OF TOOL TO Y0 CENTER OF HOLE)

G00 G53 X0 Y0 Z-11.

T0101 (ENTER YOUR BROACH TOOL # HERE)

M05
M155 (OR THE ENGAGE C AXIS M CODE)

G00 C0 (ENTER THE ANGLE OF YOUR KEY WAY HERE)

M14 (OR LOCK C AXIS M CODE)

G00 X0 Z.5 Y.552 ( Y IS THE RADIAL VALUE)
(OFF THE START OF KEY WAY FROM CENTER OF HOLE)

M08

M97 P1 L133 (L IS NUMBER OF PASSES NO DECIMAL)
G00 G99 Z1.0 M09
G00 G53 Y0
G53 X0 Z-11.
M30

(NOTE IS YOU CHANGE DEPTH OF CUT THE L MUST BE A COMON
FACTOR)
(IE.. DEPTH OF KEY WAY IS .133 L RUNS SUB 133 TIMES EACH PASS
IS V.001)
(.001 X 133 = RADIAL DEPTH OF KEY WAY .133)

N1 G00 V.001 (V IS THE DEPTH OF CUT)

(THE FOLLOWING Z VALUE)
( IS THE DISTANCE THE TOOL WILL GO IN TO HOLE)
G01 G98 Z-3.0 F250. (THIS IS THE FEED RATE IPM YOU CAN CHANGE)

G00 V-.3 (MAKE SURE THIS V IS LARGER THAN THE TOTAL DEPTH OF
KEYWAY)
Z.5
V.3 (THIS V MUST MATCH THE PREVIOUS V BUT + NOT -)
M99
%
```

