

Hexagon Socket Programming Code Examples from Machine Builders in Metric

Hex socket size : Hex 3.0mm, AF(Final "X" position) 3.46mm, Depth 3.5mm
Pilot drill diameter : 3.0mm **Starting "X" position :** 2.95mm (see chart on W3)
Insert : SSP030N1940N TM4
Parameters : Feed 3000mm/min(120 IPM), DOC(Roughing) 0.025mm, (Finishing) 0.005mm

Programming tips

● **Make a program considering final " X "position.**

- #1 Final "X" position : 3.46mm(AF)
- #2 Finishing position of roughing : $3.46 - 0.01$ (Finishing) = 3.45mm
- #3 Calculate total DOC for roughing : $3.45 - 3.0$ (Pilot hole) = 0.45mm
- #4 Determine number of cuts : $0.45 \div 0.05$ (DOC for Dia.) = 9.0 + 2 (round down to whole number and add "2" for program adjustment)
 → Roughing sequence runs 11 times
- #5 Set starting point : $3.45 - (0.05 \times (11 - 1)) = 2.95$ mm : must subtract by "1" for program adjustment

CITIZEN

Main Program Sequence

```
M25
M78 S0 .....I
Shaper T****
G50 U1.6 .....II
G0 X2.95 Z-2.0 T** .....III
M98 P2100 L11 .....IV
M98 P2200 .....V
```

```
M78 S60 .....I
G0 X2.95 Z-2.0
M98 P2100 L11
M98 P2200 } <a>
```

Repeat <a> program sequence 4 more times to complete the cuts at S120, S180, S240, S300 (represents 120°, 180°, 240°, 300°).

```
M20
G0 Z-2.0
G50 U-1.6
G0 U0 W0 T0
M1
```

STAR

Main Program Sequence

```
M25
Shaper T****
G50 U1.6 .....II
M8
G0 X2.95 Z-2.0 C0 T** .....I, III
M98 P2100 L11 .....IV
M98 P2200 .....V
```

```
G0 C60.0 .....I
G0 X2.95 Z-2.0
M98 P2100 L11
M98 P2200 } <a>
```

Repeat <a> program sequence 4 more times to complete the cuts at C120.0, C180.0, C240.0, C300.0 (represents 120°, 180°, 240°, 300°).

```
G0 Z-2.0
G50 U-1.6
G0 T0
G28 W0
M1
```

TSUGAMI

Main Program Sequence

```
M105
M150
G28 H0 .....I
M182
Shaper T****
G50 U1.6 .....II
G0 X2.95 Z2.0 T** .....III
M98 P2100 L11 .....IV
M98 P2200 .....V
M183
```

```
G0 C60 .....I
M182
G0 X2.95 Z2.0
M98 P2100 L11
M98 P2200
M183 } <a>
```

Repeat <a> program sequence 4 more times to complete the cuts at C120, C180, C240, C300 (represents 120°, 180°, 240°, 300°).

```
M151
G0 Z2.0
G50 U-1.6
G0 U0 W0 T0
M1
```

Sub-Program Sequence #1 for Roughing

```
N2100
G4 U0.02 .....A
G98 G1 Z3.5 F3000 .....B
G4 U0.02
U-0.2 W-0.018 .....C
G4 U0.02
G0 Z-2.0
G4 U0.02
U0.25 .....D
M99
```

Sub-Program Sequence #1 for Roughing

```
O2100
G4 U0.02 .....A
G98 G1 Z3.5 F3000 .....B
G4 U0.02
U-0.2 W-0.018 .....C
G4 U0.02
G0 Z-2.0
G4 U0.02
U0.25 .....D
M99
```

Sub-Program Sequence #1 for Roughing

```
O2100
G4 U0.02 .....A
G98 G1 Z-3.5 F3000 .....B
G4 U0.02
U-0.2 W0.018 .....C
G4 U0.02
G0 Z2.0
G4 U0.02
U0.25 .....D
M99
```

Sub-Program Sequence #2 for Finishing

```
N2200
G98 G1 X3.46 Z-2.0 F1000 .....E
G4 U0.02
Z3.5 F3000
G4 U0.02
U-0.2 W-0.018
G4 U0.02
G0 Z-2.0
M99
```

Sub-Program Sequence #2 for Finishing

```
O2200
G98 G1 X3.46 Z-2.0 F1000 .....E
G4 U0.02
Z3.5 F3000
G4 U0.02
U-0.2 W-0.018
G4 U0.02
G0 Z-2.0
M99
```

Sub-Program Sequence #2 for Finishing

```
O2200
G98 G1 X3.46 Z2.0 F1000 .....E
G4 U0.02
Z-3.5 F3000
G4 U0.02
U-0.2 W0.018
G4 U0.02
G0 Z2.0
M99
```

- I. Index the sub-spindle 6 times in 60 degree increments.
- II. Specify the coordinate system shift command (in X axis direction) for the tool. [2 x f, where f is tool dimension located in catalog].
 - A positive direction shift is recommended for easier programming.
- III. Execute the positioning of the tool.
 - X position should be smaller than pilot drill diameter.
 - Z position should be offset 2.0 mm from material to achieve program feed rate.
- IV. Go to the Sub-Program #1.
 - Sequence runs 11 times. First cutting point X2.95 and final cutting point X3.45, with 0.05 DOC (for diameter) each time.
- V. Go to the Sub-Program #2, for finishing sequence.
- A. Specify dwell time. This allows the program and machine to stay synchronized.
- B. Cut into part 3.5mm. F3000 is recommended feed to be used for most materials; including Titanium Alloy and Stainless Steel.
- C. This code backs off the tool with an angle greater than 6 degrees (10 degrees used in example). See page W3.
- D. Return to the X position + 0.05mm (the DOC for diameter).
- E. Finishing operation with 0.005mm DOC (X 3.46) is recommended for better surface finish.

Hexalobular Socket Programming Code Examples from Machine Builders in Metric

Hexalobular socket size : Hexalobular T15 (depth : 3.81mm)

Pilot drill diameter : 2.3mm

Insert : SSP050N43T15 TM4

Parameters : Feed 3000mm/min(120 IPM), DOC(Roughing) 0.025mm, (Finishing) 0.005mm

Programming tips

● **Make a program considering final “ X ” position.**

- #1 Final “ X ” position : $3.35\text{mm}(A)$
- #2 Finishing position of roughing : $3.35 - 0.01$ (Finishing) = 3.34mm
- #3 Calculate total DOC for roughing : $3.34 - 2.3$ (Pilot hole) = 1.04mm
- #4 Determine number of cuts : $1.04 \div 0.05$ (DOC for Dia) = $20.8 + 2$ (round down to whole number and add “2” for program adjustment)
→ Roughing sequence runs 22 times
- #5 Set starting point : $3.34 - (0.05 \times (22 - 1)) = 2.29\text{mm}$: must subtract by “1” for program adjustment

CITIZEN

Main Program Sequence

```
M25
M78 S0 .....I
Shaper T****
G50 U4.8 .....II
G0 X2.29 Z-2.0 T** .....III
M98 P2100 L22 .....IV
M98 P2200 .....V
```

```
M78 S60 .....I
G0 X2.29 Z-2.0
M98 P2100 L22
M98 P2200 } <a>
```

Repeat <a> program sequence 4 more times to complete the cuts at S120, S180, S240, S300 (represents 120°, 180°, 240°, 300°).

```
M20
G0 Z-2.0
G50 U-4.8
G0 U0 W0 T0
M1
```

STAR

Main Program Sequence

```
M25
Shaper T****
G50 U4.8 .....II
M8
G0 X2.29 Z-2.0 C0 T** .....I, III
M98 P2100 L22 .....IV
M98 P2200 .....V
```

```
G0 C60.0 .....I
G0 X2.29 Z-2.0
M98 P2100 L22
M98 P2200 } <a>
```

Repeat <a> program sequence 4 more times to complete the cuts at C120.0, C180.0, C240.0, C300.0 (represents 120°, 180°, 240°, 300°).

```
G0 Z-2.0
G50 U-4.8
G0 T0
G28 W0
M1
```

TSUGAMI

Main Program Sequence

```
M105
M150
G28 H0 .....I
M182
Shaper T****
G50 U4.8 .....II
G0 X2.29 Z2.0 T** .....III
M98 P2100 L22 .....IV
M98 P2200 .....V
M183
```

```
G0 C60 .....I
M182
G0 X2.29 Z2.0
M98 P2100 L22
M98 P2200
M183 } <a>
```

Repeat <a> program sequence 4 more times to complete the cuts at C120, C180, C240, C300 (represents 120°, 180°, 240°, 300°).

```
M151
G0 Z2.0
G50 U-4.8
G0 U0 W0 T0
M1
```

Sub-Program Sequence #1 for Roughing

```
N2100
G4 U0.02 .....A
G98 G1 Z3.81 F3000 .....B
G4 U0.02
U-0.2 W-0.018 .....C
G4 U0.02
G0 Z-2.0
G4 U0.02
U0.25 .....D
M99
```

Sub-Program Sequence #1 for Roughing

```
O2100
G4 U0.02 .....A
G98 G1 Z3.81 F3000 .....B
G4 U0.02
U-0.2 W-0.018 .....C
G4 U0.02
G0 Z-2.0
G4 U0.02
U0.25 .....D
M99
```

Sub-Program Sequence #1 for Roughing

```
O2100
G4 U0.02 .....A
G98 G1 Z-3.81 F3000 .....B
G4 U0.02
U-0.2 W0.018 .....C
G4 U0.02
G0 Z2.0
G4 U0.02
U0.25 .....D
M99
```

Sub-Program Sequence #2 for Finishing

```
N2200
G98 G1 X3.35 Z-2.0 F1000 .....E
G4 U0.02
Z3.81 F3000
G4 U0.02
U-0.2 W-0.018
G4 U0.02
G0 Z-2.0
M99
```

Sub-Program Sequence #2 for Finishing

```
O2200
G98 G1 X3.35 Z-2.0 F1000 .....E
G4 U0.02
Z3.81 F3000
G4 U0.02
U-0.2 W-0.018
G4 U0.02
G0 Z-2.0
M99
```

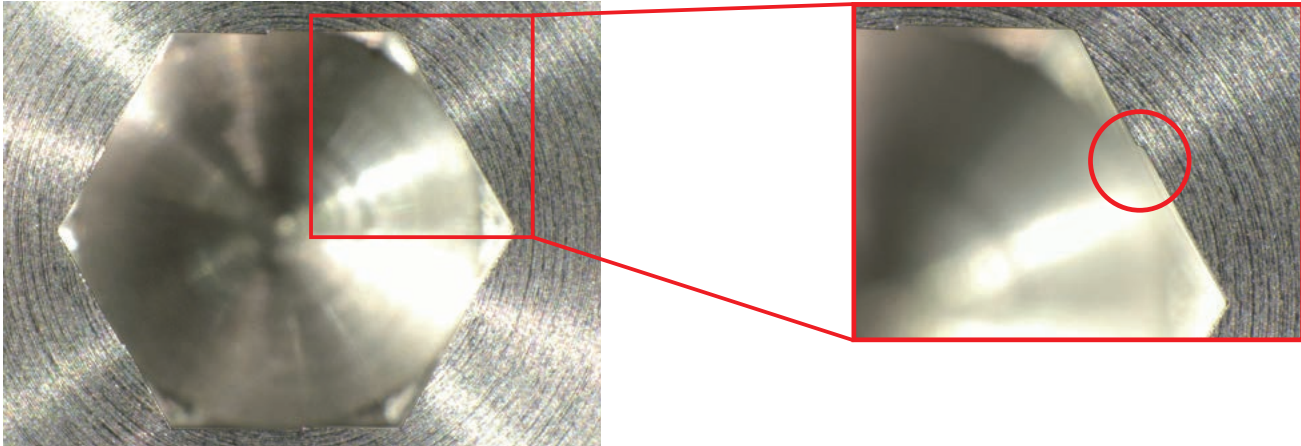
Sub-Program Sequence #2 for Finishing

```
O2200
G98 G1 X3.35 Z2.0 F1000 .....E
G4 U0.02
Z-3.81 F3000
G4 U0.02
U-0.2 W0.018
G4 U0.02
G0 Z2.0
M99
```

- I. Index the sub-spindle 6 times in 60 degree increments.
- II. Specify the coordinate system shift command (in X axis direction) for the tool. [2 x f, where f is tool dimension located in catalog].
 - A positive direction shift is recommended for easier programming.
- III. Execute the positioning of the tool.
 - X position should be smaller than pilot drill diameter.
 - Z position should be offset 2.0 mm from material to achieve program feed rate.
- IV. Go to the Sub-Program #1.
 - Sequence runs 22 times. First cutting point X2.29 and final cutting point X3.34, with 0.05 DOC (for diameter) each time.
- V. Go to the Sub-Program #2, for finishing sequence.
 - A. Specify dwell time. This allows the program and machine to stay synchronized.
 - B. Cut into part 3.81mm. F3000 is recommended feed to be used for most materials; including Titanium Alloy and Stainless Steel.
 - C. This code backs off the tool with an angle greater than 6 degrees (10 degrees used in example). See page W3.
 - D. Return to the X position + 0.05mm (the DOC for diameter).
 - E. Finishing operation with 0.005mm DOC (X 3.35) is recommended for better surface finish.

SHAPER DUO Troubleshooting

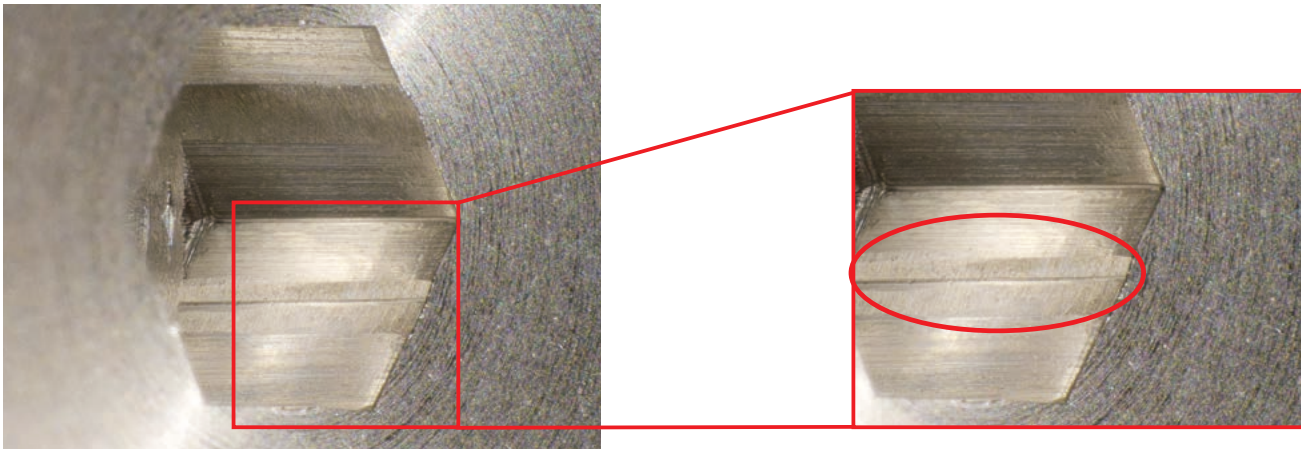
■ Problem: Step on sides



Cause: Incorrect tool set-up
(Center-line shift)

Solution: Machine one angle and make sure both [a] and [b] lengths are identical, rotating the sleeve if necessary

■ Problem: Wall dented



Cause: Pilot hole remaining

Solution: Need pilot hole tool's offset

■ Problem: Wall tapered

Solution: ● Smaller depth of cut
● Less tool overhang

■ Problem: Chuck is slipping / Insert chipped

Solution: ● Run at 3000 mm/min (120 IPM) feed rate
● Smaller depth of cut

- 3000 mm/min (120 IPM) feed rate can cover most materials including Titanium alloy and Stainless steel.
- Too slow or too fast of a feed rate may cause excessive tool pressure for the workpiece and tool.