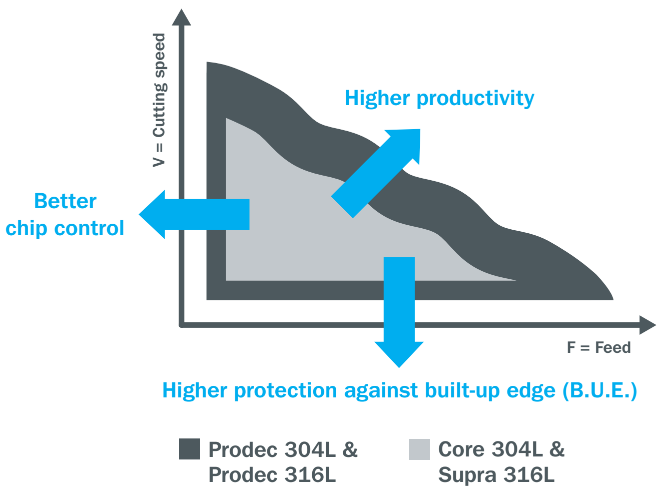
North American Version

Imperial Units

Machining guideline for Prodec 304L and Prodec 316L

Prodec 304L and Prodec 316L are special variants of standard Types 304/304L and 316/316L 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 non-Prodec treated Core 304L and Supra 316L.



Product forms

Prodec 304L and Prodec 316L are available as hexagon, square, flat and round bars, as well as rolled billets and plate.

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.

Turning

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

	Carbide Tooling				HSS Tooling		
Turning	Depth of cut or width (in)	Speed (sfm)	Feed (in/rev)	Tool Grade	Speed (sfm)	Feed (in/rev)	Tool Grade
Finishing	-0.08	850–915	0.004	M10–15	160[1]	0.004	T15
Medium	0.08–0.20	655–850	0.010	M10–25	115	0.010	T15
Roughing	0.20–0.40	160–720	0.016	M25–35	65	0.016	T15

[1] Coated tools

Milling

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

	Carbide Tooling			HSS Tooling		
Milling	Speed (sfm)	Feed (in/rev)	Tool Grade	Speed (sfm)	Feed (in/rev)	Tool Grade
Face milling	490–815	0.003–0.012	M10–30	80–130	0.003–0.008	T15
Side milling	590–785	0.003–0.012	M10–30	80–130	0.003–0.008	T15
End milling	490–720	0.002–0.008	M10–30	80–130	0.001–0.006	T15
End milling[2]	160–325	0.002–0.008	M35	–	–	–

[2] Solid cemented carbide

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.

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 chips 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

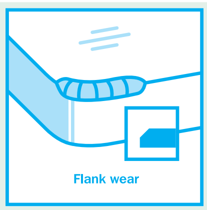
- Cutting data is very dependent on the drill design. Hence, the manufacturers recommendations must be considered.

	HSS Tooling			
Drilling[3]	Diameter (in)	Speed (sfm)	Feed (in/rev)	Speed (rpm)
	0.04	33–39	0.002	3200–3800
	0.12	50–57	0.004	1600–1800
	0.20	57–66	0.005	1080–1270
	0.40	57–66	0.006	540–640
	0.60	57–66	0.008	360–430
	0.80	57–66	0.012	270–320
	1.20	57–66	0.012	180–220

[3] HSS-5%Co

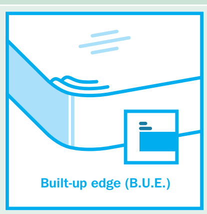
	Carbide Tooling			HSS Tooling		
Other machining operations	Speed (sfm)	Feed (in/rev)	Tool Grade	Speed (sfm)	Feed (in/rev)	Tool Grade
Cut-off	325–490	0.002–0.006	M30	80	0.002	T15
Reaming	160	0.004–0.016	M10–M30	33–50	0.004–0.016	T15
Tapping	–	–	–	15–43	–	–
Threading singel insert	295–425	–	M10–M30	50–65	–	T15
Drillling indexable insert	655–815	0.002–0.005	Center M30 Periferi M10	–	–	–

Troubleshooting

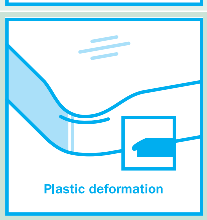


Flank wear
For longer tool life – reduce cutting speed or use a harder insert.

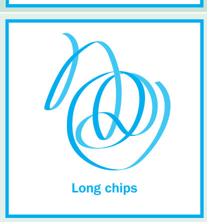
Notch wear
Notch wear is a common wear mechanism when machining stainless steel. Increased cutting speed will reduce notch but increase flank wear. If possible, use an insert with smaller entering angle 60-80 degrees or variable cutting depth or softer insert grade.



Built-up edge (B.U.E.)
Built-up edge 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
To avoid – reduce either cutting speed, feed or use a harder insert.



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

1584EN-US:1. February 2016.