

Program for CNC Keyway Broaching

For FANUC CNC 2 axis lathe

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

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

%
O0001 (CNC BROACH PROGRAM) ;
(FANUC) ;

(YOU MUST ENTER THE VALUES BELOW FOR YOUR TOOL AND PART) ;

G00 G18 G20 G40 G54 G64 G80 G99 ;
G28 U0 W0 ;

T_____ (ENTER TOOL # HERE EXAMPLE IF TOOL # IS 5 THEN T0505) ;

M05 ;

M19 ;

G00 X_____ Z_____ (X & Z STARTING POSITION) ;

M08 ;

G98 ;

(ENTER VALUES IN BOTH G76 LINES BELOW) ;

(1ST LINE VALUES) ;

(Q=DEPTH OF CUT NO DECIMAL 10 = .001 & 20 =.002 R=FINISH NO DECIMAL) ;

(2ND LINE VALUES) ;

(X=FINAL POSITION Z=FINAL POSITION Q= DEPTH 1ST PASS NO DECIMAL) ;

(P= KEY WAY DEPTH NO DECIMAL 1000 = .100 & 2000=.200 F= FEED RATE IPM) ;

G76 P000000 Q_____ R_____ ;

G76 X_____ Z_____ Q _____ P_____ F_____ ;

G00 Z1.000 M09 ;

G28 U0 W0 ;

M30 ;

%

