

PRODEC[®]

Machining guideline for Prodec[®] 304L and Prodec[®] 316L

Prodec[®] 304L and Prodec[®] 316L 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 and 316L.

Product forms

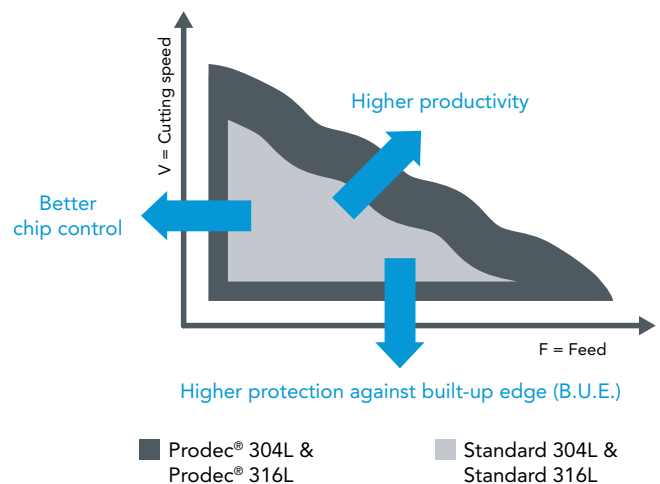
Prodec[®] 304L and Prodec[®] 316L are available as round and hexagon bars.

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

Milling

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

Turning	Carbide Tooling			
	Grade	Feed (in/rev)	SFM	DoC (in)
Finishing	M10-20	0.004	840-930	< 0.08
Medium	M10-25	0.010	650-850	0.08 - 0.2
Roughing	M20-35	0.015	150-750	0.2-0.4

Milling	Carbide Tooling		
	Grade	Feed	SFM
Face Milling	M10-25	0.004	485-825
Side Milling	M10-30	0.010	580-800
End Milling	M10-30	0.015	480-725

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

Drilling (HSS Cobalt Alloy)	HSS Tooling			
	Diameter (in)	RPM	Feed	SFM
	0.04	2850-3800	0.002	30-40
	0.12	1600-1750	0.004	50-55
	0.2	955-1050	0.005	55-65
	0.4	470-625	0.006	55-65
	0.6	350-415	0.008	55-65
	0.8	265-310	0.012	55-65
	1.2	175-210	0.012	55-65

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

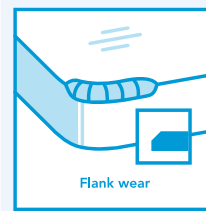
- Full profile insert for high quality thread forms
- V-profile insert – threading with minimum tool inventory
- Multipoint insert for economic threading in mass production

Drilling indexable insert

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

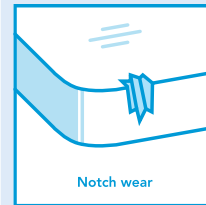
Other Machining Operations	Carbide Tooling		
	Grade	Feed	SFM
Cut-off	M25	0.002-0.006	325-500
Reaming	M10-M30	0.004-0.016	170
Tapping	-	-	-
Threading single insert	M10-M30	-	300-450
Drilling w/indexable insert	Center M30 Periphery M10	0.002-0.005	650-825

Troubleshooting



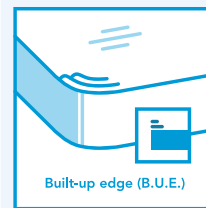
Flank wear

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



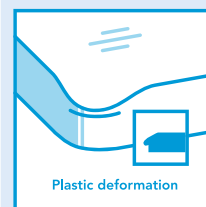
Notch wear

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.

