# STANDARD CUTTING CONDITIONS

## DSW-DE (External supply)

| ISO | Workpiece material | Brinell hardness (HB) | Cutting speed: *Vc* (m/min) | | | Feed: *f* (mm/rev) | | |
|---|---|---|---|---|---|---|---|---|
| | | | ø3 ~ ø6 | ø6 ~ ø10 | ø10 ~ ø16 | ø3 ~ ø6 | ø6 ~ ø10 | ø10 ~ ø12 |
| P | Low carbon steels (C < 0.3)<br>SS400, SM490, S25C, etc.<br>C15E4, E275A, E355D, etc. | ~ 180 | 40 - 100 | 60 - 120 | 60 - 130 | 0.15 - 0.3 | 0.15 - 0.35 | 0.2 - 0.5 |
| | Carbon steels (C > 0.3)<br>S45C, S55C, , etc.<br>C45, C55, etc. | 180 ~ 300 | 40 - 90 | 50 - 120 | 60 - 130 | 0.15 - 0.3 | 0.15 - 0.35 | 0.2 - 0.4 |
| | High alloy steels<br>SCM440, etc.<br>42CrMo4, etc. | 250 ~ 350 | 40 - 80 | 50 - 100 | 50 - 100 | 0.1 - 0.2 | 0.15 - 0.3 | 0.15 - 0.35 |
| M | Stainless steels<br>SUS304, etc.<br>X5CrNi18-9, etc. | ~ 200 | 20 - 40 | 30 - 50 | 30 - 60 | 0.05 - 0.2 | 0.1 - 0.25 | 0.1 - 0.3 |
| K | Grey cast irons<br>FC300, etc.<br>250, etc. | ~ 200 | 40 - 90 | 50 - 95 | 50 - 100 | 0.15 - 0.3 | 0.2 - 0.4 | 0.2 - 0.5 |
| | Ductile cast irons<br>FCD450, etc.<br>450-10S, etc. | ~ 300 | 30 - 80 | 40 - 90 | 45 - 90 | 0.1 - 0.3 | 0.2 - 0.4 | 0.2 - 0.4 |
| N | Aluminium alloys<br>ADC12, etc.<br>AlSi11Cu3, etc. | - | 40 - 90 | 50 - 100 | 50 - 100 | 0.15 - 0.3 | 0.2 - 0.4 | 0.2 - 0.5 |
| S | Titanium alloys<br>Ti-6Al-4V, etc | - | 20 - 40 | 20 - 40 | 20 - 40 | 0.1 - 0.2 | 0.15 - 0.25 | 0.15 - 0.4 |
| | Heat-resistant alloys, Inconel<br>Inconel 718, etc. | 250 ~ | 10 - 30 | 10 - 30 | 10 - 30 | 0.03 - 0.07 | 0.05 - 0.1 | 0.07 - 0.12 |
| H | High hardened steels<br>SKD11, etc.<br>X153CrMoV12, etc. | ~ 40HRC | 20 - 40 | 20 - 40 | 20 - 40 | 0.05 - 0.15 | 0.05 - 0.15 | 0.05 - 0.2 |

- The cutting parameters shown in the table are merely a starting guideline for general machining. Values should be varied depending on the power or rigidity of the machine to be used. Optimum conditions should be selected depending on the actual chip control or damage on edges.
- When using the smaller diameter tools in each range, set the feed "*f*" to the lower recommended values.
- The coolant supply is critical for the provision of stable machining conditions and enhanced tool life. A large coolant volume should be supplied, especially when drilling difficult-to-cut materials.
- When drilling stainless steel with low machinability such as austenitic stainless steel with a depth deeper than L/D = 3, a pecking cycle or internal coolant supply is recommended.

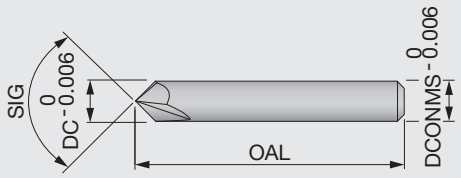
## DSW-DI (Internal supply)

| ISO | Workpiece material | Brinell hardness (HB) | Cutting speed: *Vc* (m/min) | | | Feed: *f* (mm/rev) | | |
|---|---|---|---|---|---|---|---|---|
| | | | ø3 ~ ø6 | ø6 ~ ø10 | ø10 ~ ø16 | ø3 ~ ø6 | ø6 ~ ø10 | ø10 ~ ø12 |
| P | Low carbon steels (C < 0.3)<br>SS400, SM490, S25C, etc.<br>C15E4, E275A, E355D, etc. | ~ 180 | 70 - 140 | 80 - 160 | 90 - 190 | 0.15 - 0.3 | 0.15 - 0.35 | 0.2 - 0.5 |
| | Carbon steels (C > 0.3)<br>S45C, S55C, , etc.<br>C45, C55, etc. | 180 ~ 300 | 50 - 130 | 70 - 160 | 80 - 170 | 0.15 - 0.3 | 0.15 - 0.35 | 0.2 - 0.4 |
| | High alloy steels<br>SCM440, etc.<br>42CrMo4, etc. | 250 ~ 350 | 40 - 100 | 60 - 140 | 60 - 160 | 0.1 - 0.2 | 0.15 - 0.3 | 0.15 - 0.35 |
| M | Stainless steels<br>SUS304, etc.<br>X5CrNi18-9, etc. | ~ 200 | 25 - 75 | 50 - 100 | 50 - 120 | 0.05 - 0.2 | 0.1 - 0.25 | 0.1 - 0.3 |
| K | Grey cast irons<br>FC300, etc.<br>250, etc. | ~ 200 | 80 - 140 | 100 - 160 | 100 - 180 | 0.15 - 0.3 | 0.2 - 0.4 | 0.2 - 0.5 |
| | Ductile cast irons<br>FCD450, etc.<br>450-10S, etc. | ~ 300 | 70 - 140 | 80 - 150 | 80 - 170 | 0.1 - 0.3 | 0.2 - 0.4 | 0.2 - 0.45 |
| N | Aluminium alloys<br>ADC12, etc.<br>AlSi11Cu3, etc. | - | 60 - 200 | 60 - 200 | 60 - 200 | 0.15 - 0.3 | 0.2 - 0.4 | 0.2 - 0.5 |
| S | Titanium alloys<br>Ti-6Al-4V, etc | - | 20 - 60 | 30 - 80 | 30 - 80 | 0.1 - 0.2 | 0.1 - 0.25 | 0.15 - 0.4 |
| | Heat-resistant alloys, Inconel<br>Inconel 718, etc. | 250 ~ | 10 - 30 | 10 - 40 | 10 - 40 | 0.03 - 0.07 | 0.05 - 0.1 | 0.07 - 0.15 |
| H | High hardened steels<br>SKD11, etc.<br>X153CrMoV12, etc. | ~ 40HRC | 20 - 50 | 30 - 60 | 30 - 60 | 0.05 - 0.15 | 0.05 - 0.15 | 0.05 - 0.2 |

- The cutting parameters shown in the table are merely a starting guideline for general machining. Values should be varied depending on the power or rigidity of the machine to be used. Optimum conditions should be selected depending on the actual chip control or damage on edges.
- When using the smaller diameter tools in each range, set the feed "*f*" to the lower recommended values.
- Oil holes that become blocked may cause drill breakages. A filter to prevent the circulation of chips must be used on the coolant supply system.

## DSM-CP

Centering drill for DSM drill




| Designation | DC | YH170 | DCONMS | OAL | SIG |
|---|---|---|---|---|---|
| DSM-CP90 | 3 | ● | 3 | 38.1 | 90° |
| DSM-CP140 | 3 | ● | 3 | 38.1 | 140° |

● : Line up

## STANDARD CUTTING CONDITIONS

### DSM

| ISO | Workpiece material | Hardness | Cutting speed: *Vc* (m/min) | | | Feed: *f* (mm/rev) | | | | |
|---|---|---|---|---|---|---|---|---|---|---|
| | | | ø0.1 - ø0.3 | ø0.31 - ø0.5 | ø0.51 - ø3 | ø0.1 - ø0.3 | ø0.31 - ø0.5 | ø0.51 - ø1 | ø1.01 - ø2 | ø2.01 - ø3 |
| P | Carbon steels, Alloy steels | - 300 HB | 5 - 20 | 15 - 30 | 25 - 60 | 0.001 - 0.004 | 0.002 - 0.01 | 0.005 - 0.05 | 0.03 - 0.09 | 0.05 - 0.1 |
| M | Stainless steels | - 200 HB | 2 - 12 | 6 - 18 | 10 - 20 | 0.0005 - 0.004 | 0.002 - 0.008 | 0.005 - 0.03 | 0.01 - 0.04 | 0.02 - 0.05 |
| K | Grey cast irons | 150 - 250 HB | 5 - 15 | 10 - 25 | 20 - 50 | 0.0005 - 0.004 | 0.002- 0.012 | 0.005 - 0.03 | 0.01 - 0.06 | 0.03 - 0.12 |
| | Ductile cast irons | 150 - 250 HB | 5 - 15 | 10 - 25 | 20 - 50 | 0.001 - 0.003 | 0.002 - 0.01 | 0.005 - 0.02 | 0.01 - 0.05 | 0.03 - 0.1 |
| N | Aluminium alloys | - | 10 - 20 | 10 - 30 | 20 - 50 | 0.001 - 0.01 | 0.005 - 0.03 | 0.01 - 0.05 | 0.04 - 0.15 | 0.06 - 0.2 |
| | Copper / Brass | - | 10 - 20 | 10 - 30 | 20 - 50 | 0.001 - 0.01 | 0.005 - 0.03 | 0.01 - 0.05 | 0.04 - 0.15 | 0.06 - 0.2 |
| S | Heat-resistant alloys | - 40 HRC | 2 - 6 | 5 - 10 | 8 - 20 | 0.0005 - 0.003 | 0.002 - 0.004 | 0.002 - 0.004 | 0.002 - 0.004 | ※ |
| H | High hardened steels | - 50 HRC | 4 - 8 | 6 - 10 | 6 - 16 | 0.0005 - 0.002 | 0.001 - 0.005 | 0.005 - 0.02 | 0.01 - 0.03 | 0.02 - 0.06 |

※ Not recommended

Notes: - When the drilling depth is deeper than L/D = 5, use drill pecking every 10 to 50% of the drill diameter.
- The above cutting conditions are applied to when a water soluble cutting fluid is used. For drilling a hole smaller than ø0.3 mm, use of a starting drill is recommended.
- When setting the drill, the drill runout should be within 0.002 mm on the taper. (Especially for the drill diameter smaller than ø0.5 mm)

### DSM-CP

| ISO | Workpiece material | Hardness | Cutting speed: *Vc* (m/min) | Feed: *f* (mm/rev) | |
|---|---|---|---|---|---|
| | | | | DSM-CP90 | DSM-CP140 |
| P | Carbon, Mild and Alloy steels | - 300 HB | 30 - 80 | 0.01 - 0.06 | 0.03 - 0.08 |
| M | Stainless steels | - 200 HB | 15 - 40 | 0.01 - 0.03 | 0.02 - 0.06 |
| K | Grey and ductile cast irons | 150 - 250 HB | 30 - 80 | 0.02 - 0.06 | 0.05 - 0.1 |
| N | Aluminium alloys | - | 60 - 120 | 0.02 - 0.1 | 0.05 - 0.15 |
| H | High hardened steels | - 45 HRC | 10 - 40 | ※ | 0.01 - 0.05 |

※ Not recommended

Notes: - Use DSM-CP140 for drilling hard materials and stainless steel that have work-hardening characteristic.
- The above cutting conditions are designed when using water-soluble cutting fluid, in which case, set the cutting speed to the lower side of the range.