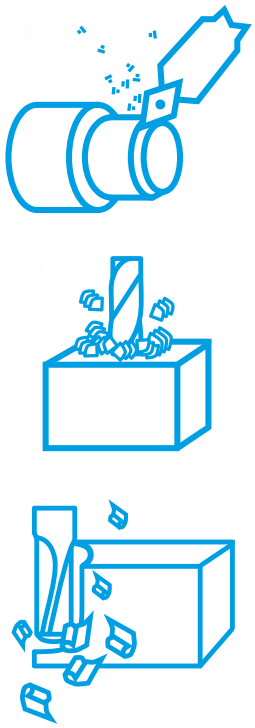
outokumpu

Machining guideline for Prodec 303/4305

Prodec 303/4305 is a fully resulfurized free-machining austenitic stainless steel. The Prodec brand name means this steel has been specially melted and treated by Outokumpu's proprietary ladle metallurgy techniques to maximize machinability while retaining good mechanical properties, corrosion resistance, and forming characteristics. This free cutting stainless steel gives you faster machining, longer tool life, better tolerances, superior machined surface quality, and reduced scrap losses compared to conventionally produced EN 1.4305.



Product forms

Prodec 303/4305 is available as hexagon, square, flat and round bars, as well as rolled billets.

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 (mm)**	**Speed (m/min)**	**Feed (mm/rev)**	**Tool Grade**	**Speed (m/min)**	**Feed (mm/rev)**	**Tool Grade**
Finishing	0.050–0.10	180–375	0.10–0.25	M10–15	40–55	0.12–0.25	T15
Roughing	0.12–5.0	90–220	0.25–0.60	M25–35	30–40	0.38–0.50	T15

Milling (only end milling)

- 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		
Milling	**Depth of cut or width (mm)**	**Speed (m/min)**	**Feed ex. (mm/rev)**	**Tool Grade**	**Speed (m/min)**	**Feed (mm/rev)**	**Tool Grade**
End milling[1]	1.0–15.0	50–250	0.050–0.20	M35	10–50	0.075–0.15	T15

[1] 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 drill as possible.

	HSS Tooling			
Drilling[2]	Diameter (mm)	Speed (m/min)	Feed ex. (mm/rev)	Rpm (rev/min)
Steel Twist Drills	1	13–16	0.065	4100–4900
	3	19–22	0.13	2000–2300
	5	22–26	0.16	1400–1650
	10	22–26	0.20	700–830
	15	22–26	0.25	470–560
	20	22–26	0.40	350–420
	30	22–26	0.40	230–290

[2] HSS-5%Co

Other machining operations

Cut-off

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

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

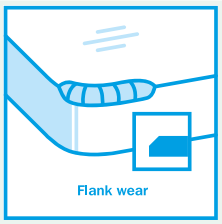
- Full profile insert for high quality thread forms.
- V-profile insert – threading with minimum tool inventory.
- Multipoint insert for economic threading in mass production.

Forming

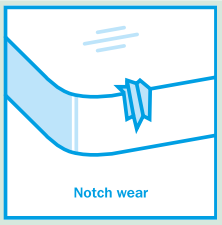
- Use coolant.
- The machine and setup must be rigid.
- Use shortest possible tool length.

	Carbide Tooling				HSS Tooling		
Other machining operations	Depth of cut or width (mm)	Speed (m/min)	Feed ex. (mm/rev)	Tool Grade	Speed (m/min)	Feed (mm/rev)	Tool Grade
Cut–off	1.5–7.0	80–200	0.040–0.15	M30	20–40	0.030–0.080	T15
Tapping	–	–	–	–	3–35	–	–
Threading singel insert	–	90–130	–	M10–M30	3–35	–	T15
Forming	7–50	40–130	0.040–0.12	M10–M30	20–40	0.040–0.40	T15

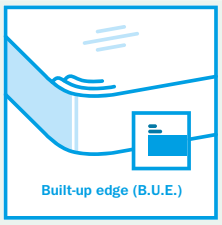
Troubleshooting



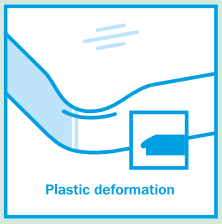
Flank wear
For longer tool life – reduce cutting speed or use a harder insert.



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.

1590EN-GB:1, March 2016, Erweko Finland.