Hi Andrew,

G-code toolpath-tolerance of 0,0005 mm are Clean Room-settings, and in everyday manufacturing environment only theoretical. With tolerances smaller as 0,1mm, the effects of tool wear and tear becomes more and more influential. A common practice is to use the older tools for roughing, and the new crisp ones for finishing.

On industrial CNC machines one can often change the tool-parameters on the fly to compensate, were as with home build machines, grbl, TinyG and mach3 this is rare.

QCAD/CAM toolpath is missing some small -but major important- features, to compensate the industrial wear and tear compensation, and for roughing and finishing toolpaths. These small features might improve CAM in a giant leap for laser, milling (and machining).
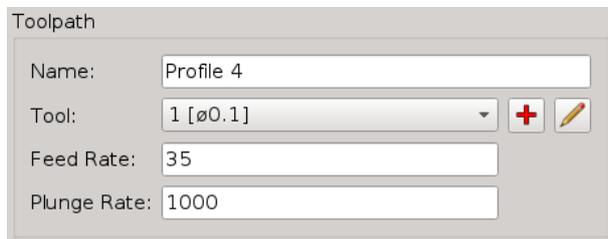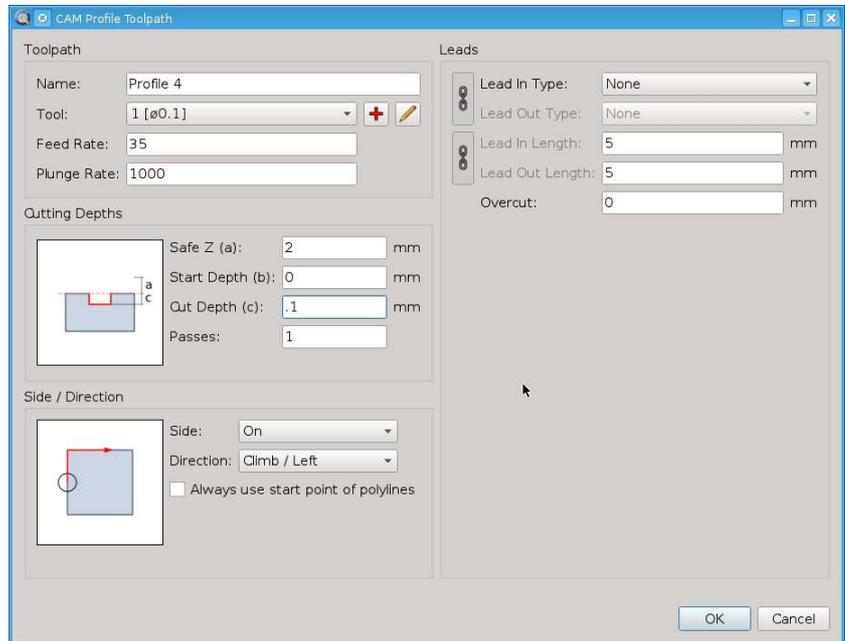
Name:	Profile 4
Path header:	;;; Rough milling inlet chamber
Tool:	1
Spindle Speed:	255
Feed Rate:	35
Plunge Rate:	1000
Path footer:	S0 M05

Adding profile name as comment in G-code.

Field Path header (max 70 char) is intended to be used for adding comments in the G-code-file, however, it can/might also been used for adding specific prefix toolpath G-code commands as Coolant On, G54 ,G90, Set extruder temperatue, or as ....

Field Spindle Speed should be added for more a lot more programming G-code flexibility.
   Example;Roughing and Finishing requires different processing speeds.
        (odd, to make multiple tools for different Spindle Speed-settings)

Field Path footer (max 70 char) ; add as toolptah subfix; see Field Path header

N10 ;Profile 4
N20 ;;; Rough milling inlet chamber
N30 T1 F35 S255 M03
N[last] S0 M05

With these added fields, one could also add empty toolpath's (G0 movements) , for inserting G-code manualy.

The major missing, most important feature are the Untouchable Preset Offset distances. With these parameters G-code for exact tolerance machining, subsequent toolpaths for roughing and finishing comes into reach.
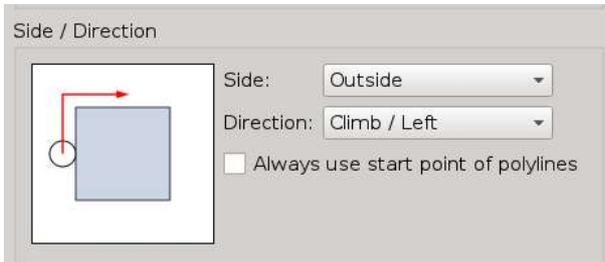
Side: Outside
Direction: Climb /Left
[ ] Always use start point of polylines
[ ] Start at selected entity point (start/mid/end)
Offset Preset UX: 0,3 mm
Offset Preset UY: 0,3 mm
Offset Preset UZ: 0,1 mm

[ ] Start at....   For milling and machining, the CNC operator might prefer a different startpoint.

Its a common CNC practice when 'roughing offset toolpaths' are programmed,
to use preset-offset-(UX, UY, UZ)-settings for leaving enough material on the workpiece
for manual finishing, or for the finishing toolpaths.

Offset Presets values are allowed as 'plus one tenth tool-radius' to 'minus one tenth tool-radius'.
By this limitation toolpath's are only preset offsetted within the toolpath-paramaters.
Example; a 10 mm mill has an Untouchable X Preset range from +1 mm ... -1 mm.

Example; With a tolerance of 0,0005 mm, one can CAM G-code exactly by correcting the
appropiate Preset Value.

Very handy would be a "Copy Toolpath" option, for copying the current activated toolpath, as a new one added to the list. One has only to change the operating toolpath-parameters from Roughing to Finishing.