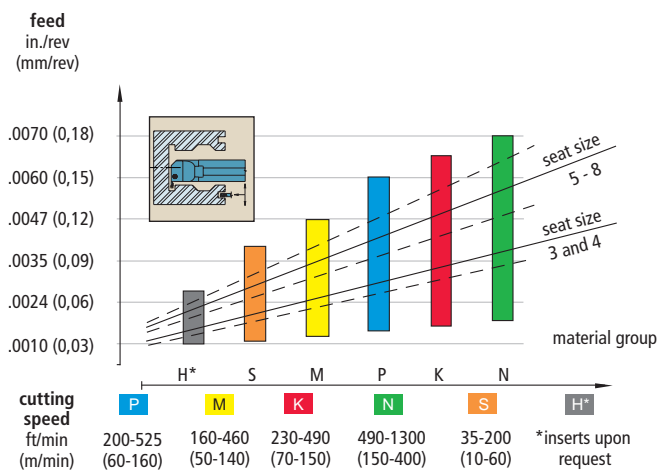
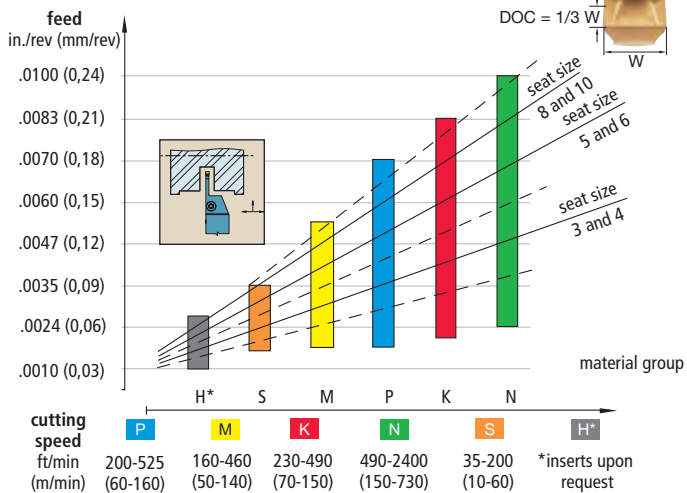


Feed and Speed Selection for ID and Face Grooving



Feed and Speed Selection for OD Grooving

For profiling, depth of cut recommendation

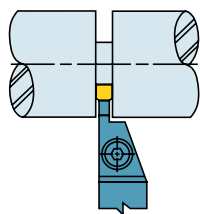


For radial grooving under stable conditions, feed can be increased by up to 50%.

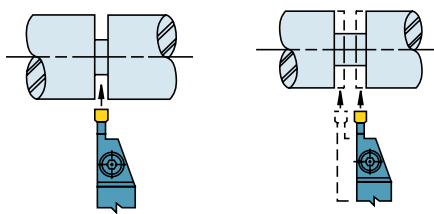
Tool Application Guidelines

- Always use good general machining practices.
- Make the machine and workpiece setup as rigid as possible.
- Integral shank toolholders offer the best rigidity. They should be your first toolholder choice, when possible.
- Use the toolholder with the shortest possible depth-of-cut for the application ("CD" dimension).
- When changing inserts, make sure the new insert locates securely against the toolholder's positive stop.
- Never tighten the clamping screw without an insert in the pocket.
- Toolholder projection out of the tool block should be as short as possible.
- Inserts should cut as close to center as possible.
- Dwell time in bottom of groove should be less than three revolutions.
- Recommended cutting speeds and feeds are a starting point. Adjust, as necessary, for optimum tool life and chip control.

Deep Grooves

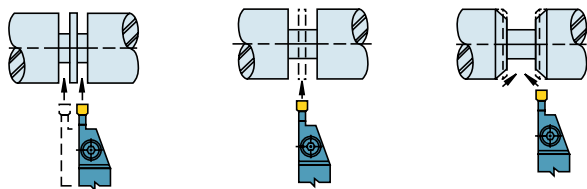


Deep Grooves Slightly Wider than the Tool



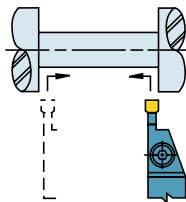
1. Plunge the center of the groove.
2. Plunge each side of the groove to get the specified width. Use a slower feed rate when cutting groove sides.

Extra Wide Deep Grooves



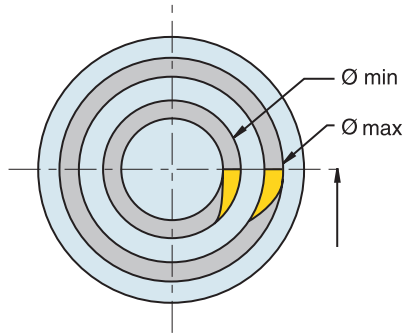
1. Plunge out both sides of the groove width.
2. Plunge center area to remove web of material remaining.
3. Plunge both sides of groove at the required angle, using approximately one-half the width of the grooving tool for maximum width of cut.

Finish Turning of the Groove / Light Profiling



1. Follow recommendations explained above.
2. To avoid insert chipping and to achieve groove wall perpendicularity, follow the tool path shown here.
3. Use the lightest depth of cut possible while still maintaining good chip breaking, tool life, and surface finish.

Face Grooving Application Guidelines



Tool Selection

- When selecting the toolholder, always start at the largest diameter possible and work toward the smaller diameter. This will allow the strongest tool to be used.

Cutting the First Groove

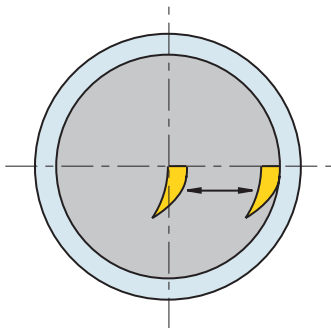
- The outside diameter of the first groove must be between the diameter minimum and diameter maximum capability of the face grooving tool (see illustration above). This creates clearance for the toolholder.

Chip Control

- Adjust speed and feed for good chip control and evacuation from the groove. Chip compaction can cause poor surface finish, tool breakage, and reduced tool life.

Tool Setting

- The tool should be set as close to the center as possible to avoid extreme formation of burrs.
- Align the cutting edge square to the workpiece.



Widening a Face Groove

- After the first groove has been cut, the groove width can be widened in either direction using the same tool. The best practice is to work from the OD to the ID.

Practical Solutions to Grooving Problems

Problem	Remedy
Burr	<ol style="list-style-type: none"> 1. Verify tool center height. 2. Use sharp tool (index more often). 3. Use positive rake PVD coated insert. 4. Use correct grade for workpiece material. 5. Use correct geometry (e.g. positive rake for workhardening material). 6. Change tool path.
Poor surface finish	<ol style="list-style-type: none"> 1. Increase speed. 2. Use sharp tool (index more often). 3. Dwell time in bottom 1-3 revolutions (max). 4. Use proper chip control geometry. 5. Increase coolant flow. 6. Verify proper setup (overhang, shank size). 7. Use correct geometry (e.g. positive rake for workhardening material).
Groove bottom not flat	<ol style="list-style-type: none"> 1. Use sharp tool (index more often). 2. Dwell time in bottom 1-3 revolutions (max). 3. Reduce tool overhang (increase rigidity). 4. Reduce feed rate at groove bottom. 5. Use a wider insert. 6. Verify tool center height.
Poor chip control	<ol style="list-style-type: none"> 1. Use sharp tool (index more often). 2. Increase coolant concentration. 3. Adjust feed rate (usually increase first).
Chatter	<ol style="list-style-type: none"> 1. Reduce tool and workpiece overhang. 2. Adjust speed (usually increase first). 3. Adjust feed (usually increase first). 4. Verify tool center height.
Insert chipping	<ol style="list-style-type: none"> 1. Use correct grade for workpiece material. 2. Increase speed. 3. Reduce feed. 4. Use a stronger grade. 5. Increase tool and setup rigidity.
Built-up edge	<ol style="list-style-type: none"> 1. Use positive rake PVD coated insert. 2. Increase speed. 3. Reduce feed. 4. Increase coolant flow/concentration. 5. Use cermets.
Side walls not straight	<ol style="list-style-type: none"> 1. Check tool alignment for square. 2. Reduce workpiece and tool overhang. 3. Use sharp insert (index more often).