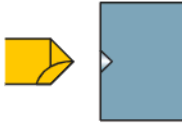


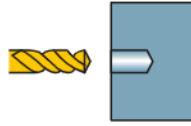
## Process Chart

### ① Center drilling



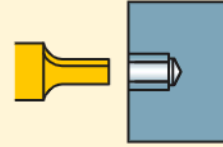
Make a center hole which is smaller than pilot hole drill.

### ② Drilling (Pilot hole)



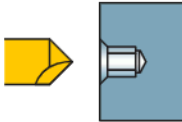
Select a drill with same or smaller (0~0.1mm) dia. as AF and machine a bit deeper because burrs may cause chipping on shaper insert

### ③ Shaper tool



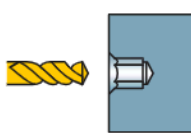
Machine socket rotating 60 degrees 6 times

### ④ Chamfering



Chamfer with the same pilot hole drill as ①

### ⑤ Deburring



Finish and deburr with the same drill as in process②  
☆Reduce cutting conditions due to heavy interruption

## SHAPER DUO Process Chart -Hexalobular-

Socket Size	Tool	Pilot bore Dia. (mm)	Starting "X" position (mm)	Number of passes			Estimated cycle time *		
				Final "X" position (mm)	Roughing pass 0.025mm	Finishing pass 0.005mm	ISO10664 Standard depth of Hexalobular hole (mm)	Whole process ①-⑤	Process④ Shaper
T6	SSP050N25T06	1.15	1.14	1.75	13	1	1.82	51 sec	23.2 sec
T7	SSP050N31T07	1.38	1.35	2.06	15	1	2.44	59 sec	28.2 sec
T8	SSP050N36T08	1.62	1.59	2.40	17	1	3.05	67 sec	33.8 sec
T10	SSP050N41T10	1.92	1.89	2.80	19	1	3.56	75 sec	39.5 sec
T15	SSP050N43T15	2.30	2.29	3.35	22	1	3.81	84 sec	46.2 sec
T20	SSP050N46T20	2.71	2.69	3.95	26	1	4.07	94 sec	55.4 sec
T25	SSP050N50T25	3.13	3.09	4.50	29	1	4.45	105 sec	63.8 sec
T27	SSP050N55T27	3.52	3.51	5.07	32	1	4.70	115 sec	71.8 sec
T30	SSP050N55T30	3.91	3.89	5.60	35	1	4.95	125 sec	80.2 sec

\* Using Carbide drill

\* Shaper cutting conditions

Feed : 3000 mm/min

DOC : 0.025 mm (Roughing), 0.005 mm (Finishing)

## SHAPER DUO Process Chart -Hexagonal-

HEX Standard	Tool	Pilot bore Dia. (mm)	Starting "X" position (mm)	Number of passes			Estimated cycle time *		
				Final "X" position (mm)	Roughing pass 0.025mm	Finishing pass 0.005mm	ISO 2936 standard depth of Hex hole (mm)	Whole process ①-⑤	Process④ Shaper
HEX 1.5	SSP020N1130H	1.5	1.47	1.73	6	1	2	39 sec	14 sec
HEX 2.0	SSP020N1430H	2.0	1.95	2.31	8	1	2.5	44 sec	16 sec
HEX 2.5	SSP030N1940H	2.5	2.48	2.89	9	1	3	50 sec	20 sec
HEX 3.0	SSP030N1940H	3.0	2.95	3.46	11	1	3.5	55 sec	23 sec
HEX 4.0	SSP040N2450H	4.0	3.96	4.62	14	1	5	73 sec	33 sec
HEX 5.0	SSP050N3260H	5.0	4.96	5.77	17	1	6	90 sec	46 sec
HEX 6.0	SSP060N42120H	6.0	5.97	6.93	20	1	8	117 sec	63 sec
HEX 8.0	SSP080N62160H	8.0	7.98	9.24	26	1	10	155 sec	92 sec

\* Pilot bore diameter is same or smaller(0-0.1mm) as AF.  
\* Using Carbide drill

\* Shaper cutting conditions

Feed : 3000 mm/min

DOC : 0.025 mm (Roughing), 0.005 mm (Finishing)

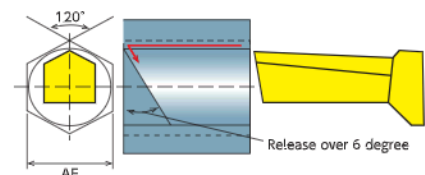
## Recommended Cutting Conditions

Feed : 3000 mm/min (120 IPM)

DOC : Roughing ... 0.025 mm (.0010") + Finishing ... 0.005 mm (.0002")

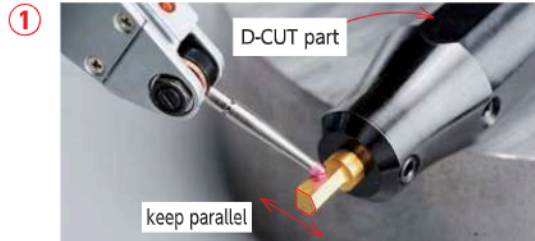
(Based on machining 303 stainless)

Program Example → 6 · 7



## SHAPER DUO Set-up Instructions -Hexagonal

### Outside machine

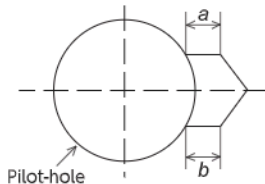


- Set the insert bar in the sleeve and check the parallelism of the flat portion of the sleeve and the insert bar.
- Minimize the overhang of the insert.

### Inside machine



- Set the sleeve into the tool post and make sure the sleeve is set parallel.
- Minimize sleeve overhang.



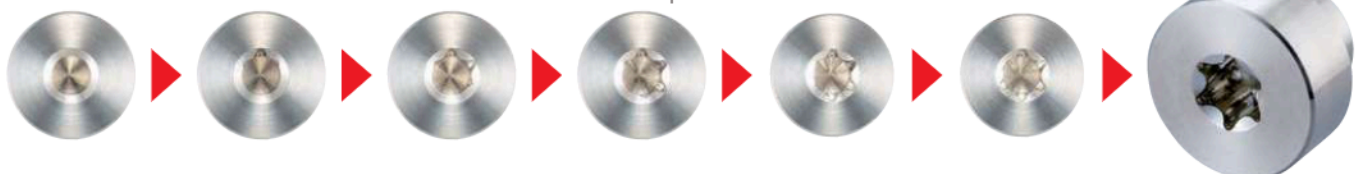
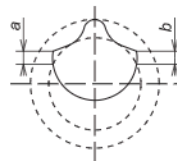
- Increase the number of machining passes with smaller depth of cut if the insert chips with large depth of cut. (0.025mm×5pass is recommended)  
No chamfering process is required for measuring purpose.
- Measure the length of both [a] and [b] with comparator or magnifier.
- Adjust centerline height by rotating the sleeve until you get the same length for [a] and [b]. (The difference should be less than .0008")  
\*If the straight is not seen with increased passes, please reset the insert and the sleeve.  
Please make sure both the insert and the sleeve are set up correctly.

### ④ Machine Hexagonal shape

- Run full HEX machining program.



### For Hexalobular machining Basically same as Hexagonal socket



## 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 :  $3.45 - (0.05 \times (11 - 1)) = 2.95\text{mm}$  : must subtract by "1" for program adjustment

#### CITIZEN

##### Main Program Sequence

```

M25
M78 S0 .....I
Shaper T****
G50 U1.5 .....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-1.5
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 #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
    
```

#### STAR

##### Main Program Sequence

```

M25
Shaper T****
G50 U1.5 .....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-1.5
G0 T0
G28 W0
M1
    
```

##### 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 #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
    
```

#### TSUGAMI

##### Main Program Sequence

```

M105
M150
G28 H0 .....I
M182
Shaper T****
G50 U1.5 .....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-1.5
G0 U0 W0 T0
M1
    
```

##### 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

```

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.

- 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).
- V. Go to the Sub-Program #2, for finishing sequence.
- E. Finishing operation with 0.005mm DOC (X 3.46) is recommended for better surface finish.

NOTE: Programming code update **G50 in red**

## 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 : 3.35mm(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 .....I
M98 P2100 L22 .....I
M98 P2200 .....I
```

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 .....I
M98 P2100 L22 .....I
M98 P2200 .....I
```

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 .....I
M98 P2100 L22 .....I
M98 P2200 .....I
M183
```

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.

- 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).
- V. Go to the Sub-Program #2, for finishing sequence.
- E. Finishing operation with 0.005mm DOC (X 3.35) is recommended for better surface finish.