

153 MA™ and 253 MA® Machining Guideline

The aim of this machining guide is to be an introduction to fabricators machining in 153 MA™ and 253 MA®. The machining parameters in this guideline will work under normal machining conditions.

Machining heat resistant austenitic stainless steel

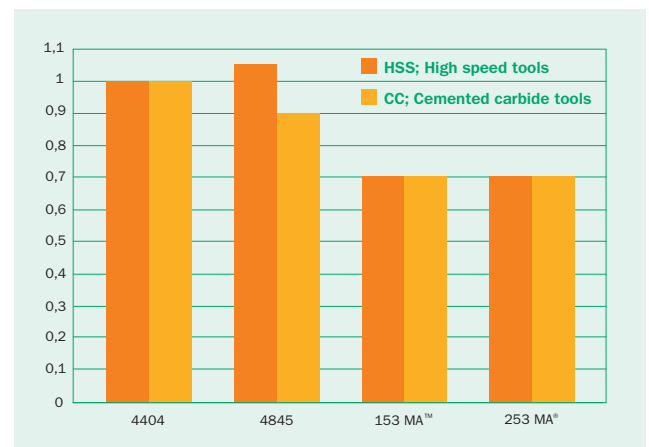
Heat resistant austenitic stainless steels, such as Outokumpu 153 MA™ and 253 MA®, are generally more difficult to machine than conventional austenitic stainless steels, e.g. Outokumpu 4404. The alloying with nitrogen and rare earth metals causes both higher cutting forces and more severe tool wear than that experienced when machining lower alloyed stainless steel grades.

- Ensure a stable setup – Higher cutting forces compared to standard austenitic grades
- Use sharp tools in order to generate less heat and minimize work hardening
- Coolant – Less heat results in a longer tool life

Machinability ranking

The machinability of different stainless steel grades can be illustrated by a machinability ranking. This ranking, where a higher figure means better machinability, is based on a combination of test data from several different machining operations. The ranking shows that 153 MA™ and 253 MA® is more difficult to machine in relation to 4404 and that the machinability decreases with increasing alloy content. Please note the relative ranking of steel grades and that the scale is very different between Cemented Carbide CC and High Speed Steel HSS tools which is demonstrated in the tables below.

Machining index



Turning

- The machine and setup must be rigid
- Use carbide grade M20-M25 or P20-P25
- Always use coolant
- Use smallest possible nose radius to avoid vibrations

Cutoff turning

- The machine and setup must be rigid
- Use carbide grade M20-M25 or P20-P25
- Always use coolant

	Cemented carbide		HSS
	Roughing	Finishing	Finishing
Cutting speed v_c (m/min)	90-120	120-160	14-18
Feed f_z (mm/rev)	0.3-0.6	0.05-0.3	0.05-0.2
Depth of cut a_p (mm)	2.0-5.0	0.5-2.0	0.5-2.0

	Cemented carbide	HSS
Cutting speed v_c (m/min)	70-90	14-18
Feed f_z (mm/rev)	0.08-0.10	0.05-0.08

Milling

- Use shortest possible tool length
- Avoid cutting through holes/cavities
- Ensure good chip evacuation. Recutting of chips may cause tool damage

	Cemented carbide		HSS	HSS
	Roughing	Finishing	Slot milling	Face milling
Cutting speed v_c (m/min)	80-110	110-140	8-12	8-10
Feed f_z (mm/tooth)	0.2-0.4	0.1-0.2	0.01-0.09	0.1
Depth of cut a_p (mm)	2-5	1-2		

Drilling

- Always use coolant
- If possible use internal coolant through the drill
- Stable setup is very important when drilling through holes

	Cemented carbide	HSS
	Indexable drill	Twist drill
Cutting speed v_c (m/min)	90-130	5-11
Feed f_z (mm/rev)	0.08-0.15	0.04-0.30

Threading

- Use of a plunge infeed normally gives the best result
- For internal threading reduce the cutting speed by about 30%
- Use coolant

	CC	HSS	HSS
	Single point	Single point	Tapping
Cutting speed v_c (m/min)	50-70	10-15	5-7
Number of passes	7-12	10-16	

Working towards forever.

We work with our customers and partners to create long lasting solutions for the tools of modern life and the world's most critical problems: clean energy, clean water and efficient infrastructure. Because we believe in a world that lasts forever.