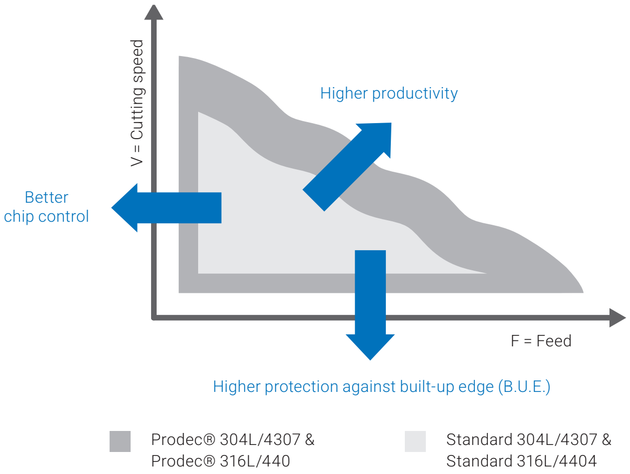
Machining guideline

For Prodec® 1.4307/1.4301, 304/304L and Prodec® 1.4401/1.4404, 316/316L

Prodec® 304L/4307 and Prodec® 316L/4404 are special variants of standard Types 304 (UNS S30400) / 304L (UNS S30403) and 316 (UNS S31600) / 316L (UNS S31603) respectively with enhanced metallurgy for better machinability. The general rules for machining stainless steel also apply to the Prodec® grades.The difference is that Prodec® grades enable a longer tool life and/or tougher machining conditions. The machining window illustrated on the right gives a demonstration of this.

Other fabrication operations such as welding, hot working and cold working can be performed in the same way as for standard 304L/4307 and 316L/4404.



Product forms

Prodec® 304L/4307 and Prodec® 316L/4404 are available as round, hexagon and square bars, as well as wire rod and concast billets.

Turning

- The machine and setup must be rigid
- Use shortest possible tool length
- Use coolant
- Use smallest possible nose radius to avoid vibrations

Milling

- Avoid cutting through holes/cavities
- Ensure good chip evacuation, recutting of chips may cause tool damage

Machining guidelines

The cutting parameters in this guideline will work under normal cutting conditions. It is suggested to begin with cutting parameters in the ranges indicated in the tables and then to improve parameters by moving to higher or lower speed, feed or depth of cut until best performance is reached.

It is possible to end up in a range somewhat outside the values indicated in the tables depending on the actual machine set-up. A guide for further optimization of cutting parameters can be found under the "Troubleshooting" section on the next page.

	Carbide Tooling				HSS Tooling		
Turning	**Depth of cut or width (mm)**	**Speed (m/min)**	**Feed (mm/rev)**	**Tool Grade**	**Speed (m/min)**	**Feed (mm/rev)**	**Tool Grade**
Finishing	-2	260–280	0.10	M10–15	50 [1]	0.10	T15
Medium	2–5	200–260	0.25	M10–25	35	0.25	T15
Roughing	5–10	50–220	0.4	M25–35	20	0.4	T15

[1] Coated tools

	Carbide Tooling			HSS Tooling		
Milling	**Speed (m/min)**	**Feed (mm/rev)**	**Tool Grade**	**Speed (m/min)**	**Feed (mm/rev)**	**Tool Grade**
Face milling	150–250	0.08–0.30	M10–30	24–40	0.08–0.20	T15
Side milling	180–240	0.08–0.30	M10–30	24–40	0.08–0.20	T15
End milling	150–220	0.05–0.20	M10–30	24–40	0.025–0.15	T15
End milling [2]	50–100	0.05–0.20	M35	–	–	–

[2] Solid cemented carbide

Want to know more? Get in touch with our Sales-Team.

Damstahl GmbH Raiffeisenstr. 6-8 40764 Langenfeld Germany
Phone: +49 2173 8983 200 Mail: prodec@damstahl.com

DAMSTAHL + PRODEC®

Drilling – high speed steel twist drills

- Use coolant
- If possible use internal coolant through drill
- Use of cobalt high alloyed drills is preferred
- With PVD-coated HSS drills
- the cutting speed can be increased by 10%
- Use as short a drill as possible

	HSS Tooling			
Drilling [3]	**Diameter (mm)**	**Speed (m/min)**	**Feed (mm/rev)**	**Rpm (rev/min)**
	1	10–12	0.05	3200–3800
	3	15–17	0.1	1600–1800
	5	17–20	0.12	1080–1270
	10	17–20	0.15	540–640
	15	17–20	0.2	360–430
	20	17–20	0.3	270–320
	30	17–20	0.3	180–220

[3] HSS-5%Co

	Carbide Tooling			HSS Tooling		
Other machining operations	**Speed (m/min)**	**Feed (mm/rev)**	**Tool Grade**	**Speed (m/min)**	**Feed (mm/rev)**	**Tool Grade**
Cut-off	100–150	0.05–0.15	M30	24	0.05	T15
Reaming	50	0.10–0.40	M10–M30	10–15	0.10–0.40	T15
Tapping	–	–	–	5–13	–	–
Threading singel insert	90–130	–	M10–M30	15–20	–	T15
Drillling indexable insert	200–250	0.06–0.12	Center M30 Periferi M10	–	–	–

Other machining operations

Cut-off

- Reduce feed by 50% approximately 6mm from the center

Reaming

- Type of coolant: emulsion or cutting oil

Tapping

- For blind holes use spiral flute grinding for good chip evacuation
- For through holes use spiral point grinding with gun nose to push the hips forward

Threading single insert

- Full profile insert for high quality thread forms
- V-profile insert – threading with minimum tool inventory
- Multipoint insert for economic threading in mass production

Drilling indexable insert

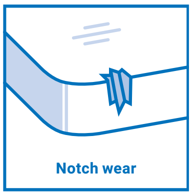
- Cutting data is very dependent on the drill design. Hence, the manufacturers recommendations must be considered

Troubleshooting


Flank wear

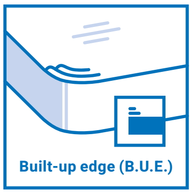
Flank wear

For longer tool life – reduce cutting speed or use a harder insert.


Notch wear

Notch wear

Notch wear is a common wear mechanism when machining stainless steel. Increased cutting speed will reduce notch but increase flank wear. If possible, use an insert with smaller entering angle 60-80 degrees or variable cutting depth or softer insert grade.


Built-up edge (B.U.E.)

Built-up edge (B.U.E.)

Built-up edge occurs when the cutting speed is too low and the stainless steel tends to stick to the tool (in milling the chips stick to the tool). To avoid – increase cutting speed or use another coating.


Plastic deformation

Plastic deformation

To avoid – reduce either cutting speed, feed or use a harder insert.


Long chips

Long chips

To avoid – increase feed or use an insert with smaller chip breaker.

Want to know more? Get in touch with our Sales-Team.

Damstahl GmbH Raiffeisenstr. 6-8 40764 Langenfeld Germany
Phone: +49 2173 8983 200 Mail: prodec@damstahl.com