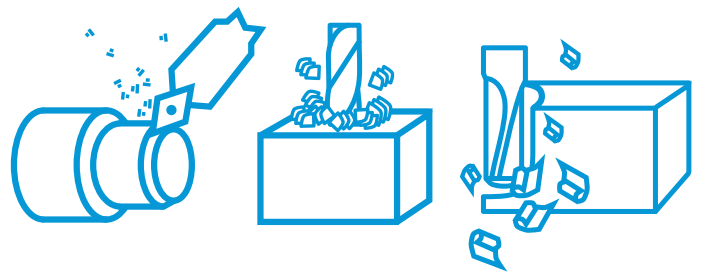


PRODEC®

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 Marcegaglia'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 round, hexagon and square bars as well as wire rod and concast 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

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

Turning	Carbide Tooling				HSS Tooling		
	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	Carbide Tooling				HSS Tooling		
	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

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

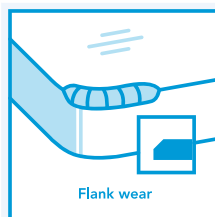
Contact sales at
barfinishingsales@marcegaglia.com

Drilling ²	HSS Tooling			
	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

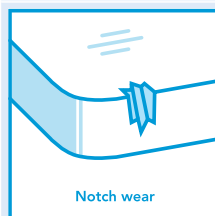
Other machining operations	Depth of cut or width (mm)	Carbide Tooling			HSS Tooling		
		Speed (m/min)	Feed ex. (mm/rev)	Tool Grade	Speed (m/min)	Feed (mm/rev)	Tool Grade
Cut-off	1.5–7.0	80–200	0.040–0.15	M30	20–40	0.030–0.080	T15
Tapping	–	–	–	–	3–35	–	–
Threading single insert	–	90–130	–	M10–M30	3–35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

Troubleshooting



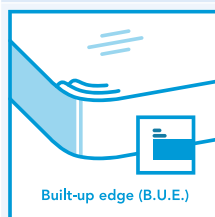
Flank wear

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



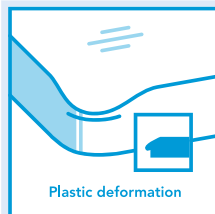
Notch wear

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.