Laurent NAERT
Ingénieur en productique

Operating manual for software for controlling machining units PHOENIX and ORION

LN 13/11/2016

72, rue de la colonie - 75013 Paris - Tél. 01 45 88 89 06 - Fax 01 45 81 56 39
Email support@naert.com - Siret 345 244 826 00045 - APE 6201Z

Warning

Depending on the configuration of the machine, certain functions in this manual could be modified or invalidated.

This software and its documentation are referenced by the Agence pour la Protection des Programmes [Program Protection Agency] under the number IDDN FR.001.040018.001.R.P.2005.000.30600

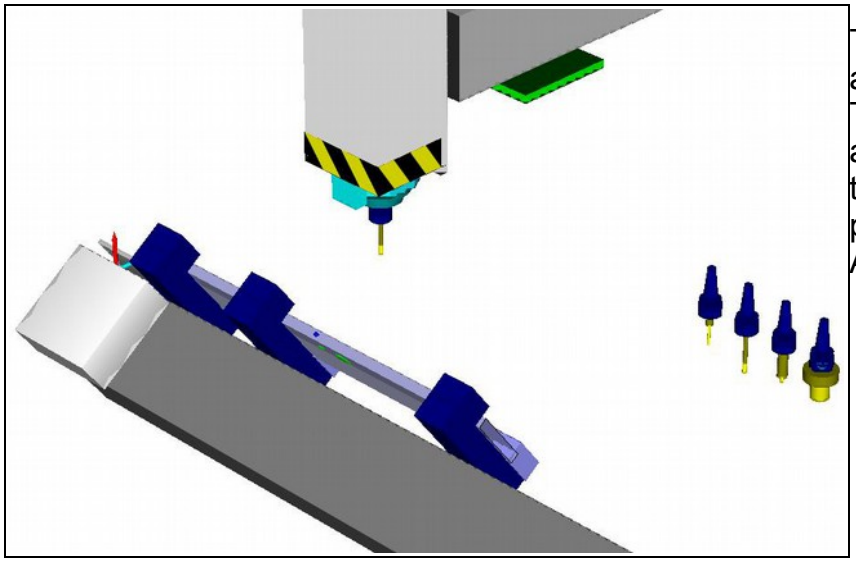
Date	***Author***	***Modifications***
04/02/12	LN	Correction of the “tool-file” diagram
28/03/20	LN	New tool file
28/03/20	LN	Enriched batch-file (format no.3)
13/11/20	LN	New type of machining (Notch)

1 Présentation

The control software has two main functions:

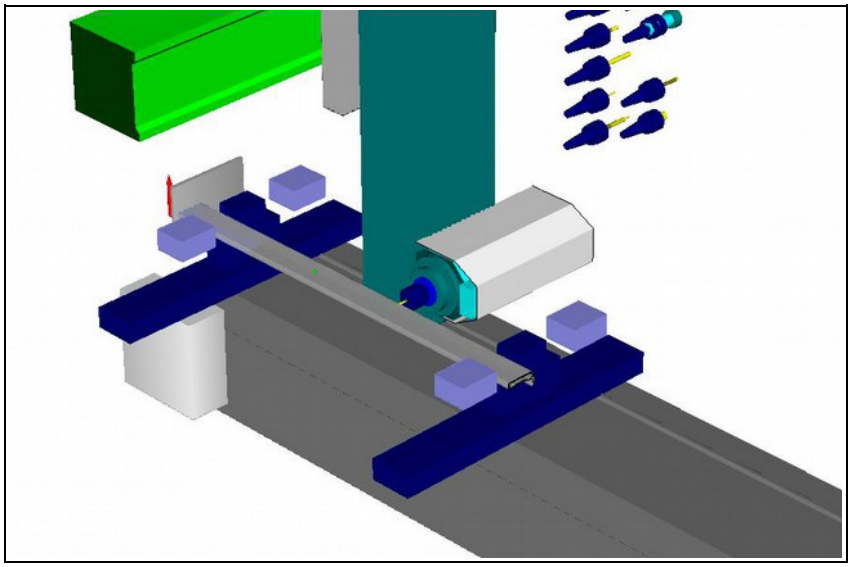
- The man-machine interface displays the state of the machine (position of axes, status of actuators, program being executed, etc.) and transmits the orders of the operator (manual movements, program execution, etc.) to the robot
- The post-processor prepares a machining program from the description of the part to produce and a library of tools, profiles and machining operations.

It controls the 4 axis Phoenix or Orion LGF machining units.



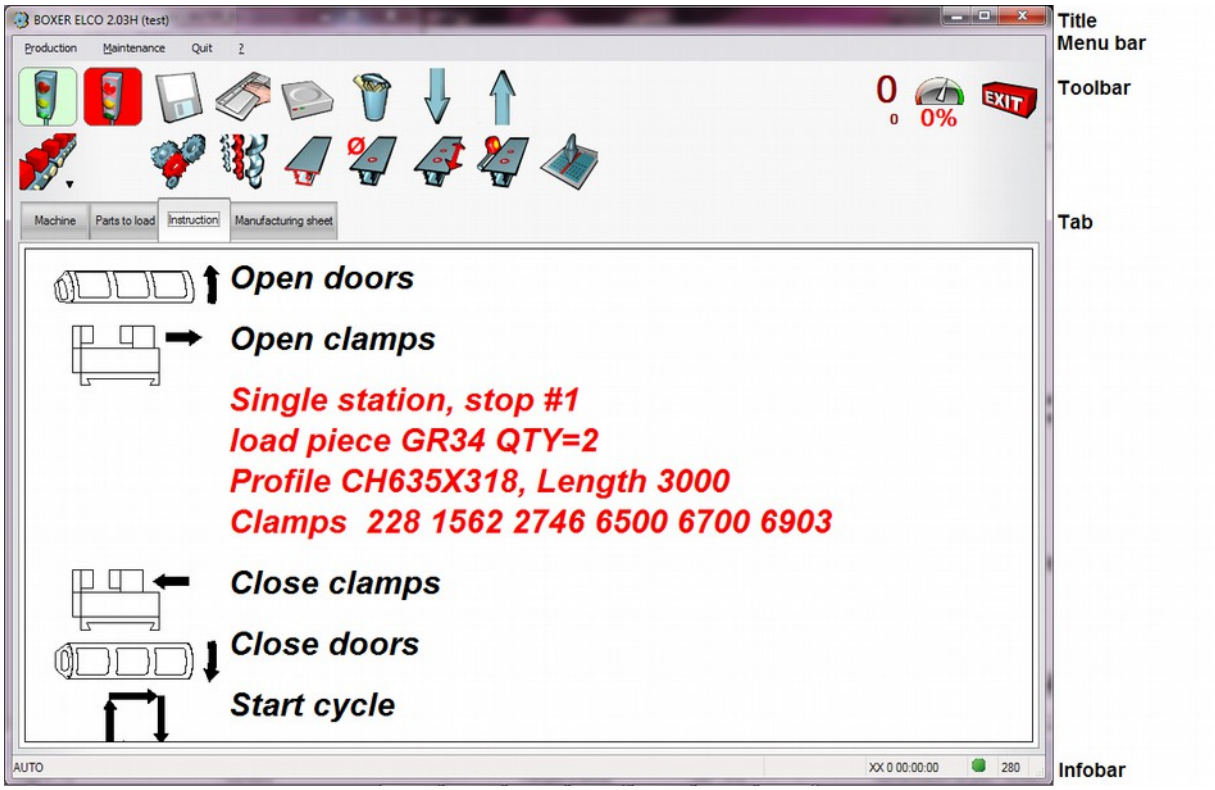
The **PHOENIX** machines are single station units. The spindle is carried by a 3 axis (XYZ) carriage, the part is fixed on a pivoting table around the A axis.

The **ORION** machines are single or double station units. The spindle is carried by an XYZA carriage, the part is fixed. The capacity of the Orion units is greater than that of the Phoenix units. They cab be equipped with a double vice to machine 2 parts in parallel or a wide vices for machining panels.



On the PHOENIX machining units, the first generation piloted by a DOS PC, the man-machine interface is provided by a specific software called TECMOTO.

2 Main window of the control software



2.1 Title bar

The title bar indicates the version number of the software and its operating mode ‘the indication “test” indicates a demo version which cannot be connected to the machine)

2.2 Menu

The software functions are classified in two menus, “production” (see §4) and “maintenance” (see §5).
Access to the maintenance menu is protected by a password, entered in the Software Information Window (see §6).

2.3 Toolbox

The buttons of the toolbox provide direct access to the main software functions. From top to bottom and from left to right, you will find:

- Green and red lights to start and stop the cycle
- Buttons allowing you to constitute a list of parts to machine
- The parts counter and the feed potentiometer
- The software closing button
- The choice of working mode (manual, automatic)
- Access to machine parameters, tools, profile, machining, etc.

2.4 Tabs

During the use of the machine, different information is available. It is displayed in 4 tabs:

- Machine = Man-Machine Interface (MMI) in manual mode
- Loader = list of parts to machine
- Instructions = operator messages
- Fabrication file = detail of part selected on the loader

2.5 Message bar

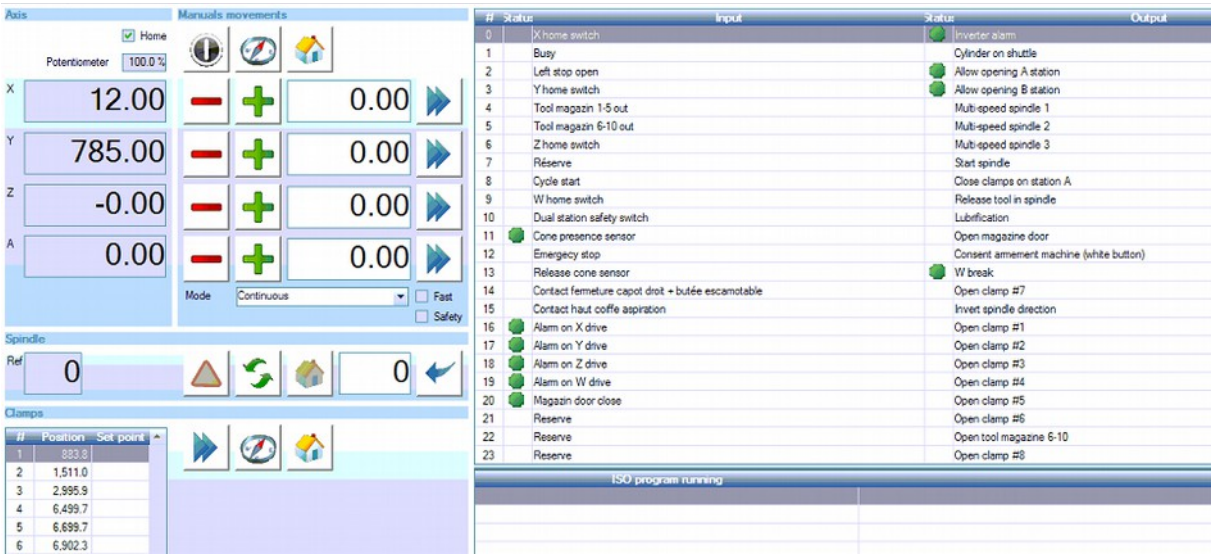
At the bottom of the screen, you will find from left to right:

- The help Message
- The counter and chronometer
- Sign-of-life (indicator of communication with the robot)

•

3 Man-Machine Interface

The Machine tab displays the status of the machine and allows you to command the elementary movements in manual mode.
This tab is active on machines equipped with an ELCO robot. It is hidden on those piloted by a DOS PC (First generation Phoenix)



The “Axis” frame provides the status of the POM indicator (a tick indicates that the origin has been set), of the feed potentiometer and the current position of the 4 axes of the machine.

The frame “Manual movements” allows the axes to be commanded:


Authorises the resetting after a loss of power;


Launches the origin setting cycle


Brings the carriage to the parking position


Moves the axis by one step in the negative direction. The step value is chosen by the “mode” selector (unlimited, step of 10mm, 1mm, 1/10mm or 1/100mm);


Moves the axis by one step in the positive direction.


Move the axis up to the position entered;

The “rapid” box allows the speed to be selected (1m/min if not ticked, 5m/min if ticked);
The “secu” box allows the outputs to be controlled (if ticked, a double click on the output, changes its status);

The “spindle” frame assembles the functions related to the spindle and to the tool magazine Number of tool in the spindle

 Release the tool (this function requires the “secu” mode)

 Open or close the magazine

 Place the tool in the magazine. This function is only valid if the POMs have been done.

 Force the tool no. into the spindle

The frame ‘Vices” groups together the functions related to the positioning of the vices
The current position table for each vice (measured from the machine origin to the centre of the vice) and the requested position

 Sending of the vice to the requested position

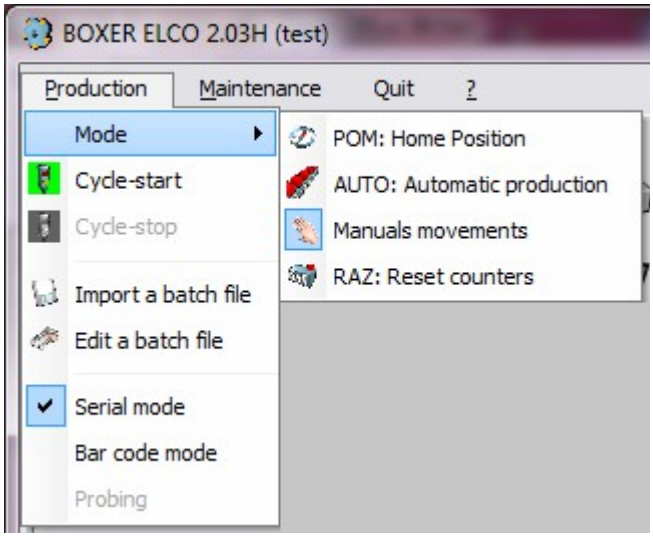
 Start the vice detection cycle

 Open the vices and confirm the closing push-buttons

The input-output frame allows the input-output status of the robot to be displayed. A green circle  indicates that the input or the output is active. A double click inverts the output status (this function requires the “safe” mode).

The frame “ISO program” displays the last program transferred to the robot. The line selected is that being executed.

4 Menu Production



4.1 Mode

The working mode is selected from the main menu (production/mode menu) or by clicking on the “mode” button: in this case, the different possible modes are proposed successively.

4.1.1 POM (origin setting) mode

The green-light (start-cycle) starts an origin setting cycle. The 4 axes XYZ and A rejoin an electrical contact and the carriage U detects the position of the vices. At each starting of the software, an origin setting is required before launching the automatic cycle.

4.1.2 RAZ (reset) mode

The green light resets parts counter to zero.
The RAZ mode is only validated if the parts counter is displayed (machine parameter no.207).

4.1.3 MANU mode

The green light activates the commands manually (“machine” tab)

4.1.4 AUTO mode


The green light launches the production cycle. The system will machine the parts listed in the “loader” tab.

4.2 Start and stop-cycle

Once the mode is selected, the green light launches the cycle and the red light stops it. The background of the button indicates the state in progress:

Illustration 1: Program running the cycle

Illustration 2: Program stopped .

4.3 Importing a remote batch file

The batch file describes the parts to produce. They are described in job-programs upstream of the control software. The possible formats are described in the appendix.

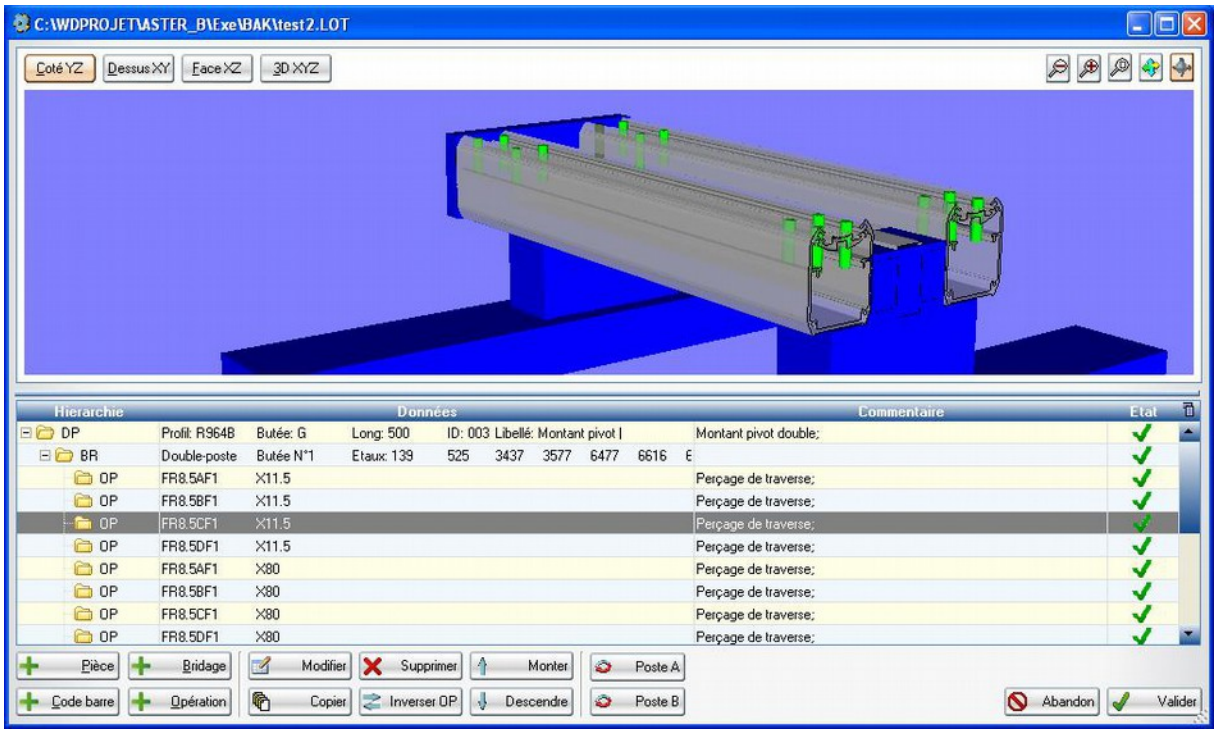
During the importing of the batch file, the control software performs the following operations

- Reading of the batch file in a remote dossier
- Calculation of the positions of the machining operations if the batch file calls on configured parts
- Calculation of the vice positions
- Reading of the detailed batch file in a local file
- Preparation of a manufacturing file per part
- Display of the list of operations not able to be done (clamping stresses not complied with)
- Addition of parts of the batch at the end of the “loader” list if the software is not working in bar code mode

4.4 Editing a local batch

This function allows you to create or modify a batch in the local work directory.

The batch is displayed in a hierarchical table. The buttons under the table allow it to be modified. The part selected is represented graphically above the table.



4.4.1 PD= Part data:

Profile: Name of profile in which the part is machined.
Stop: side in stop (L = Left, R = Right, X = double loading, one part on the left, the other on the right)
Length: length of the part
ID= part identifier, used for calling it in bar code mode Description: description of the part (free text intended for the operator)

4.4.2 BR = Clamping

Job: single or double station. The double station is only possible on Orion units on condition that the length of the part is less than the length of the station. It allows a station to be loaded during machining on the opposite station.
Stop: number of the supporting stop
Vices: position of the vices (measured between the origin of the machine and the centre of the vice)

4.4.3 OP = Machining Operation:

Name: Name of the operation or group of operations
X: longitudinal position of the operation (measured between the left end of the part and the X origin of the operation)

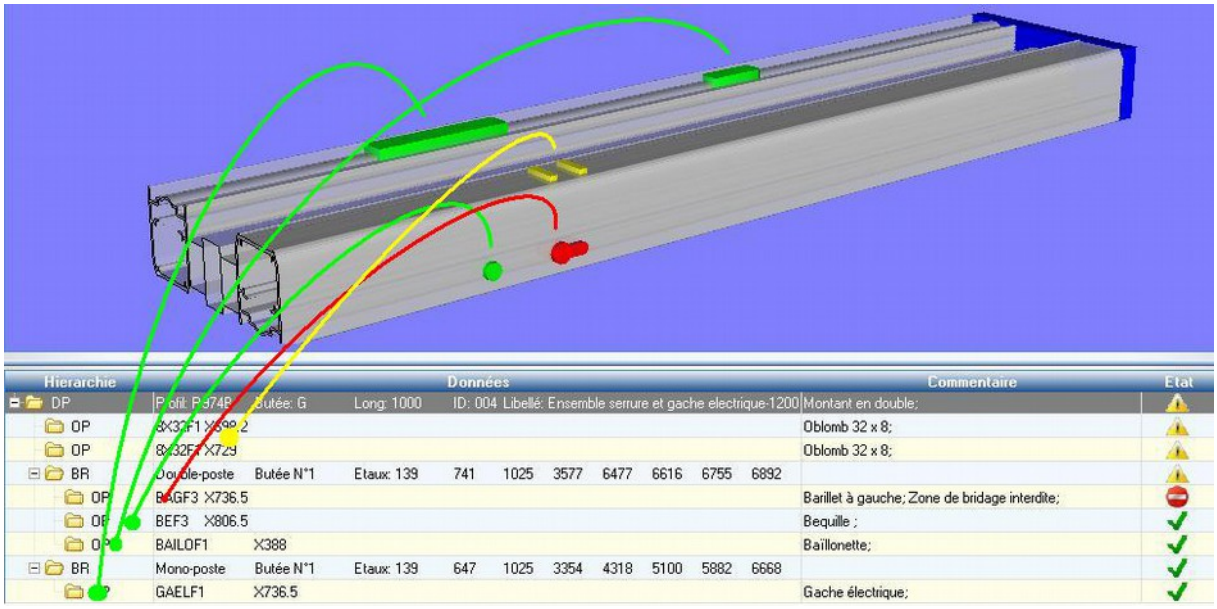
4.4.4 Hierarchy:

The clamping and the operations are associated with a part.
The operations can be associated with a clamping or directly to the part. The latter will not be done by the machine. To do this, a machining operation must be performed to a clamping and must comply with the conditions specified in the machining declaration (min/max distance between the operation and the vices).

The order of clamping operations and operations will be respected by the program. The software sorts the operations so as to minimise the number of movements of the vices, then the
number of tool changes. The arrows allow the programmer to modify the order of the original sorting.

The status indicates if a part or a machining can be done -green tick = OK, orange or red symbol = danger or interdiction). If it cannot be done, the comments can explain why.

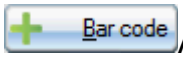
E.g.



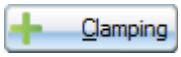
The part is produced in a profile R974B of 1m in stop on the left. The view was turned to display the operations performed on the rear face;
The 2 yellow oblongs 32x8 are not assigned to a clamping. They will not be machined;
The vices position themselves at 138, 741, 1025 etc. for the first clamping; the red barrel cannot be machined as too close to the vice; The strut and the green bayonet are machined;
The vices position themselves at 138, 647, 1025, etc. for the second clamping; The green electrical strike plate is machined.

4.4.5 Actions on the batch file:

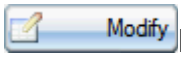
Add a part. You chose the profile or a configured part in a hierarchical family/profile/configured-part table. You can chose a profile or a configured-part. You then enter the length, you chose the stop and the values of the possible parameters. If you have chosen a configured-part, the machining operations are positioned automatically. Otherwise, you must add them one by one.



Add a part from amongst those in the memory (selection by identifier)

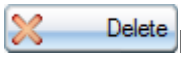


Add a clamping. You select the stop and position the vices. Add an operation from amongst those declared for the profile.

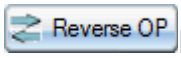


Modify the line selected (part, clamping or operation)

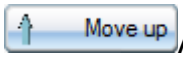
Copy the line selected. A multiple copy is proposed if you select an operation.



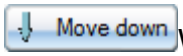
Delete the lines selected



Invert the positions of the operations on the line selected. The inverted position is equal to the length of the part minus the original position. E.g. On a part of 1m, OP AAA X200 becomes OP AAA X800. If you select a part (line PD), all the operations on the part will be inverted.

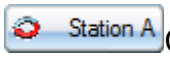


Advance the line selected. The order of clamping operations and operations of the batch will be respected by the program. The arrows allow the programmer to modify the order of the original sorting.

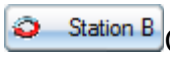


Withdraw the line selected.

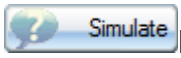
Calculate the clamping (Phoenix)



Calculate the clamping on station A (Orion)



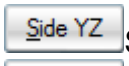
Calculate the clamping on station B (Orion)



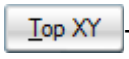
Launch the program simulation.

4.4.6 Actions on the graphical representation:

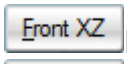
The part displayed is that on which the cursor in the hierarchical table is found
If the cursor is on the PD line, all the operations of the part are shown. The vices are not shown.
If the cursor is on a BR line, the vices and the clamping operations are shown. The clamping operations are those made without moving the vices. In the hierarchical table, it is the OP lines which depend on the BR line.
If the cursor is on an OP line, the vices and the pass operations are shown. The operation selected is displayed highlighted.



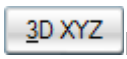
Side view. The profile is at the end.



Top view;



Face on view;



Perspective view;

Zoom in; each press “advances” the image;

Zoom out; Each press “withdraws” the image;

Frame zoom; The frame is defined by its diagonal using the mouse; Translation;

The translation is defined using the mouse;

Rotation; The rotation is defined using the mouse;

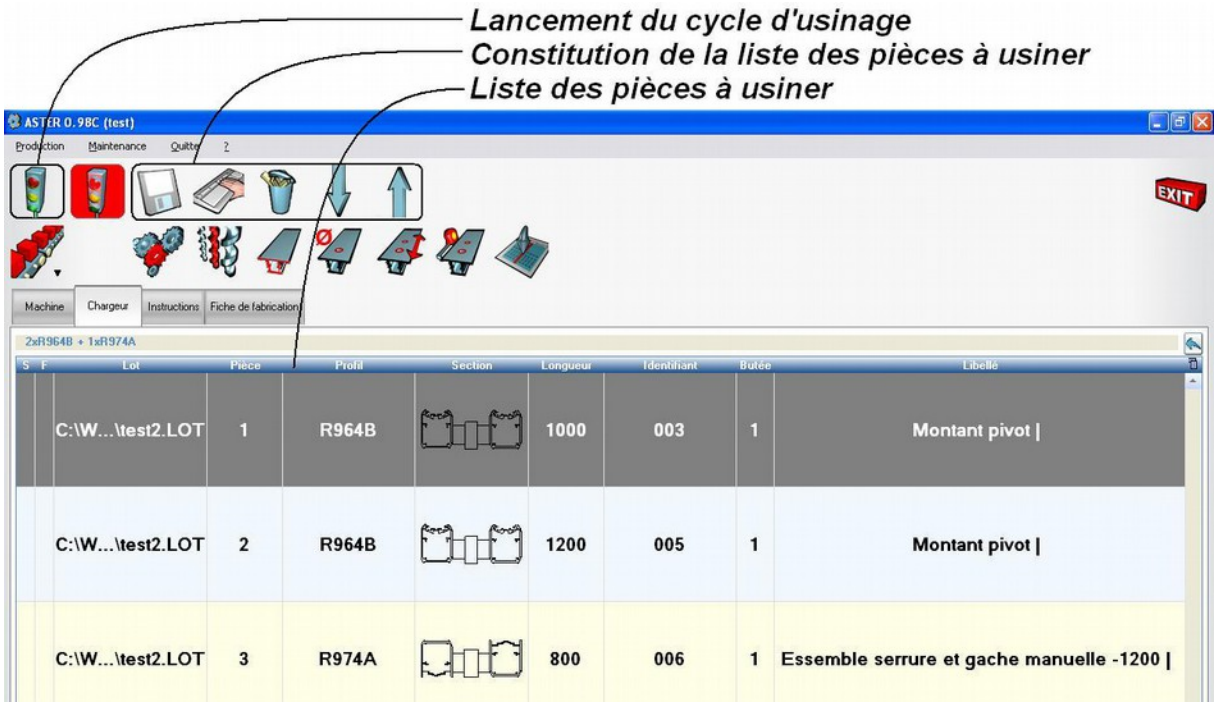
4.4.7 Confirmation or abandon

Ok Confirm the entry. The batch file displayed is saved. If a batch of the same name exists, it will be overwritten.

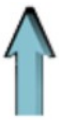
Cancel Abandon the entry. All modifications made will be lost.

4.5 Constitution of the list of parts to machine

“Loader” tab shows the list of parts to machine.



In “bar code” mode, you supply the list by scanning the identifier of the parts to add.
In “file” mode, you supply this list by adding batches of parts using the buttons  (see chapter: Importing a remote batch file) and  (see chapter Edit a local batch).

The arrows  and  allow you to modify the order of the list.

The waste-basket  delete the line(s) selected.

A double click on the loader line applies a stop point (red point ● in the first column).
The cycle will stop before machining the part.

4.5.1 “Series” mode

The maintenance menu allows you to select “series” mode or not to select it.
In “series” mode, the program loops on the parts of the loader. Once machined, the part at the front of the list passes to the end.
In “diffuse” mode, the parts machined are deleted from the list.

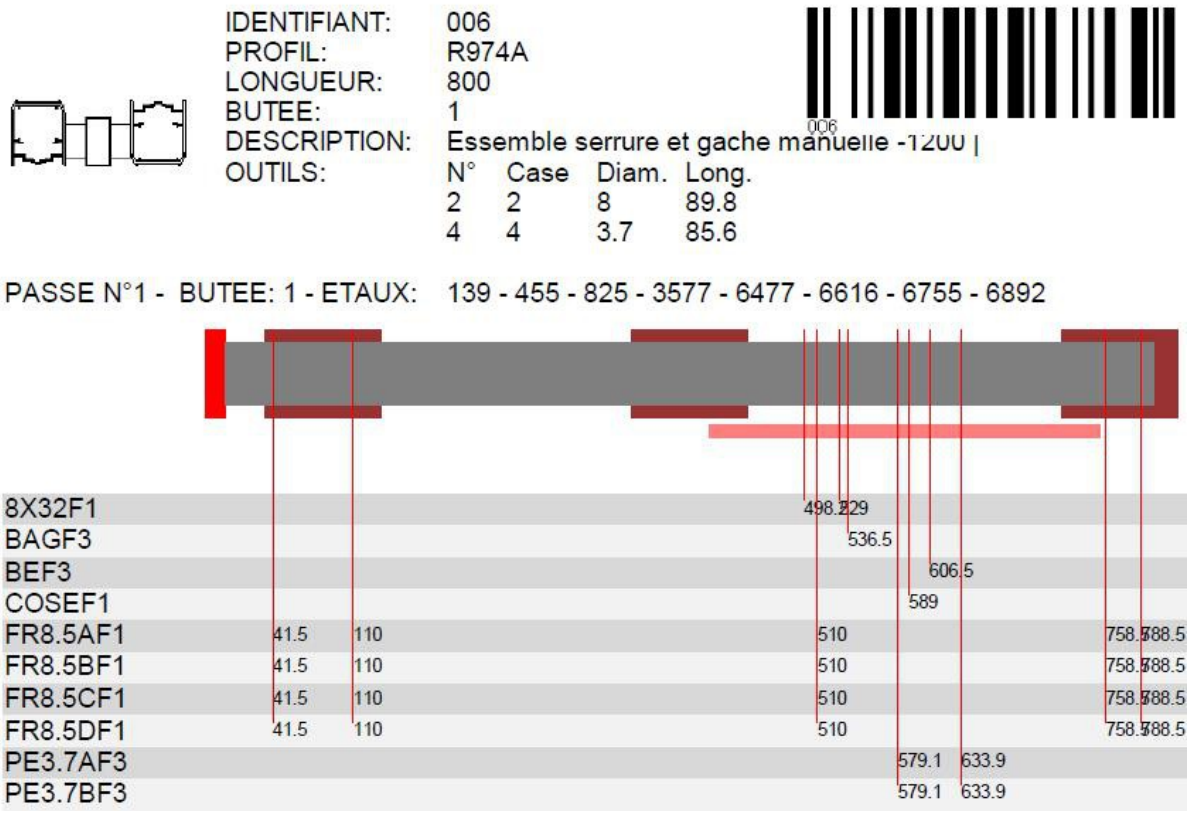
4.5.2 ‘Bar code” mode

The maintenance menu allows you to select “bar code” mode or not to select it.
In “bar code” mode, you supply the list by scanning the identifier of the parts to add.
In “file” mode, you supply the list by adding batches of parts using

4.6 Program execution

4.6.1 Manufacturing file

By clicking on the line of the list of parts to machine, the list of machining operations of the corresponding part can be displayed in the “manufacturing file” tab



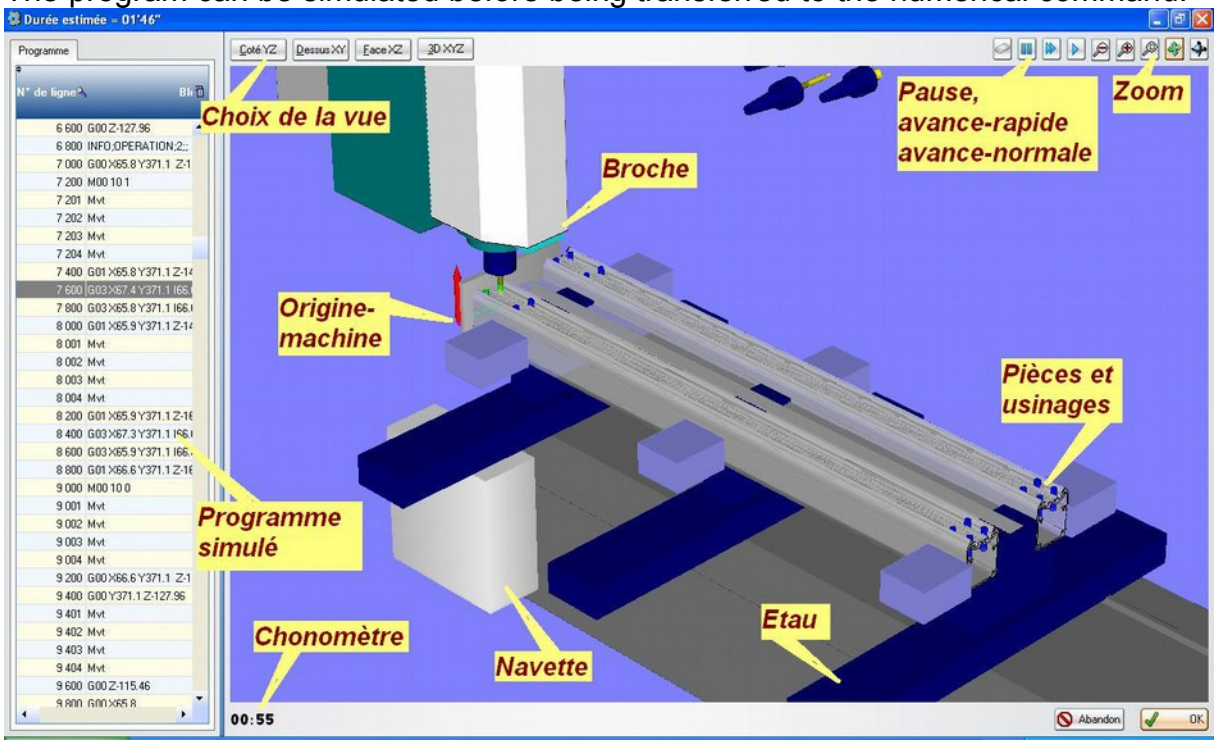
This file is saved in PDF format in the temporary directory.
You can chose to display or not display the manufacturing file using the machine parameter no.102.

4.6.2 Operator - instructions

During the cycle, when the operator needs to intervene to load or unload a part, specific instructions can be displayed in the “instruction” tab.
You can chose to display or not display the machining range using the machine parameter no.103.

4.6.3 Simulator

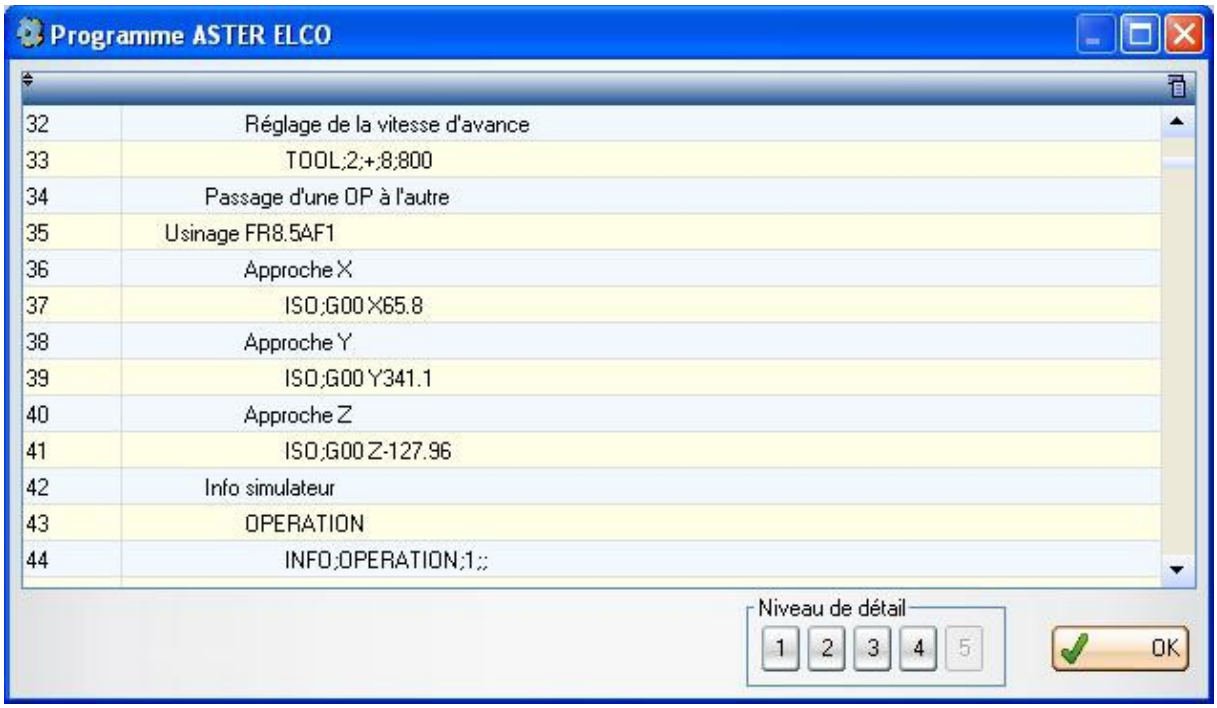
The program can be simulated before being transferred to the numerical command.



You can chose to simulate or not simulate the program using the machine parameter no.105.

4.6.4 Machining range

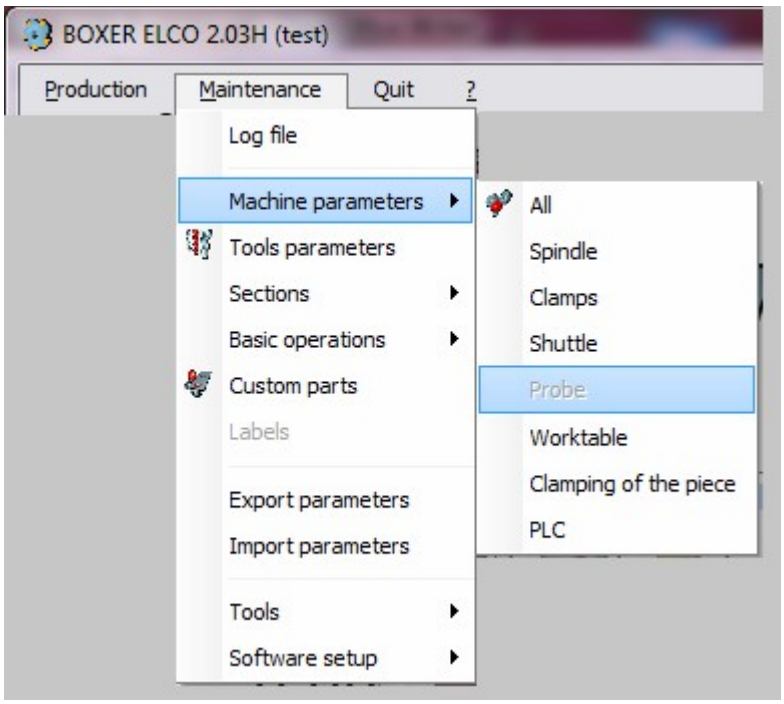
The commented program (machining range) can be displayed before being transferred to the numerical command. The different levels of detail allow you to obtain more or fewer comments.



You can chose to display or not display the machining range using the machine parameter no.101.

5 Maintenance menu

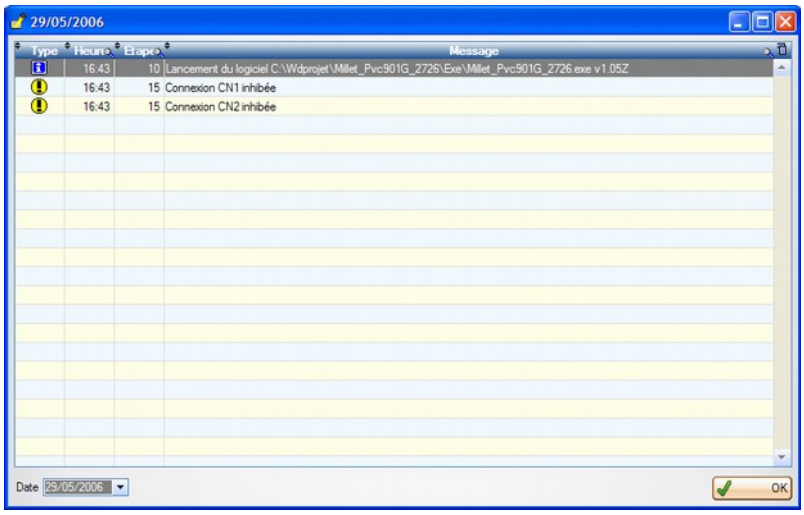
The Maintenance menu assembles the software parametrising functions. It is accessible only if the maintenance mode is active (see §Erreur : source de la référence non trouvée).



5.1 Journal

The software memorises certain events. They are classified by day, time and type:

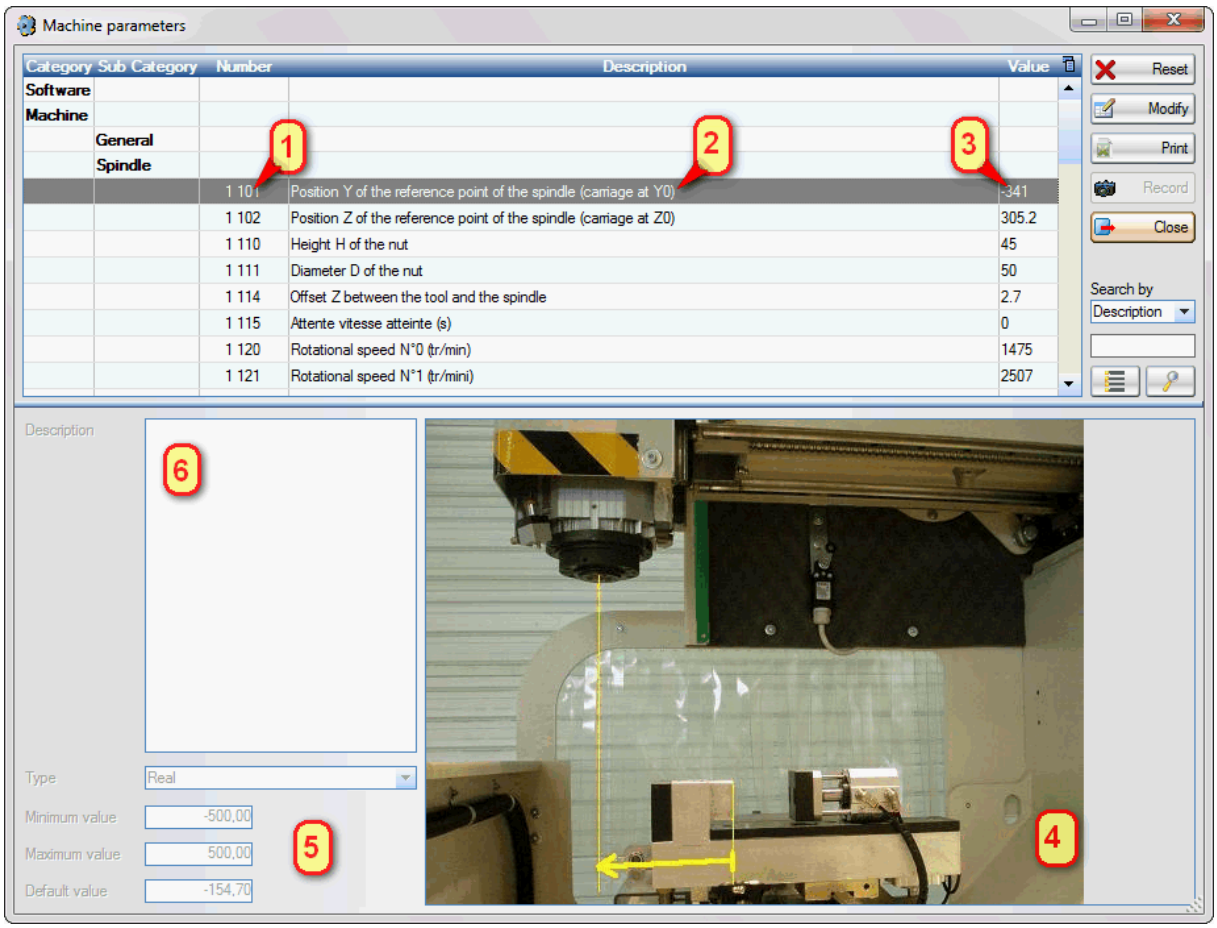
Info: start-up, stop, saving and restoration of parameters ...
Warning
Error



The journal lists these events. It supplies assistance for problem solving in case of anomaly.

5.2 Machine- parameters

The button  allows you to access the list of machine parameters. This appears in the window above.



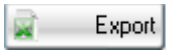
Each parameter is defined by

- A category and a sub-category, used for classifying the parameters by function.
- A unique number (1).
- A Wording (2).
- A type: Boolean, integer, real or chain (5).
- An interval and a default value (for numerical parameters).
- A value chosen by the user (3).
- An optional description, entered by the user (6). This description can be used to clarify the title or provide the history of the modifications.
- An optional associated image (4). This must be in the drawings directory. It carries the name " CMxxxx.JPG", xxxx being the parameter number.

The value of the parameter used by the software is found in the “value” column. If the value entered is less than the minimum value or greater than the maximum value, it is replaced by the default value.
The min., max. and default values are only accessible in the parameter setting mode (see §6).
The category, sub-category, number and title fields cannot be modified.

Delete the parameter selected. This will be initialised to its default value in the next start-up of the software;

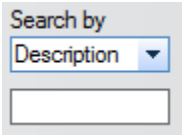
Accesses the description (customisable title), min., max. and default values;



Exports the table displayed in an Excel table;

Closes the window;

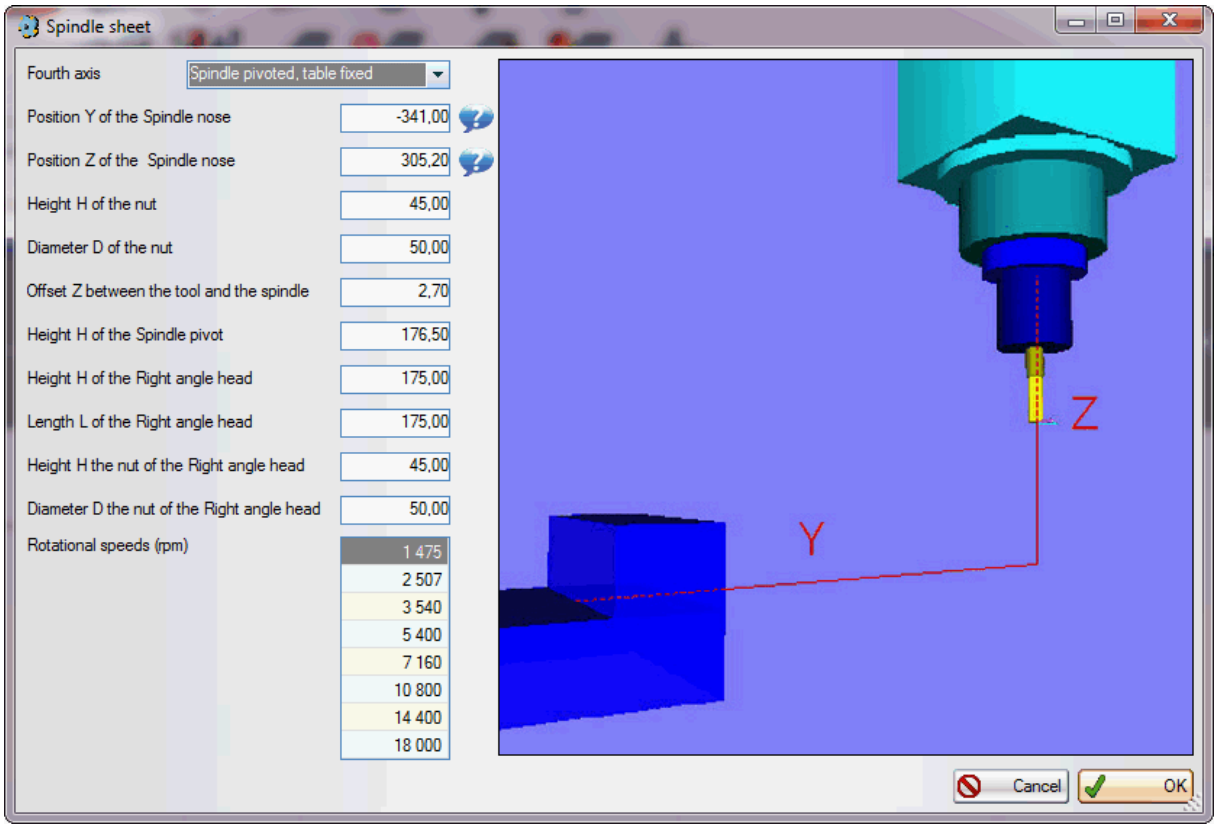

Develops the levels Category/Sub-category/Parameters. A double-click on a category or a sub-category only develops the line selected;



+  Searches a parameter by its number or its title. When several parameters correspond to the search, the key [F3] allows you to highlight the following;

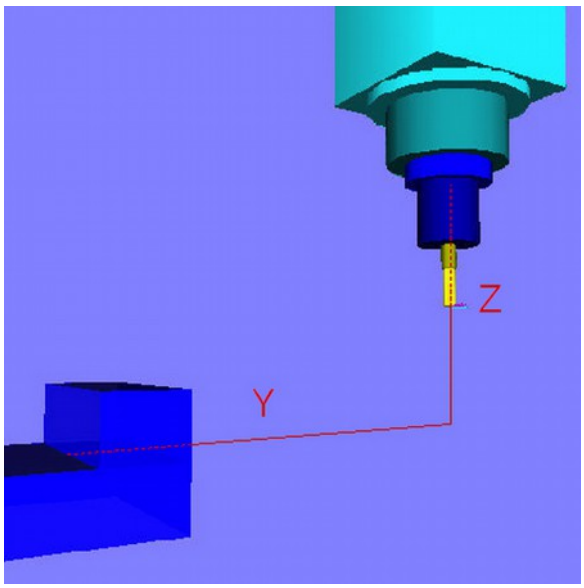
5.2.1 Spindle

The “spindle” file collects together the machine-parameters related to the spindle.

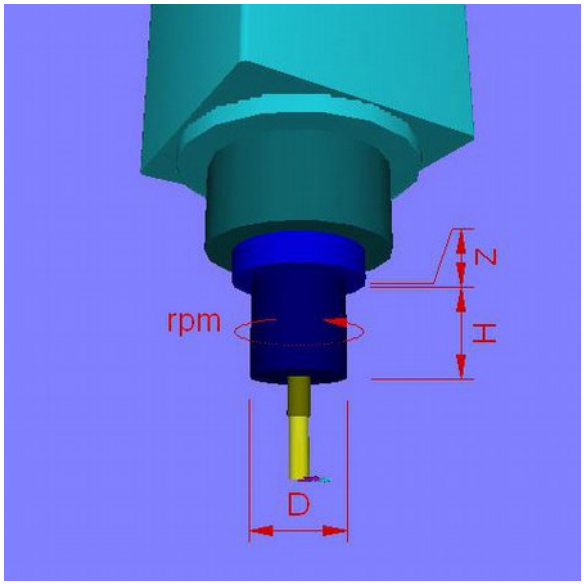


4th axis: To machine on the side of a profile, you can switch over the spindle (case of the Orion unit = spindle on pivot, table fixed) or the part (case of Phoenix unit = spindle fixed, table on pivot)

Position Y and Z of the spindle tip: All the axes being at zero, the position of the spindle tip from the machine origin. The machine origin is usually positioned at the origin of the standard vices.



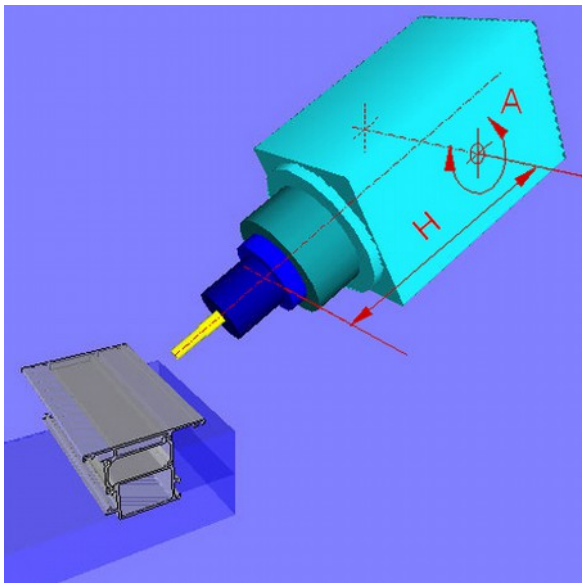
Height H and diameter D of the screw. The screw is modelled as a cylinder (see diagram opposite)



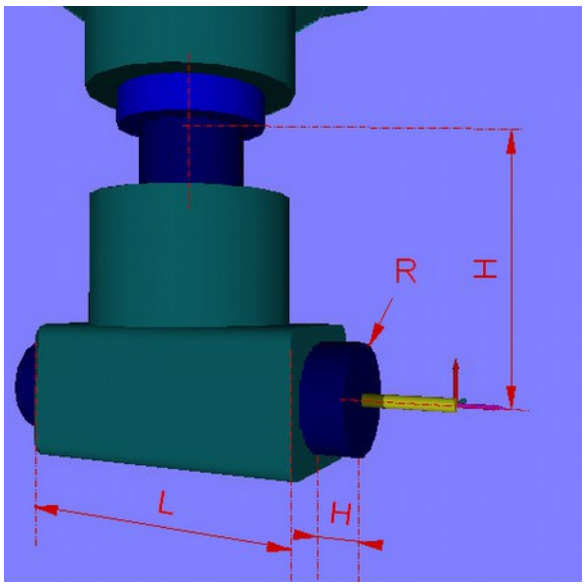
Offset Z tool/spindle. When the screw is in the spindle, it is not resting on the spindle tip. This distance must be indicated to be able to easily measure the tools on the assembly bench, independently of the spindle.

Rotation speeds. 8 speeds are registered on the speed controller. Each tool is associated with one of these speeds

Height H of the pivot of the spindle. Distance between the spindle tip and the axis of the pivot. This parameter is only active on a machine whose spindle inclines.



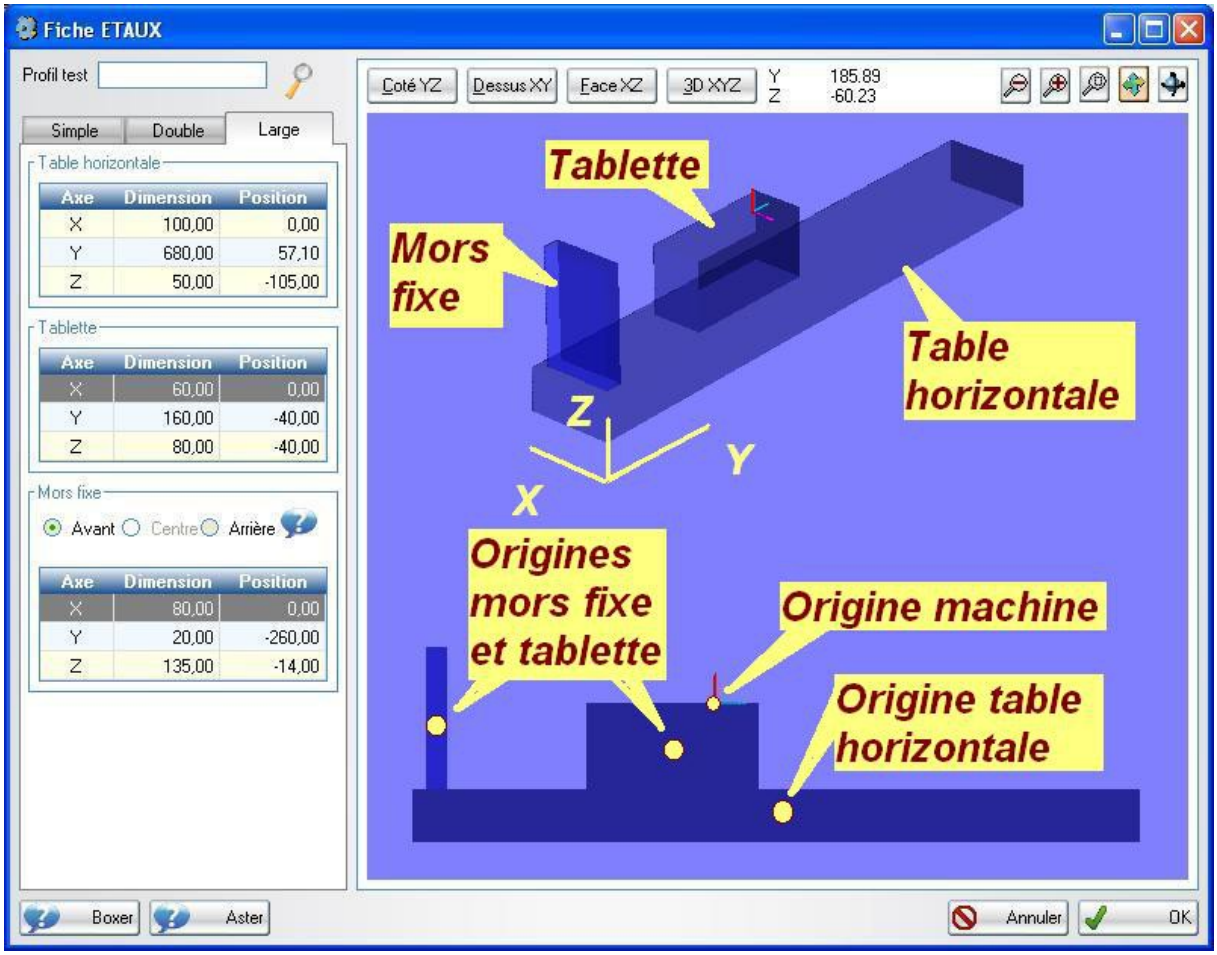
Angle drive:
H is the distance between the axis of the tool and the tip of the spindle
L is the length of the angle drive. It is centred with respect to the spindle.
H and R define the screw.



5.2.2 Vices

The vice file allows the vices used to be defined.

On the Phoenix unit, a single model of vice is managed.
On the Orion unit, the vices are modular. The software manages 3 variants, names single, double and wide.



Each model of vice is composed of a horizontal table, a plate and a fixed bit. The horizontal table is common to the 3 variants. Each element is modelled by a parallelepiped for which you define the dimensions and the position of the centre.

By convention, the YZ machine origin is positioned on the origin of the standard vice (Y = fixed bit side, Z = top of plate)

5.2.3 Carriage U

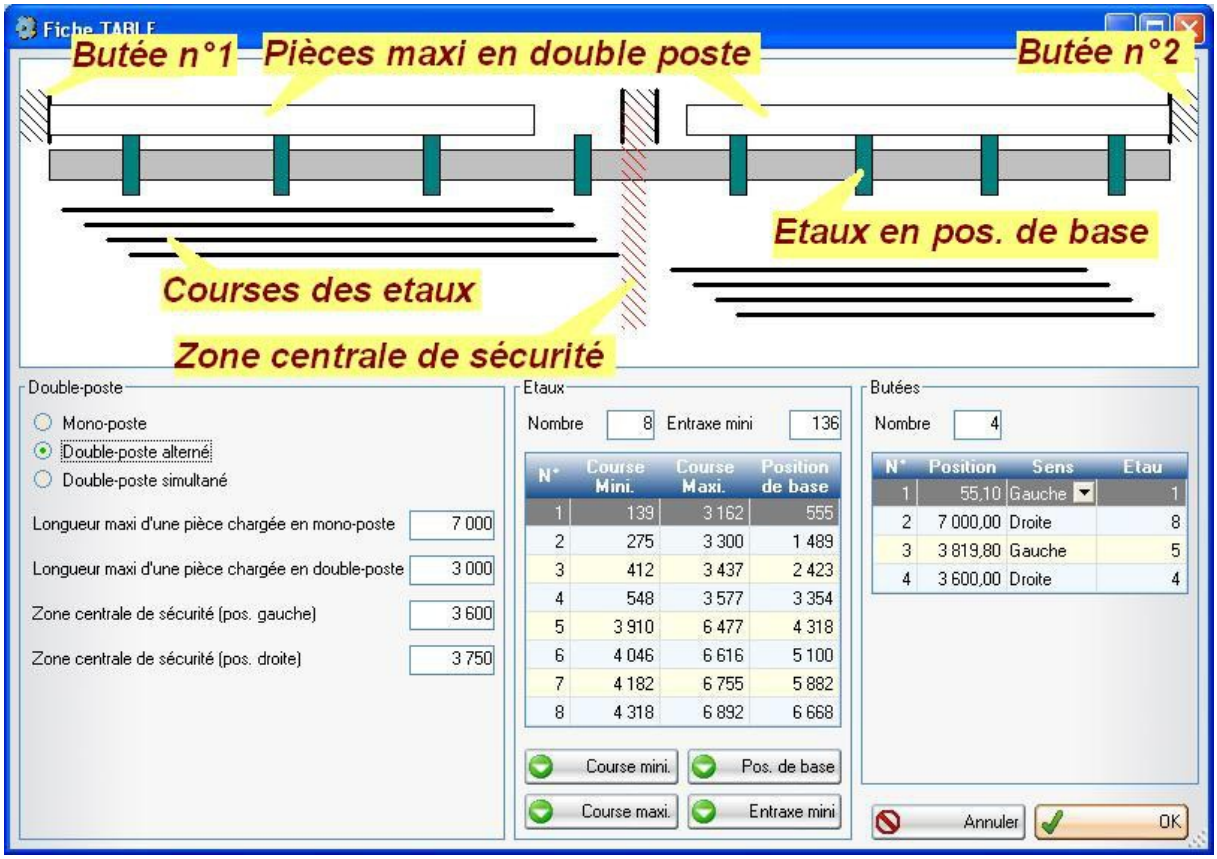
The “carriage U” file assembles the carriage U adjustment parameters for setting the position of the vices.

The “detection” frame allows you to set the cycle for measuring the positions of the vices. The higher the speed, the less accurate the detection. The mechanical system for setting the vices in place authorises an error of several millimetres (radius of the centring taper on the positioning pin).
The difference is the distance in millimetres between the value sent by the detection system and the real position of the centre of the vice.
The button (?) launches the measurement cycle for the vice and allows the difference to be compensated for.

The “hooking” frame allows you to set the cycle for moving the vices. The difference corresponds to the distance between the centre of the vice and the positioning pin.
The button (?) launches a movement cycle for the 1st vice. It allows the cycle to be tested and the difference to be compensated for.

5.2.4 Work table

The “work table” file assembles the machine-parameters related to the stops and the vices



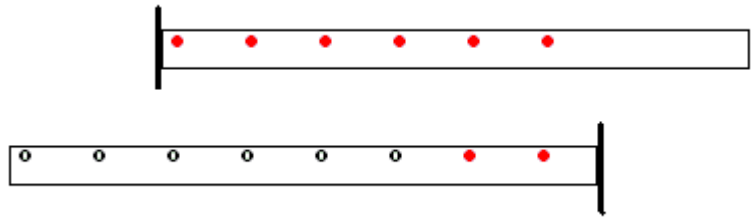
The “double-station” frame defines the mode for loading the parts.
In single-station, the operator cannot access the loading table while the system is machining.
In double-station, the loading table is divided into two zones, names station A and station B. While the system works in one zone, the operator can access the other. This allows you to load and unload during the hidden time.
The double station requires specific equipment, available on option on Orion.

To generate a “double station” program, you must select the double station option and machine a part with a length less than the set threshold.
If one of these conditions is not met, the software prepares a “single station” program.

The software assigns the same number of vices to each station. A double station machine must have an even number of vices.

For a question of safety, you can reduce the stroke of the central vice when it works on a double station. To do this, a central safety zone, forbidden to the vices, is defined.

In single-station, you can machine a part longer than the machine on condition to have a stop at each end of the table.
If the part is longer than the defined threshold, it is machined in 2 passes.



The frame “vices” defined the number and the strokes of the vices.
You can note these values directly on the machine (with practice) using the buttons [Min. Strokes], [Max. Strokes], [Base position] and [Min. centre-line distance].
The strokes and the positions are measured between the machine origin X and the centre of the vice.

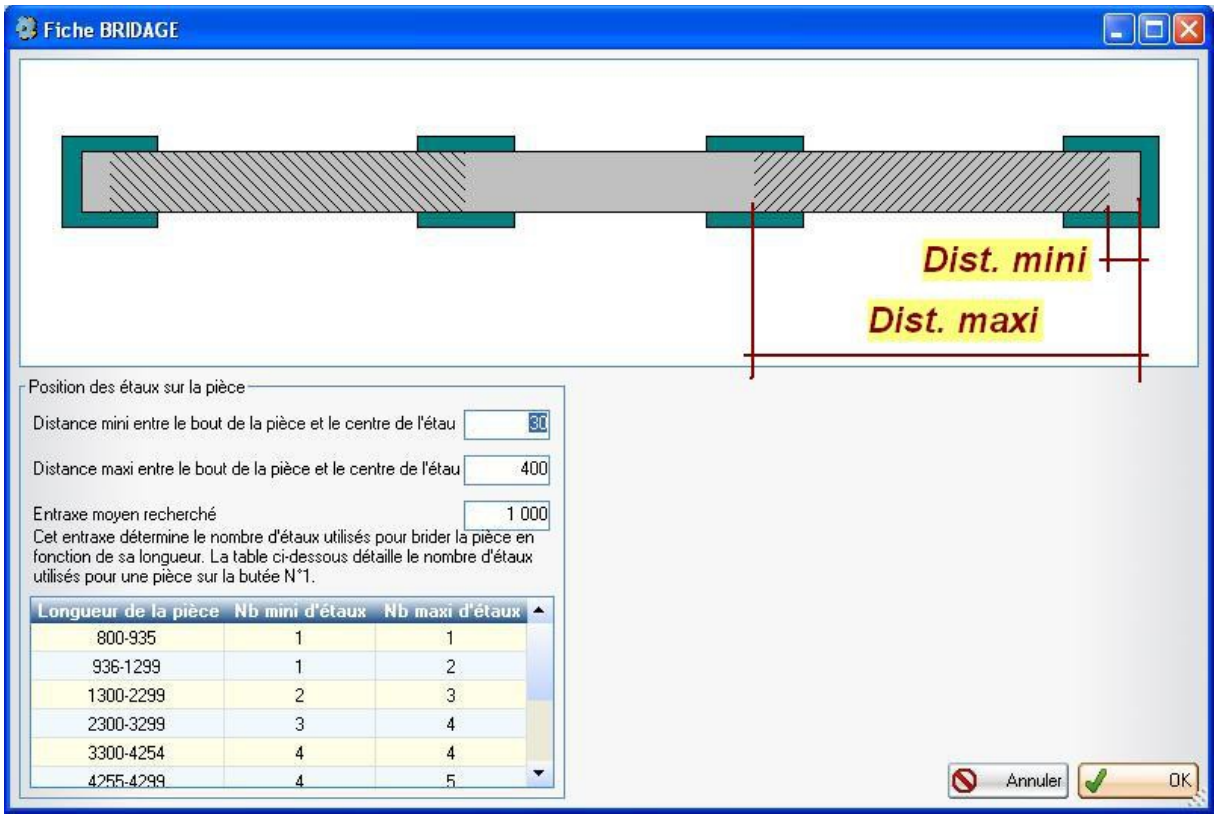
The frame “Stop” defines the positions and the orientations of the stops
By convention, station A is on the left of the machine and station B on the right. Stop 1 is the first on the left of the machine (X side min.)
Stop 2 is the last on the right of the machine (X side max.) Stop 3 is the left stop of station B
Stop 4 is the right stop of station A
The positions are measured between the machine origin X and the active face of the stop.

A vice is associated with each stop. It will systematically be used when a part will be in reference against this stop.

5.2.5 Clamping of the part

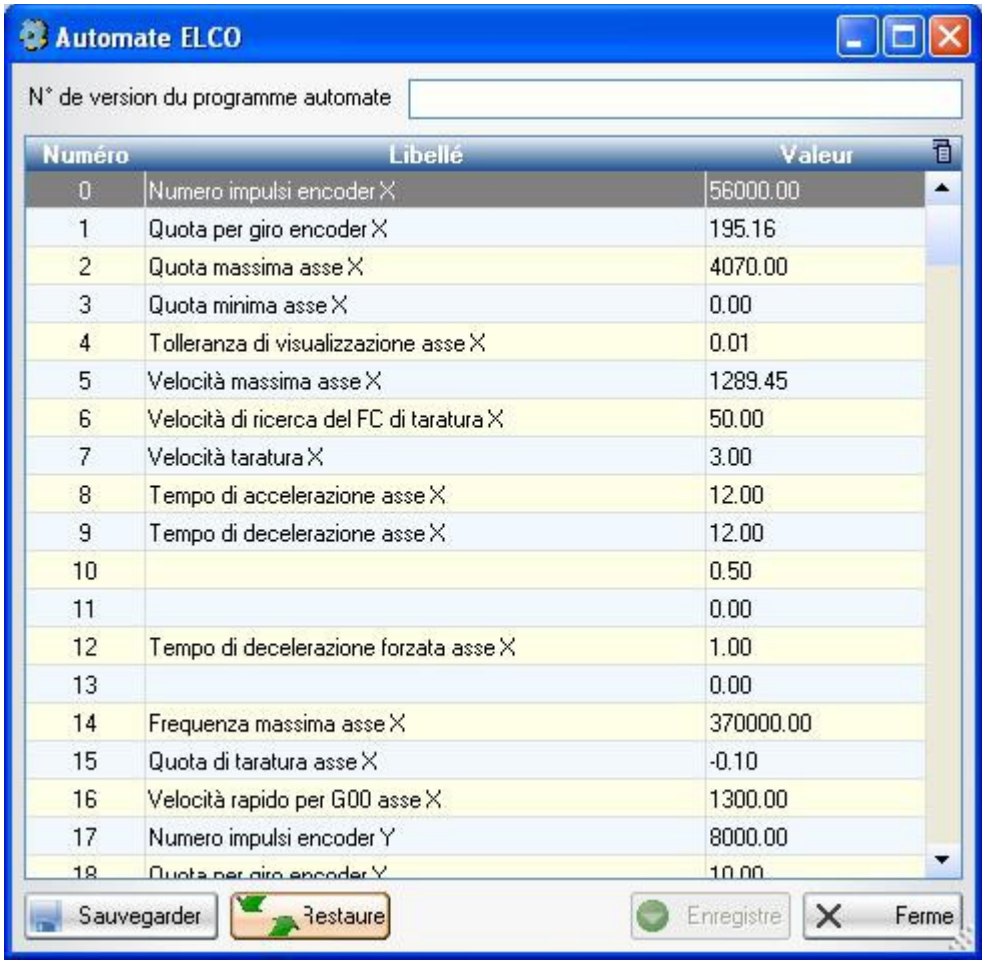
The “clamping” file defines the retaining of the part.
You choose the maximum cantilever at the end of the part as well as the “ideal” centre-line distance between the vices.

The software will determine the number of vices used as a function of this centre-line distance, the length of the part and the stresses related to machining



5.2.6 PLC

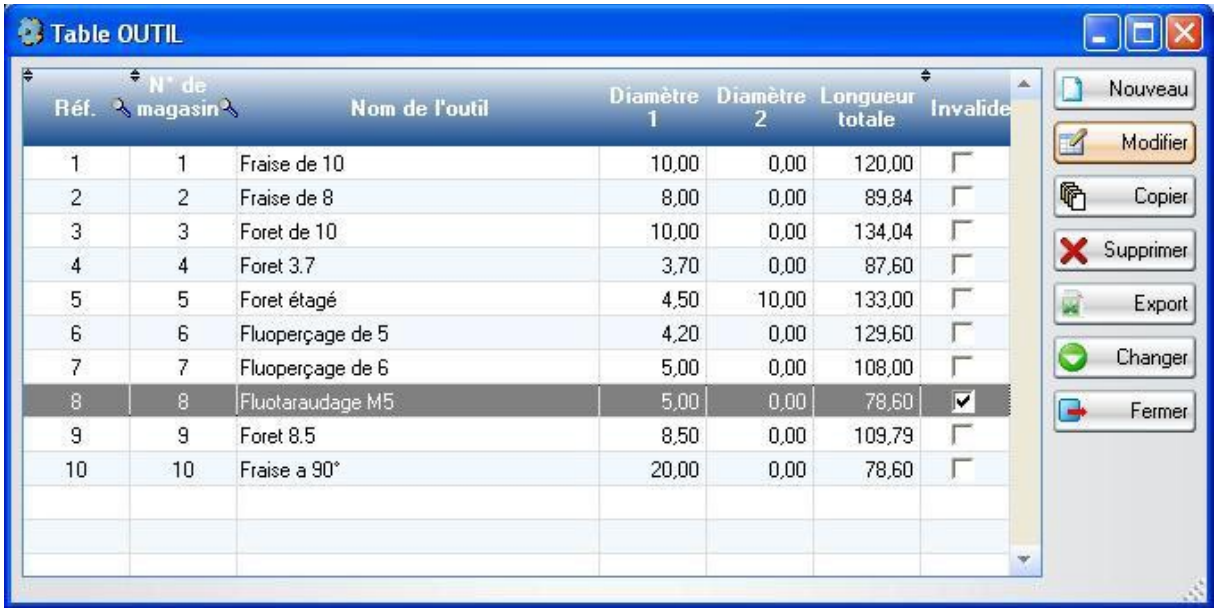
The window “ELCO PLC” allows you to access the internal parameters of the PLC.



Its use is reserved to the manufacturer.

5.3 Tools-Parameters

The button  allows you to access the list of tools.
This appears in the table above.



[New] allows you to create a new tool. It opens a blank entry file. [Modify] opens the file of the tool selected. A double-click on the line has the same effect.
[Copy] allows you to create a new tool from the tool selected. [Delete] deletes the line selected.
[Export] exports the table displayed to an Excel file
[Change] sets the chronometer to zero after a change of tool (optional function]
[Close] closes the window

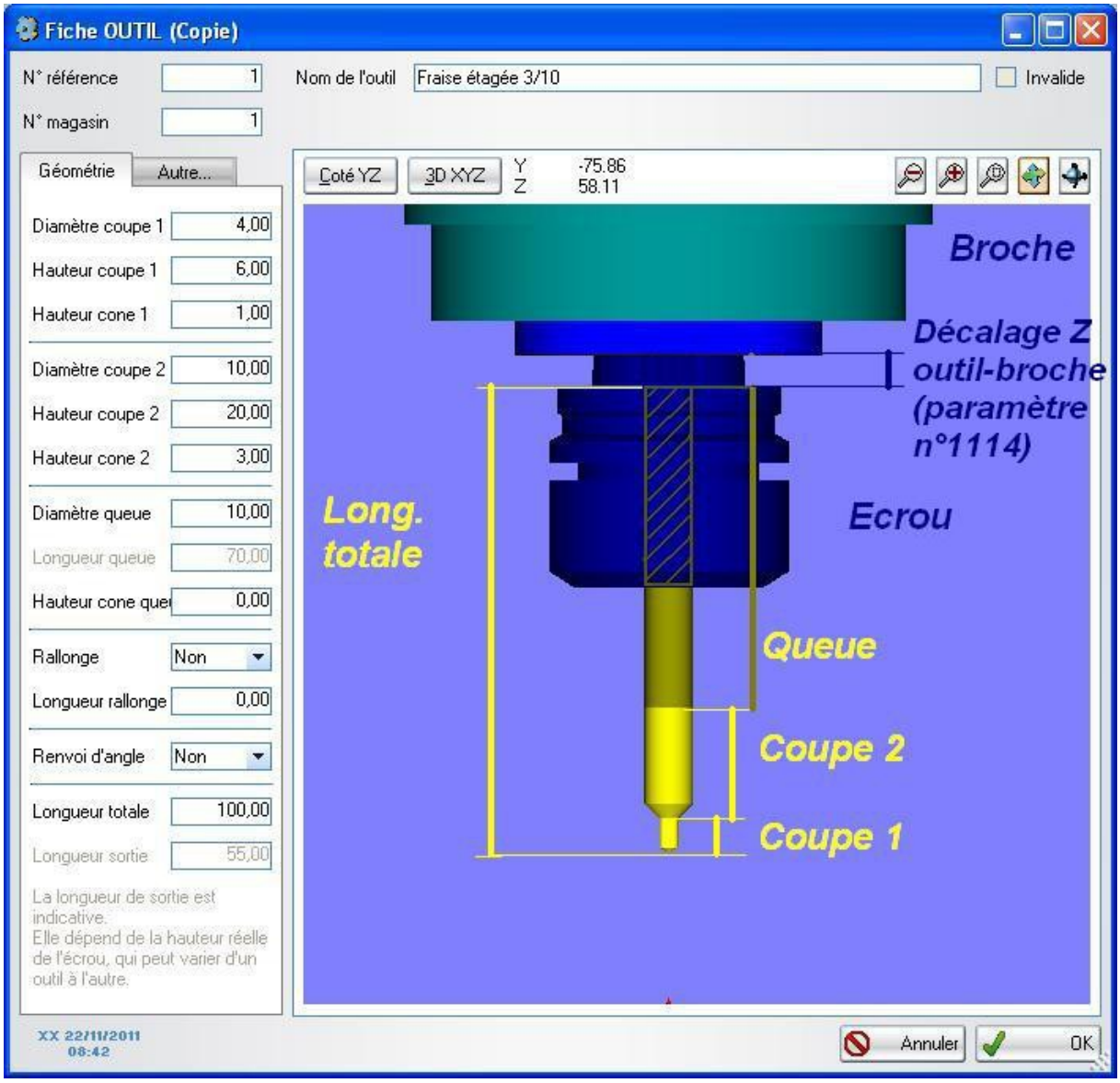
The reference No. is a unique identifier.
This number is used to assign a tool to each machining operation. When the order of machining is not imposed, the operations are sorted by increasing reference No.

The position is the number of the box (physical position) of the tool in the magazine. Several tools can be assigned to the same position, on condition that a single one is validated.

Diameter 1 is the useful diameter of the tool. Diameter 2 is used in case of a stepped tool.

The total length is measured between the point of the tool and the reference face of the tool holder taper.

“Geometry” tab

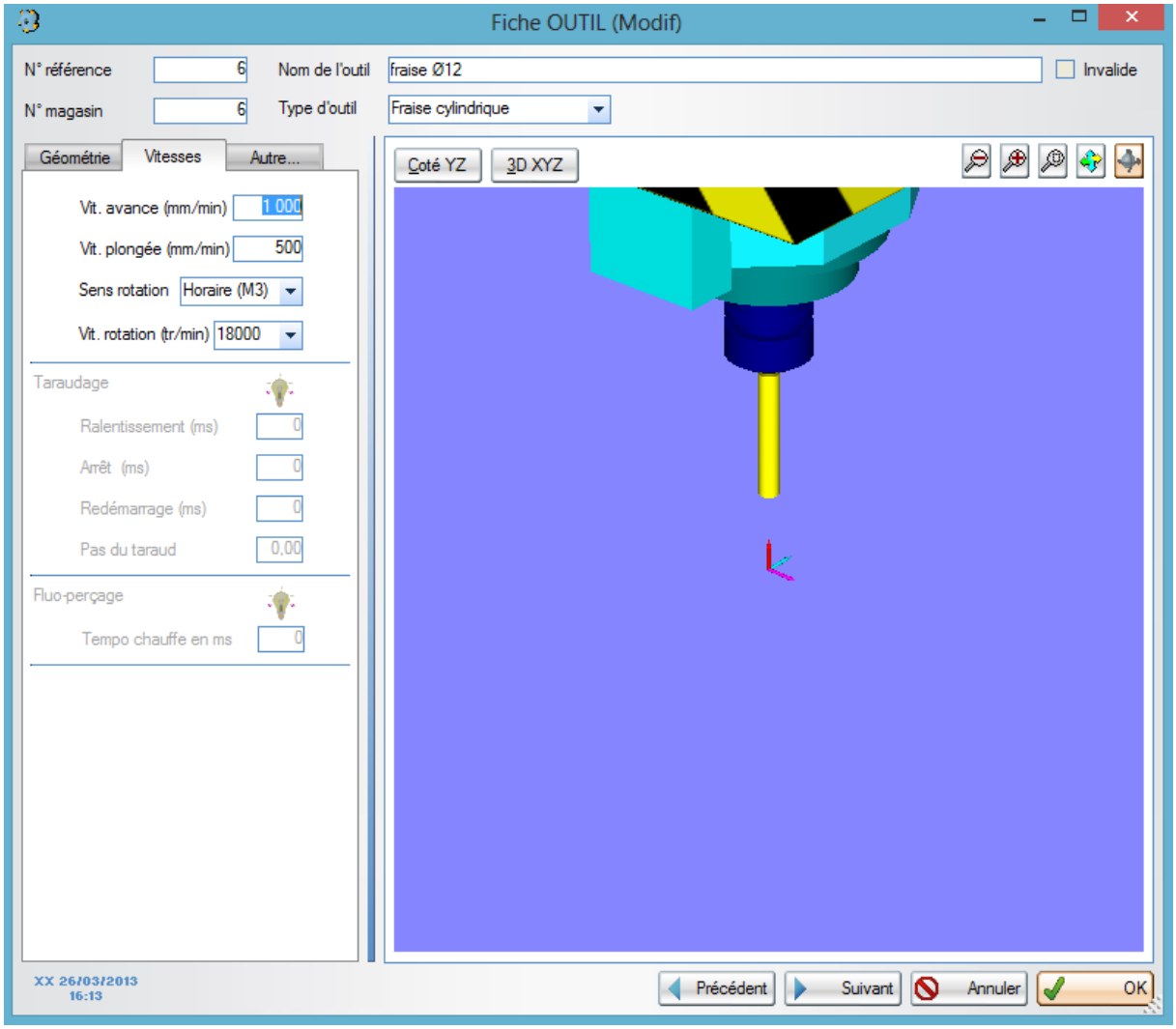


The tool is modelled by 2 cutting zones and a shank. Each zone is composed of a conical part and a cylindrical part. The cylindrical part is defined by a diameter and a length and a tapered part by a height.

A single mill-bit will be defined by a diameter and a cutting height, a shank diameter and a total length.
A stepped drill-bit will have two diameters and two cutting heights, two taper heights, one shank diameter and a total length.

If the tool is mounted in an extension, you can declare it. This is optional, this is only used for the graphical representation.

“Speed” tab



The feed and plunge speed in mm/min are used by the assistants during the creation of the machining operations.

The direction of rotation is clockwise. Exceptionally, you can mount a tool whose direction of rotation is reversed (trigonometric direction).

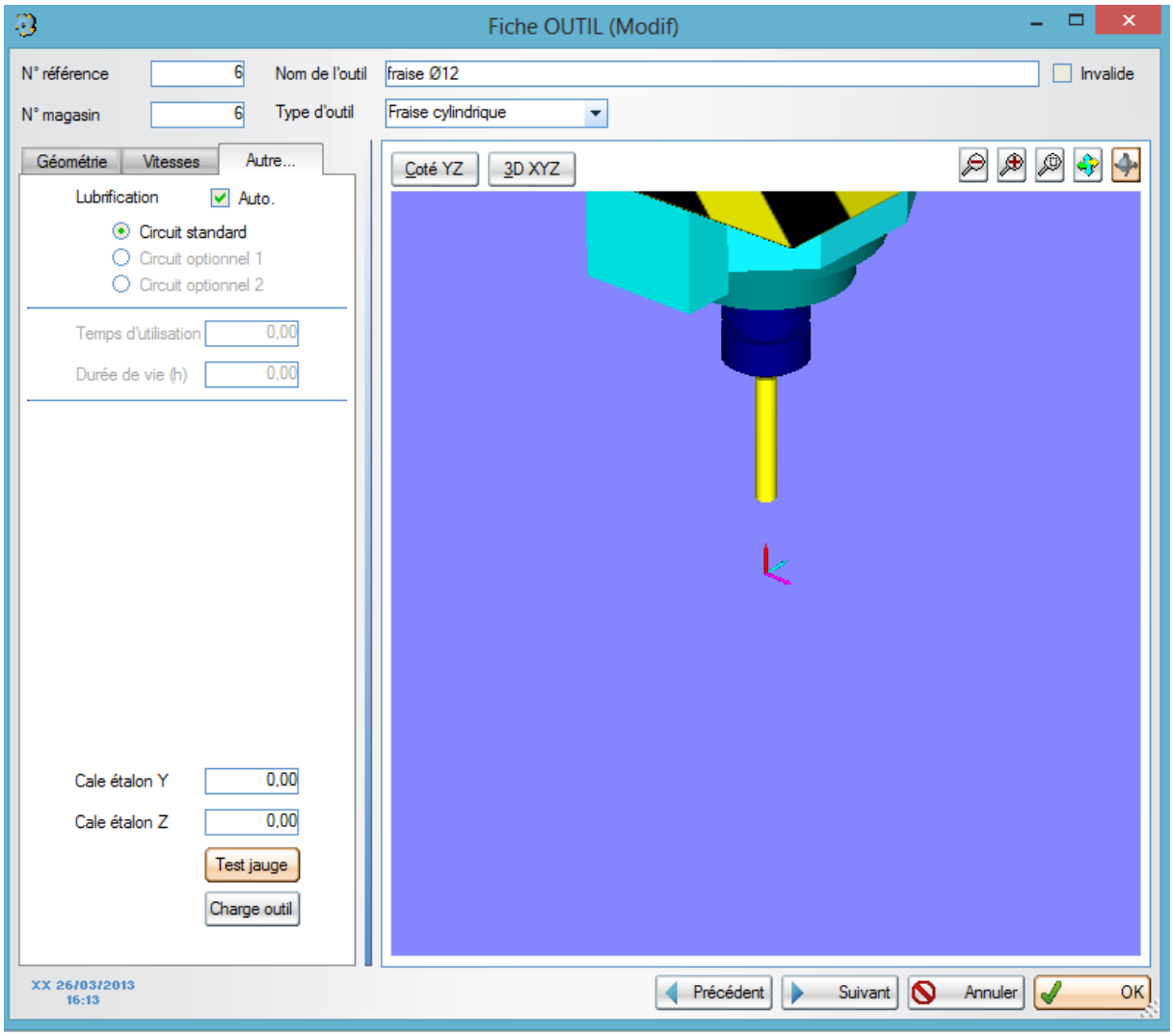
The rotation speed is chosen amongst the pre-programmed speeds in the speed controller.

For a tap, you can adjust the time outs used in the tapping cycle:
The tool approaches the top of the hole, spindle stopped.
The spindle is started at the programmed speed (clockwise)
It plunges to a programmed depth. The spindle is stopped N ms before reaching this depth.
The cycle is stopped for J ms (J=400 by default) the spindle is started in the opposite direction. After K ms, the spindle returns up to the release dimension and then is stopped.
Values of N=350ms, J=400ms and K=80ms in general give the correct result. These values can be adjusted as a function of the inertia of the spindle.

The thread pitch is used to calculate a feed speed appropriate to the rotational speed.

For a flow-drilling tool, you can adjust the heating time-out (moment where the point of the tool presses on the partition to drill, without advancing)

“Other” tab



The lubrication can be started systematically during the whole machining cycle or only upon request from the operator.

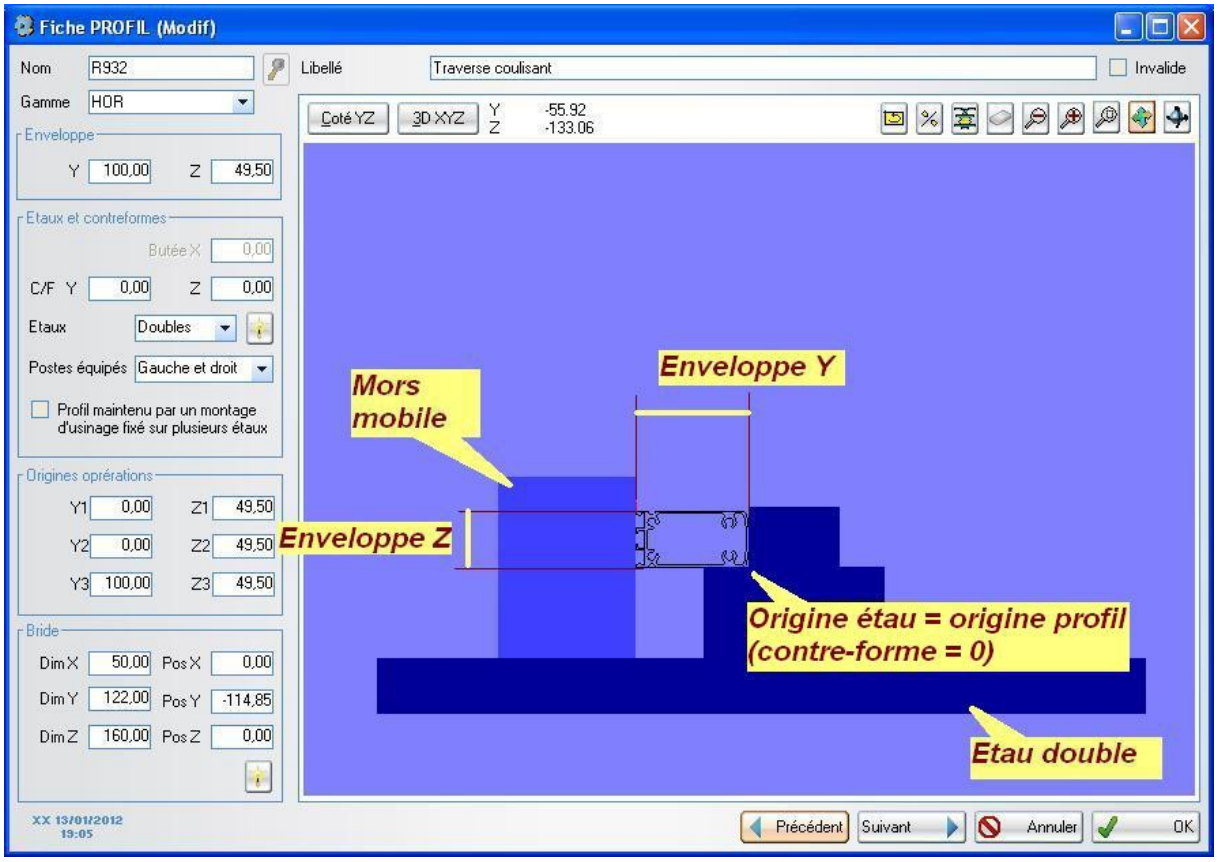
Several lubrication circuits are installed on certain machines. One circuit is associated with each tool.

The calibrated shim is used to test a gauge. The [Gauge test] button positions the tool on the shim. If the gauge is correct, the point must be exactly at the dimension programmed.

The [load tool] button allows a tool to be taken from the magazine and loaded into the spindle

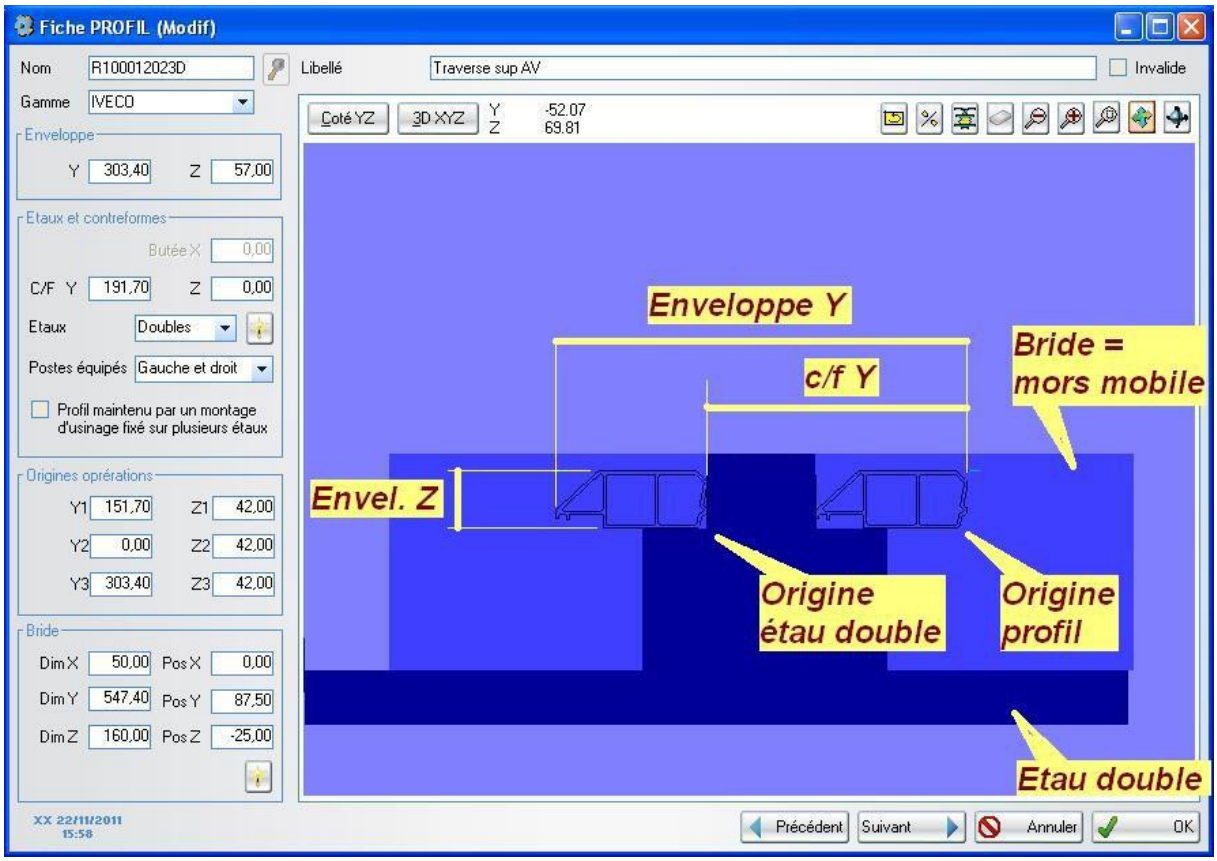
5.4 Profile-Parameters

A profile corresponds to an extruded (aluminium, PVC) or profiled (wood) cross-section. The list of profiles covers all the profiles declared by the user.
A profile-file details all of the parameters related to the profile.



The name of the profile is its identifier. Associated with a length, it defines a part.
The title is free text used to describe the profile.
The range is used to classify the profiles by family, e.g. “Opening strike”, “dormant strike”, “sliding”, etc.
The tick “invalid” allows a profile to be inhibited. The parts whose profile is invalid will not be machined on the machine. This allows the passage of a profile undergoing parameter setting to be forbidden, for example.
The envelope is the overall dimension of the profile expressed in millimetres. The envelope Y corresponds to the overall width of the profile, the envelope Z to the overall height.
The stop X is the distance which allows a profile to be separated from the stop

The counter-form allows the origin of the profile to be offset with respect to the origin of the vices (e.g. See below, the parametrizing of a profile pair on double vices).



The option “Machining fixture” must be ticked when you place the part in a fixture, itself fixed on the vices. In this case, the vices are positioned by hand by the operator, they are not moved by the shuttle.
The fixture must be designed to avoid collisions during the machining.
The approaches and the clearings are managed by the post-processor. It clears the tool outside of this rectangle defined by the clamp to pass from one operation to another.

The origins of the operations are positioned by the programmer on the characteristic points of the profile. They will be used to position the machining operations. It is helpful to place them on the departure point(s) of dimensioning.

The clamp in general represents movable jaws. If you use a machining fixture, the clamp is the rectangle in which the fixture is inscribed (see machining fixture option above).

The cross section of the profile is displayed if a file in DXF format bearing the name of the profile is found in the directory of drawings, defined in the maintenance menu (see §3.3.1). The three buttons below act on the DXF.

 Rotation by 90°

 Symmetry of the drawing

 Compression of the drawing (reducing the size of the drawing to accelerate its display)

5.4.1 Archiving of profiles

The function “profile archiving” backs up and then deletes from the working database, the profiles selected and their machining.

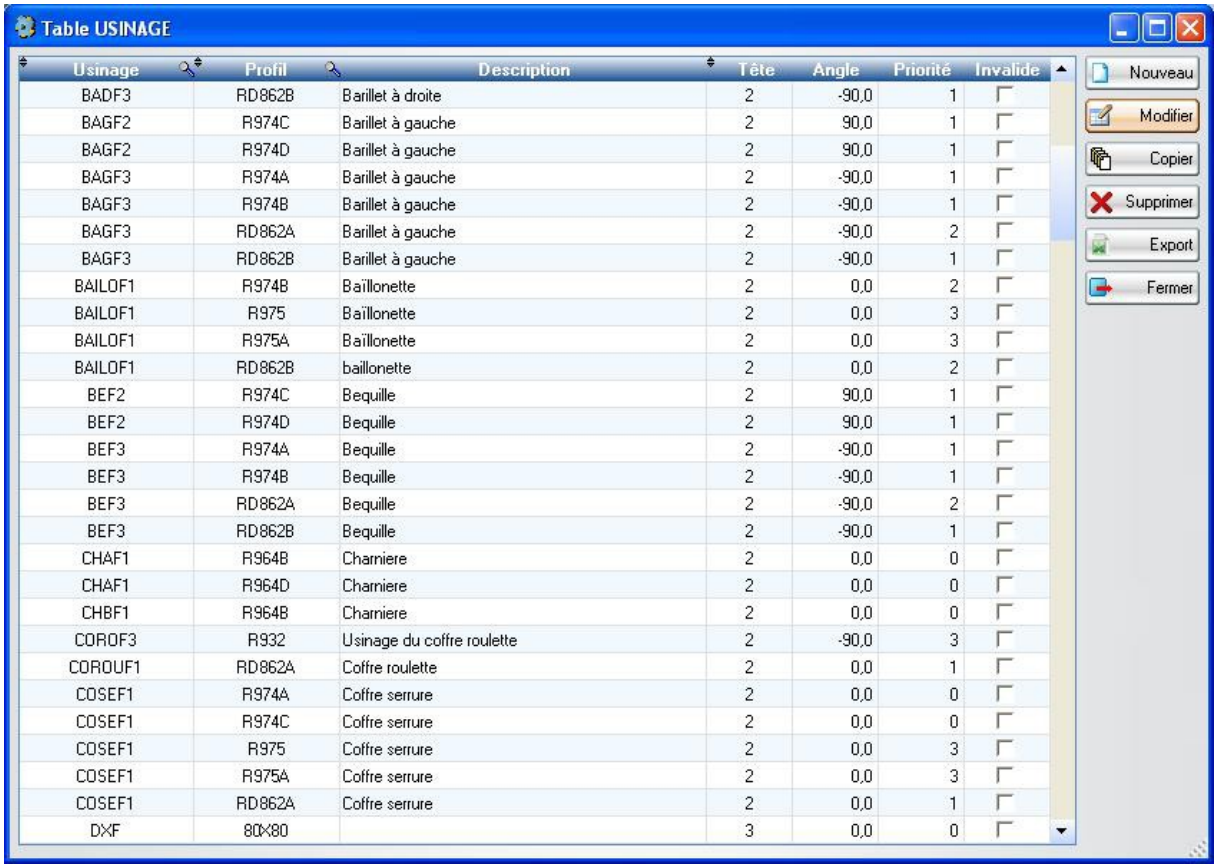
5.4.2 Restoration of profiles

The function “profile restoration” allows previously archived parameters to be restored.

5.5 Machining-Parameters

5.5.1 Basic machining

The machining parameters describe machining cycles by operation and by profile.



- Machining: name of machining operation.
- Profile: name of the profile associated with the machining.
- Description: title of the machining.
- Head/Tool: number of the tool which will be used for performing the machining.
- Angle: Inclination of the head or the table (+90° = front face, 0° = top,
- -90°=Rear face)
- Priority: Machining priority number, used to define the order of execution of operations of each part
- Invalidation: if this option is ticked, the machining will be ignored.

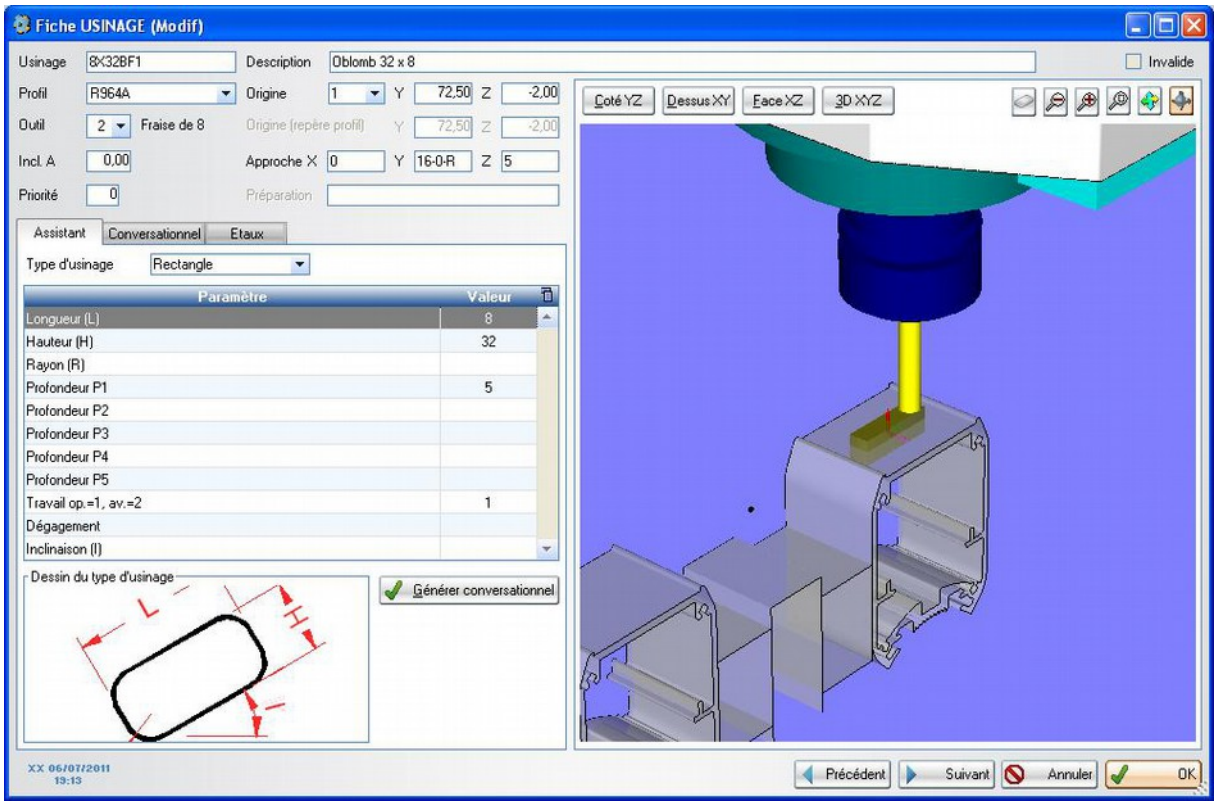
The list of machining operations covers all the machining operations declared by the user. The window above allows you to create, modify, copy or delete a machining operation of the existing list. To modify the parameters of a machining operation, you simply double-click on the machining desired, or select the line in the table then click to modify it.

General parameters:

- Machining: name of machining operation.
- Description: title of the machining.
- Invalidation: if this option is ticked, the machining will be ignored.
- Profile: name of the profile associated with the machining.
- Origin: the “zero point” of the operation is defined from a offset on Y and Z from one of 3 operation origins of the profile. In general, you place this origin at the centre of the operation, on the machined wall of the profile.
- Head/Tool: number of the tool which will be used for performing the machining.
- Inclination/Angle: Inclination of the head or the table (+90° = front face, 0° = top, -90°= rear face)
- Approach: position of the approach of the point of the tool, from the origin. The reference frame is oriented by the spindle. The Z axis is parallel to the tool. You can enter the dimensions (e.g. Z=5), or formulae which call upon variables (Y=16-R, R being the radius of the tool).
- Priority: Machining priority number, used to define the order of execution of operations of each part. The operations are sorted by pass No.(=position of the vices), Priority No., Tool No., Position X, Inclination, Operation No.

Assistant tab:

The assistants facilitate the programming of the current operations: drilling operations, circular milling operations, rectangular milling operations, text engraving, etc.



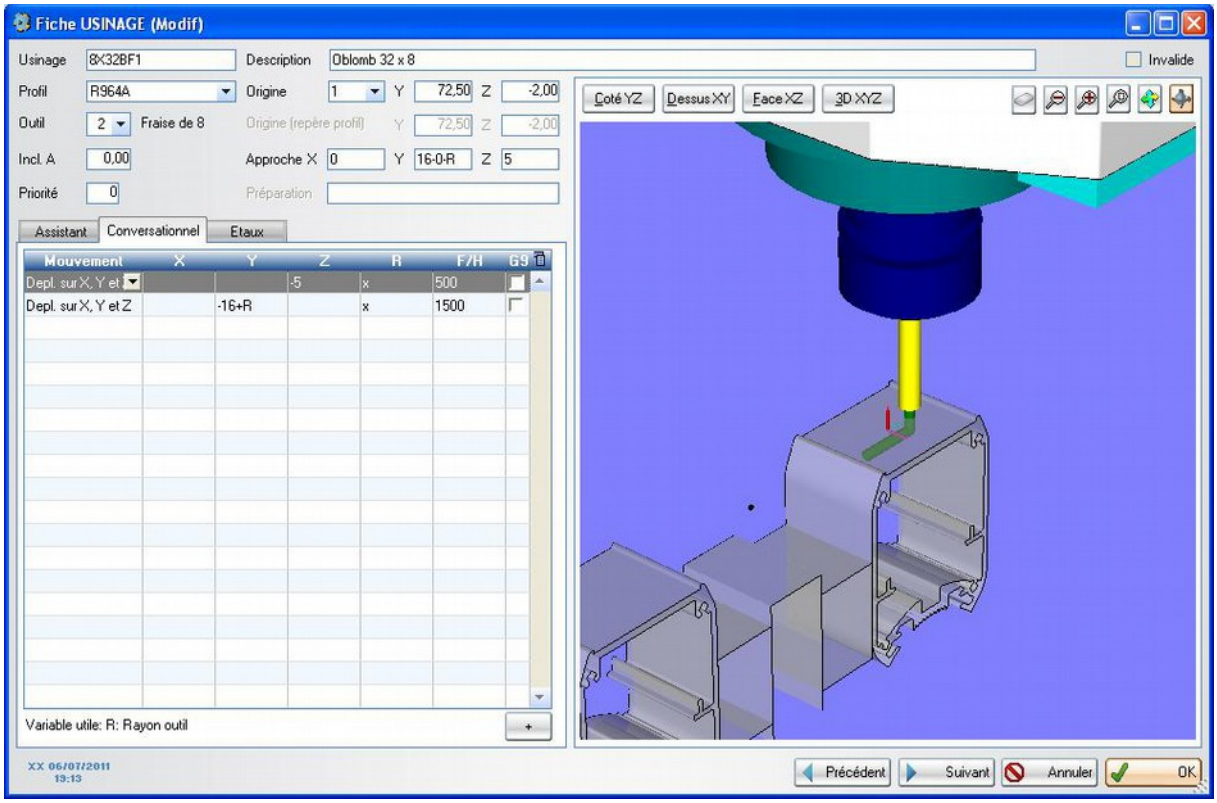
To use an assistant,

- Enter the general parameters (profile, origin, tool no., inclination),
- Choose the type of machining (circle, rectangle, etc.)
- Enter the dimensional parameters (diameter, depths, etc.)
- Click on the button [generate the dialogue]

The approach position and the detail of the trajectory (“dialogue” tab) are then calculated.

Dialogue tab:

The trajectory of the point of the tool when the machining is described step-by-step in the “movement” table

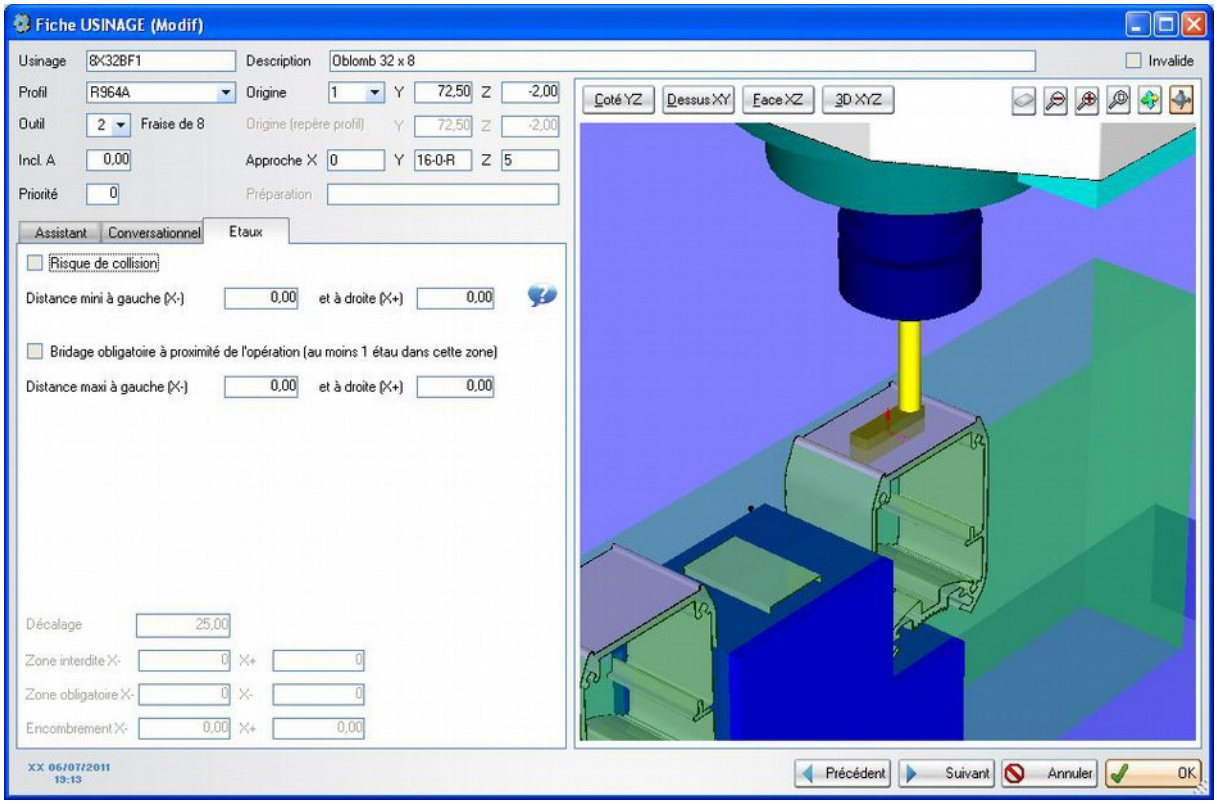


The trajectory is composed of a succession of linear (G1 in ISO), circular (G2 or G3 in ISO) movements and time-outs (G4 in ISO).
The coordinates of the arrival points X, Y and Z are given as absolute with respect to the operation origin. The unit is the millimetre.
The radius R of the arcs of the circles and dimension in millimetres. The speeds F are given in millimetres per minute. The time-outs H are given in seconds.

If a value is not entered on a line, the previous value is used.

The coordinates (origin, approach and trajectory) can be expressed as a mathematical formula using the variables R or D (radius or diameter of the tool).

Vices tab:



Risk of collision: this option must be ticked if there is a risk of collision between the spindle and the vices. In this case, you must specify the zone forbidden to the vices.

Min. distance on the left (X-) and on the right (X+): forbidden zone on either side of the origin X of the machining. It is situated between the vices (in light red on the drawing). It is measured between the origin of the machining and the edge of the vice. It is a function of the footprint and of the depth of the machining, the inclination of the table and the length of the tool, etc.

N.B. Take the footprint of the spindle into account when the table is inclined.

The drawing allow the absence of collision to be checked.

Classic examples:

- Horizontal table (inclination = 0°) and blind machining => no risk of collision
- Horizontal table and through machining => risk of collision. Min dist = dist between the origin and the edge of the machining
- Inclined table => risk of collision. Min dist = dist between the origin and the edge of the machining + ½ the width of the spindle + margin of security.

Clamping compulsory near machining: allows the placing of the vice to be forced around a configurable operation zone (compulsory zone shown in green).

Max. distance on the left (X-) and on the right (X+): compulsory zone on either side of the origin X of the machining. It appears in green in the drawing. This zone must

be larger than the forbidden zone, at least 200mm on either side. In fact, the more the forbidden zone is reduced, the less the software has the possibilities to place the vices.

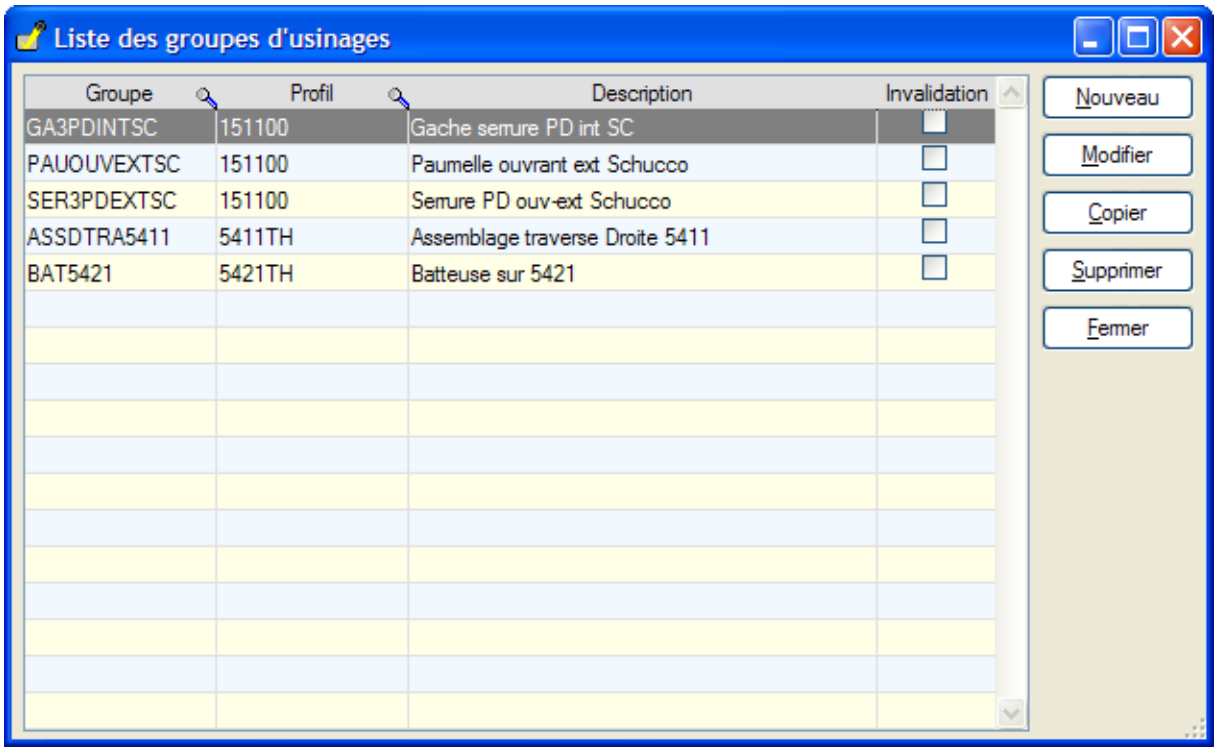
Drawing legend:

The envelope of the profile is in grey if the DXF format drawing does not exist.
The origin of the operation is shown by 3 arrows. The violet arrow represents the X axis, the blue, the Y axis and the red, the Z axis.
The tool is shown in yellow. The spindle is shown in blue
The tool trajectories are shown in green. The vices are shown in light red.
If you click on a line of the table of movements, the tool goes to the corresponding position and the trajectory followed appears in light green.

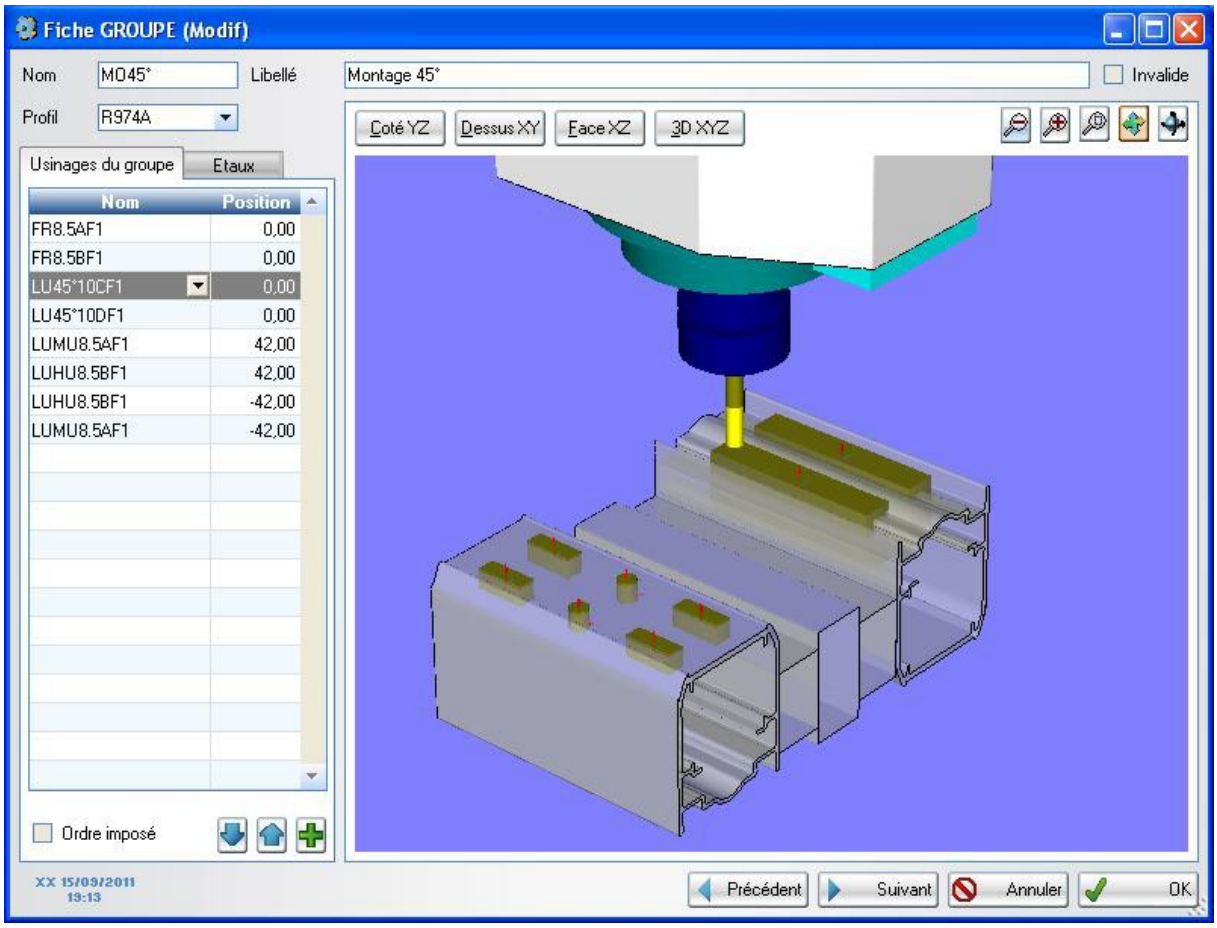
Manipulation of the drawing See chapter 4.4.6

5.5.2 Machining groups

A group allows several machining operations to be associated . The call of the group in the batch file triggers the execution of all of the associated operations.



To access a group, you simply double-click on the line of the table or select the line by a single click and press the modify button.



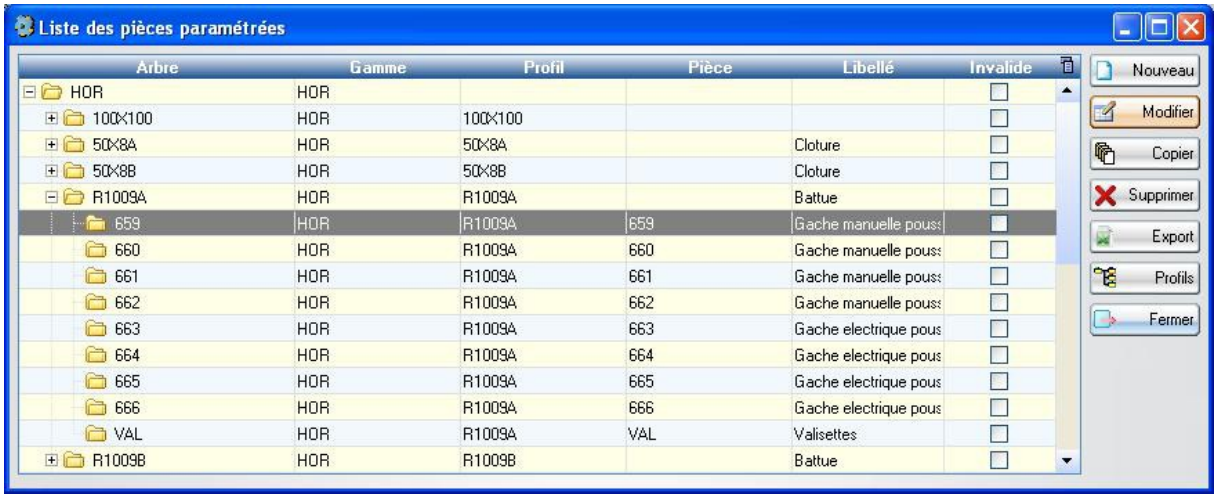
Group name: name of the machining group. Wording: clear description of the machining group. Profile: profile associated with the machining group.
Imposed order: forces the machining order (e.g. Roughing then finishing).

For each group operation, you specify the position on X of the operation relative to the position on X of the group declared in the batch file.
E.g.
You declare a group SERD3 composed of the operations SQUARE01 to 0, HOLE01 to
-21.5 and +21.5

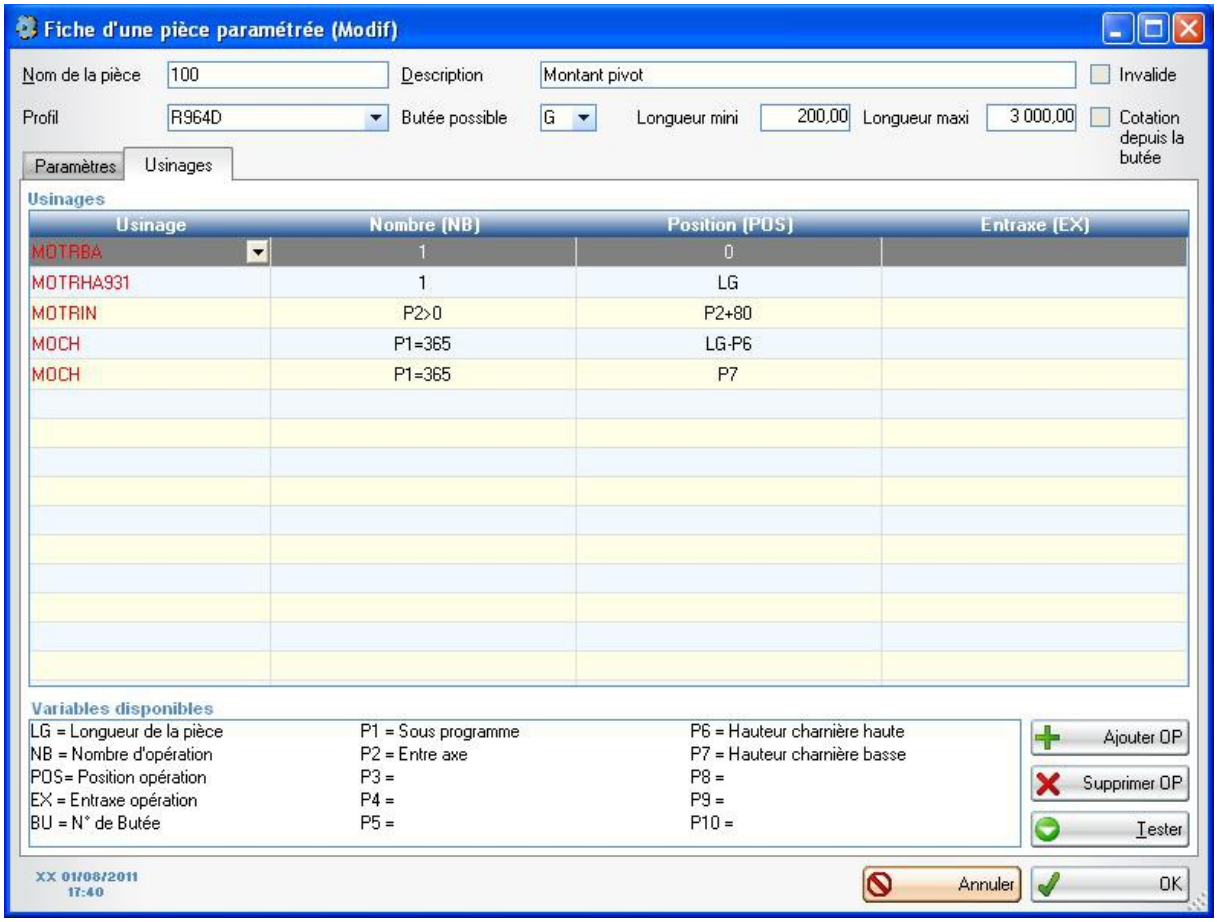
If the group SERD3 is requested at 500 mm, the operations are found at the following positions:
SQUARE01 => 0 + 500 = 500
HOLE01 => -21.5 + 500 = 478.5
HOLE01 => +21.5 + 500 = 521.5
Etc.

5.6 Configured-Parts

On a configured part, you are going to define the machining operations to perform as a function of the length and several parameters.
These parts are classified in a hierarchical table, by range, profile and part name.



To access a model, you simply double-click on the line of the table or select the line by a single click and press the modify button.



The user gives a name to the part, a description, a profile to use, the range of acceptable length (min. And max.), the possible stop(s).

In the parameter tab, you list the parameters which the user must enter. Each parameter is defined by a name and a range of acceptable values (min./max.).
The number of parameters is limited to 10 per part.

In the machining tab, you list the machining operations.
Each machining operation is defined by its name, the number of the operation to perform, the position of the first operation and the centre-line distance between the following operations.
The number of different machining operations is not limited.

The position, the number and the centre-line distances can call for calculation formulae. E.g.
Number: NB = 2+(LG>1000)+(LG>2000) = 2 if LG≤1000, 3 if 1000<LG≤2000, 4 if LG>2000
Position: POS = LG/2+50 = the half-length of the part + 50mm
Centre-line distance: (LG-2*POS)/(NB-1) = the centre-line distance between NB operation regularly distributed between POS and LG-POS.

The [Test] button allows these formulae to be tested on different part lengths or with different parameter values.

5.7 Export/Import of parameters

The export allows you to save the configuration of the software (machine parameters, profiles, operations, etc.) in a single file.
This file is called: NameMachine_Date_Time.ZIP
The directory in which it is written is chosen by the user.

The import allows you to restore the parameters from a back-up file chosen by the user.

 ***It is important to regularly perform back-ups (exports) and to keep the files in a safe place.***

5.8 Tools

5.8.1 Purge of old batches and programs

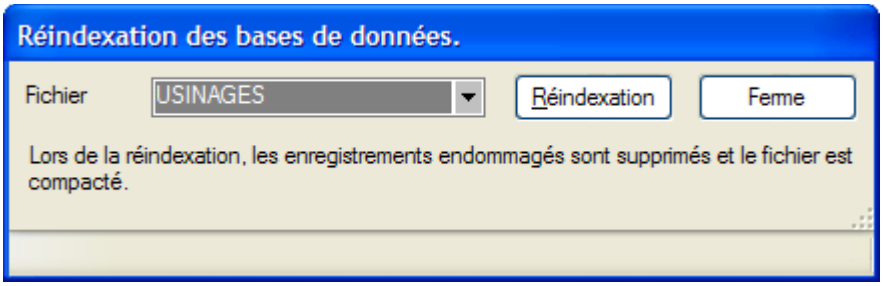
The purge allows you to free up space on the disc by deleting old batches (*.LOT) and programs (*.PRG, *.DES).

Configuration of the “purge” function
The files purged are those which have mire than N days, N being set by a machine parameter No.800
The dossiers purged are chosen using the machine parameter No. 802
By set to 1, the machine parameter No.801, the purge is automatically launched at each start-up of the software.

5.8.2 Database repair

Re-indexing can be necessary following the loss or alteration of index files due to a malfunctioning of the PC or a power cut, for example. The recordings non damaged will be re-indexed, the others will be deleted.

Using the following window, you simply choose the file to re-index in the drop-down list, then click “Re-index”:



If a recent backup of parameters (parameter export) was performed before the problem, it is not necessary to re-index the files. A simple import of parameters is preferable.

5.8.3 Technical assistance contact

This tool facilitates the sending of a request by e-mail to the technical assistance (support@naert.com). It requires internet access from the machine station.

5.8.4 Exploring the software directory

Allow you direct access to the software installation directory.

5.8.5 Importing of LGF parameters

This function must be used during the initial installation. It enters the machine parameters and tools from the files QUOTE and UTENSILI supplied by the manufacturer.

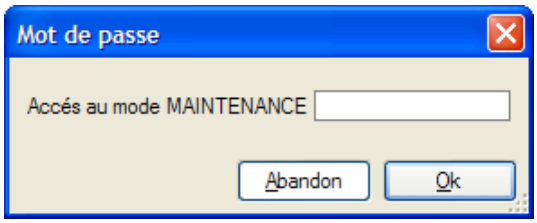
5.8.6 Recovery of parameters from an old version

This function is used when an update of an old Phoenix software version is performed

5.9 Software settings

5.9.1 Password

The Maintenance Mode provides access to all the functions of the Maintenance menu. It is possible to restrict access to this menu using a password.



If a password is entered, only the Production menu will then be accessible at the next start-up of the software.
To re-establish access to the Maintenance menu, the option Maintenance Mode must be ticked in the Software Information window and a password declared.

5.9.2 Setting of directories

The setting of Directories is the first step to perform after installation of the software. This setting allows you to define the different working directories of the software.

Remote directory of batch files: reading path of files coming from the GP. The files present in this directory will be proposed during the importing of batches.
Local directory of batch files: reading path of files already imported. The files present in this directory will be proposed during the conversion of the batches into programs.
Program directory: directory for storing generated programs. If the MMI supplied by the LGF site is used, the program directory must be C:\LGF.
Drawings directory: storage directory for profile drawings (*.DXF) and machine parameters (*.BMP).

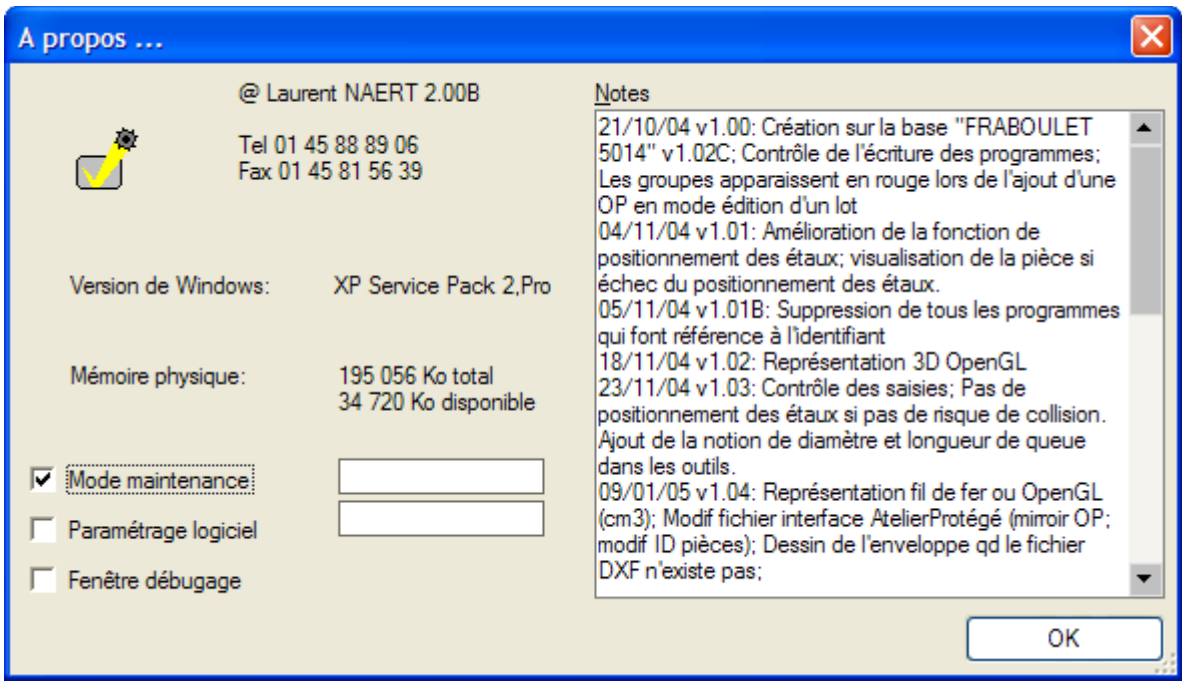
5.9.3 Configuration of the communication

The software communicates with the ELCO robot by an RS232 link. This function allows you to chose the port used by the PC.

5.9.4 Update installation

This function automates the installation of updates.
It performs a back-up of the current version in the dossier \ARCHIVE.

6 Software Information Window



This window is accessible by clicking on the symbol ? In the menu bar then Information

It gives the history of modifications made to the software and allows you to access the maintenance (configuration of machine, profile, machining parameters, etc.) and configuration (adjustment of acceptable ranges for the machine parameters) modes.

7 Appendices

7.1 Format of batch file no.1 (*.TXT) – Call for a part configured on the machine

The different models of the parts to produce are defined in the machine. The machining operations to perform are chosen and positioned from several parameters supplied by the software which prepares the file.

ASCII File Name: name of batch Extension: .TXT
The saved files are separated by a change of line (characters CR+LF) The fields are separated by a semi-colon

Each saved file describes a part

Profile; Stop; Length; Identifier; Comment; Model; Param_1; Param_2; ...; Param_10

- Profile: Name of the profile of the part. 4 to 20 alphanumeric characters (capital letters or numbers).
- Stop: side of the part (left or right) at stop on the machine reference. A letter, L or R.
- Overall length of the part in millimetres. The decimal places are taken into account (e.g. 1234.5).
- Identifier: unique identifier of the part. The identifier is used to name the programs generated. This identifier can be used to load the program from the bar codes. 5 to 16 alphanumeric characters (capital letters or numbers)1
- Comment: description of the part, operator instructions, etc. (200 characters max.)
- Model: Name of part model. This name corresponds to a program saved in the machine. 4 to 20 alphanumeric characters (capital letters or numbers).
- Param_1 to Param_10: 10 numerical parameters to pass to the program saved in the machine.

7.2 Format of batch file No.2 (*.LOT) – Provision of a list of operations to perform

The operations to perform are selected and positioned by the software which prepares the file. The tools and trajectories associated with each operation are defined and stored in the machine.

ASCII file
Name: name of batch (8 characters) Extension: .LOT
The saved files are separated by a change of line (characters CR+LF) The fields are separated by a semi-colon

Types of saving:

New part
DP; Profile; Stop; Length;Identifier; Comment

- Profile: Name of the profile of the part. 1 to 12 alphanumeric characters (capital letters or numbers).
- Stop: side of the part (left or right) at stop on the machine. A letter, L or R.
- Overall length of the part in millimetres. The decimal places are taken into account (e.g. 1234.5).
- Identifier: unique identifier of the part. The identifier is used to name the programs generated. 5 to 16 alphanumeric characters (capital letters or numbers)2
- Comment: description of the part (optional field, 200 characters max.)

Machining
OP; Machining; Position_in_the_part

- Machining: name of machining operation. The machining parameters (tool(s) used, approach dimensions, position(s) of origin, machining cycle(s) are defined in the post-processor for each machining operation in each profile. The same name can correspond to 2 different cycles in 2 different profiles. On the other hand, symmetric machining operations (locks on left and right uprights for example), must be named differently. 1 to 12 alphanumeric characters (capital letters or numbers).
- Position: distance in millimetres between the left end of the part and the machining reference. The decimal places are taken into account (e.g. 1234.5).

N.B. On this machine, 3 faces are accessible to the tool. If the machining operations must be done on the 4th face of a profile, you must consider that it is a new part in a different profile.

7.3 Installation of the Ithea key

The Ithea key is a device allowing you to protect the software against copies. Without the key, the software cannot transfer the program to the machine.

Illustration 3: The driver and the key are correctly installed

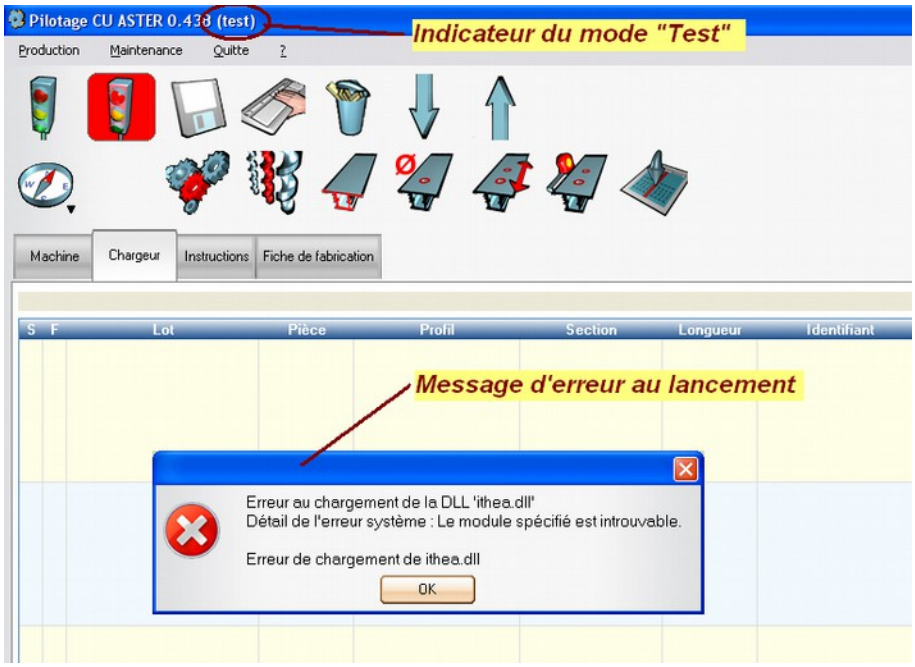


Illustration 4: The driver or the key are not correctly installed

It connects to a USB port of the PC and requires the installation of a driver.

7.3.1 Installation procedure:

1) Execute the program SETUP.EXE This program can be found on the CD supplied with the software or downloaded from the Internet.

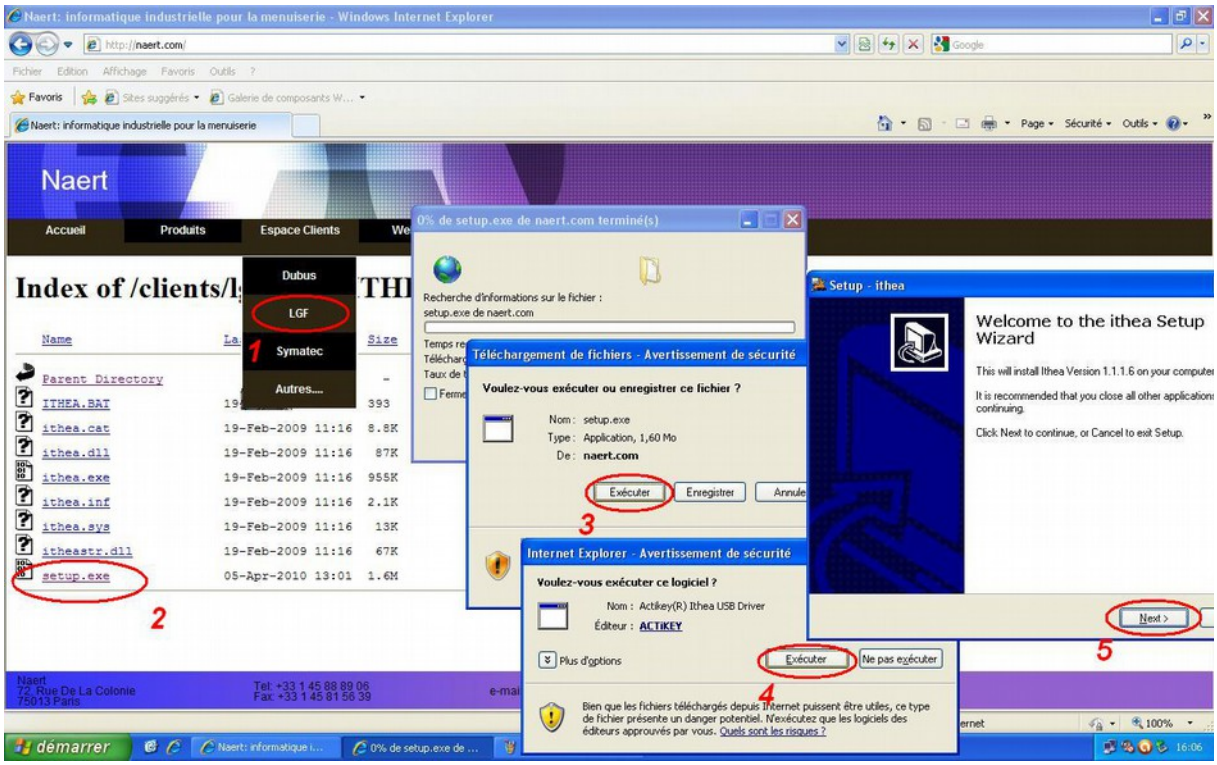


Illustration 5: Downloading and execution of the Setup.exe program

2) Connect the key to the USB port of the PC.

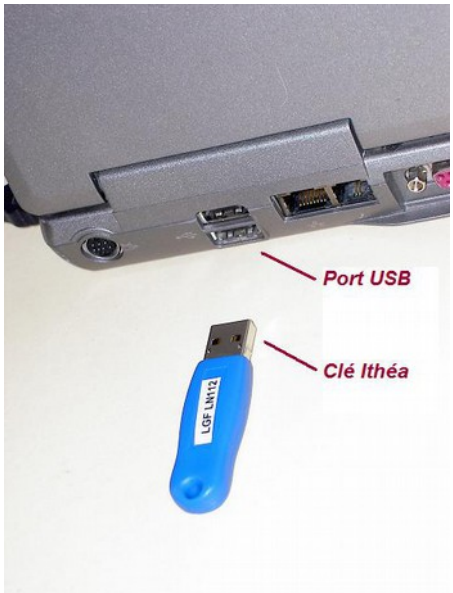


Illustration 6: Connection of the key on the USB port of the PC

7.3.2 Uninstallation procedure:

1) Close the program if it has been started

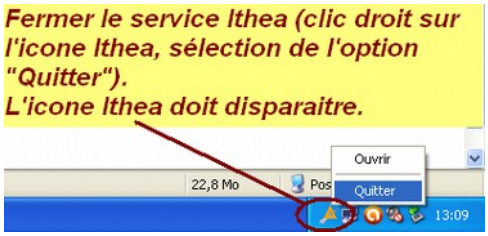


Illustration 7: program stop

2) Uninstall the programs

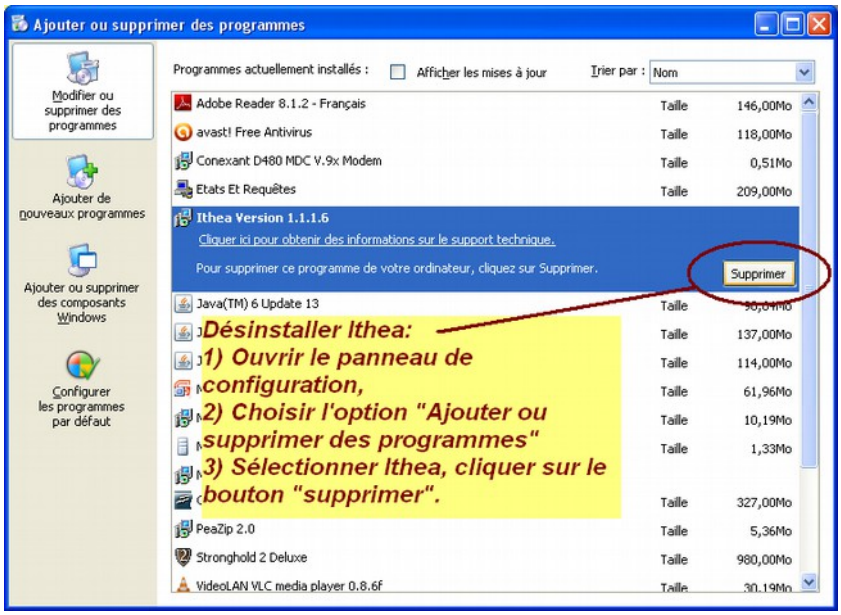


Illustration 8: Uninstalling the program

3) Uninstall the driver

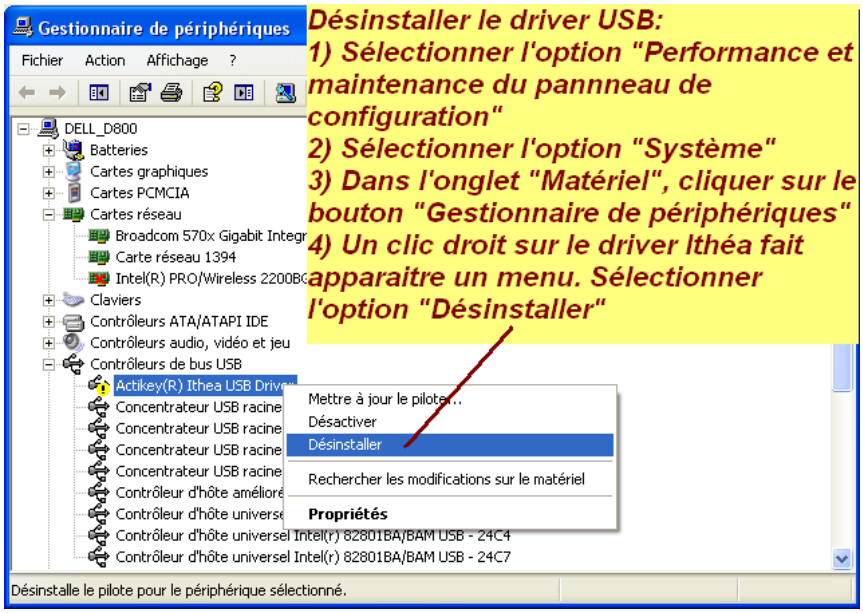


Illustration 9: Uninstalling the driver