easyGRIND

Software changes

VERSION 6.13

Machining 2
- Helical Fellow cutter 2
- Mill 3
- Hob 3

Probing 4
- Probing macro 4
- Wheel probing 4
- P-Plate length probing 4
- Safety on the total length 4
- Reamer probing with blade probe 4
- Helix probing 5
- Helix and diameter probing on multi-tooth tools 5
- Inserts: Setup probing on an offset flat 5

Miscellaneous 5
- 3D Simulation 5
- General use 5
- Machine 6

Fixes 6
- v.6.13.0 6
- Patch 6.13.1 **Erreur ! Signet non défini.**

Machining

Helical Fellow cutter



Major improvement to **face sharpening** for the following jobs:
- "Rake Face": new job to use
- "FACE by Peri CYL"
- "FACE by Front Bushing"

- **regarding probing**: the %134 program has been rewritten. It now returns the probed length at the middle of the tooth on the outer diameter, and the setup at the middle of the tooth along the length.

The following 3 parameters have been added:
- "probe pos on face": designates the radius at which the probe will position itself to probe the setup between 2 teeth (default 1 mm).
- *"radius offset 1"*: this is an offset / outer diameter for probing the first total-length point. By default, set to 0. If >0, it will probe below
- "radius offset 2": this is an offset / outer diameter for probing the second total-length point. If 0 -> 2 mm.

When probing the length via a plate, if the Y and Z positions are =0, the offset is applied by the tool radius without displaying an error message

- **regarding parameter setup**:
The parameters "X Offset", "A Offset", "(2) A Offset" are hidden.
The parameter "Primitive helix angle" now appears and allows the helix pitch to be calculated reliably if it is not directly marked on the tool.
The parameters "Rake Face OFFSET" and "Tooth Root OFFSET" in the "Machining" tab allow the toolpath to be adjusted in the same way as by modifying the wheel W and D.

- **regarding machining**:

The global depth of cut caused a rotation in A of the tool (this was reversed)

The default machining position of a left-hand cutter, method D, is now correct

Activation: Numaffut.ini, [CONFIG_OUTILS], Fellow_New_Gestion=1 (1 by default)

Mill

Farewell to "Bench Length"

It is now possible to replace the "Bench Length" parameter with the 2 parameters "Setup dist." & "Setup Angle" for all mills.

Activation: Numaffut.ini, [CONFIG_OUTILS] FRAISE : SUPPRIMER_Longueur_Banc = 1

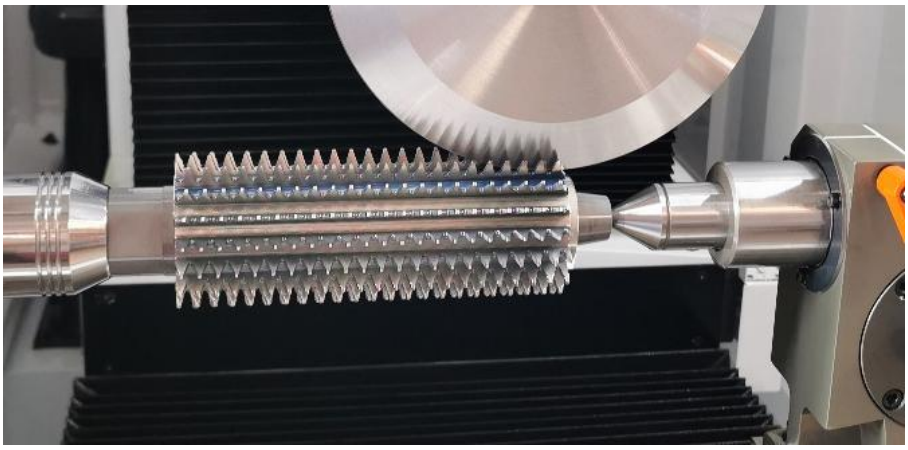
Bug fix on "Radiused tip relief"

The handling of the wheel incline was incorrect (in version 6.12). It is back to being identical to 6.11

Bug fix on the type S "End Flute"

In simulation, type S end flutes could be incorrect (linked to a "large" wheel radius). Note that for this job only, the simulation and the machining in the machine do not

Hob



The many probing and checking options (with a control report printout) are now available as standard in the software.

A machining sequence has been developed, which, thanks to the check performed in the machine without removing the part, makes it possible to very quickly obtain the **expected precision class**.

Improvements have been made to **straightness of cut probing**, and to the handling of measured deviations.

A few fixes:
- when a hob has zero helix, a helix direction equal to the cutting direction is generally declared, with a helix angle equal to 0.001 (right helix) or -0.001 (left helix). During probing, the left helix changed sign, which changed the machining position.

- during straightness-of-cut probing, the reference setup position is critically important for the result obtained, particularly for calculating the corrections to apply. Previously, either the pre-machining setup or the setup probed after machining was used. However, the setup of the last pass must be used to calculate a relevant value

Probing

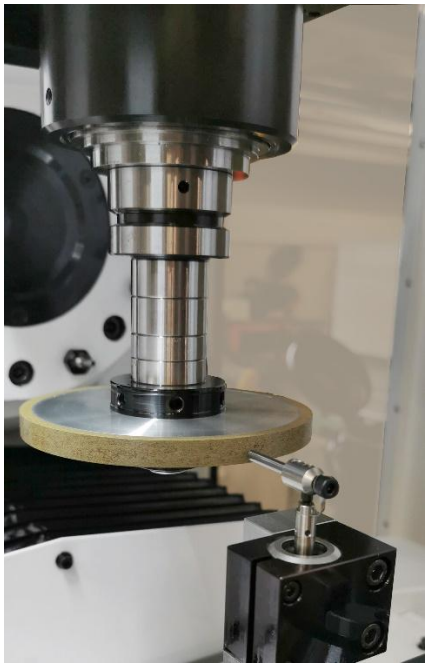
Probing macro

In some cases, this could take more than 20 seconds to generate the probing program. To **remove this waiting time**, the code has been completely reworked.

activation: Numaffut.ini, [Moulinette_Mnemo] Activer=1

Wheel probing

The wheel can now be probed in Y, B at 90°



P-Plate length probing

Ability to specify "Use tool setup". Checking this parameter will add an approximate length probing, which will serve as a basis for all other probing (setup, helix, diameter...). Then, at the end, the probe will use the probed setup (and the helix) to position itself on the tooth and probe the length precisely

Safety on the total length

The safety on the "Start X Pos." parameter, which positions the probe in front of the tool, is based on:
- either a minimum value from the configuration
([MODULE LONGUEUR], Secu_PosX_Mini, default 100mm)
- or the length of the selected tooling +20 mm (arbitrary).

If the "Start X Pos." parameter is less than one of these 2 values, probing cannot be edited.

Reamer probing with blade probe

%113 has been modified to allow probing Length, setup and eccentricity with the blade probe. The correct program can be generated by choosing which probe %113 should

be edited for. By default, with a blade probe, the setup is probed by going in 0.5 mm. Caution: only right-hand OR left-hand cutting reamers can be probed.

Helix probing

Since 6.01, the consistency of the probed helix angle is checked against the declared helix direction. From now on, if the helix angle is small it is considered consistent, and is initialized to 0.001 (right helix) or -0.001 (left helix)

Helix and diameter probing on multi-tooth tools

When the diameter entered in the parameters was very far from the tool's actual diameter, a deviation could occur in the probed helix angles. This problem has been fixed

Inserts: Setup probing on an offset flat

It is now possible to probe the setup on a flat that is offset relative to the A axis

Miscellaneous

3D Simulation

Save the 2D model

It is now possible to save the 2D profile of the machined blank. This can be used to start a drawing
Accessible from the menu linked to the [Simulation] button - Save 2D profile)

Customize the wheel hub

It is now possible to give certain dimensions for the wheel hub:
> Rear diameter
> Front diameter
> Front length

This notably allows better simulation of the space taken up by balancing rings

General use

Broach pitch and progression table

Clicking on the chart at the bottom of the table positions the cursor on the clicked tooth. This makes navigation easier, especially when the broach has many teeth

Saving a tool

For a tool with many parameters (for example a drill with many stages and jobs), the last parameters were never stored. On loading, the following message was displayed: "Generer_OutChainS - The saved parameters may be corrupted. Contact SMP"

Machine

Maximum spindle load

It is possible to specify a maximum spindle load beyond which the machine goes into fault.
Menu->Machine parameters->Machine-> check "max allowable spindle power"

Dressing

The variables containing the wheel diameter wear could conflict during a setup probing on all teeth.

Creation of the entry [DIAMANTAGE] Usure_E83000 = 0, 1 or 2

= 0: wheel wear stored in the E800xx.
the only possible case before this version

= 1: wheel wear stored in E830xx.
This option is recommended for ALL new machines

= 2: wheel wear stored in E800xx and in E830xx
This option allows a transition for customers who already use dressing, but want to make their process more secure.

-Fixed: coolant did not restart after a cycled dressing when the general coolant option "Coolant only at the start of the toolpath" was checked (created for CA7U)

Fixes

v.6.13.0

Zero feed rate bug

A tool opened & saved with version v.6.12.3, then reopened with version v.6.11 could have lost its feed rates

The software freezes!

Commands were lost when editing in the CNC.

Also included …

Sign of the depth of cut in drills
Particularly, in the spiro-conical stage job, the depth of cut could be applied in the wrong direction

RUNTIME ERROR:
Sometimes, the "ValS_I()" message appears and completely blocks the software. This message has been suppressed; it no longer blocks use, but it still appears in the log.

3-LIP FORM MILL (with helix)
The profile was poorly positioned in the flute with the new relief machining methods

CHAMFER MILL
When "Chamfer wheel incl." equals 0, it uses 45 for machining and then modifies the parameter value. Now, it is only initialized to 45 on creation. If the operator sets it to 0, that value will be kept

Tapered mill: EXT roughing form
Used for chip breaker; bug no longer worked since v6.11.5 (E61001 or E62001 were overwritten by the multi-pass)

FORM MILL,
Bug on approach. The approach travel was 0 in FORM MILLS (other form tools were OK), v.6.12 only

FORM GRINDING
The value of the "Continuous A Rotation" parameter entered in the tool parameters was always overwritten by the value entered in the job (even if it was equal to 0)

GRINDING,
Machining with wheel dressing along the Y axis did not work

TRS,
Access violation when closing the software while the TRS option is enabled

CNC Link,
When the varCN.ini file contained a duplicate entry, it crashed the software

3D Simulation,
FEPA toric 1F1R wheel with a small diameter was simulated incorrectly

3D Simulation,
The NCSimul button does not display if the software is not installed on the PC (even if the license is activated)

UI,
When "No. of teeth/No. of passes" was reversed, the counters were incorrect