Shank types: straight, morse taper, etc.

Flute form and web thickness

Surface treatment and coating

Point shapes: design, angles, thinning

Number of teeth: from 1 to 4

Tool material

Helix angle

Dimensions
WHICH HSS FOR MAXIMUM EFFICIENCY?

<table>
<thead>
<tr>
<th>HSS</th>
<th>HSS-E 5% cobalt</th>
<th>HSS-E 8% cobalt</th>
<th>HSS-PM (powder metallurgy)</th>
</tr>
</thead>
<tbody>
<tr>
<td>For conventional use</td>
<td>Basic choice for industrial applications</td>
<td>For drilling of difficult-to-machine materials</td>
<td>For high performance machining</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>Combines the performance of carbide with the toughness of HSS</td>
</tr>
</tbody>
</table>

**TOOL MAKER’S TIP**
Discover the performance of coated HSS-PM drills, especially when carbide drills are not successful.

**SUCCESS STORY**

- **Operation**
  - Drilling of holes Ø 8.25 mm, depth 80 mm
- **Tool**
  - Coated HSS-PM drill
- **Cutting data**
  - \( v_c \) 60 m/min, \( f_z \) 0.25 mm /rev.
- **Tool life**
  - More than double compared with carbide (812 holes vs. 375)
- **Cost per hole**
  - Divided by 2 compared with carbide

Grey cast iron
### Tool Maker's Tip

Chip removal is easier with coated drills, due to lower friction and increased cutting data. DLC coatings can also be used to drill non-ferrous materials.

<table>
<thead>
<tr>
<th>Surface Treatment</th>
<th>Description</th>
</tr>
</thead>
</table>
| Steam oxide       | Popular surface treatment  
                   | For ferrous materials only|
| Nitride           | Seldom used  
                   | For cast iron and aluminium |
| TiN Gold          | Conventional multipurpose coating  
                   | Cost efficient  
                   | Medium performance |
| TiCN Grey-violet  | High wear resistance  
                   | For steels |
| TiAlN or TiAlCN Black-violet | High performance multipurpose coating, for higher cutting speeds  
                   | For ferrous alloys (steels, cast iron), hard or abrasive materials  
                   | Suitable for dry machining |
| MoS₂ or WC-C Grey-black | Good anti-welding properties, reduces friction  
                   | Used in combination with other coatings  
                   | Suitable for dry machining |
A DRILL AROUND THE WORLD
French: un foret
German: ein Bohrer
Italian: una punta
Spanish: una broca

<table>
<thead>
<tr>
<th>Margin width</th>
<th>Margin</th>
<th>Lip relief width</th>
<th>Body diameter clearance</th>
<th>Radial point thinning</th>
<th>Body clearance</th>
<th>Flute width</th>
</tr>
</thead>
<tbody>
<tr>
<td>Lip relief</td>
<td>Core</td>
<td>Web</td>
<td>Flute run-out</td>
<td>Overall length</td>
<td>Shank length</td>
<td>Lip relief angle</td>
</tr>
<tr>
<td>Radial point thinning</td>
<td>Drill diameter</td>
<td>Point angle</td>
<td>Helix angle</td>
<td>130°</td>
<td>Radial point thinning</td>
<td>Shank diameter</td>
</tr>
</tbody>
</table>
Flute length is one of the most critical determinants of tool life: for longer tool life, the flute length should be as short as possible. Longer flute length results in a lower rigidity of the drill, which causes unstable drilling.

In most operations, the flute length can be calculated as follows:
- Depth of hole
- + bush length
- + distance between bush and workpiece
- + 2x diameter (clearance for chip ejection)
- + resharpening length
- + penetration length
**Tool Maker’s Tip**
Solid twist drills are also suitable for high performance drilling, thanks to advanced designs, coatings and HSS-PM.

### Solid twist drill
- Multi-purpose tools
- The widest range of diameters (from 0.05 up to 80 mm or more)
- Available in 4 lengths: extra short, short, long, extra long
- The longest tools (for instance 1000 mm, in diameter 10 mm)

### Drill with HSS indexable inserts
For drilling large holes, especially above 20 mm, or for combined operations
- No resharpening needed (throw-away inserts)
- Multi-purpose tool holder, for several insert diameters
- Auto-centering point and sharp edges for lower cutting forces, compared with carbide inserts
- Can be used in stacked plates and holes > 50 mm
- More fragile and not cost efficient in small diameters
<table>
<thead>
<tr>
<th>Drill Type</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Oil-hole drill</td>
<td>For high performance and deep hole drilling</td>
</tr>
<tr>
<td>Double margin drill</td>
<td>For improved hole quality</td>
</tr>
<tr>
<td>Parabolic drill</td>
<td>For deep holes</td>
</tr>
<tr>
<td>Pivot drill</td>
<td>Increased rigidity for small diameter holes</td>
</tr>
<tr>
<td>Core drill</td>
<td>Basic choice to enlarge a hole</td>
</tr>
<tr>
<td></td>
<td>Highly productive for IT8 hole quality</td>
</tr>
<tr>
<td>Step drill</td>
<td>For combined operations in one pass</td>
</tr>
<tr>
<td>Center drill</td>
<td>To produce the center holes needed for turning or grinding operations</td>
</tr>
<tr>
<td>Starting drill</td>
<td>For spotting and chamfering</td>
</tr>
</tbody>
</table>
CONVENTIONAL
• For general purpose
• Wide chip room
• Small web thickness: 0.10~0.25 D

MEDIUM WEB
• High rigidity against high feed rate. Thinnings for low thrust load are necessary
• Used for steels and cast iron
• For high efficiency drilling and longer tool life
• Web thickness: 0.20~0.35 D

PARABOLIC TYPE
• High rigidity with smooth chip removal
• Used for aluminium alloys and stainless steels
• Long tool life
• For deep holes, to prevent drill breakage or curved hole
• Web thickness: 0.30~0.45 D

TOOL MAKER’S TIP
The web is the drill’s backbone
TOOL MAKER'S TIP
Do not forget that on a drill the helix angle is the cutting angle.

**Low helix angle**
Use: hard materials, bronze, brass
Also recommended for small diameter drills, to improve tool rigidity
+ Increases cutting edge resistance
- Increases cutting forces

**Standard helix angle**
Basic choice
The most popular design

**High helix angle**
Use: soft materials (aluminium, copper)
+ Reduces cutting forces
- Decreases cutting edge resistance

30°
Small angle
Small angles: 90°
For soft materials

Standard angle
General purpose
Note: point angle has an impact on thrust load and torque as well as on the length of the cutting edge and the thickness of the chips

Large angle
Large angles: 130°, 135° or 140°
For hard materials
Prevents drill deviation under special drilling conditions (deep holes, crossing holes, lock pin holes, angular holes, etc.)
Boost drilling quality and performance: choose the right point design

**Tool Maker's Tip**

**Conical**
- Conventional drill
- General purpose

**Four facet**
- For precise hole tolerances
- Recommended for small holes
- Easy to resharpen

**Spiral**
- Good centering
- Reduce burrs
- Use: aluminium

**Center point**
- Easy drill positioning
- Prevents burrs and vibrations when used for drilling thin sheets and pipes
- Use: structural steels

**Double angle**
- High corner resistance
- Use: hardened material, abrasive material, cast iron

The most popular
Examples of point thinnings

- **No thinning**
  - General purpose

- **Three-rake**
  - Precise cutting edge
  - For difficult-to-machine materials or deep holes

- **W type**
  - For heavy drilling and hard materials
  - Effective for prevention of chipping

- **Radial point**
  - For heavy drilling. Good biting action
  - Produces short, broken chips
  - Effective for reduction of thrust load

**Tool maker’s tip**
Advanced HSS drills are autocentering; starting drills are not required.
**TOOL MAKER’S TIP**

Point thinning reduces thrust force for improved cutting data, hole accuracy and tool life.

**BENEFITS OF POINT THINNING**

### Standard drill geometry (without point thinning)

- **A-A Cross Section**
  - Positive angle
- **B-B Cross Section**
  - Positive angle
- **C-C Cross Section**
  - No cutting, only deformation

### Advanced drill geometry (with point thinning)

- **A-A Cross Section**
  - Positive angle
- **B-B Cross Section**
  - Positive angle
- **C-C Cross Section**
  - Positive angle
- **C-C Cross Section**
  - Positive angle
<table>
<thead>
<tr>
<th>TYPE</th>
<th>WEB</th>
<th>HELIX ANGLE</th>
<th>POINT ANGLE</th>
<th>USE</th>
</tr>
</thead>
<tbody>
<tr>
<td>N</td>
<td>Standard web</td>
<td>Standard (30°)</td>
<td>Standard (118° or 120°)</td>
<td>Ferrous materials Cast iron</td>
</tr>
<tr>
<td>H</td>
<td>Small web</td>
<td>Small (12 or 16°)</td>
<td>Standard (118° or 120°)</td>
<td>Short chip materials Bronze Brass</td>
</tr>
<tr>
<td>W</td>
<td>Small web</td>
<td>High (35-40°)</td>
<td>Large (130°)</td>
<td>Long chip materials Aluminium alloys Copper</td>
</tr>
<tr>
<td>Parabolic</td>
<td>Large web or small web</td>
<td>High (35-40°)</td>
<td>Standard (118° or 120°)</td>
<td>Easy-to-machine materials Long chip materials</td>
</tr>
<tr>
<td>Highly rigid</td>
<td>Large web</td>
<td>Medium (20-35°)</td>
<td>Large (130°)</td>
<td>Difficult-to-machine materials (stainless steels, heat resistant steels, spring steels)</td>
</tr>
</tbody>
</table>
**Tool Maker’s Tip**

*Improve hole accuracy with advanced HSS drills*

**Standard geometry**
- Tool diameter: 10 mm
- Hole diameter: 10.07 mm
- Poor localization: AV 0.15 mm
- IT12

**Advanced geometry**
- Tool diameter: 10 mm
- Hole diameter: 10.025 mm
- Improved localization: AV 0.045 mm
- IT9
**Clamping Drills**

**Straight Shank**
- Basic choice

**Straight Shank with an Inclined Flat**
- For large diameters (between 6 - 20 mm)
- For oil-hole drills
- Prevents the drill from slipping out of the chuck

**Straight Shank with a Tang**
- For quick tool change
- Simple holder with high rigidity
- Large runout

**Morse Taper Shank with a Tang**
- For large diameters
- For quick tool change
- High rigidity

**Tool Maker’s Tip**
Drills are also available:
- With a reinforced shank for higher rigidity and small tools
- Or with a smaller shank for bar turning lathes

---

17 CLAMPING DRILLS
Drilling is a machining operation in which the tool rotates with an axial displacement, except when mounted on lathes, where the drill is fixed and the drilled bars rotate.

In drilling, the cutting speed varies along the cutting edge. At the drill point, the cutting speed is zero. The point does not cut, but pushes the metal.

TOOL MAKER’S TIP
Did you know? There are drilled holes in 75% of all mechanical parts
TYPICAL CUTTING SPEEDS

- Nickel alloys >850 Mpa (Feed No. 2)
- Titanium alloys (type TA6V) (Feed No. 2)
- Duplex / highly alloyed stainless steels (Feed No. 4)
- Nickel alloys < 850 Mpa (Feed No. 3)
- Ferritic, martensitic, ferritic-austenitic stainless steels (Feed No. 4)
- Cu Al Fe (Feed No. 5)
- Austenitic stainless steels (Feed No. 4)
- Hardened cast iron > 270 HB (Feed No. 4)
- Nodular graphite cast iron (Feed No. 7)
- Steels: 850 - 1200 Mpa (Feed No. 3)
- Lamellar graphite cast iron (Feed No. 7)
- Thermosetting plastics (Feed No. 5)
- Pure Nickel (Feed No. 4)
- Steels < 550 Mpa (Feed No. 5)
- Pure Titanium (Feed No. 2)
- Magnesium (Feed No. 6)
- Aluminium Si <5% (Feed No. 7)
- Copper alloys - short chips (Feed No. 5)
- Copper alloys - long chips (Feed No. 5)
- Graphite (Feed No. 5)
- Cu Al Ni (Feed No. 4)
- Nickel alloys < 850 Mpa (Feed No. 3)
- Aluminium Si 5-10% (Feed No. 7)
- Aluminium Si > 10% (Feed No. 6)
- Steels < 550 Mpa (Feed No. 6)
- Copper alloys - short chips (Feed No. 5)
- Copper alloys - long chips (Feed No. 5)
- Plastics (Feed No. 5)
- Nodular graphite cast iron (Feed No. 7)
- Pure copper (Feed No. 5)
- Copper alloys - short chips (Feed No. 5)
- Copper alloys - long chips (Feed No. 5)
- Aluminium Si > 10% (Feed No. 6)
- Duplex / highly alloyed stainless steels (Feed No. 4)
- Nodular graphite cast iron (Feed No. 7)
- Copper alloys - long chips (Feed No. 5)
- Aluminium Si <5% (Feed No. 7)
- Copper alloys - short chips (Feed No. 5)
- Copper alloys - long chips (Feed No. 5)
- Plastics (Feed No. 5)
- Nodular graphite cast iron (Feed No. 7)
- Copper alloys - long chips (Feed No. 5)
- Aluminium Si <5% (Feed No. 7)
- Copper alloys - short chips (Feed No. 5)
- Copper alloys - long chips (Feed No. 5)
- Plastics (Feed No. 5)
- Nodular graphite cast iron (Feed No. 7)
- Copper alloys - long chips (Feed No. 5)
- Aluminium Si <5% (Feed No. 7)
- Copper alloys - short chips (Feed No. 5)
- Copper alloys - long chips (Feed No. 5)
- Plastics (Feed No. 5)
- Nodular graphite cast iron (Feed No. 7)
- Copper alloys - long chips (Feed No. 5)
- Aluminium Si <5% (Feed No. 7)
- Copper alloys - short chips (Feed No. 5)
- Copper alloys - long chips (Feed No. 5)
- Plastics (Feed No. 5)
- Nodular graphite cast iron (Feed No. 7)
- Copper alloys - long chips (Feed No. 5)
- Aluminium Si <5% (Feed No. 7)
- Copper alloys - short chips (Feed No. 5)
- Copper alloys - long chips (Feed No. 5)
- Plastics (Feed No. 5)
- Nodular graphite cast iron (Feed No. 7)
- Copper alloys - long chips (Feed No. 5)
- Aluminium Si <5% (Feed No. 7)
- Copper alloys - short chips (Feed No. 5)
- Copper alloys - long chips (Feed No. 5)
- Plastics (Feed No. 5)
- Nodular graphite cast iron (Feed No. 7)

Cutting speed in m/min
<table>
<thead>
<tr>
<th>Drill Ø mm</th>
<th>1</th>
<th>2</th>
<th>3</th>
<th>4</th>
<th>5</th>
<th>6</th>
<th>7</th>
<th>8</th>
<th>9</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>F (mm/rev.)</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>0.50</td>
<td>0.004</td>
<td>0.006</td>
<td>0.007</td>
<td>0.008</td>
<td>0.010</td>
<td>0.012</td>
<td>0.014</td>
<td>0.016</td>
<td>0.019</td>
</tr>
<tr>
<td>1.00</td>
<td>0.006</td>
<td>0.008</td>
<td>0.012</td>
<td>0.014</td>
<td>0.016</td>
<td>0.018</td>
<td>0.020</td>
<td>0.023</td>
<td>0.025</td>
</tr>
<tr>
<td>2.00</td>
<td>0.020</td>
<td>0.025</td>
<td>0.032</td>
<td>0.040</td>
<td>0.050</td>
<td>0.063</td>
<td>0.080</td>
<td>0.100</td>
<td>0.125</td>
</tr>
<tr>
<td>2.50</td>
<td>0.025</td>
<td>0.032</td>
<td>0.040</td>
<td>0.050</td>
<td>0.063</td>
<td>0.080</td>
<td>0.100</td>
<td>0.125</td>
<td>0.160</td>
</tr>
<tr>
<td>3.15</td>
<td>0.032</td>
<td>0.040</td>
<td>0.050</td>
<td>0.063</td>
<td>0.080</td>
<td>0.100</td>
<td>0.125</td>
<td>0.160</td>
<td>0.160</td>
</tr>
<tr>
<td>4.00</td>
<td>0.040</td>
<td>0.050</td>
<td>0.063</td>
<td>0.080</td>
<td>0.100</td>
<td>0.125</td>
<td>0.160</td>
<td>0.200</td>
<td>0.200</td>
</tr>
<tr>
<td>5.00</td>
<td>0.040</td>
<td>0.050</td>
<td>0.063</td>
<td>0.080</td>
<td>0.100</td>
<td>0.125</td>
<td>0.160</td>
<td>0.200</td>
<td>0.250</td>
</tr>
<tr>
<td>6.30</td>
<td>0.050</td>
<td>0.063</td>
<td>0.080</td>
<td>0.100</td>
<td>0.125</td>
<td>0.160</td>
<td>0.200</td>
<td>0.250</td>
<td>0.315</td>
</tr>
<tr>
<td>8.00</td>
<td>0.063</td>
<td>0.080</td>
<td>0.100</td>
<td>0.125</td>
<td>0.160</td>
<td>0.200</td>
<td>0.250</td>
<td>0.315</td>
<td>0.315</td>
</tr>
<tr>
<td>10.00</td>
<td>0.080</td>
<td>0.100</td>
<td>0.125</td>
<td>0.160</td>
<td>0.200</td>
<td>0.250</td>
<td>0.315</td>
<td>0.400</td>
<td>0.400</td>
</tr>
<tr>
<td>12.50</td>
<td>0.080</td>
<td>0.100</td>
<td>0.125</td>
<td>0.160</td>
<td>0.200</td>
<td>0.250</td>
<td>0.315</td>
<td>0.400</td>
<td>0.500</td>
</tr>
<tr>
<td>16.00</td>
<td>0.100</td>
<td>0.125</td>
<td>0.160</td>
<td>0.200</td>
<td>0.250</td>
<td>0.315</td>
<td>0.400</td>
<td>0.500</td>
<td>0.630</td>
</tr>
<tr>
<td>20.00</td>
<td>0.125</td>
<td>0.160</td>
<td>0.200</td>
<td>0.250</td>
<td>0.315</td>
<td>0.400</td>
<td>0.500</td>
<td>0.630</td>
<td>0.630</td>
</tr>
<tr>
<td>25.00</td>
<td>0.160</td>
<td>0.200</td>
<td>0.250</td>
<td>0.315</td>
<td>0.400</td>
<td>0.500</td>
<td>0.630</td>
<td>0.800</td>
<td>0.800</td>
</tr>
<tr>
<td>31.50</td>
<td>0.160</td>
<td>0.200</td>
<td>0.250</td>
<td>0.315</td>
<td>0.400</td>
<td>0.500</td>
<td>0.630</td>
<td>0.800</td>
<td>1.000</td>
</tr>
<tr>
<td>40.00</td>
<td>0.200</td>
<td>0.250</td>
<td>0.315</td>
<td>0.400</td>
<td>0.500</td>
<td>0.630</td>
<td>0.800</td>
<td>1.000</td>
<td>1.250</td>
</tr>
<tr>
<td>50.00</td>
<td>0.250</td>
<td>0.315</td>
<td>0.400</td>
<td>0.500</td>
<td>0.630</td>
<td>0.800</td>
<td>1.000</td>
<td>1.250</td>
<td>1.250</td>
</tr>
<tr>
<td>63.00</td>
<td>0.315</td>
<td>0.400</td>
<td>0.500</td>
<td>0.630</td>
<td>0.800</td>
<td>1.000</td>
<td>1.250</td>
<td>1.600</td>
<td>1.600</td>
</tr>
<tr>
<td>80.00</td>
<td>0.400</td>
<td>0.500</td>
<td>0.630</td>
<td>0.800</td>
<td>1.000</td>
<td>1.250</td>
<td>1.600</td>
<td>1.600</td>
<td>2.000</td>
</tr>
</tbody>
</table>
- Cutting fluids are essential for heat evacuation, chip removal and lubrication, to prevent adhesive wear at the drill point (where the cutting speed is zero).
- In drilling, soluble oils are usually preferred but oil can also be used.
- Soluble oils with additives significantly prolong the tool life of HSS drills.
- The cutting fluid must be supplied directly to the cutting edge.
- The amount of lubricant required depends on drill diameter, hole depth and cutting data.
TOOL MAKER’S TIP

Oil-hole drills are essential for longer tool life, higher speeds and deep holes.

Benefits of oil-hole drills and high pressure coolant
+ prevent chip welding
+ prevent damaging chemical reactions that occur at high temperatures
+ prolong tool life (up to 300%)
+ allow an increase of cutting speeds by more than 30%
+ improve surface finish

SUCCESS STORY

Operation
• Drilling of holes Ø 8.25 mm, L 80 mm in an automotive part

Tool
• HSS-PM 9% Co. Drill with oil holes + coating + special geometry

Cutting data
• $v_c \ 60 \text{ m/min}, \ f \ 0.25 \text{ mm/rev}$

Tool life
• more than doubled compared with a carbide drill (812 holes instead of 375 holes)

Costs per hole
• divided by 2 compared with a carbide drill

Cast iron
<table>
<thead>
<tr>
<th>Flank wear</th>
<th>Crater wear</th>
<th>Chipping</th>
<th>Deformation</th>
<th>Built-up edge</th>
</tr>
</thead>
<tbody>
<tr>
<td>• Normal wear pattern</td>
<td>• To be avoided</td>
<td>• To be avoided</td>
<td>• To be avoided</td>
<td>• To be avoided</td>
</tr>
<tr>
<td>• Increase the cutting speed ( (v_c) ) and/or the feed ( (f_z) )</td>
<td>• Decrease the cutting speed ( (v_c) ) and/or the feed ( (f_z) )</td>
<td>• Decrease the cutting speed ( (v_c) ) and/or the feed ( (f_z) )</td>
<td>• Increase the cutting speed ( (v_c) ) and/or the feed ( (f_z) )</td>
<td>• Increase the cutting speed ( (v_c) ) and/or the feed ( (f_z) )</td>
</tr>
<tr>
<td>• Increase the effective cutting angle</td>
<td>• Use a coated tool and a harder HSS material</td>
<td>• Decrease the cutting speed ( (v_c) ), increase the coolant pressure</td>
<td>• Use a coated tool and a harder HSS material</td>
<td>• Use a coated tool and a harder HSS material</td>
</tr>
<tr>
<td></td>
<td></td>
<td>• Use a tougher HSS material</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
• Long curly chips stick to the flute and prevent the coolant from going through the hole. The result is tool melting or breakage.

• When too short, chips are difficult to evacuate and sometimes pack in the flute. This worsens the quality of the drilled hole and shortens tool life.

Drilling chip shapes

Chip form at each stage of the drilling process when a drill with point thinning is used
<table>
<thead>
<tr>
<th>Problem</th>
<th>Causes</th>
<th>Solutions</th>
</tr>
</thead>
<tbody>
<tr>
<td>Oversize hole</td>
<td>Loose hold, unequal point angle, varying length of lip</td>
<td>Check holder and run-out&lt;br&gt;Regrind and check precision</td>
</tr>
<tr>
<td>Irregular hole size</td>
<td>Loose hold, unequal point angle, large length of lip, excessive feed rate, poor lubrication</td>
<td>Check holder and run-out&lt;br&gt;Regrind and check precision&lt;br&gt;Decrease feed rate&lt;br&gt;Use an oil-hole drill</td>
</tr>
<tr>
<td>Low position accuracy</td>
<td>Spindle run out&lt;br&gt;Low alignment accuracy&lt;br&gt;Run out when cutting</td>
<td>Check holder and run-out, check alignment, select a low cutting resistance thinning, use a drill bush, or use centering</td>
</tr>
<tr>
<td>Poor hole perpendicularity</td>
<td>Excessive tool wear, unequal point angle, drilling surface not horizontal, poor alignment (on lathes)</td>
<td>Regrind and check precision, check the workpiece position, make a center hole</td>
</tr>
<tr>
<td>Bad cylindrical accuracy</td>
<td>Unequal point angle, loose hold, relief angle too wide, low drill rigidity</td>
<td>Regrind and check precision&lt;br&gt;Check holder and run-out&lt;br&gt;Use a large web drill</td>
</tr>
<tr>
<td>Poor surface finish</td>
<td>Poor regrinding, cooling problem, loose hold, excessive feed rate, chip packing</td>
<td>Regrind correctly, increase coolant volume and improve quality, reduce feed rate, select a wide flute, high helix oil-hole drill</td>
</tr>
<tr>
<td>Drill breakage</td>
<td>Low rigidity, excessive feed rate, tool wear, chip packing, difficult entering</td>
<td>Increase rigidity, reduce feed rate, select a wide flute, high helix oil-hole drill, use a drill bush or centering</td>
</tr>
<tr>
<td>Tang breakage</td>
<td>Slipping chuck, defect (damage, swarf) of the inner surface of morse taper</td>
<td>Modify the surface of the holder or change the holder</td>
</tr>
<tr>
<td>Drilling of inclined surface</td>
<td>Cross-hole and non-symmetric hole</td>
<td>Drilling sheets</td>
</tr>
<tr>
<td>-------------------------------</td>
<td>----------------------------------</td>
<td>----------------</td>
</tr>
<tr>
<td>• Mill a flat surface before drilling</td>
<td>• To be avoided</td>
<td>• Use a back-up plate</td>
</tr>
<tr>
<td>• Make a pre-hole with a center or starting drill</td>
<td>• Use a highly rigid drill or a double margin drill</td>
<td>• Use a brad point or a step drill</td>
</tr>
<tr>
<td>• Use a guide bush</td>
<td>• Reduce the feed rate</td>
<td>• Reduce the feed rate</td>
</tr>
<tr>
<td>• Use a highly rigid drill</td>
<td>• Fill the hole with the same material in order to balance the cutting</td>
<td></td>
</tr>
<tr>
<td>• Decrease the feed rate</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Symbol</td>
<td>Unit</td>
<td>Name</td>
</tr>
<tr>
<td>--------</td>
<td>------</td>
<td>-----------------------------------</td>
</tr>
<tr>
<td>D</td>
<td>mm</td>
<td>Tool diameter</td>
</tr>
<tr>
<td>I</td>
<td>mm</td>
<td>Hole depth</td>
</tr>
<tr>
<td>L</td>
<td>mm</td>
<td>Total run: approach run + hole depth + length of point</td>
</tr>
<tr>
<td>N</td>
<td>rev/mm</td>
<td>Revolution per minute</td>
</tr>
</tbody>
</table>

- **\( v_c \)**: Cutting speed
- **\( v_f \)**: Feed per minute
- **\( f \)**: Feed per revolution
- **\( T \)**: Machining time