FILOU NC12

for WinPC-NC

Introduction to FILOU-NC for WinPC-NC

Contents

Why FILOU-NC, I already have WinPC-NC?

WinPC-NC is a machine controller. This means WinPC-NC controls a machine with a NC-program.

This NC-program consists of commands (switching on the spindle, etc.) and a description of the toolpath. WinPC-NC reads and interprets this to drive the machine.

One can write NC-code manually with a text editor. This requires good knowledge of CNC language and mathematics. It's more comfortable to create the NC-program using CAD and CAM programs, such as FILOU-NC.

CAD	CAM	CAD
e.g. AutoCAD, Rhino, CorelDraw etc.	FILOU-NC for WinPC-NC	WinPC-NC light or Economy

The workpiece to be manufactured is defined as geometry in a CAD program. CAD drawings are generally made at a scale of 1:1 and therefore they are perfectly suitable for creating NC-programs with a CAM program.

Many CAM programs offer CAD functions, the geometry can be directly used in the CAM system. FILOU-NC is such a CAM system.
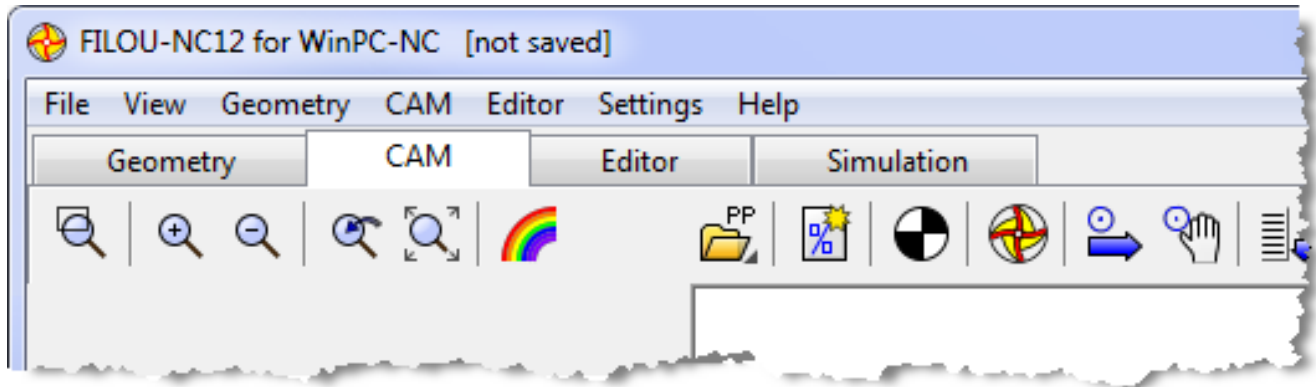
However, it is also possible to import existing CAD files, e.g. in DXF format, and to use them for creating NC-programs. Of course, this imported CAD file can be processed and optimized in FILOU-NC.

The produced CAD drawing defines the workpiece to be manufactured by measurements and outlines. Parameters for the following production are not included. Information about milling distances, speeds, spindle speed, etc. are not available in the CAD file.
This is the role of the NC-program.

This is the reason why direct conversion of CAD geometry into NC-code seldom makes sense, except for some cases, as it would create an incomplete NC-program.

FILOU-NC, as CAM program, does those complex calculations and creates the complete NC-program according to your specification. This works quick and precise.

After completing this procedure the NC-program is simulated in FILOU-NC. Errors which may occur can easily be noticed and the user saves time and money.

As a result you receive validated NC-code. Contrary to NC-codes which are automatically created, this code provides considerably improved and optimized milling paths. Time savings to up to 60 % are usual.

How does FILOU-NC for WinPC-NC work?

FILOU-NC for WinPC-NC means four programs which are closely associated: *Geometry* (CAD), *CAM*, *Editor* and the 3D *Simulation*.

A special feature is that any of the four programs or better areas are independent of each other. You can switch to an other area at any time. For example if you note during programming in the CAM area that the geometry is not complete, it is absolutely no problem to switch back to the *Geometry* area to complete the missing geometry. Characteristic of the four areas:
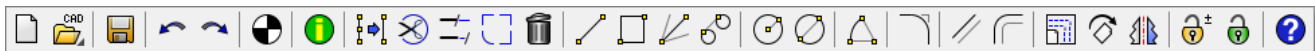
On the *Geometry* page the CAD geometry can be imported, created, processed, optimized and of course exported. The geometry is the base for creating NC-programs.

In the CAM area the toolpaths are derived from the geometry. Furthermore it is possible to generate additional NC-code by macros and cycles.

A NC-program does not only mean coordinates. In order to create the suitable format, there are postprocessors.
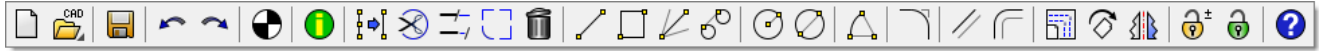
The generated NC-code is written by the CAM area directly into the Editor.
The Editor holds the NC-code through the whole process.

Besides standard edit functions the Editor provides additional special NC functions such as *Renumber* and *Extract tools*.

The Editor becomes highly efficient through the integrated *Work plan.* That makes an immediate and simple amendment of a finished NC-program possible.

```
(94: Contour pocket -30)
N1799 G0 X22.664 Y23.091
N1800 G0 Z1
N1801 G0 Z-22
N1802 G1 Z-30 F400
N1803 G1 X21 Y23.091 F800
N1804 G1 X21 Y23.524
N1805 G3 X22.664 Y23.091 I54 J204.064
N1806 G3 X127.371 Y23.1 I52.336 J204.497
N1807 G1 X129 Y23.1
N1808 G0 Z1

(95: Tool Fasenfräser 8)
N1809 G0 Z100
N1810 M9
N1811 T01 M6 (GUID:F472FFB5-23C3-4F4A-A6C4-3FC!
N1812 S5000 F800 M3
(96: Contour -2)
N1813 G0 X82.113 Y-4.04
N1814 G0 Z1
N1815 G1 Z-2 F2000
N1816 G3 X75 Y-1 I-7.113 J-6.8 F800
```

The 3D Simulation makes the NC-program visible as it would be milled.
It doesn't matter whether the NC-program has been created in FILOU-NC or an existing NC-code has been loaded into the Editor.
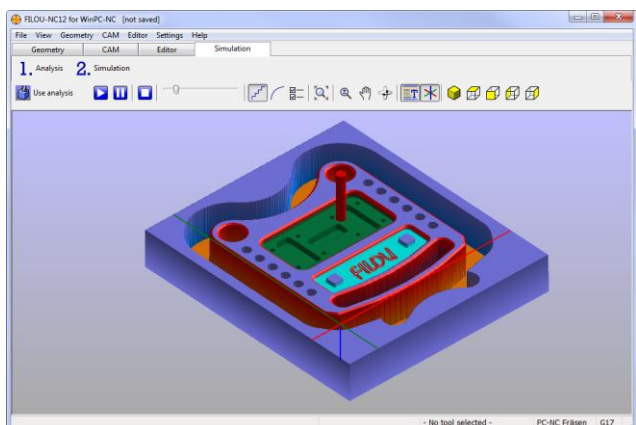
The FILOU-NC-Simulation supports three simultaneous axes.

Support and Help

One hint about the program's help. We have put much work into the help. By various links and animated graphics it is more comfortable than any printed manual.

Furthermore you can use the function key F1 at any time in the program. This immediately displays the corresponding help topic on the screen. A manual is not able to provide such convenience.

This introduction is meant to explain some basics.
You'll find a detailed example in the help menu - First steps.

More help is available on the internet:
http://www.filou.de/service/kontakt/support-anfrage.html

Or visit the FILOU forum to exchange with other users:
http://www.filou.de/index.php?id=76

In three steps to the NC-program

Step 1 - Geometry

First you have to create a CAD drawing with the geometry of the workpiece to be manufactured. It is important that the geometry defines the finished workpiece and not the toolpath to be performed.

The geometry can easily be created in FILOU-NC or imported by a CAD system as AutoCAD, TurboCAD, Rhino, etc. However, it is essential that the geometry has been created at a scale of 1:1.

Creating the geometry, color and layer don't play a role. There is also no requirement for closed contours or for contour elements arranged into one direction.

Important is, if not intended, to have no gaps in a contour.

Smaller gaps can be neglected by a *Gap tolerance* in FILOU-NC.

A well-thought layer structure provides advantages with extensive drawings. The layers can be switched on and off by the Layer manager in FILOU-NC. This brings a better overlook.

If you create the geometry in a CAD program, it is recommended to export it as a DXF file. FILOU-NC has a DXF import filter importing merely lines, circles, arcs and polylines. Other elements are ignored as they are useless for creating NC-programs.

First the imported drawing is cleared up from unnecessary or disturbing elements by the *Clean up drawing* function. This automatic procedure works quick and efficient.

After the geometry is prepared it should be saved as DXF with the *Save drawing* command.

Later on, during the subsequent programming you still have the possibility to correct or change the geometry. This is possible because the NC-code is *derived* from the geometry, and the NC code is not the geometry or connected to it.

Step 2 - CAM

Following statements basically apply to DIN/ISO programs, but most of it applies to HPGL as well.

In the beginning there is the origin.
Why ? The program's origin, which must be identical with the workpiece origin on the milling machine, should not be changed in a single NC-program (exceptions for experts only).

The origin for the X and Y axes is defined in FILOU-NC. If you are asked for values concerning the axis Z later on in the program, these values refer to the Z origin as defined on the machine for the workpiece.

Usually this is the upper plane of the workpiece. However, some users use the lower plane. Generally it does not matter which plane is used, but it must be the same through the whole NC-program. This way you can avoid collisions through unexpected movements.

Using WinPC-NC it has to be noted that the Z-axes is in reversed position contrary to the standard, i.e. minus is on the top and plus is at the bottom. Therefore the Z-axis in WinPC-NC must be turned around (menu Parameter / Data format / DIN-ISO-Z-invert axis).

The next step is to select the postprocessor. This means selection of the format for creating the NC-code.

Now you have the choice between HPGL or DIN/ISO. If you are not sure what that means, see later in this introduction to read about the differences.
Please note that WinPC-NC *Light* can only read and process HPGL programs.

Structure of the program

NC-programs must have a defined structure. The controller must be able to read and understand the content. Usually it begins with *Program start* and ends with *Program end*.

Now we select *New NC program*. A dialog pops up where you can enter a certain program number and a comment (e.g. description of the program). Now this, the beginning of the program, the *Program start*, is ready. Have a look into the Editor to prove it.

The next step is launching the tool. This can be done by activating the tool manager or inside of the milling functions (contour, pockets, etc.).

Now we work on the geometry with the milling functions. The NC-code in the Editor is created automatically. Finally we insert the *NC-Program end*.

Ready.

You can inspect the program in the Editor now.

Editor and work plan

When working in the *CAM* area FILOU-NC registers each process in the Editor together with a corresponding comment. These comments are automatically extracted to the work plan.

It is possible to give these entries meaningful names, to move or delete them. If the comments are not wanted they can be stripped of automatically when saving the file.

3D Simulation

The NC-program in the Editor can be simulated at any time, regardless whether it has been created in FILOU-NC or an existing NC-code has been loaded. First of all the NC code will be analyzed, then the stock part dimensions are defined and the used tools will be assigned. After this the NC-code's operations can be simulated.

Step 3 - Send to WinPC-NC

After the NC code has been completed and the Simulation is ok, the project can be saved. Then the NC-program can be directly sent to WinPC-NC.
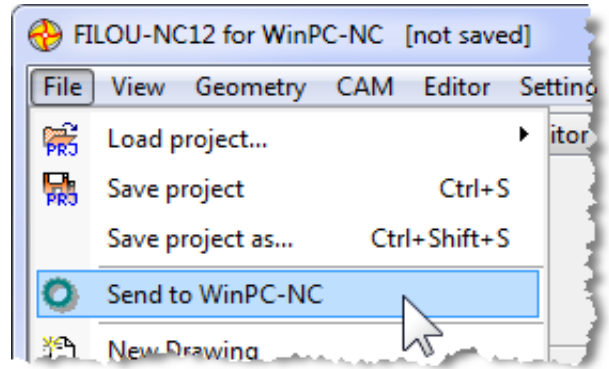
Teamwork: WinPC-NC and FILOU-NC

Z-axis

The standard setting for the Z axis in WinPC-NC is in reversed position, i.e. minus is top and plus is at bottom. Therefore the Z axis has to be turned around in WinPC-NC.
(Menu: Parameter / Data format / DIN-ISO-Z-invert axis)

Tool numbers

Please note that WinPC-NC accepts only tool numbers from 0 to 9. This is important to know when you create new tools. Other settings in the WinPC-NC menu Parameter / Data format should remain deactivated.

DIN-ISO and WinPC-NC *Light*

Attention: Only HPGL programs can be processed by WinPC-NC *Light*.
To create NC-code for WinPC-NC *Light* select a HPGL postprocessor.

Feed and spindle speed with DIN-ISO code

Using the calculation of cutting data in the FILOU NC tool data, the spindle speed is revolutions per minute.
The feed rate is in mm/min.

The units mm and mm/min. have to be set up in WinPC-NC accordingly.
(Menu: Parameter / Coordinates / Units)

Units

The unit for DIN/ISO code has to be 1/1000 mm.
(Menu: Parameter / Coordinates / Units)

Example: Import and prepare geometry

1. ⌷ Load the drawing
Select the desired file format and load the file SAMPLE.DXF from the samples folder.

2. ⌷ Clean up drawing
This step should be done generally when a drawing is loaded. This function performs four tasks:

- Remove identical elements
  Deletes elements of exactly the same size. Such errors may occur during copying or alike in the CAD program.

- Remove overlaying elements
  Elements next to each other or overlapping elements are reduced to a single element. This is often the case with drawings derived from 3D.

- Full circle arcs to circles
  If an arc includes 360° it will become a circle.

- Remove small elements
  Elements below a minimum length are deleted from the drawing. This is also often the case with drawings derived from 3D.

3. ⌷ Close gaps
You can close larger gaps with the *Trim* function.

Example: Milling contours

1. 📂 Load and prepare the drawing
See Import and prepare geometry.
Switch to the *CAM* page.

2. 📂 Select postprocessor
First you have to select the desired format: DIN/ISO or HPGL.
In this case we'll use DIN/ISO.

3. 📊 Create a new program
Please enter a program number as well as a brief description of what the program does.
Click *OK* to write the program start along with your data into the Editor.

4. ⊕ Set the origin
Set the origin for the X and Y axis. Activate an *Object snap* you want to use and select the found point with a left click.

5. ✪ Insert a tool
Call up the *Tools* and select the required cutter. If the material you want to mill is already defined you can also auto-calculate the resulting feed rates and spindle speed.

6. ➡ Contour trace
When tracing contours the toolpath will be calculated from the part's geometry and the used cutter. FILOU-NC finds the possible path automatically. Please have a look at the left of the status bar, the messages come there.

Important
The *Contour trace* function is able to process open and closed curves.
Contour processing starts always at the end or the mid point of an element, i.e. at the end of a line or an arc. From this position FILOU-NC moves to the other end of the element and checks whether there is an adjacent element with one of its ends. If this is the case, the element is automatically adopted, etc. If FILOU-NC finds several fitting elements, the message *Click on following element* is displayed in the status bar.
Click to the correct element, and the contour trace continues running as far as possible.

The *Contour trace* function always wants to route all elements of a contour. This means that there are two reasons why a contour can't be processed: The cutter is larger or of the same size as an element of the contour or an element can't be processed at all. Then a dialog pops up: *Contour is not possible*.
Select a smaller cutter or change the contour.

Hint: Contour gaps
The *Tolerance for following elements* can be adjusted in the *Settings* under *Contour tracing*. To find adjacent elements they have to have a common point. A small deviation is allowed.

Additional settings:

- Confirm
  If this option is active as recommended you have to confirm every traced path to write the resulting NC-code into the Editor.

- Release used elements
  Used elements are locked during contour tracing. These elements are displayed in bright green and are invisible for further functions. If the option *Release used elements* is active, these elements are immediately released after contour tracing.

- Offset correction
  To obtain a workpiece corresponding exactly to the size of the drawing, the cutter has to stay besides the contour. This procedure is called *Offset correction*. There are three possibilities in FILOU-NC:

  > to the left of the contour
  > onto the contour (without correction)
  > to the right of the contour
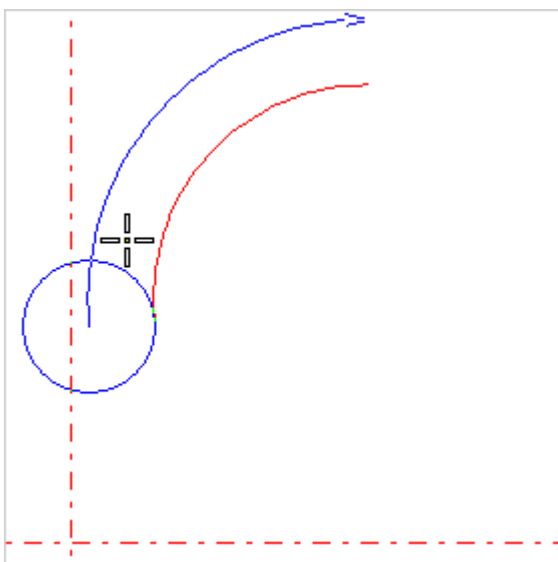
Right and left is referring to the milling direction which is defined by the starting point of the contour as already explained. Move the cursor over the geometry. FILOU-NC displays the current path correction. A click with the left mouse button confirms your selection.

Once more we want to remind you of the program's help which comes up by pressing F1 and provides you with direct support. Even the graphics you see on the left is from the help.
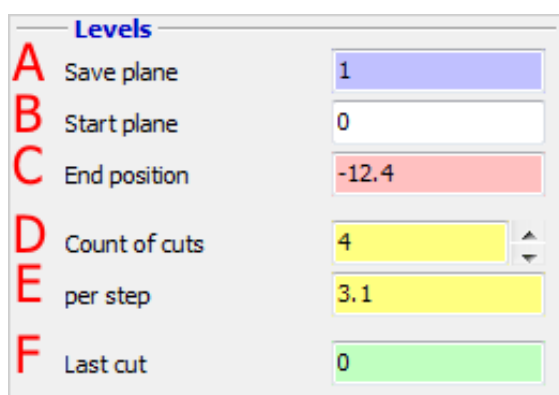
Until you start the Contour trace by the left click you can change all the settings as you like.
If all is set, give a left click and do as described before.

Then you can generate more NC-code. After all is finished call the dialog *NC-program End*.

Ready.

Simulate the created program

The *Simulation* analyzes the NC-code in the Editor. Then the milling process will be shown like a movie.
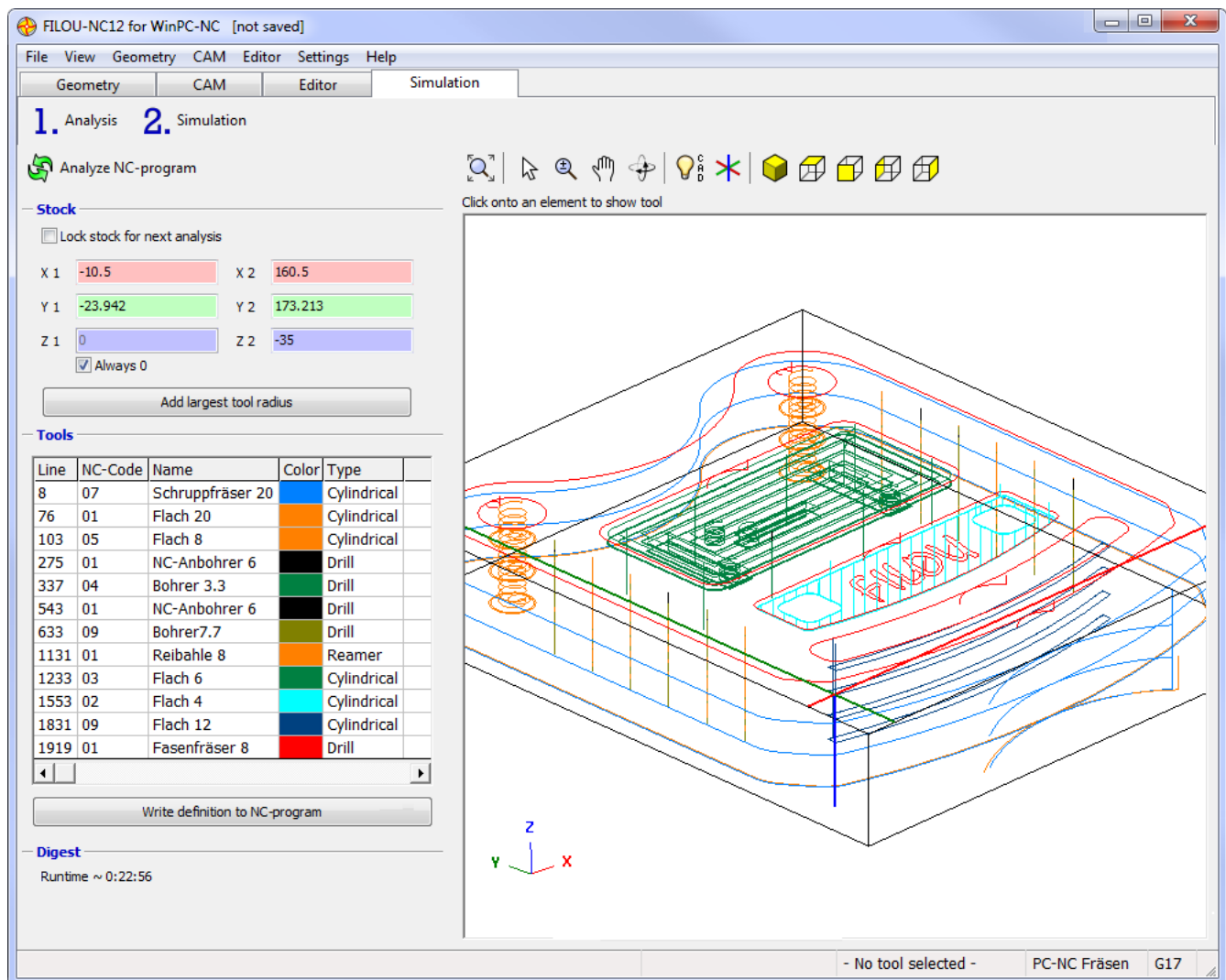
1. Step: Analysis of the NC program

First click Analyze NC-program. Then FILOU-NC displays the toolpath. On the left side you can enter the size of the stock material.

FILOU-NC calculates the stock from the toolpath extents. The coordinates refer to the origin.

When a tool was not detected or you want to change it you can click the *Change* button in the *Tools* list. This may be necessary when the NC-code to simulate was not built with FILOU-NC.

2. Step: Simulate NC-Program

When all is set select *Simulation*.
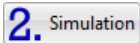
In the Simulation window click *Use Analysis*.
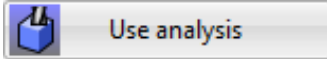
The time required for calculating the 3D simulation depends on the size of the workpiece and the adjusted precision. The smaller the workpiece, the finer the precision may be.
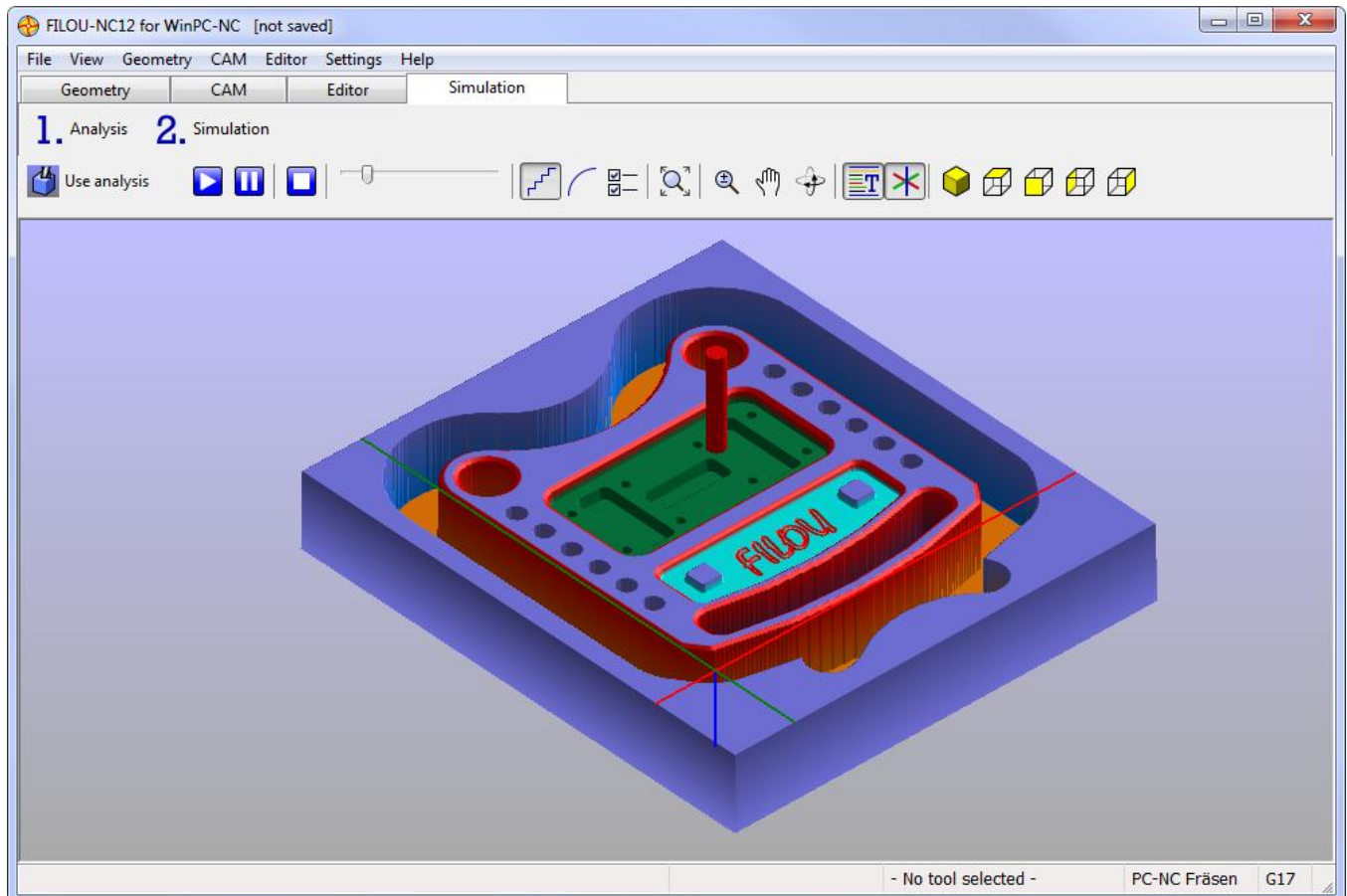
Existing NC-programs can also be simulated.

To do so, load the NC-program into the Editor and assign the tools manually.

It is possible to simulate NC-programs with max. three simultaneous axes.
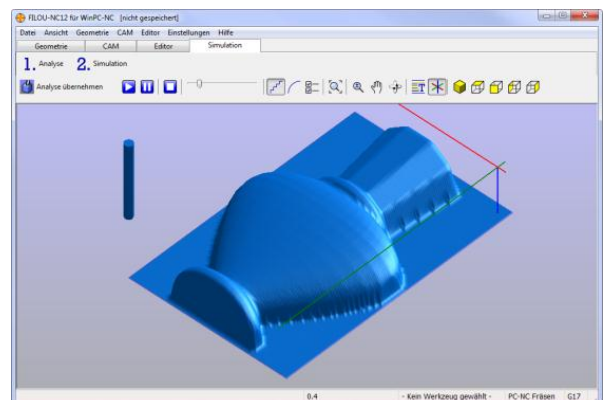
DIN-code or HPGL?

We are often asked: What's the difference between these two languages when it comes to NC machining? Some general and also subjective information and thoughts:

HPGL was born as a language for pen plotters only. This is why it's called HPGL (Hewlett-Packard-Graphic-Language). It is a description of pure 2D geometry, with the only option pen up or pen down for the third axis.
Arcs and circles are split into many small lines. The plotter draws, depending on the color, with a certain pen. Later on HPGL/2 came up with some more comfort and possibilities, but in principle it is the same. As the plotter pens are directly drawing the geometry, there is no way for path or tool correction.

One characteristic is worth mentioning: The smallest resolution is 1/40 mm respectively 0.025 mm. Some rare programs or machines provide / require a HPGL file with the smallest possible resolution of 0.0254 mm which is 1/1000 inch.

Simple creation of the HPGL format is a great advantage. It can be produced by nearly all available graphic programs supporting a (HPGL) plotter. A disadvantage is the restriction to 2D paths only. Drilling is not possible with pure HPGL code.

Relatively coarse resolution, working in one plane only, missing correction of the cutter radius and sometimes the quite irregular connections of the contours, unnecessary pen change is normal. Therefore it makes sense to optimize the HPGL code with FILOU-NC.

Some manufacturers of controllers have modified HPGL in a way that 3D can be used. However this process is very specific to the controller and cannot be produced by CAD or graphic programs.

The values in HPGL range from plus/minus 32768 (16 bit) and results in a max. traveling path of 2 x 32768 / 40 = 1638.4mm per axis.

This should cause no problems, as actually there are a lot of controllers accepting larger numbers.

DIN code, also called G-code, is defined according to DIN 66025 and is considered as standard in professional NC programming. Many industrial controllers use DIN programming or elements of it as basis, completed by individual functions.

There are so-called G functions, e.g. for coordinates, cycles, selection of the plane, path correction, etc.
M commands are help functions mostly, they switch the cooling fluid on and off or alike.

This is enriched by sub-routines and 3D processes and much more. Generally programming is done in resolution of 0.001mm.

DIN code offers a wider range of possibilities than HPGL
DIN code provides much more possibilities than the relatively simple HPGL format and does not have such disadvantages as HPGL. However, creating NC-code in the DIN format requires basic knowledge of the language. The DIN format is clearly structured and easy to learn.

Conclusion
HPGL is ok for simple 2D contours. For a better control (e.g. think of the depth) and absolute accuracy, the DIN programming is recommended.

FILOU-NC and WinPC-NC
By special postprocessors FILOU-NC is adapted to WinPC-NC as stepmotor-controller. This is the case for HPGL as well as for the DIN code. FILOU-NC generates NC-Code which can be read and processed by WinPC-NC. FILOU-NC provides the whole functionality which can be used by WinPC-NC.

FILOU-NC und HPGL
FILOU-NC creates an optimized HPGL code.

It is possible to trace contours, completely or partly, in a defined direction. The toolpath with corresponding path correction can be created for the selected tool immediately. By contour tracing the contours are automatically combined and unproductive moves are minimized. This way all contours for a defined tool can be machined. Afterwards you can create the toolpath for the next tool. No unnecessary tool changes.

DIN/ISO commands

Following DIN/ISO commands are supported by WinPC-NC:

G-commands
G00	Linear interpolation in rapid speed
G01	Machining, linear interpolation
G02	Clockwise circle interpolation
G03	Counter-clockwise circle interpolation
G04	Dwell time, time with F-command
G17	Plane XY
G18	Plane XZ
G19	Plane YZ
G70	Measurements in inch
G71	Measurements in mm
G90	Absolute position information
G91	Relative position information
G98	Define subroutine

M-functions
M00	Program halt
M02	End of program
M03	Spindle on, clockwise rotation
M04	Spindle on, counter-clockwise rotation
M05	Spindle off
M06	Change tool
M07	Mist on
M08	Coolant on
M09	Coolant off
M16	Wait for input, input number in F
M30	End of program
M66	Change tool, same as M06
M70	Set output 100 to HIGH
M71	Set output 101 to HIGH
M72	Set output 102 to HIGH
M73	Set output 103 to HIGH
M74	Set output 104 to HIGH
M75	Set output 105 to HIGH
M76	Set output 106 to HIGH
M77	Set output 107 to HIGH

Commands
S	Spindle speed
F	Input number (with M16 command)
F	Dwell time (with G04 command)
F	Feed (Unit mm/min, mm/s, or inch/min)
I,J,K	Circle parameters
T	Select tool
X,Y,Z	Coordinates
U,V,W	Coordinates for 4th axis, if activated
A,B,C	Coordinates for 4th axis, if activated
R	Radius of arcs
L	Call subroutines (1...20)

Other
N   Block number

DIN/ISO example program

%Name	Start of program
N001 G90	Absolute coordinates
N002 G71 T1 M6	Tool change
N003 G00 X110 Y100 Z10	Rapid move to first position
N004 G01 Z-1	Plunge movement with Z
N005 G01 X190	Feed movement in straight line
N006 G03 X200 Y110 J10	Arc about center point
N007 G01 Y190	etc...
N008 G03 X190 Y200 I-10	
N009 G01 X110	
N010 G03 X100 Y190 J-10	
N011 G01 Y110	
N012 G03 X110 Y100 I10	
N013 G01 Z10	
N014 G00 X0 Y0 Z0	Rapid speed to the origin
N015 M30	End of program

HPGL commands

Following HPGL commands are supported by WinPC-NC:

PA [x,y[,x,y]]	Move to absolute position(s)
PR [x,y[,x,y]]	Move to relative position(s)
PU [x,y[,x,y]]	Lift and move tool
PD [x,y[,x,y]]	Lower and move tool
SP x	Select tool x
CI r[,a]	Draw complete circle
AA x,y,a[,c]	Absolute arc
AR x,y,a[,c]	Relative arc

HPGL example	
PU	Start by lifting tool
PA1000,1000	Then move to 1000/1000
SP1	Select tool 1
PD	Lower tool
PA1000,2000	Outline square...
PA2000,2000	
PA2000,1000	
PA1000,1000	
PU	Lift tool

Version comparison

	Basic	Expert	Open	PCNC-Basic	PCNC
Geometry	yes	yes	yes	yes	yes
Editor	yes	yes	yes	yes	yes
Backplot	---	yes	yes	---	---
Transfer	---	yes	yes	---	---
3D Simulation	yes	yes	yes	---	yes
Detailed analysis	---	yes	yes	---	---
Macros	---	yes	yes	---	yes
Own Macros	---	---	yes	---	---
Cycles	---	yes	yes	---	yes
Own Cycles	---	---	yes	---	---
Postprocessor customization	---	---	yes	---	---
Own Postprocessors	---	---	yes	---	---
CAM					
Contour trace	yes	yes	yes	yes	yes
All Contour pockets	partly	yes	yes	partly	yes
Rectangular pocket	---	yes	yes	---	yes
Circular pocket	---	yes	yes	---	yes
Pattern milling	---	yes	yes	---	yes
Batch processing	---	yes	yes	---	yes
Oscillating	---	yes	yes	---	yes
Milling text	---	yes	yes	---	yes
All drilling strategies	partly	yes	yes	partly	yes
Free milling	yes	yes	yes	yes	yes
Batch	---	yes	yes	---	yes
Pattern milling	---	yes	yes	---	yes
Tool database	yes	yes	yes	yes	yes
Send to PCNC	---	---	---	yes	yes
Send to other program	yes	yes	yes	---	---
Postprocessors					
All	yes	yes	yes	---	---
PCNC	yes	yes	yes	yes	yes
HPGL	yes	yes	yes	yes	yes

Glossary

CAD
Computer Aided Design (CAD)

CAD 2D
2D CAD systems are vector orientated drawing programs. Drawing elements are points, lines, polylines, circles, arcs, splines. The process is similar to the conventional drawing board work (outdated). FILOU-NC imports 2D DXF files and creates 2D geometry.

CAD 2½D
This is not real 3D and is also called wireframe model with spatial arranged vectors, e.g. a 3D polyline. FILOU-NC imports 3D polylines from DXF files.

CAD 3D
The geometry is defined by surfaces and solids.

CAM
Computer Aided Manufacturing (CAM) means fabrication supported by computers. CAM is a program which makes it possible to create NC code from the CAD geometry in order to actuate machines. Therefore the NC code is derived from the CAD geometry. FILOU-NC is a CAM program.

CAM 2½D
2½D CAM programs are calculating the tool distance parallel to the 2D geometry. The tool is working in Z-direction. Two axes are moving simultaneously at a given height. The Z-height is defined by the user. FILOU-NC provides 2½D milling routines.

CAM 3, 4 or 5-axes simultaneously
To machine 3D freeform surfaces a CAM software is required which is able to create NC-code for three, four or five simultaneous axes.
For this application we offer DeskProto and madCAM.

CNC, NC
A Computerized Numerical Control (CNC) means electronic and/or software for controlling machines. CNC comes from NC. Today both terms, CNC and NC, are equally used. CNC controllers are differentiated by the number of simultaneously movable axes.

Postprocessor
The postprocessor is responsible for creating the correct formatted NC-code. In FILOU-NC for WinPC-NC the post processor is selected in order to create a NC-program either in DIN/ISO or HPGL format. In FILOU-NC the postprocessor is selected at the start of the NC-programming.