

# Maestro

cnc

Expand your creativity



# User manual

# SUMMARY

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# 1 Description of the software

The software described in this manual allows the definition of workpieces and the related machining operations so that they can be machined on numeric control machining centers.

The setting of all workpieces occurs in a three-dimensional CAD environment very useful for the user in order to check immediately his schedule.

Working processes are divided in two different steps: firstly, by setting the geometries, secondly by applying machine tools like in a CAM system.

After setting workpiece and working processes, all of the information will be passed to the machine, which will carry out the machining.

## 1.1 Installation

### 1.1.1 PC minimum requirements

The software requires the following minimum requirements for installation:

- Operating system: Windows XP Professional (SP2), Windows Vista or Windows 7
- Processor: Intel compatible, minimum 2GHz;  
recommended multicore processor
- Memory: minimum 1GB  
recommended 2GB
- Space on disc: 5GB
- Graphical board: OpenGL compatible

The following minimum PC requirements are needed for the "Maestro3D" option for machining surface or with a machine with "edge banding unit":

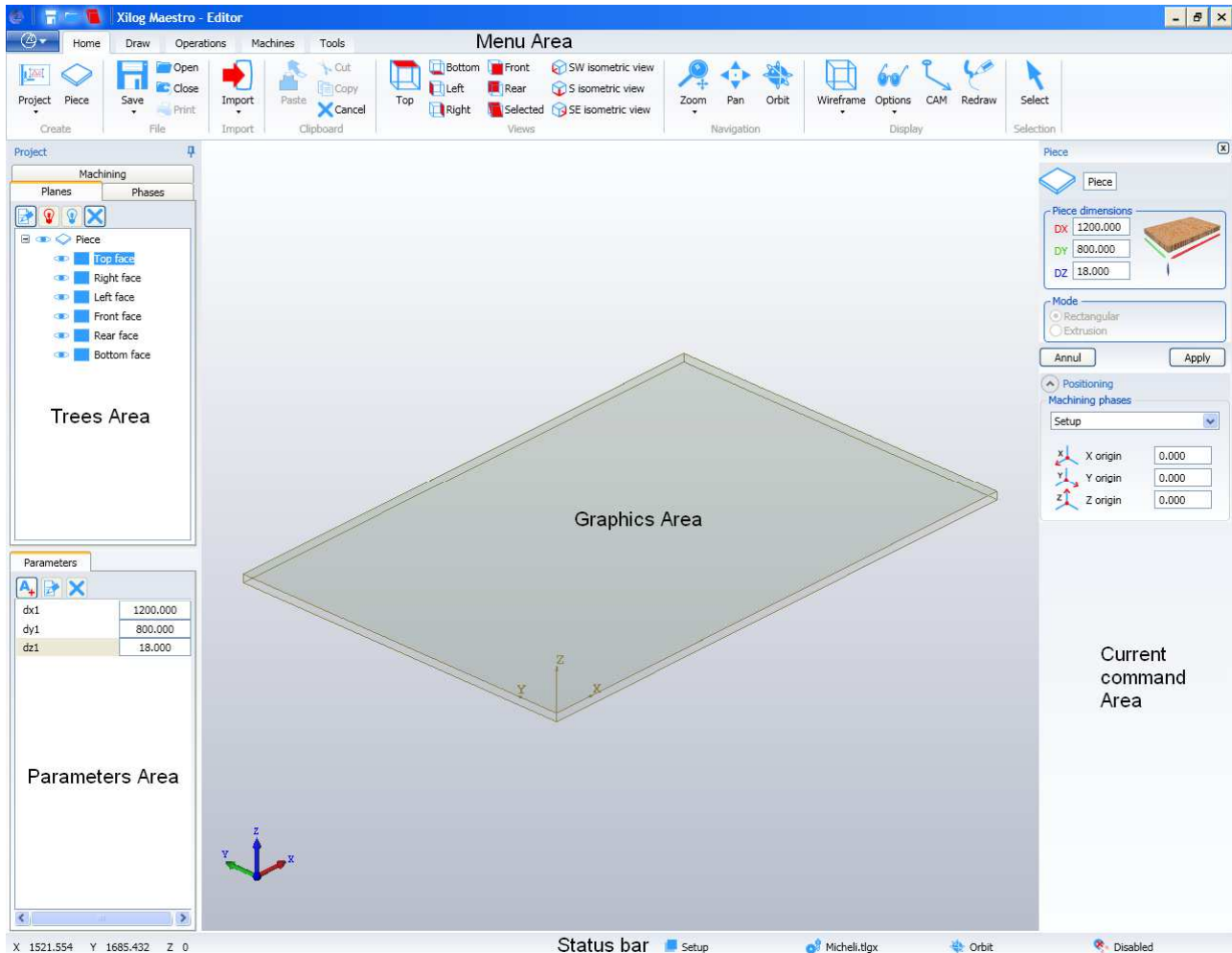
- Operating system: Windows XP Professional (SP2), Windows Vista or Windows 7
- Processor: Intel® I5 - 3.2 GHz
- Memory: 4GB (8GB recommended for 64bit operating systems)
- Free space: 30 GB (recommended HD with 7200 rpm)
- Graphic board: specific (e.g. NVIDIA GFORCE, ATI)

Note: a graphic board integrated in the motherboard could compromise the performance of the application: not recommended

- Monitor Screen resolution 1024x768 colour 32-bit

## 2 Operator interface

Maestro interface is structured as follows:



There are six areas:

Menu

Trees

Parameters

Graphics

Current command

Status bar

Following chapters describe all commands included in the menus.

# 3 Project

Project refers an object that can contain the definition of one or more workpieces with 2D geometries and associated machining.

The project contains all the information needed to define a machining process. This process will be illustrated in the following chapters:

- 1) Project management
- 2) Workpiece
- 3) Worktables
- 4) 2D geometries
- 5) Machining operations
- 6) 3D machining
- 7) Macro and Subprograms
- 8) NC functions
- 9) Work phases
- 10) Workpiece setup in machine
- 11) Blocking system
- 12) Workpieces to be machined contemporaneously
- 13) Projects optimisation
- 14) Mathematical functions

## 3.1 Project management

The "Project" file - with a .pgmx size- must contain at least one workpiece and it might include - but not necessarily- also working processes.

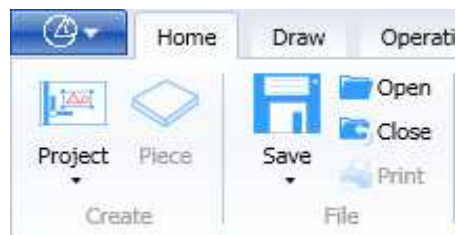
It is a compressed file including:

tool.tlgs	Data Base Tools
progetto.xml	Instructions to set the geometry.
progetto.epi	Clamp/suction pad definition

A project can be:

- Created
- Started
- Ended
- Saved

By using these icons in the Menu area:



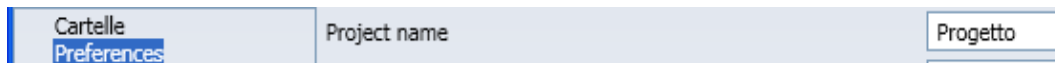
### 3.1.1 Project default

You may change project defaults by clicking on the icon "Options" in the folder "Tools".



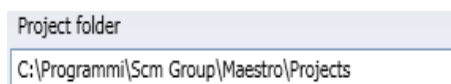
#### 3.1.1.1 Default: "Project name"

If you wish to modify the default name of a new project:  
Click on the icon "Options"  
Fill the field "Project name" .



#### 3.1.1.2 Default "Project folder"

If you wish to change the name of the folder where projects are started/saved:  
Click on the icon "Options"  
Fill the field "Project folder".





### 3.1.1.3 Save the configuration parameters

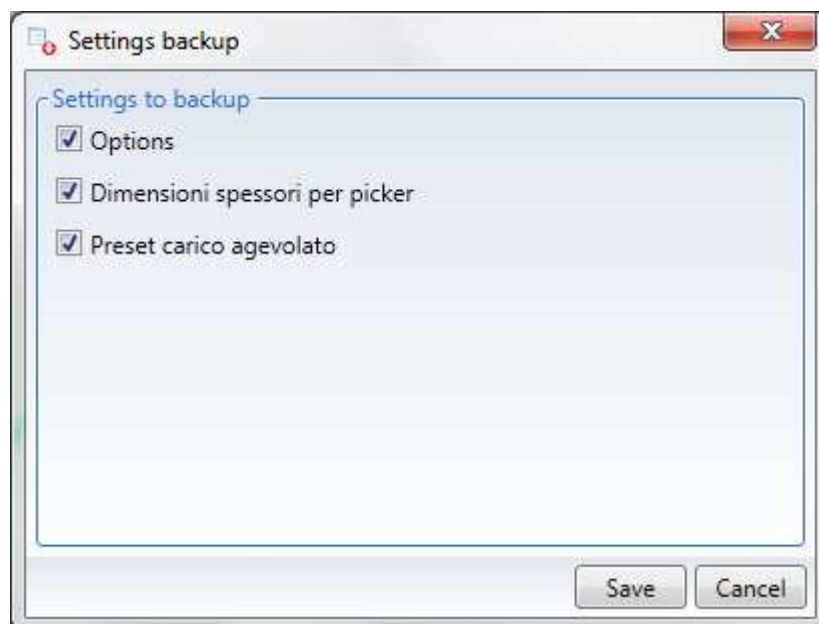
As well as the default settings described above, there are many other configurations possible in the “Options” session, described below.

If a new Maestro Release is installed all the configuration parameters set will be overwritten.

This is why we recommend saving all the configurations set with the “Backup” key in the “Tools” folder.



After having selected the command, the following menu will be displayed, to select which parameters to save.



Press “Cancel” to end the operation.

Press “Save” to save a .settingsx file with all the configuration information.

### 3.1.1.4 Restore the configuration parameters

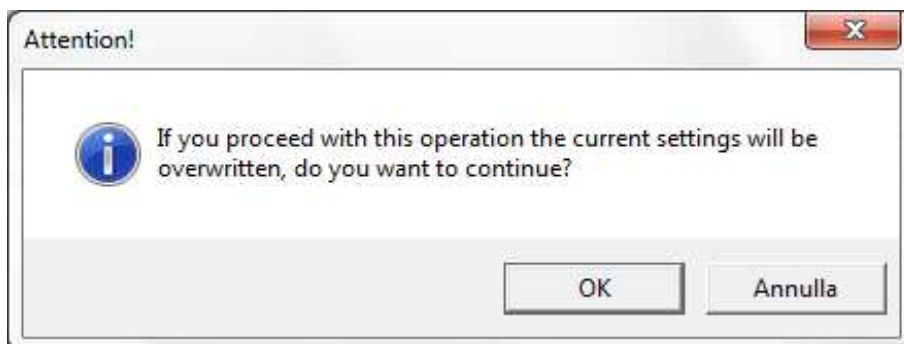
To restore the previously saved configuration parameters use the “Restore” command in the “Tools” folder.



After having selected the command, the following menu will be displayed, to select the type of restore desired.

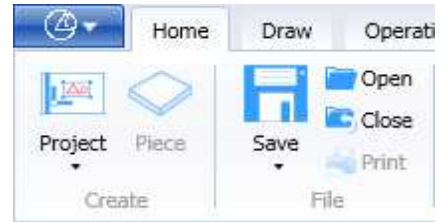


Press “Cancel” to end the operation.  
Press “Open” to display the following confirmation window.



Press “Cancel” to end the operation.  
Press “OK” to select the previously saved .settingsx file and then restore the parameters that will take place after closing Maestro.

### 3.1.2 New project setting



If you wish to set up a new project, click on the icon "Create project"

The name of such new project will be defined as described in paragraph

#### 3.1.1.1 Default "Project name"

Since a project must include at least one workpiece, whenever you create a new Project, you activate the option "Create Workpiece".

If a project was already been started, following option would automatically start. "Closing project"

### 3.1.3 Open an existing project

If you wish to open an existing project, click on the icon "Open".

The resulting window shows you the list where you may select a pgmx file.

The default folder will be defined as described in paragraph

#### 3.1.1.2. Default "Project folder"

Please select a pgmx file and click on "Open".

Or click twice on the pgmx file.

For a project that has already been started, following option would be automatically performed:

"Closing project"

### 3.1.4 Import Pgm/Xxl

This function is used to convert a .pgm or .xxl file into a project.

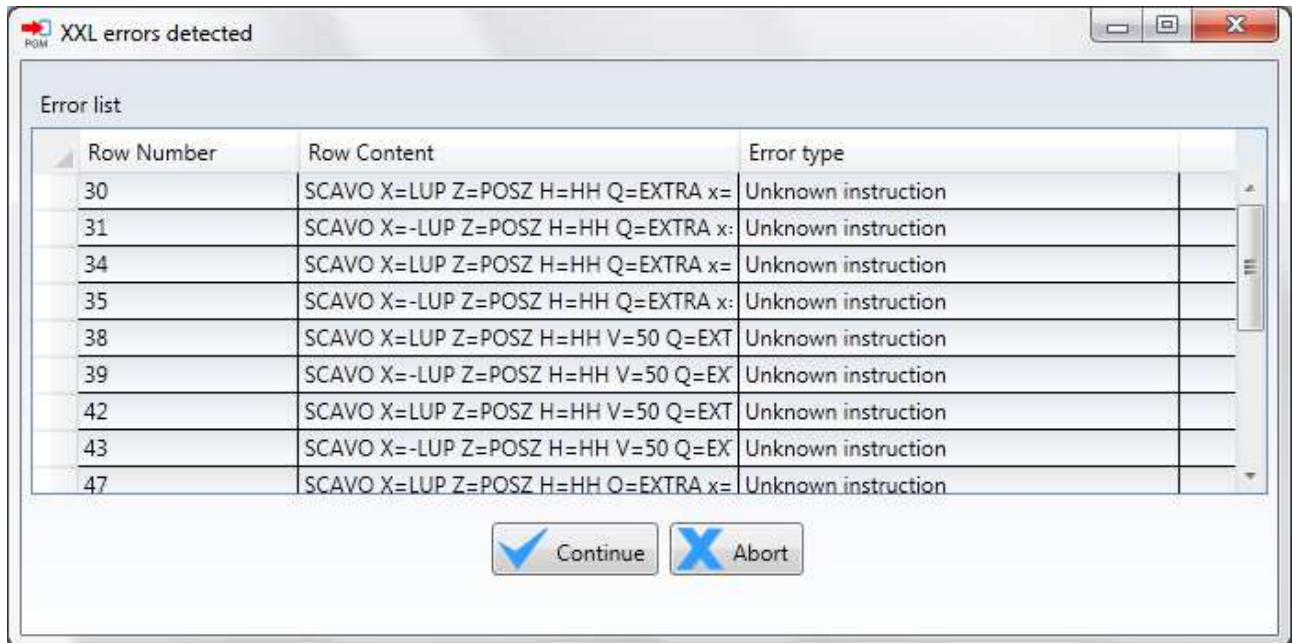
To import a .pgm or .xxl file click on the “Import Pgm/Xxl” icon in the “Import” menu.

You will be asked to close any open project as the Import Pgm/Xxl function is equivalent to opening an existing project.

A window will be displayed from which you can select a file with extension .pgm or .xxl.

The default folder will be the one defined, as described in section 13.1.2 Post-Processor Options which describes the “Output folder” Then select an .xxl file and then press “Open”.

Any unrecognised instructions will be displayed in a “XXL errors found” screen with the line number of their location in the input life.



If there are no errors the “Parameters found” screen is displayed with a list of the known parameters in the file to import.

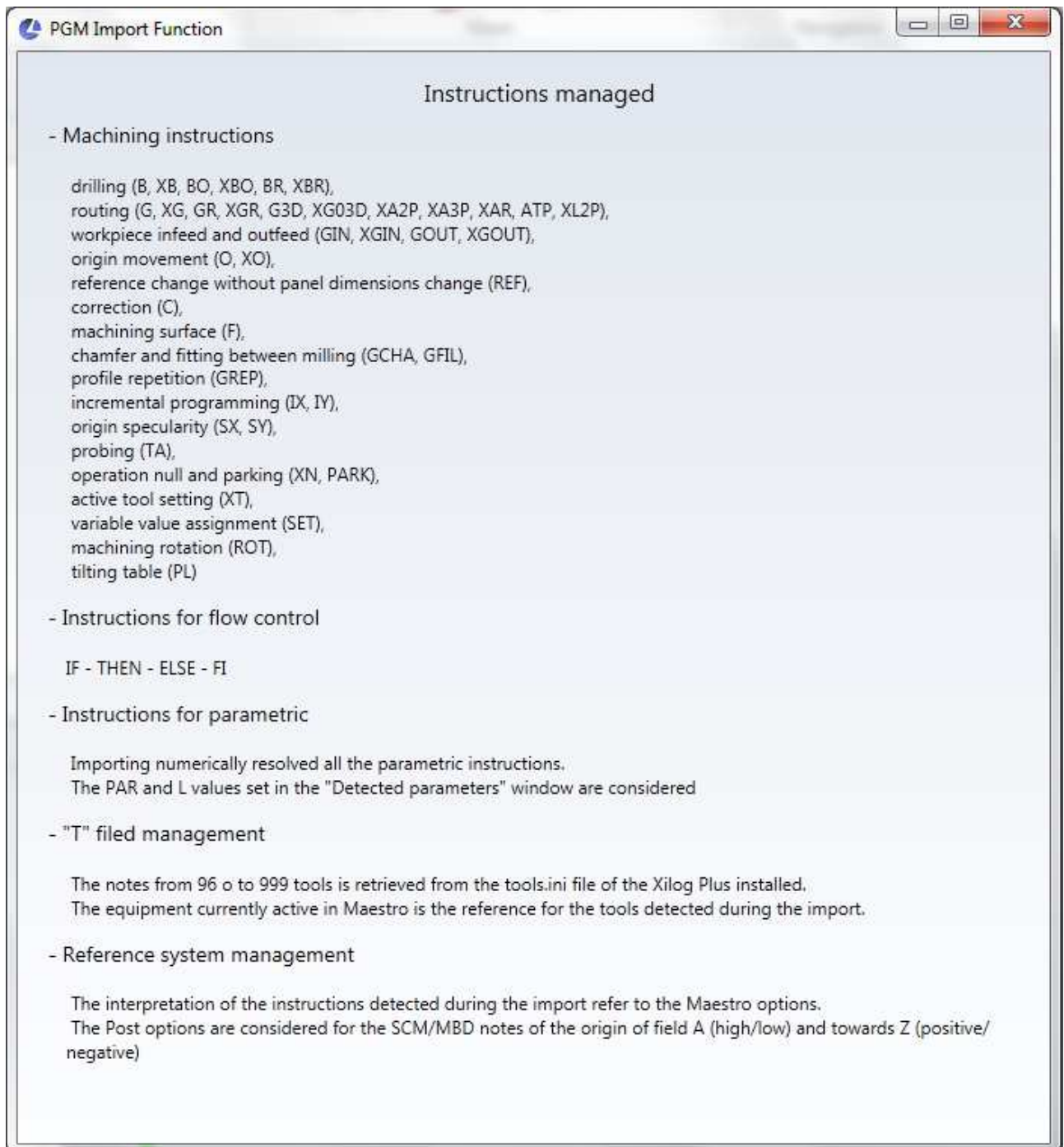
The following keys are found on the bottom:



To proceed click on “OK” and the input file will be imported as if it were an existing project.

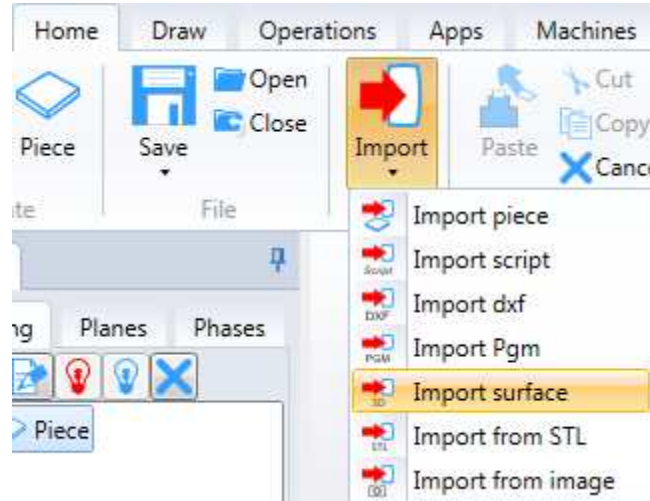
To interrupt the import click on “Cancel”

For a list of the instructions click on "Help".



### 3.1.5 Import 3D template

It is possible to import a 3D model using the IGES formats (\*.igs, \*.Iges) or the STEP (\*.stp, \*.Step) formats using the "Import 3D model" command in the "Home-Import" menu.



After selecting the command, simply select the file to be imported. If the model is imported when there is no part in the project, a new part will be created with the minimum dimensions needed to contain the imported model.

### 3.1.6 Import from STL

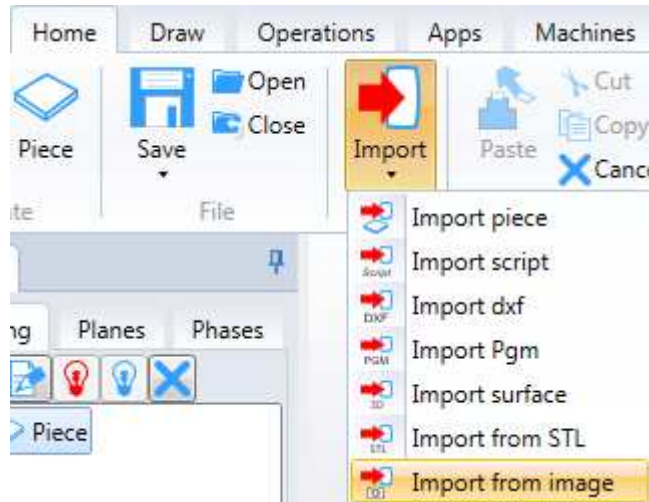
You can also import a model from an STL file (\*.stl) using the "Import from STL" command in the "Home-Import" menu.



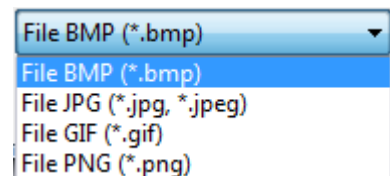
After selecting the command, simply select the file to be imported. If the model is imported when there is no part in the project, a new part will be created with the minimum dimensions needed to contain the imported model.

### 3.1.7 Import Image

You can import an image and turn it into three-dimensional geometry using the "Import image as a surface" command in the "Home-Import" menu.



After selecting the command, you can select a file to be imported with the following formats:

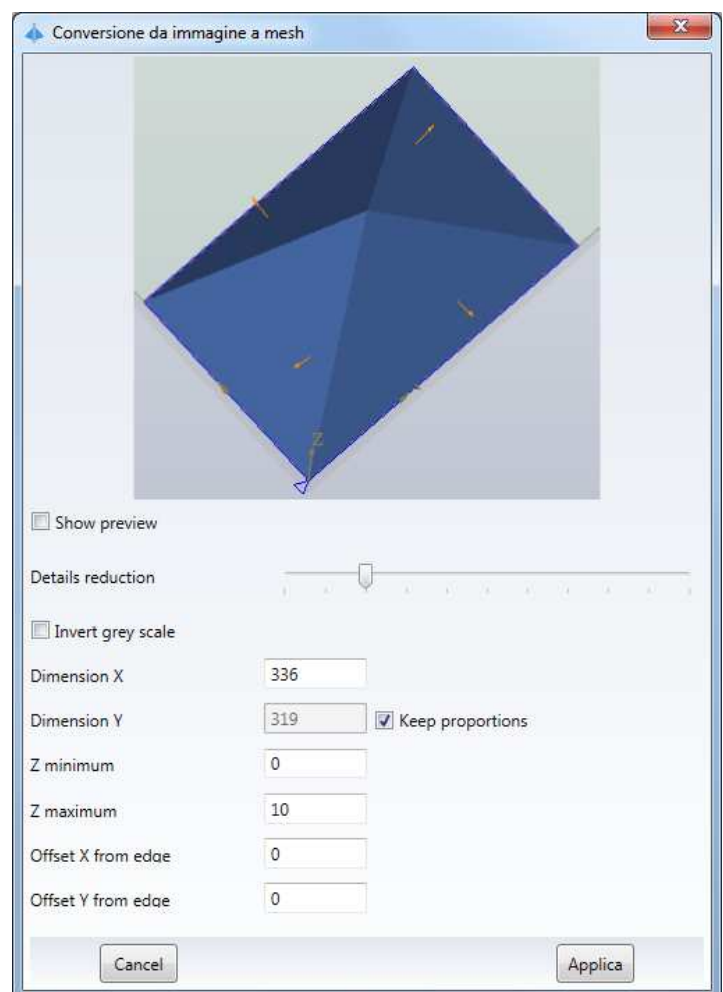


After selection, the following dialog box will appear so you can manage the following characteristics of the surface that will be generated:

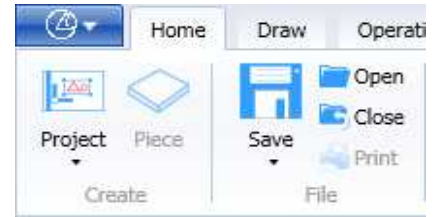
- Dimensions (X, Y)
- Maximum and minimum size of Z
- Distance of the surface from part edge

The generated surface will be lower for darker pixels and higher for light pixels. This rule can be reversed by checking the "Invert grey scale" item

With the "Detail reduction" potentiometer, you can soften the surface by blurring the grey tone of the image.

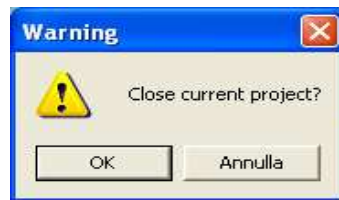


### 3.1.8 End project

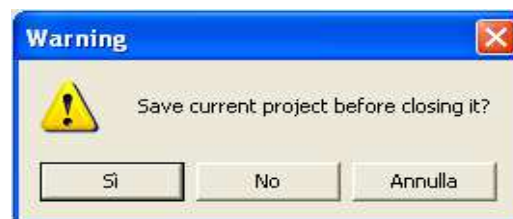


If you wish to end a project click on the icon "Close".

If your project is saved, following question will appear:



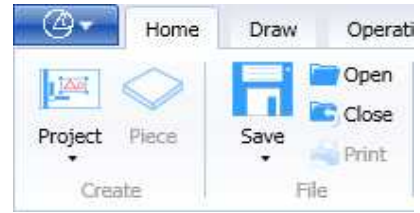
If your project has not yet been saved, following question will be displayed:



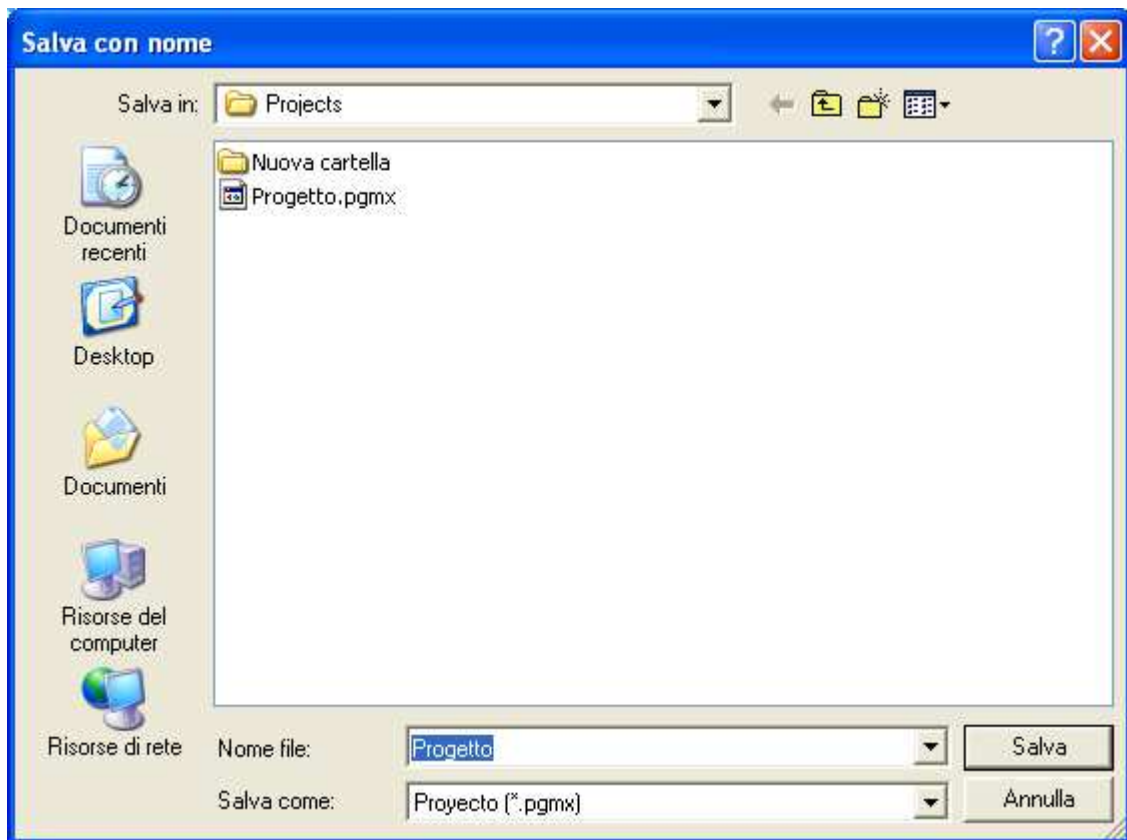
If you click on "Yes", following option will be automatically performed in order to save your project: "Automatic Saving"



### 3.1.9 Save Project



If you wish to save an open project, click on the icon "Save"  
If the project was already named, no question would be asked.  
If the project was already created, the option "Save with name" would be performed.



You can specify the project name in the box "File name".  
then click on "Save".

### 3.1.10 Project Graphic display

The project is displayed in the main area at the centre of the screen.

It is a three-dimensional representation.

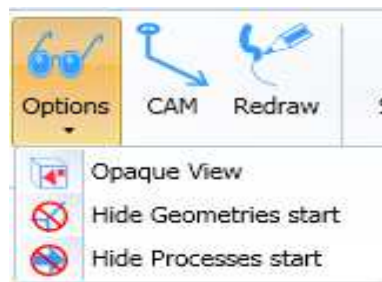
The operator can choose what, where and how to see interesting project workpieces.

#### 3.1.10.1 What you see of the project

In order to select the working processes to display among all those included in a sole project, you can use the commands available:

- In the plane tree (see paragraph 3.3.5.4)
- in the working process tree (see paragraph 3.5.19.3)
- in the menu of the step tree:

Or use the "Display" "Options" menu "Opaque View" control."



The command "Dim" displays only the working processes of the active plane.

If one or more working processes of the active plane have been switched off with the tree menu, these will not be displayed.

### 3.1.10.2 Where to see the project

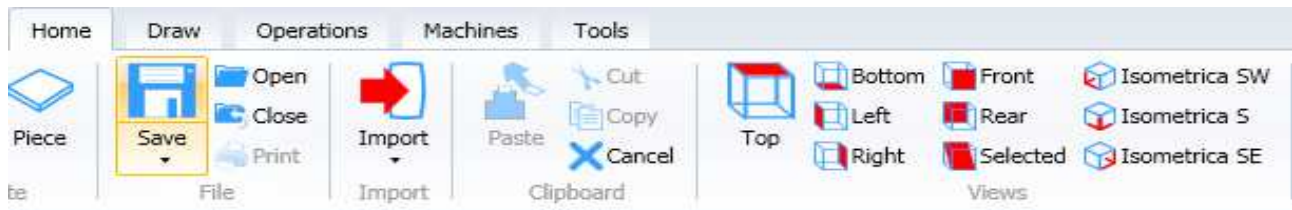
If you wish to see the project, it is possible to set the point from which you display it.

The main references to this purposes are the "Standard faces".

Immediate commands are available to see the different faces.

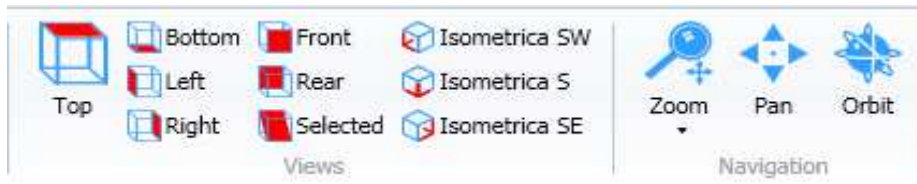
In the menu "Home" sub-menu "Views"

Clicking on the commands with the Face names, you will only display the face related to the selected view.

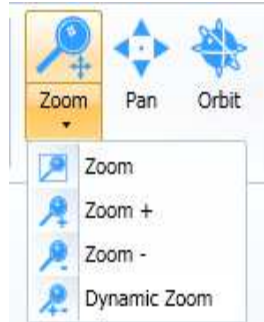


In the same menu, you can also see three commands related to three isometric views.

After selecting one out of ten views, you can zoom, size, shift or rotate the project view by using the menu "Surfing" to the right of the menu "Views".



### 3.1.10.2.1 Zoom



The main icon of the Zoom displays the whole screen.

The icon "Zoom" allows you to zoom through a rectangle. You can select it by clicking on the left button of the mouse.

The icon "Zoom +" allows you to size the view whenever you click with the left button of the mouse.

You can use this option also by rotating the mouse roller forward.

The icon "Zoom" allows you to size the view when you click with the left button of the mouse.

You can select this option by rotating the roller backwards.

The icon "Dynamic Zoom" allows you to zoom by clicking with the left button of the mouse on the area to be zoomed and then to move in order to zoom/size the view.

You can select this option by rotating the roller backwards and forwards.

The commands "Zoom +", "Zoom -" and "dynamic Zoom" can be disabled if you click on the "Esc key".

### 3.1.10.2.2 Pan

Pan allows you to shift the view by keeping the left button of the mouse pressed and then to shift with the mouse.

You can disable the command with the button "Esc" or by modifying the command.

The same function is always available if you press the central button of the mouse and then you move with the mouse.

### 3.1.10.2.3 Orbit

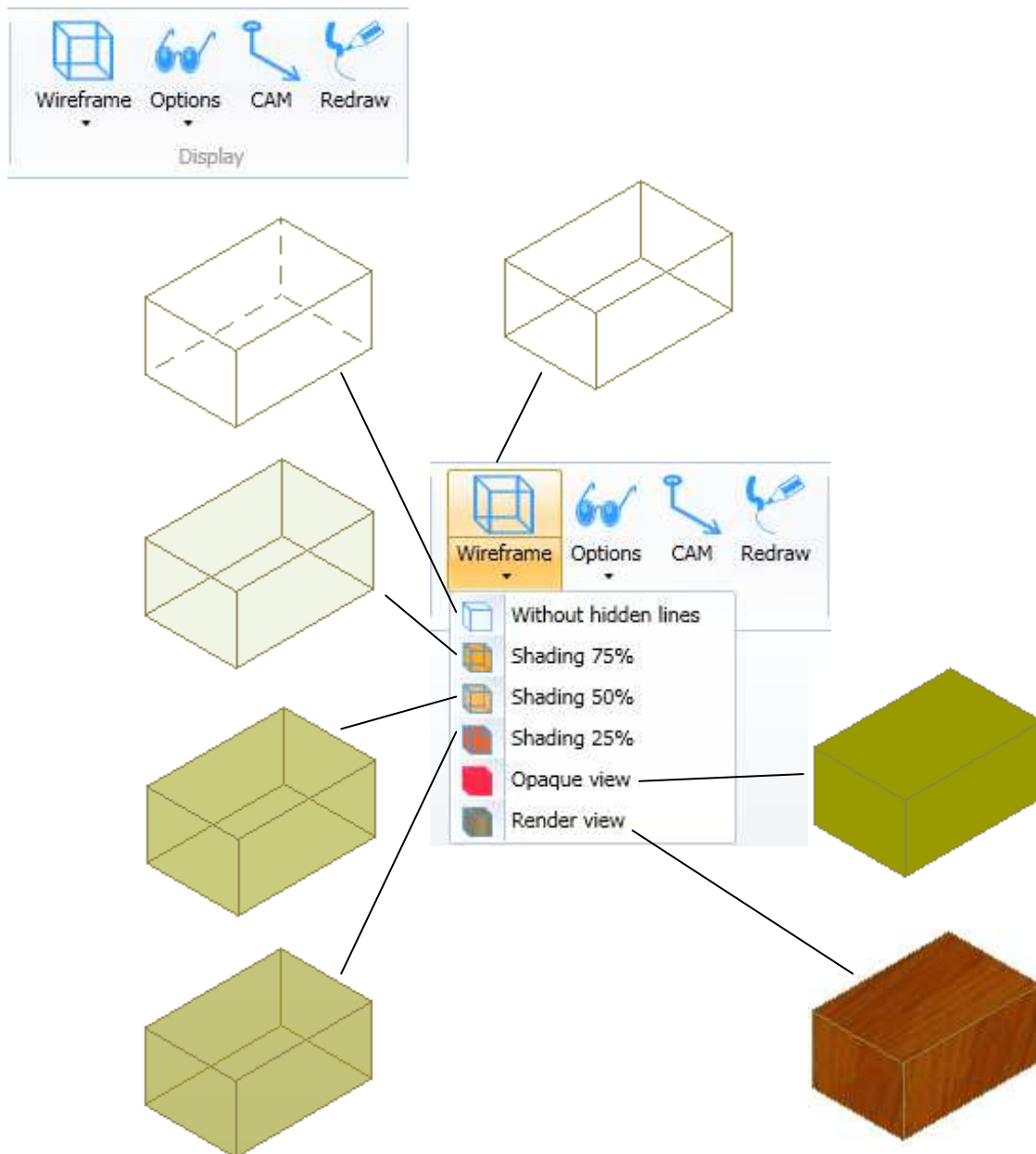
This option allows you to rotate the view by keeping the left button pressed and then move with the mouse.

You can disable the command with the button "Esc" or by modifying the command.

This option is also available if you press the button "Ctrl", the central button of the mouse while moving the mouse.

### 3.1.10.3 How to see the project

All project elements can be displayed in different ways. In order to choose the way to see such elements, the commands of this menu allow you to produce the different representations that have been shown.



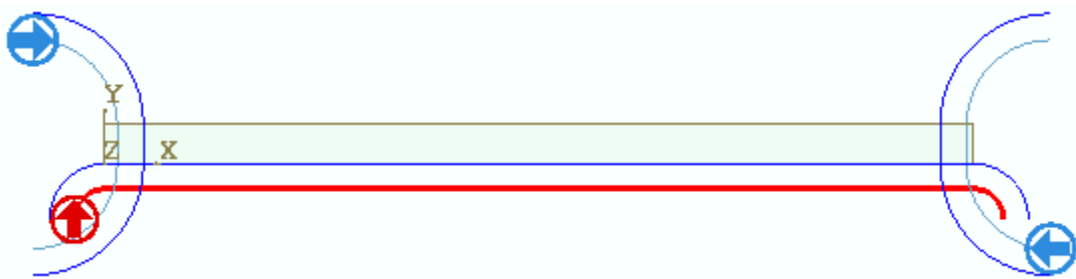
The command CAMCAM allows you to display the tool path and the side of correction.

Here you can see a workpiece with a profiling and two tenonings in Shading representation (25%)



The same workpiece in CAM representation shows you:

- the basic geometries of the profile
- paths in tool centre
- the circle with the arrow that indicates the initial point of the working process and the direction



The command "Redraw" upgrades the whole graphic area.

## 3.2 Workpiece

Workpieces must be defined as "Finalised workpiece" and optionally also as "Rough workpiece". You may insert more workpieces in one project.

### 3.2.1 Rough workpiece

Function implemented only in "Script" mode (see MSL manual)

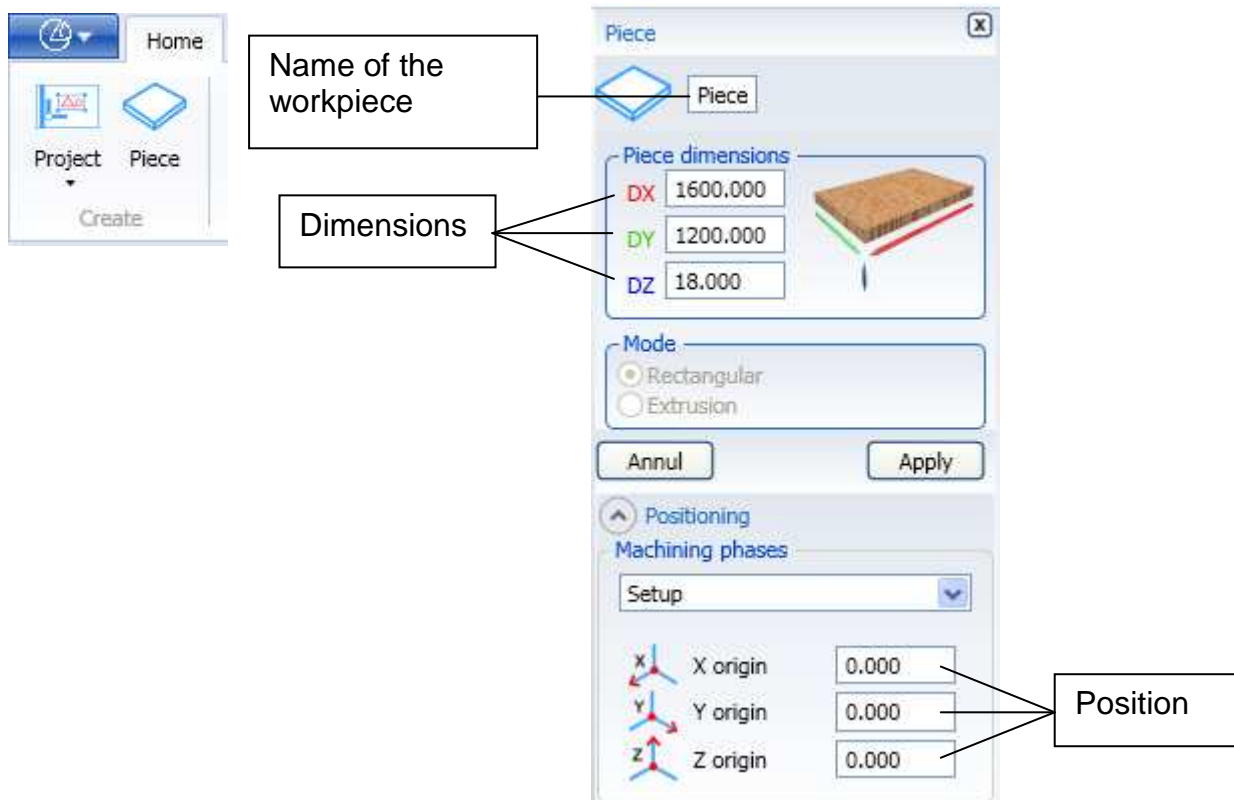
### 3.2.2 Finalised workpiece

The finalised workpiece can have several geometric shapes:

#### 3.2.2.1 Parallelepiped shaped workpiece

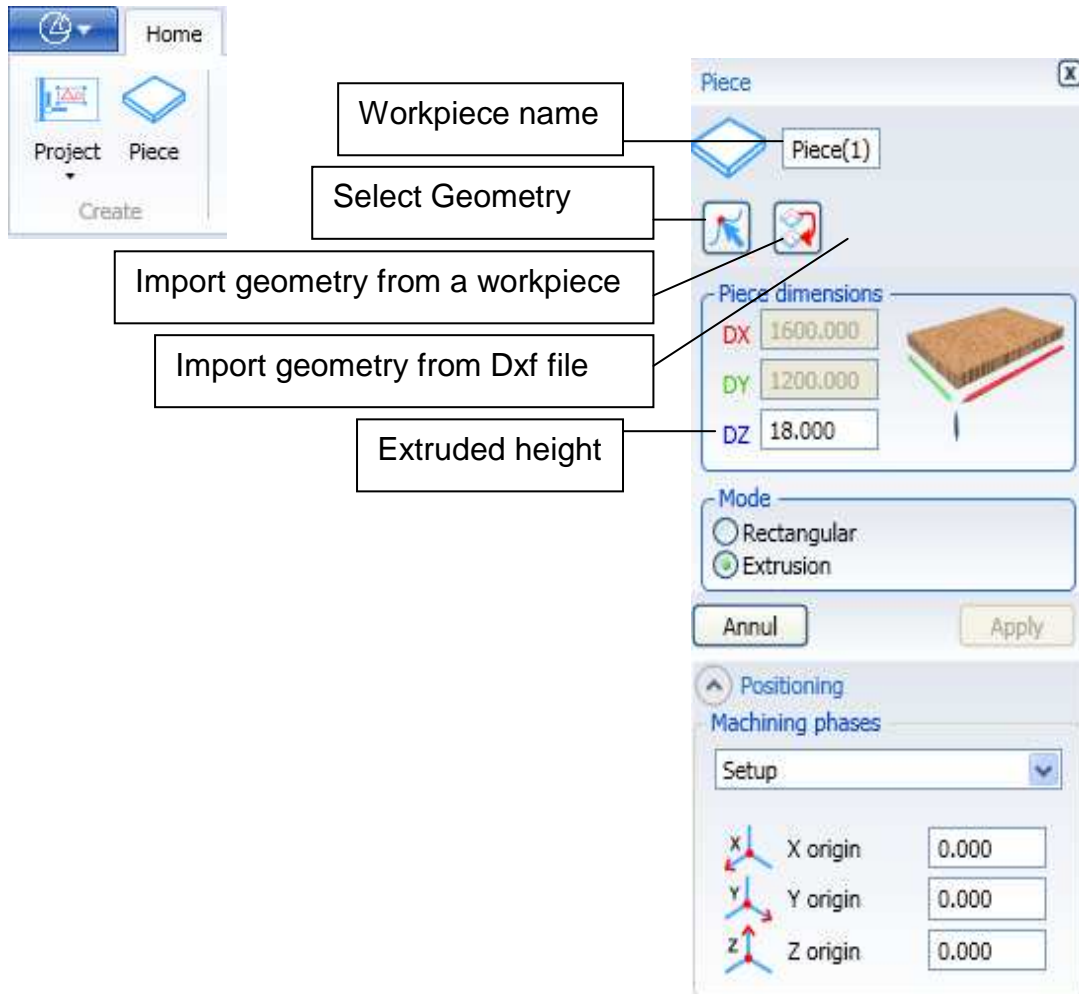
If you wish to set a parallelepiped shaped workpiece click on the menu "Create workpiece".

Introduce the values in the menu on the right.



You can change your field values after creating the workpiece.

### 3.2.2.2 Extruded workpiece



A closed geometry, that can be obtained in 3 ways, is required to make the extruded workpiece:

- 1) Creating a new geometry
- 2) Importing geometry from another workpiece
- 3) Importing geometry from dxf

Click on "Select Geometry" then on an available closed geometry

Click on "Apply" to create the "Extruded Piece" contained by a parallelepiped representing its clearance.

The various fields' values can also be changed after the piece is created.



### 3.2.2.2.1 New geometry creation for extruded

Use the "Draw" menu controls (see paragraph 3.4) to create a new closed geometry.

### 3.2.2.2.2 Geometry importing for extruded from another workpiece

Use the "Import geometry from a workpiece" control to have the closed geometry belonging to a project piece. Select a project from those in the memory.

If the selected project only has one piece, its closed geometry will appear.

If the selected project has more than one piece, select the piece which geometry is to be imported. The closed geometry of the selected piece will appear.

### 3.2.2.2.3 Geometry importing for extruded from dxf

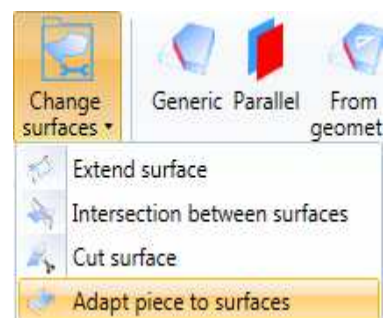
Use the "Import geometry from Dxf file" control to have the closed geometry contained in a Dxf file. From the memory select a Dxf file containing a closed geometry.

## 3.2.3 Change workpiece dimensions

If three-dimensional geometries have been imported or completed for an existing project, the part dimensions can be automatically updated to contain the selected geometries.

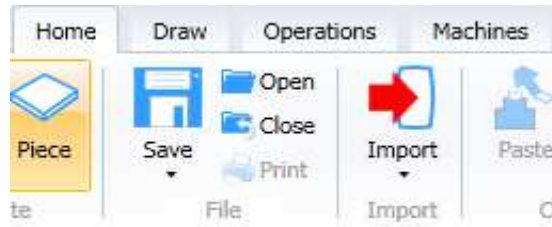
The function is activated using the "Fit part to surfaces" command in the "Draw-Modify Surfaces" menu

After selecting the surface (s), the X, Y, Z super-material to be added is requested in terms of the margins of the selected area (s).



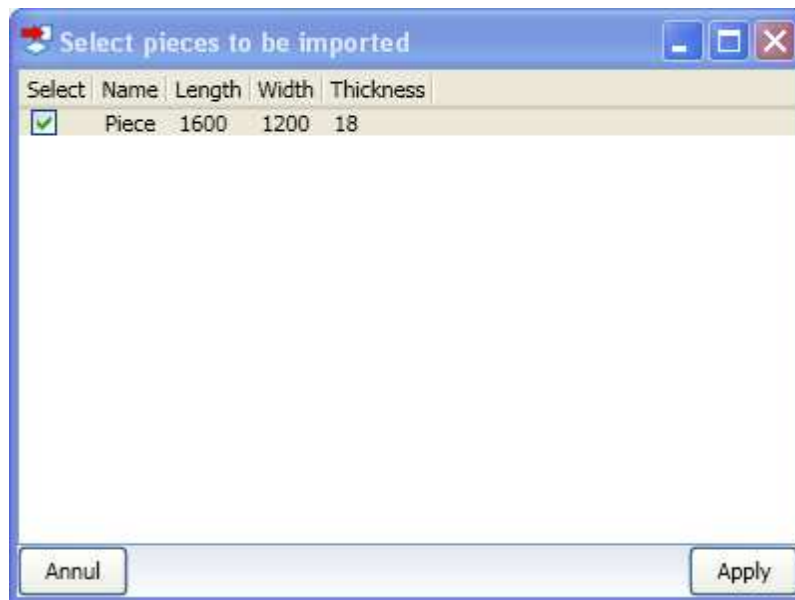
### 3.2.4 Import the workpiece

You may also import a workpiece to another project. To this purpose you can use the command "Import workpiece" of the menu "Home" "Importa".



The resulting window shows you the list of projects where you can select the workpiece to be imported.

After clicking on the icon "Open", the list of workpieces in the selected project will be showed:



You can select one or more workpieces to import and click on the icon "Open". In the tree of our project the selected workpieces will be added.

### 3.2.5 workpiece Removal

The workpiece can be removed from the working processes. (see paragraph3.5.19 )

In order to remove a workpiece:

- select the workpiece to remove from the tree of the working processes.
- click on the command "Remove" in the tree commands or in the context menu of the workpiece

the workpiece removal implies also that all related workings processes are removed as well.

### 3.3 Working planes

The "working plane" is a plane with Cartesian axis where you can add shapes of plane geometry like line, arc, circle and so on (see paragraph **3.4**)

The work tops are created in four ways:

- automatically during the creation of a new workpiece (Standard Faces)
- by creating a plane in parallel with an existing plane (Parallel Plane)
- by setting the rotations around axis Z and around axis X (Inclined Plane)
- Cut with blade (Generated work top)

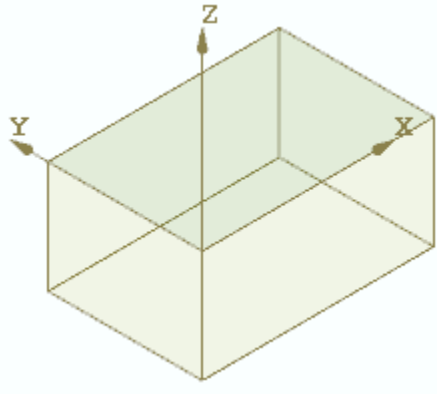
In case of Parallel and Inclined Planes you can also modify - after the creation - any value in the window - the name of the plane as well.

With regard to all standard faces, all fields remain unvaried.

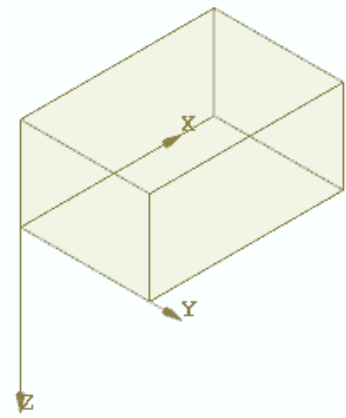
### 3.3.1 Standard Faces

A workpiece has six faces in which six working planes called "Standard Faces" have been created when a new workpiece is created.

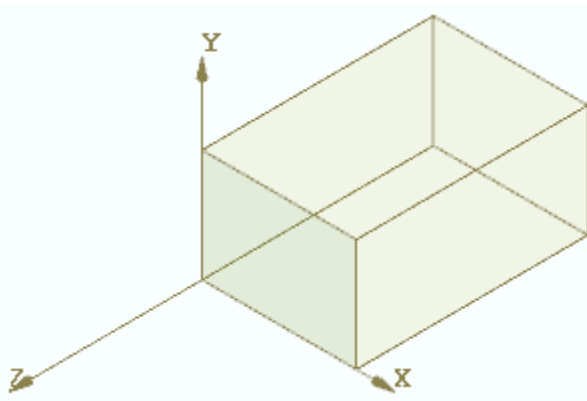
Names and positions of the "Standard Faces" are as follows:



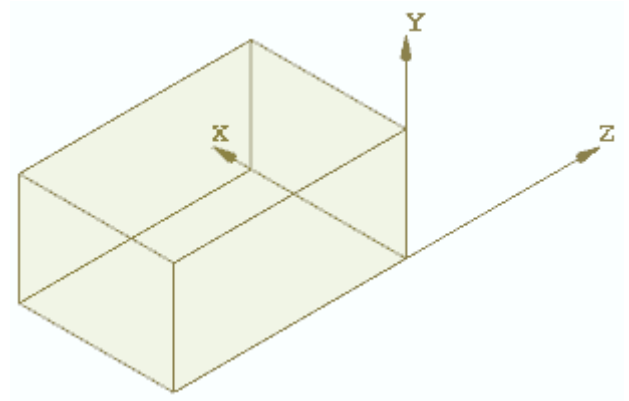
Upper Face



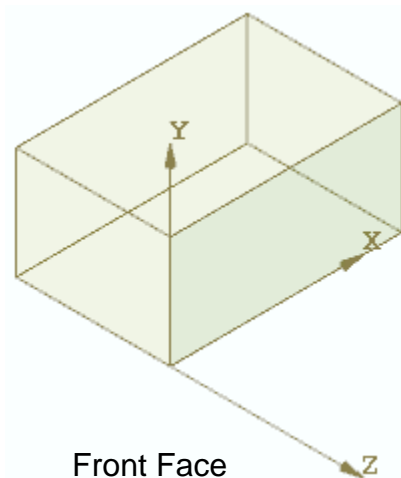
Lower Face



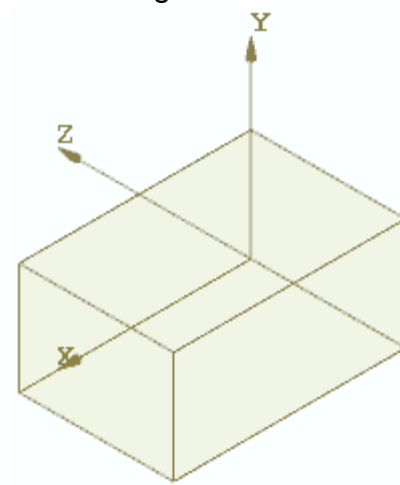
Left Face



Right face



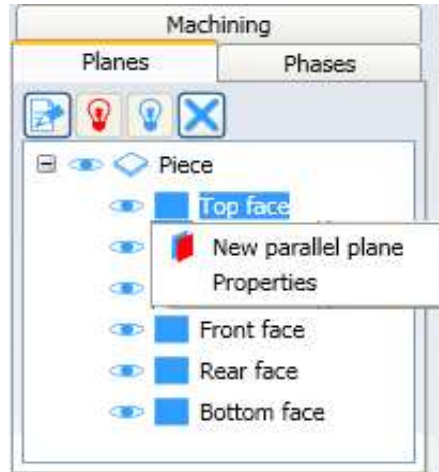
Front Face



Back Face

### 3.3.2 Parallel Plane

A parallel plane is a new plane in parallel with an existing plane in the project. There are two possibilities in order to create a parallel plane:  
by using the tree of planes



By clicking - with the right button of the mouse - on the selected face, the resulting menu allows you to create a "New parallel plane"

by using the menu "Draw" "Inclined Plane" "Parallel Plane"



In both cases a window will be displayed:

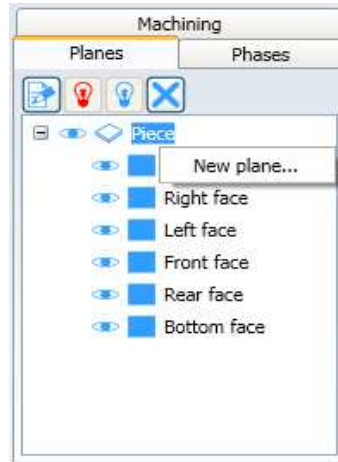


By setting following options "Distance from the plane" and "Apply" (apply), a new parallel Plane will be created.

Note: with positive value it is created on the workpiece, with negative it is created inside the workpiece.

### 3.3.3 Inclined Plane

An inclined plane is a general plane created by the user with origin and inclination. Two possibilities apply for every plane:  
by using the plane tree

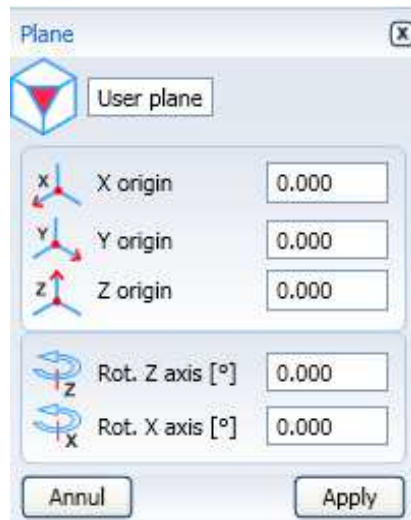


By clicking - with the right button of the mouse - on "Workpiece", the heading "New plane" will appear.

by using the menu "Draw" "Inclined Planes" "Generic"



In both cases a window will be displayed:



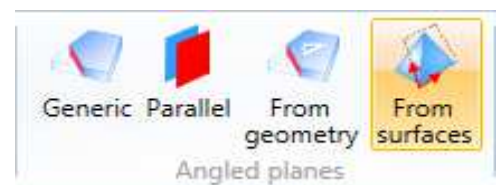
The definition of Inclined Plane starts with the threesome related to the workpiece placed at the lower corner on the left.

By setting X values as origin, Y origin and Z origin, the origin of the new plane is established.

By setting the rotating angles of Z-axis and X-axis, you also set the plane direction.

You must also take into account that both rotation angles are sequential, that is to say: only after the rotation around the Z-axis, the rotation around the new X-axis position will be performed.

### 3.3.4 Plane from surface area



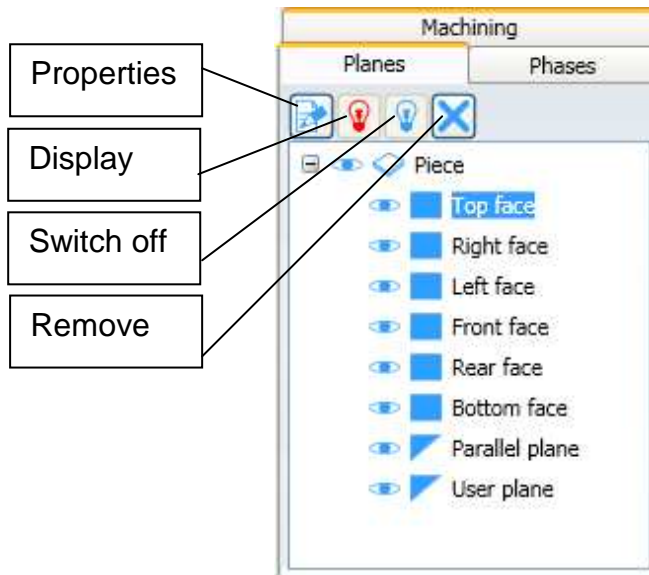
It is possible to create a coplanar plane to a surface using the "From surface" command in the "Draw-inclined planes" menu.

Once the command has been selected, you simply select a surface and a coplanar user plane to the selected surface will be set up.

Obviously the surface must be exclusively flat.

### 3.3.5 Work plane tree

The work plane tree collects all planes set for a certain project.



This tree - a workpiece from displaying existing planes- allows you to:

- activate an existing plane
- display and change the properties of existing planes
- remove a plane
- create parallel planes (see paragraph 3.3.2)
- create an inclined plane (see paragraph 3.3.3)

Buttons with light bulbs (on or off) allow you to display or switch off the working processes (see paragraph 3.5.19.3)

#### 3.3.5.1 Work plane selection

The work plane selection will identify the active plane, that is to say, the plane where the two-dimensional geometry will be drawn.

In order to reactivate an existing plane, click - with the left button of your mouse- on the name of the plane in the tree.

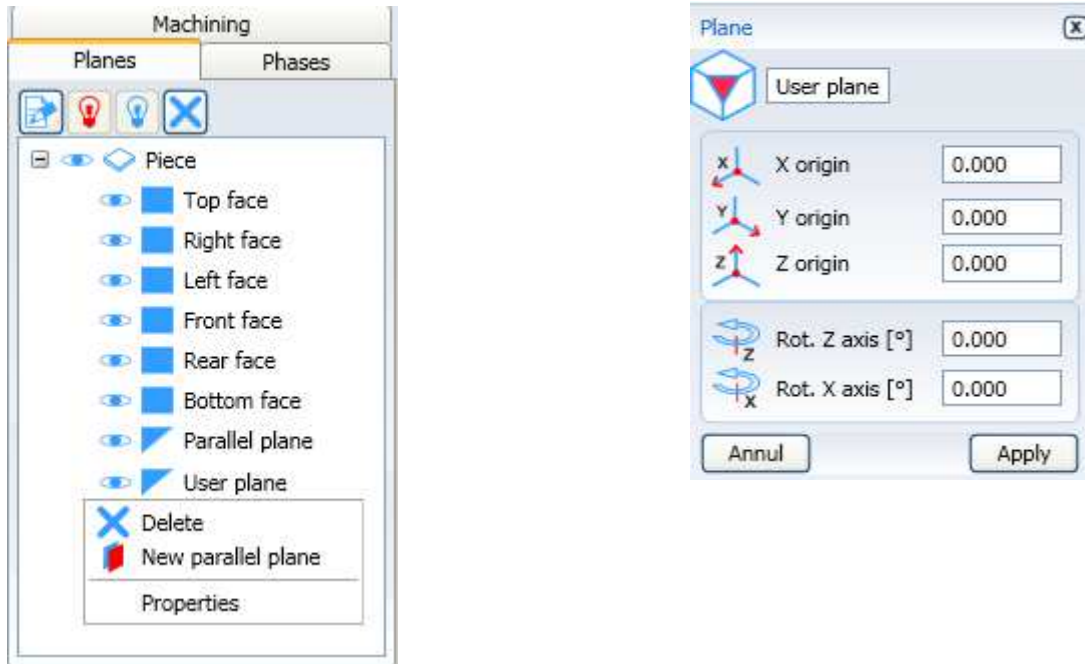
If you wish to know the active plane, check the plane highlighted in light blue.



### 3.3.5.2 Properties and plane modification

In order to see or modify the properties of a plane you can activate a plane then click on the button "Properties" or click with the right button on the name of that plane and on the heading "Properties" of the plane menu.

On the right side of your screen the menu will show you all current properties of the plane. If you wish to modify the plane, click on "Apply" in order to change its values.



### 3.3.5.3 Plane removal

If you wish to remove a plane click with the right button on the plane name, then click on "Remove" of the plane menu.

You can delete only the Parallel Planes or Inclined Planes. You are not allowed to delete the Standard Faces.

### 3.3.5.4 Displaying working processes on planes

You are free to decide to display -or not- the working processes on the plane tree. To the left side of every work plane there is the icon which stands for the face type. To the left of the icon there is the symbol which represents an "eye". By clicking with the left button on the eye, you change its status from open to closed. If the eye is open, it means that all plane working processes will be displayed. If the eye is closed, it means that all working processes will not be displayed.

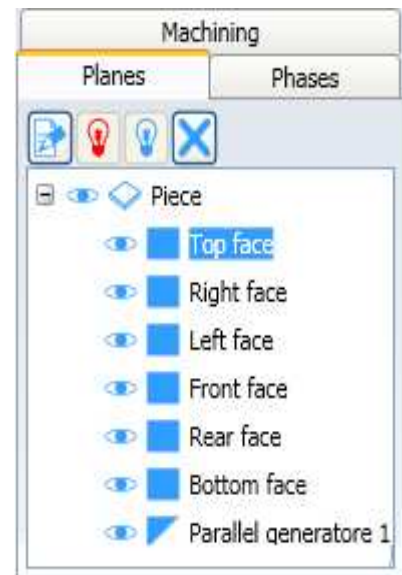
In this example, you see a project in which the "Upper Face" is active but all working processes on the same "Upper Face" have been switched off.

If you wish to switch all working processes off you can:

- 1) close all eyes to the left of the planes.
- 2) close the eye to the left of the workpiece
- 3) or click on the light bulb that is off.

If you wish to display all working processes you can:

- 1) open all eyes to the left of the planes
- 2) open the eye to the left of the workpiece
- 3) or click on the light bulb that is on.



The selection on the tree has an immediate impact on the graphic area.

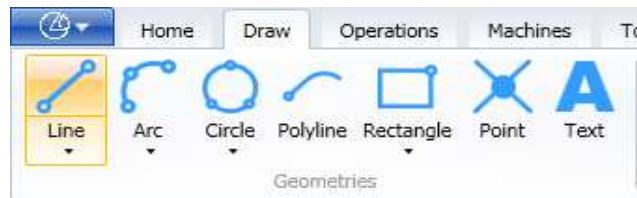
## 3.4 Geometries

### 3.4.1 Enter new 2D geometries

The 2D geometry is used in order to set the profile of an extruded workpiece as well as to set a profile (open or closed) of a working process

Before introducing any two-dimensional geometric element, you must set and activate a working plane.

The icons about the commands to insert the basic geometries are displayed in the menu "Draw" "Geometries"



All commands to create 2D geometry can be interrupted if you select a new command or click on "Esc key".

### 3.4.1.1 Type a new point

Every command aimed at introducing a new geometric shape has to identify one or more points by the operator.

Firstly, it is important to explain the meaning of "type a new point".

"Type a new point" means that the operator creates a new point or sets an existing one within the project.

The point can be created with Cartesian Coordinates (X, Y) or with Polar Coordinates (Length, Angle).

If you wish to type a new point with Cartesian Coordinates (X; Y) you can:

- move the mouse (X and Y coordinates are upgraded).
- click with the left button of the mouse or press Enter key.

Or:

- type the value of the X coordinate
- move your mouse (Y coordinate is upgraded)
- press Enter key.

Or:

- type the value of the Y coordinate
- move the mouse (X coordinate is upgraded)
- press Enter key.

Or:

- type the value of the X coordinate
- type the value of the Y coordinate
- press Enter key.

In order to type a new point with **Polar Coordinates** (Length, Angle) you can:

- move the mouse (values corresponding to Length and Angle are upgraded)
- click with the left button of the mouse or press Enter key.

Or:

- type the Length value
- move the mouse (the Angle value is upgraded)
- press Enter key.

Or:

- type the Angle value
- move the mouse (the Length value is upgraded)
- press Enter key.

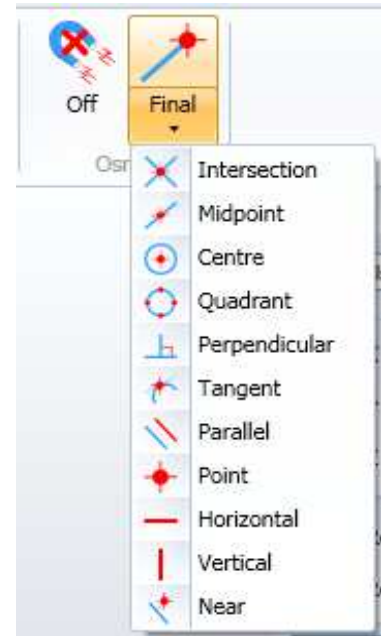
Or:

- type the Length value
- type the Angle value
- press Enter key.

### 3.4.1.2 Type an existing point by using the Osnap

In order to type a point, it is often desirable to use the points of the shapes that are already on our plane.

To this purpose, the Osnap (Object Snap) in the following menu are a useful support.



For an existing point:

- click on one command Osnap
- move the mouse (a red cross appears on the geometry)
- click with the left button of the mouse or press Enter key.

**Final**→ It is linked to the final point of any graphic shape.  
In a polyline it is linked to the final point of every entity that is contained in its structure.

**Intersection**→ It is linked to the intersection of two entities.

**Middle**→ It is linked to the middle point of any entity (also a circle)

**Centre**→ It is linked to the centre of arcs and circles.

**Quadrant**→ It is linked to the X and Y points of maximum and minimum in the circles.

**Perpendicular** → It is linked to the point perpendicularly projected to a segment.

**Tangent** → It is linked to the tangent point of an arc or a circle.

**Parallel**→ It is linked to the point projected on the parallel to a segment.

**Point** → It is linked to an entity "Point".

**Horizontal** → It keeps the Y coordinate of the previous point.

**Vertical** → It keeps the X coordinate of the previous point.

**Near** → It is linked to the point nearer to the point of the mouse.

### 3.4.1.3 Point

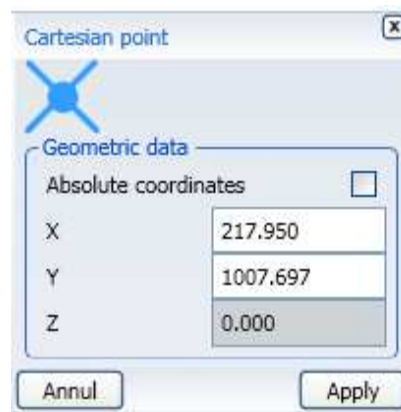
It introduces a geometric element: "Point"

In order to introduce a point:

- click on the command "Point"
- type a point

In the graphic area the point is represented with this symbol: ☒

Point properties



### 3.4.1.4 Line

It inserts a line

A line can be inserted in two ways:

- 1) Line for 2 points
- 2) Length and angle line



In order to introduce a line for 2 points:

- click on the command "Line"
- type the initial point of that line
- type the final point of that line

In order to introduce a Line with a certain length and a certain angle.

- click on the command "Line length and angle"
- type the initial point of that line
- type the final point of the line (with Polar Coordinates)

Line properties

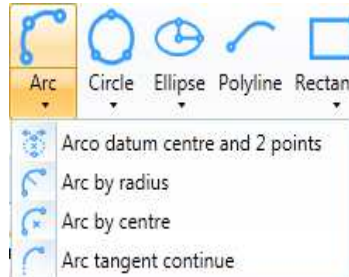


### 3.4.1.5 Arc

It inserts an arc.

An arc can be inserted in 5 ways:

- 1) Arc for 3 points
- 2) Centre data arc and 2 Points
- 3) Arc given radius
- 4) Arc given centre
- 5) Continuous tangent arc



In order to insert an **Arc for 3 points**:

- click on the command "Arc"
- type the initial point of the arc
- type the intermediate point of the arc
- type the final point of the arc

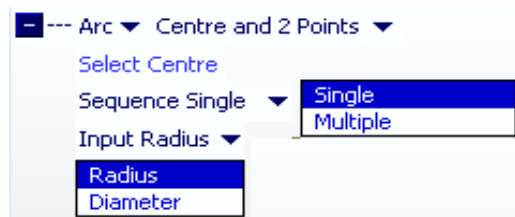
In order to insert an Arc whose centre radius initial and final angle are known:

- click on the command "Arc Centre and 2 Points"
- type the point of the arc centre
- insert the radius value and press Enter key
- insert the value of the initial angle and press Enter key  
or move the mouse (the radius value is upgraded) click on the left button of the mouse (also the initial angle is captured)
- Insert the angle value at the centre of the arc  
or move the mouse (the angle value in the centre is upgraded) click on the left button of the mouse

This command has two options at disposal:

- Sequence (Single/Multiple)
- Input (radius/Diameter)

that can be changed by the dynamic menu



The multiple sequence allows you to insert a further concentric arc to the previous one.

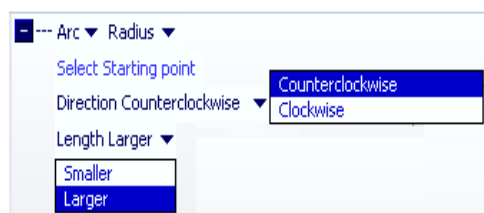
In order to insert an **Arc whose final point and radius are known**:

- click on the command "Arc given radius"
- type the initial point of the arc
- type the final point of the arc
- insert the radius value and press Enter key  
or move the mouse (the value of the radius is upgraded) click on the left button of the mouse

This command has two options at disposal:

- Direction (clockwise-anticlockwise)
- Arc length (Shorter/Longer)

that can be changed by the dynamic menu

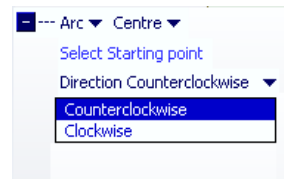




In order to insert an **Arc whose centre and final point are known:**

- click on the command "Arco dato centro" (arc given centre)
- type the initial point of the arc
- type the point of the arc centre
- type the final point of the arc

This command has following option at disposal:  
Direction (clockwise-anticlockwise)  
that can be changed by the dynamic menu

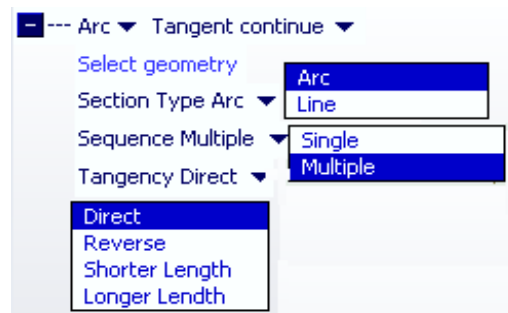


In order to insert a tangent **Arc in the previous element in a continuous manner:**

- click on the command "Continuous tangent arc"
- select a geometric element
- type the final point of the arc
- click on the left button of the mouse

This command has three options:

- Type of feature (Arc/Line)
- Sequence (Single/Multiple)
- Tangency (Direct
  - Inverse
  - Shorter length
  - Longer length)



that can be changed from the dynamic menu

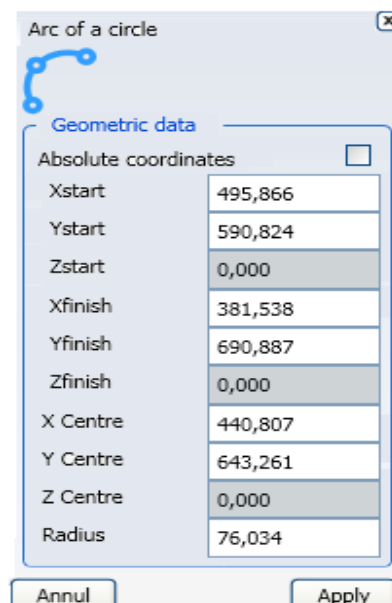
The multiple sequence allows you to make tangent elements with the previous one.

The direct tangency does not create a sharp edge

The inverse tangency creates a sharp edge

The length options create the workpiece of Shorter or Longer arc.

Arc properties



### 3.4.1.6 Circle

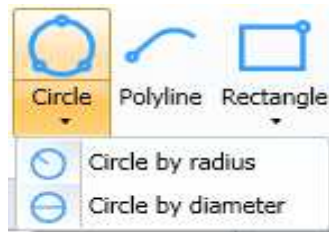
It introduces a circle:

A circle can be inserted in 3 ways:

1) Circle for 3 points

2) Circle given radius

3) Circle given diameter



In order to insert a **Circle for 3 points**:

- click on the command "Circle"
- type the initial point of the circle
- type the intermediate point of the circle
- type the final point of the circle

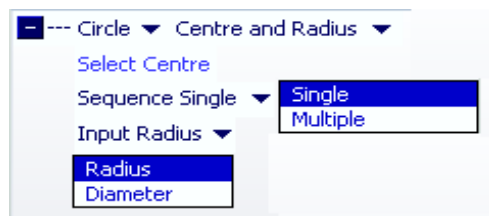
In order to insert a **Circle whose centre and radius are known**:

- click on the command "Circle given radius"
- type the point at the centre of the circle
- insert the radius value and press Enter key  
or move the mouse (the value of the radius is upgraded) click on the left button of the mouse

This command has two options at disposal:

- Sequence (Single/Multiple)
- Input (radius/Diameter)

that can be changed by the dynamic menu



The multiple sequence allows to insert a further concentric circle to the previous one.

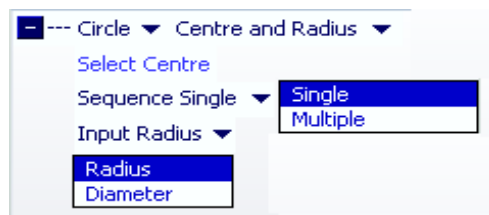
In order to insert a **Circle whose centre and diameter are known**:

- click on the command "Circle given diameter"
- type the point at the centre of the circle
- insert the value of the diameter and press Enter key  
or move the mouse (the value of the diameter is upgraded) click on the left button of the mouse

This command has two options at disposal:

- Sequence (Single/Multiple)
- Input (radius/Diameter)

that can be changed by the dynamic menu



The multiple sequence allows to insert a further concentric circle to the previous one.

It is the same command used for the circle given radius, in which the default "Input" has been turned into "Diameter"

Circle properties



### 3.4.1.7 Polyline

It introduces a polyline that is a continuous sequence of arcs and/or open or closed segments.

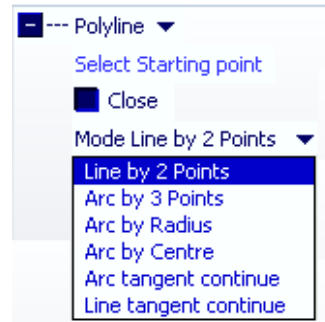
It can be drawn using the commands for the input of lines and arcs.

In order to insert a polyline:

- click on the command "Polyline"
- follow the instructions related to the command set in the option "Mode"

This command has two options at disposal:

- Closed (option that can be ticked off)
- Mode (Line given 2 Points  
Arc given 3 Points  
Arc given radius  
Arc given Centre  
Continuous tangent arc  
Continuous tangent line)



that can be changed from the dynamic menu

End: it is an option that allows you to make an open or closed polyline

Modus describes the command to use in order to describe the next geometric element. The six commands in the menu modus have been described in the previous paragraph about lines and arcs.

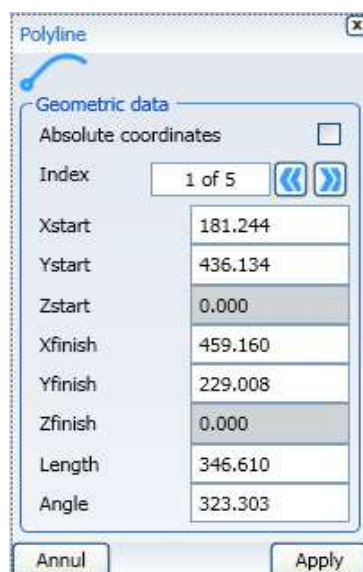
Polyline properties are made of several tables that stand for the properties of all elements in the structure (Lines or Arcs).

You can surf the tables by clicking on the left/right arrows.

If the field "Length" appears, the element described is a Line.

If the field "Radius" appears, the element described is an Arc.

Polyline properties



### 3.4.1.8 Rectangle

It introduces a rectangle that is performed with a closed polyline made of 4 lines.

The rectangle can be inserted in two ways:

- 1) By inserting 2 vertex:
- 2) Typing the dimensions



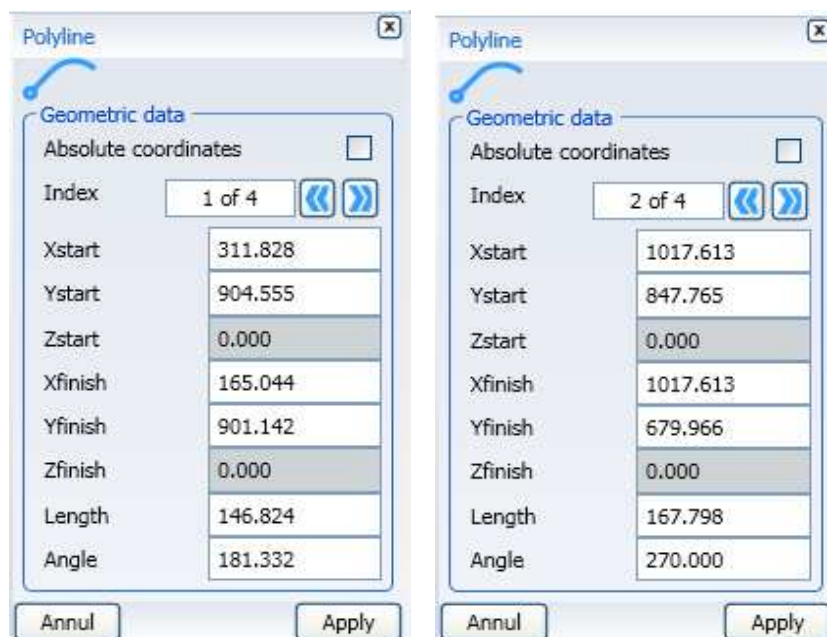
In order to insert a rectangle given 2 vertex:

- click on the command "Rectangle 2 points"
- type the point of the first vertex
- type the point of the second vertex

In order to insert a rectangle by typing the dimensions

- click on the command "Rectangle dimensions"
- type the central point of the rectangle
- Insert the Width value and press Enter key  
or move the mouse (the Width value is upgraded) click on the left button of the mouse.
- Insert the length value and press Enter key  
or move the mouse (the Length value is upgraded) click on the left button of the mouse.
- Insert the Angle value and press Enter key  
or move the mouse (the Angle value is upgraded) click on the left button of the mouse

Rectangle properties



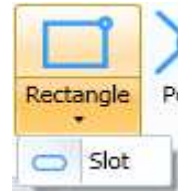
### 3.4.1.9 Buttonhole

It inserts a buttonhole performed with a closed polyline of 4 elements two lines and two arcs.

Arcs have a diameter like their width and a breadth of 180°.

In order to insert a buttonhole:

- by clicking on "Buttonhole"
- type the central point of the buttonhole
- Please insert the Width value and press Enter key or move the mouse (the Width value is upgraded) click the left button of the mouse.
- Please insert the length value and press Enter key or move the mouse (the Length value is upgraded) click on the left button of the mouse.
- Please insert the Angle value and press Enter key or move the mouse (the Angle value is upgraded) click on the left button of the mouse



Buttonhole line properties

Polyline	
Geometric data	
Absolute coordinates <input type="checkbox"/>	
Index	1 of 4
Xstart	280.000
Ystart	1013.853
Zstart	0.000
Xfinish	611.560
Yfinish	1013.853
Zfinish	0.000
Length	331.560
Angle	0.000
Annul	
Apply	

Polyline	
Geometric data	
Absolute coordinates <input type="checkbox"/>	
Index	2 of 4
Xstart	376.106
Ystart	1134.799
Zstart	0.000
Xfinish	379.644
Yfinish	841.173
Zfinish	0.000
X Centre	377.875
Y Centre	987.986
Z Centre	0.000
Radius	146.823
Annul	
Apply	

### 3.4.1.10 Loop line



It inserts an arc of circle or a tangent circle to 2 existing entity.

In order to insert a loop line:

- click on the command "Loop line"
- type the radius value
- move with the mouse (a loop line next to the cursor)
- click on the left button of the mouse

Or:

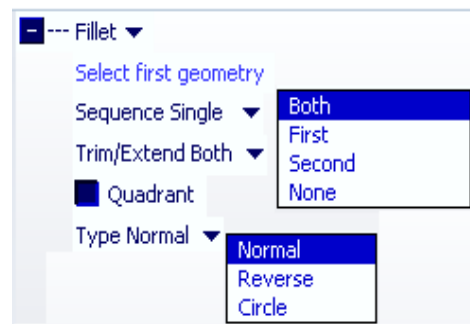
- click on the command "Loop line"
- type the radius value
- select the first geometry
- move with the mouse (a loop line next to the cursor)
- click on the left button of the mouse

Or:

- click on the command "Loop line"
- type the radius value
- click on the option "Quadrante" (quadrant)
- select the first geometry
- select the second geometry
- move with the mouse (a loop line next to the cursor)
- click on the left button of the mouse

This command has three options:

- Cut/Widen (both  
    First  
    Second  
    Nothing)
- Quadrant
- Type (Usual  
    Inverted  
    Circle)



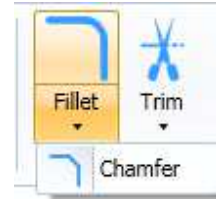
Cut/Widen serves as option to widen or cut the selected elements

Quadrant serves to insert the loop line in the third mode (2 geometries and the quadrant)

Type establishes the type of desired loop line.

The properties of the loop line are the same of the Arc.

### 3.4.1.11 Smooth



It inserts a segment that smooths 2 existing entities.  
The smooth can be set in 4 modes (see option "Type")

In order to insert a Smooth:

- click on the command "Smooth"
- type the values to set the smooth
- move with the mouse (a smooth will appear next to the cursor)
- click on the left button of the mouse

Or:

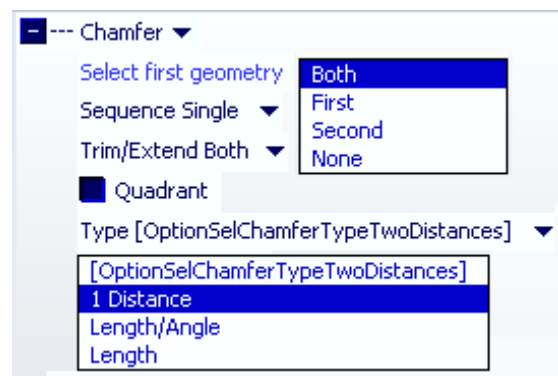
- click on the command "Smooth"
- type the values to set the smooth
- select the first geometry
- move with the mouse (a smooth will appear next to the cursor)
- click on the left button of the mouse

Or:

- click on the command "Smooth"
- type the values to set the smooth
- click on the option "Quadrant" (quadrant)
- select the first geometry
- select the second geometry
- move with the mouse (a smooth will appear next to the cursor)
- click on the left button of the mouse

This command has three options:

- Cut/Widen (both  
    First  
    Second  
    Nothing)
- Quadrant
- Type (1 Distance  
    2 Distances  
    Length/Angle  
    Length)



The Cut/Widen option serves as option to widen or cut the selected elements

The Quadrant option aims at introducing the smooth in the third mode (2 geometries and the quadrant)

The Type option establishes how to insert the data about the loop line

The values inserted in 1 Distance or 2 Distances are measured along the selected geometries.

The length represents the length of the new smooth segment inserted.

The smooth properties are the same of a Line.



### 3.4.1.12 Offset



It creates one or more geometries that have a steady value compared to an existing selected geometry.

The new entities will look like the selected ones.

In case of a polyline, the new entities will always be considered as polylines but the number of elements in their structure can be higher or lower than the elements of the selected polyline.

In order to create an Offset geometry:

- click on the command "Offset"
- select one or more geometric elements
- insert your offset value
- set the number of repetitions
- select the reference point to establish where to create the offset

This command has two options at disposal:

- Repetitions
- Preview



The option "Repetitions" allows you to create one or more offset geometries

The option Preview - if it is marked - allows you to see - in Preview- the geometries that will be created by moving the mouse.

It is possible to set off the initial and final tracts of an open break, when they are inferior to a certain length because of the offset.

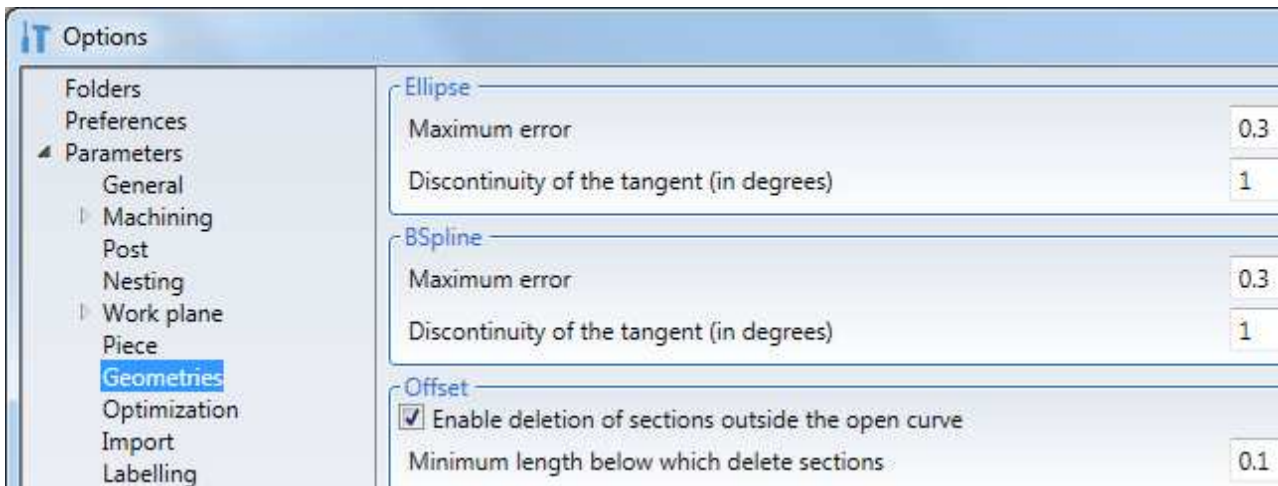
To do this you put the flag on the parameter

"Enable elimination of the outer tracts of the open curve" visible in the path:

Tools-Options-Parameters-geometries.

By ticking the parameter, the second parameter will also appear:

"Minimum length below which to delete traits"



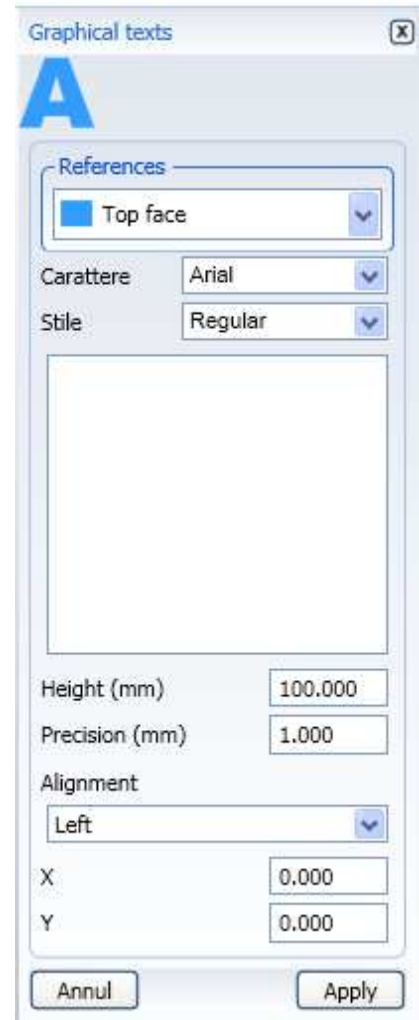
### 3.4.1.13 Text

It inserts a text made of many polylines (one or more for every character)

The text can be inserted in several fonts and heights.

In order to insert a text:

- click on the command "Text"
- select the reference plane
- selected the desired character
- select the desired style
- insert the text
- type the height
- type a precision
- select the desired alignment
- type the position in X and Y of the text
- Click on the button "Apply"



## 3.4.2 Inserting blocking profile

The blocking profile consists of a geometry that indicates the part of the piece to be blocked by the clamps.

To insert a blocking profile, you must first create a geometry with the instructions seen in the chapter "**Errore. L'origine riferimento non è stata trovata.**" and then use the "Blocking profile" command in the "Operations" menu.

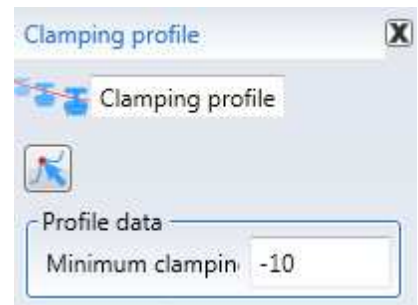
The following rules must be respected:

- the clamps are always to be positioned to the right of the geometry created.
- create the blocking profile in the correct work phase.



If, for example, you have to work an element to be blocked with the clamps to be profiled on both sides, you must make a project with two work phases (Setup and Swap), with the blocking profile to be inserted in each phase.

After selecting the command, this dialogue window appears:



where you can select the geometry and set a minimum clamping.

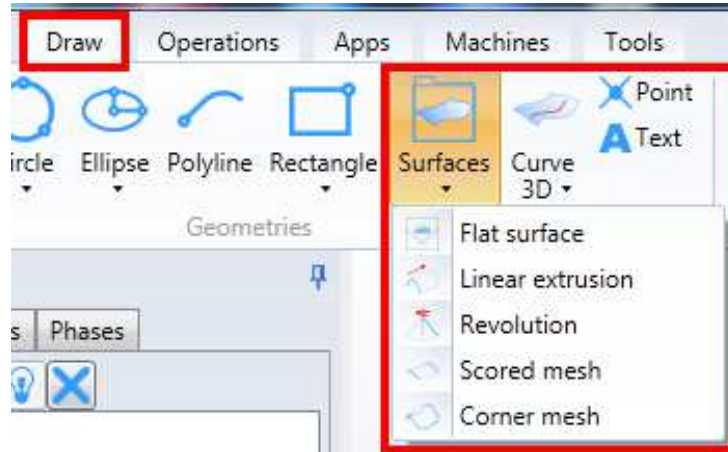
Minimum clamping means the minimum quantity of wood that can be clamped during the clamping phases with the clamps.

In the "Minimum clamping" field, you can enter a negative value or the name of a previously-set parameter.

If, in generating the geometry, considerable points of the workpiece were used, the profile is completely parametric.

### 3.4.3 Enter new surfaces

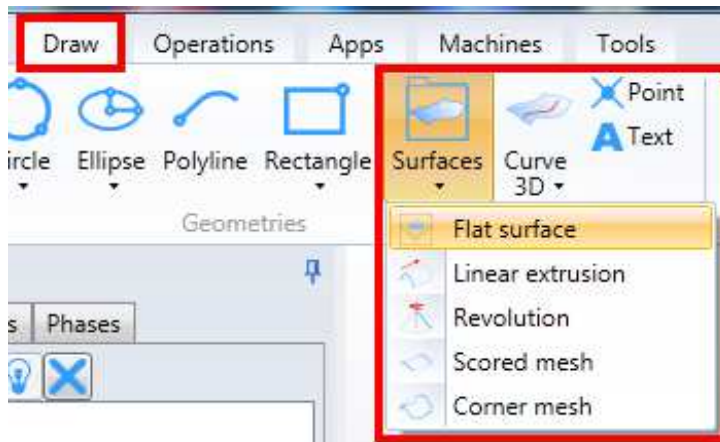
Maestro provides functions to create new surfaces. To access the commands enter the “Draw” area of the application menu and press “Surfaces”.



The methods available and their main features are described below.

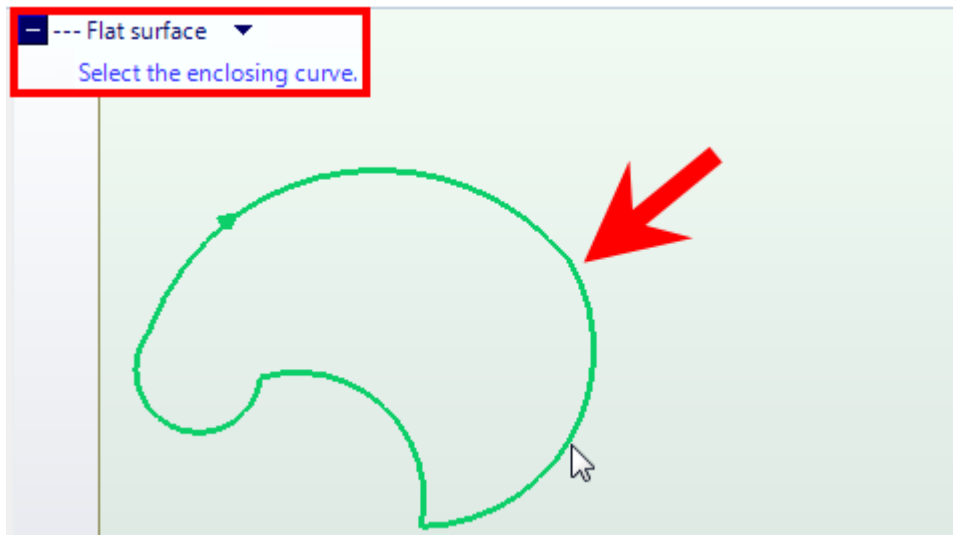
### 3.4.3.1 Flat surface

To create a flat surface click on “Flat surface”.



Then proceed as follows:

- select a closed 2D curve;

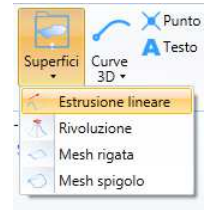
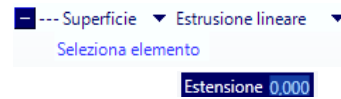


### 3.4.3.2 Linear extrusion

To create an extrusion surface click on “Linear Extrusion”.

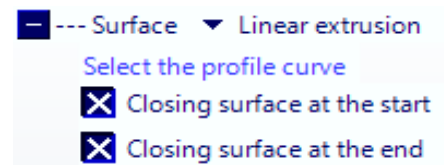
Then proceed as follows:

- select a geometry on a plane;
- enter the geometry extrusion length.



The geometry to select can be on any project plane, on the workpiece faces and created by the user (e.g. tilting plane). The length must be entered after having selected the geometry. The extrusion is performed along the axis z direction with the Cartesian reference system local to the plane. The sign determines the extrusion direction along the direction.

The created surface can be closed by two other surfaces by checking the corresponding flags, provided the section is closed

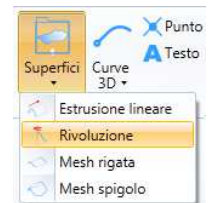


### 3.4.3.3 Revolution

To create an extrusion surface click on “Revolution”.

Then proceed as follows:

- select a geometry on a plane;
- select a revolution axis;
- enter a revolution angle.



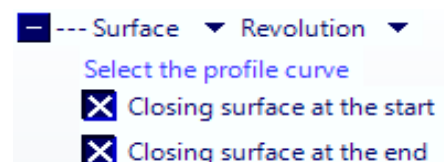
The geometry selected can be on any project plane.

The revolution axis must be a linear geometry not at right angles to the plane on which the revolution geometry is drawn.

The revolution angle value determines the rotation quantity around the axis, whilst the sign determines the clockwise/counterclockwise direction, usually clockwise.

The created surface can be closed by two other surfaces by checking the corresponding flag, provided that:

- the section is closed
- the angle is less than 360 °

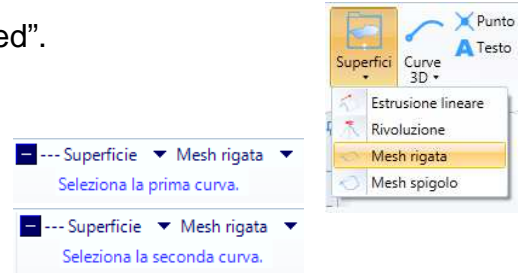


### 3.4.3.4 Mesh lined

To create a surface with mesh lined click on “Mesh lined”.

Then proceed as follows:

- select a first geometry on a plane;
- select a second geometry on a plane.



The geometries selected cannot be closed geometries.

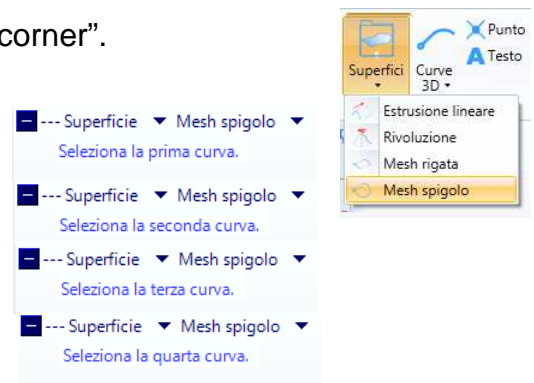
The surface is created by connecting the points of the first geometry with the points of the second, with lines. Usually the start point of the first geometry is connected to the start point of the second, the same applies to the end points. As this approach can create surfaces that collapse in the middle, if the directions of the curves used are in conflict, the algorithm inverts the start point and end point of one of the two geometries.

### 3.4.3.5 Mesh corner

To create a surface with mesh corner click on “Mesh corner”.

Then proceed as follows:

- Select the first corner geometry;
- Select the second corner geometry;
- Select the third corner geometry;
- Select the fourth corner geometry.



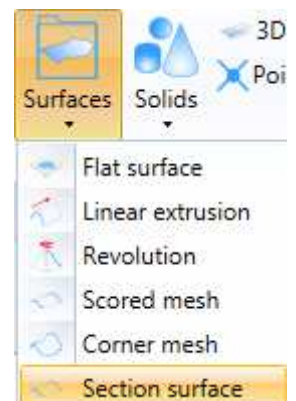
The corner geometries must have matching points, so that the total contour, formed by the four elements, is closed.

### 3.4.3.6 Section surface

To create a surface that joins multiple curves that will form the sections of the super surface, click on "Surface from Sections".

The function has the following requests:

- Surface ▾ Surface from sections  
Select first curve.
- Surface ▾ Surface from sections  
Select second curve.
- Surface ▾ Surface from sections  
Select third curve.
- Surface ▾ Surface from sections ▾  
Select the next curve or press ESC to end.



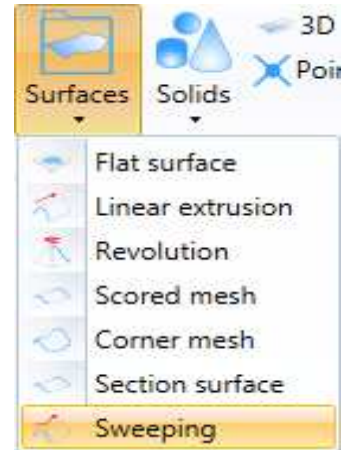
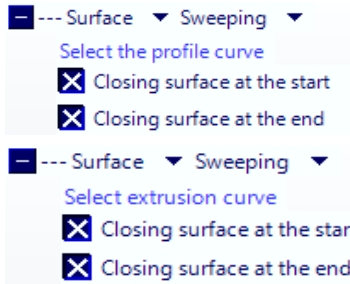
The selection of three curves is mandatory while the selection of additional curves is optional.

### 3.4.3.7 Sweeping

This function helps create a set of surfaces that form a tube with a constant section following a trajectory.

The input will consist of two curves.

The function has the following requests:

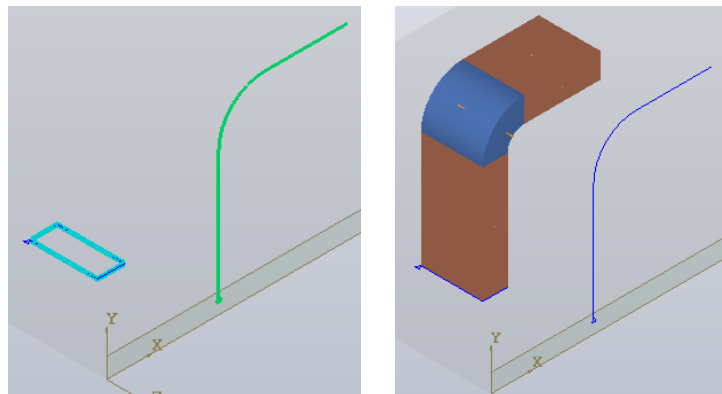


The extrusion curve must have no edges and it is advisable that the first section forms an angle of  $90^\circ$  with respect to the plane in which the profile curve has been created.

Generation occurs by making the initial points of the two curves coincide.

It is possible to plug the tube by ticking the related items "Closing surface"

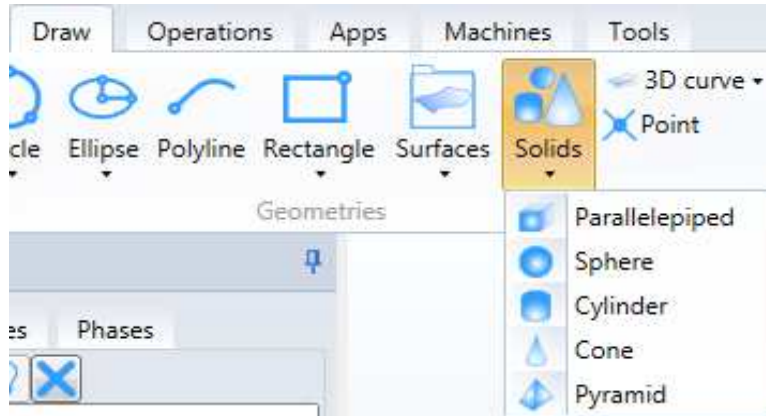
Esempio :





### 3.4.4 Insertion of solids

Solids are inserted using the "Draw - Solids" menu.



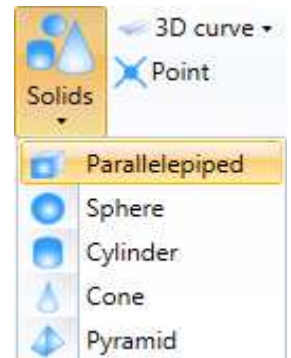
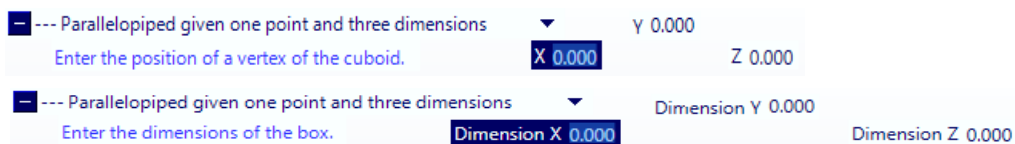
The solids created are a set of multiple surfaces.

The methods available and their main features are described below.

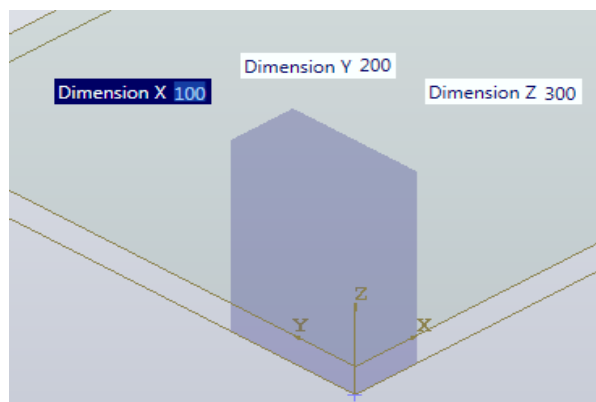
#### 3.4.4.1 Parallelepiped

This function allows you to create a parallelepiped consisting of 6 surfaces.

The function has the following requests:



Example:



### 3.4.4.2 Sphere

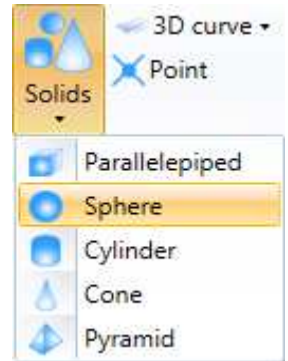
This function allows you to create a sphere or a portion of a sphere made up of one or more surfaces.

The function has the following requests:

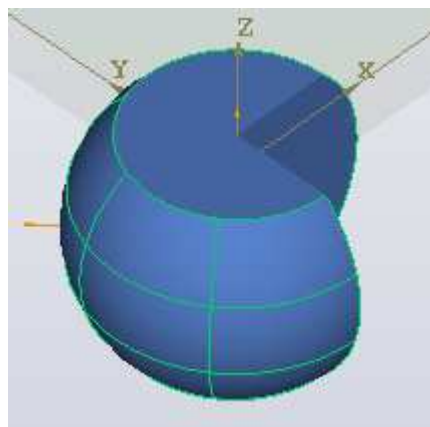
--- Spherical data center and radius Y 0.000  
 Enter the position of the centre of the sphere. X 0.000 Z 0.000

--- Spherical data center and radius End angle 360.000  
 Insert the angle at the centre on the XY plane. Starting angle 0.000

--- Spherical data center and radius End angle 45  
 Insert the angle at the centre on the XZ plane. Starting angle -45



Example:



### 3.4.4.3 Cylinder

This function enables users to create a cylinder or a cylinder portion formed by one or more surfaces.

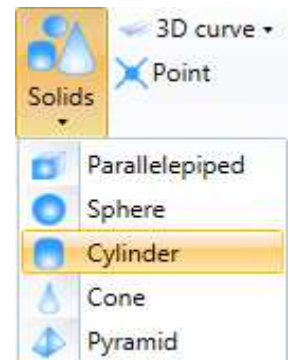
The function has the following requests:

--- Cylindrical data centre, radius and height Y 0.000  
 Enter the position of the centre of the base. X 0.000 Z 0.000

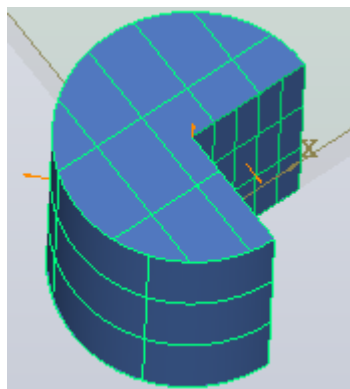
--- Cylindrical data centre, radius and height  
 Enter the radius of the base Radius 100

--- Cylindrical data centre, radius and height  
 Enter the height. Thickness 0.000

--- Cylindrical data centre, radius and height End angle 270  
 Insert the angle at the centre on the XY plane. Starting angle 0.000



Example:

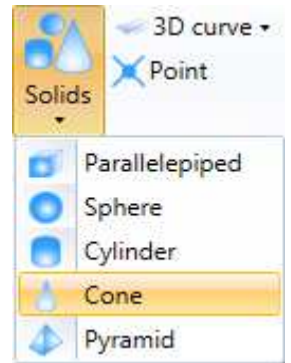


### 3.4.4.4 Cone

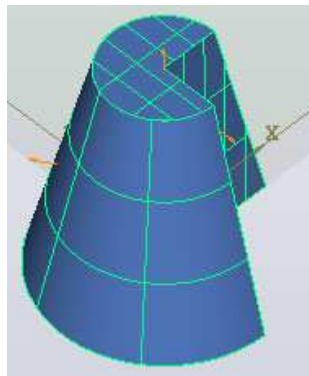
This function enables you to create a cone or cone portion formed by one or more surfaces.

The function has the following requests:

- Conical data centre, radius and height Y 0.000  
Enter the position of the centre of the base.
- Conical data centre, radius and height  
Enter the radius of the base
- Conical data centre, radius and height  
Enter the height.
- Conical data centre, radius and height  
Insert the angle at the centre on the XY plane. Starting angle 0.000



Example:

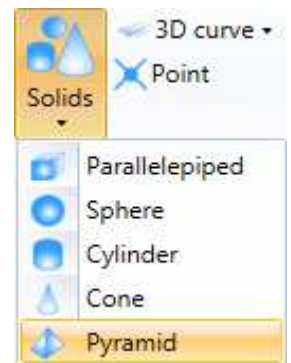


### 3.4.4.5 Pyramid

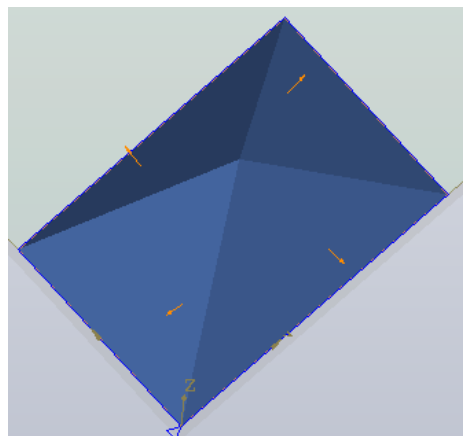
This feature means you can create a pyramid having a closed, unjoined geometry as a base. It will be made up of one or more surfaces. The vertex of the pyramid will be the projection in height at the centre of gravity of the base.

The function has the following requests:

- Pyramidal data on base and height  
Select the geometry of the base
- Pyramidal data on base and height  
Enter the height.

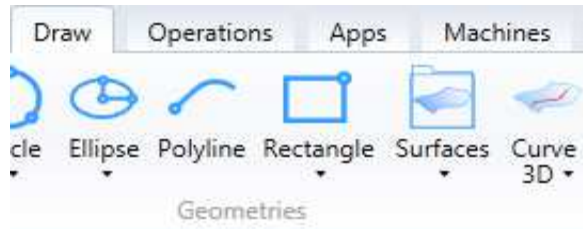


Example:



### 3.4.5 Enter new 3D curves

Maestro provides functions to create 3D curves. To access the commands enter the “Draw” area of the application menu and press “3D Curves”.



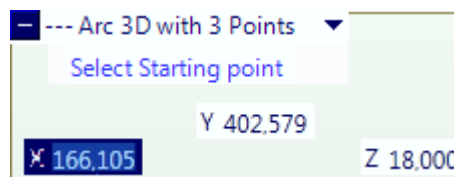
The methods available and their main features are described below.

#### 3.4.5.1 3D arc

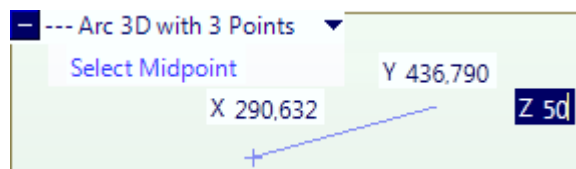
To create an arc in the space click on “3D arc”.

Then proceed as follows:

- Select the start point



- Select the intermediate point



- Select the end point



This will result in an arc that lies on the plane through the 3 points described.



Note that:

- The points can be entered by combining the mouse and keyboard.
- The arc generated is shown in red to distinguish it from an arc that has been defined on a plane

### 3.4.5.2 3D composite line

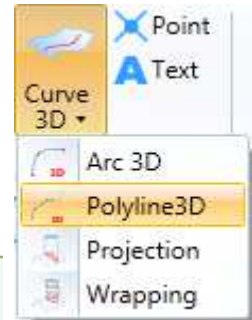
To create a break in the space click on “3D composite line”.

Then proceed as follows:

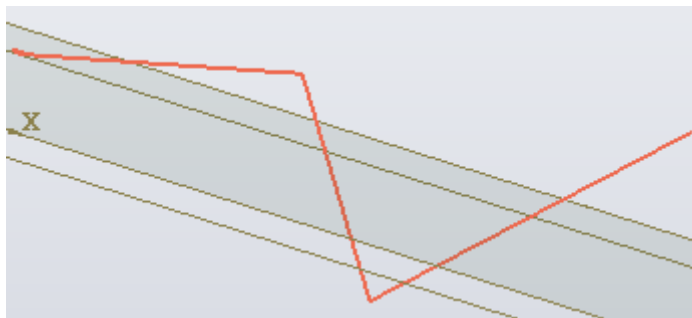
- Select the start point



- Select a series of subsequent points



To complete the composite line press “Esc”.



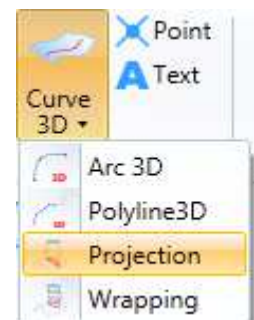
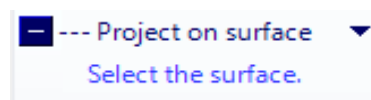
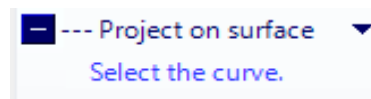
Note that the line is not part of the active plane.

### 3.4.5.3 Projection

To create a 3D curve by projecting a geometry on a surface, click on "Projection".

Then proceed as follows:

- select the geometry to project;
- select the target surface.

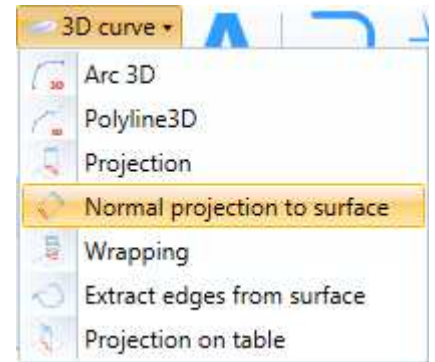


### 3.4.5.4 Normal surface projection

To create a 3D curve by projecting a curve normally on a surface, click on "Normal surface projection".

The function has the following requests:

- Project normal on surface  
Select the curve.  
 Keep original
- Project normal on surface  
Select the surface.  
 Keep original



The selected curve will normally be projected onto the surface by generating a 3D curve

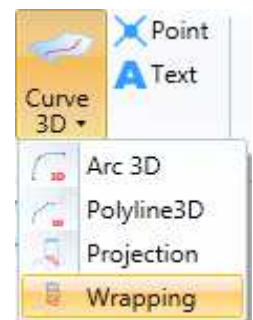
### 3.4.5.5 Wrapping

To create a 3D curve by wrapping a geometry on a surface, click on "Wrapping".

Then proceed as follows:

- select the wrapping geometry;
- select the target surface.

- Wrap curve on surface ▼  
Select the curve.
- Wrap curve on surface ▼  
Select the surface.

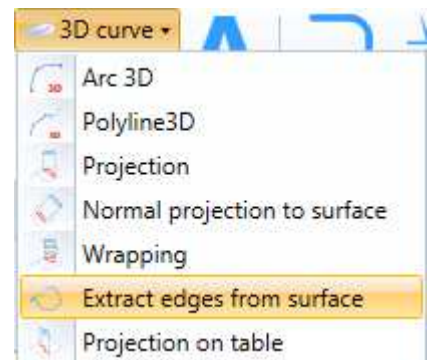


### 3.4.5.6 Extract surface edges

To get the 3D curves that make up the surface perimeter, click on "Extract Surface Edges".

The function does the following:

- Extract edges from surface  
Select the surface.

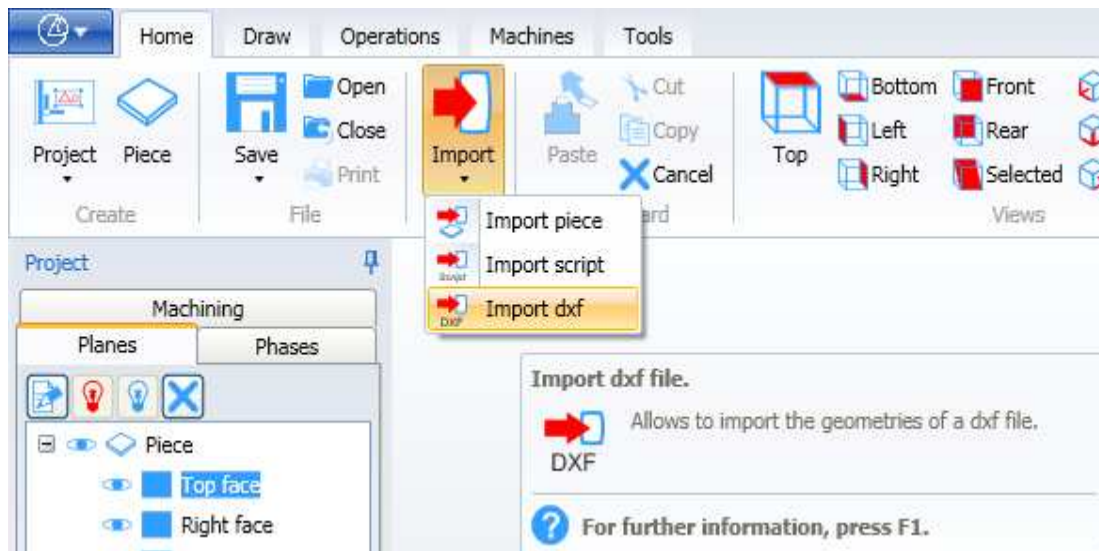


After selecting the surface, the 3D curves that form the surface perimeter are generated.

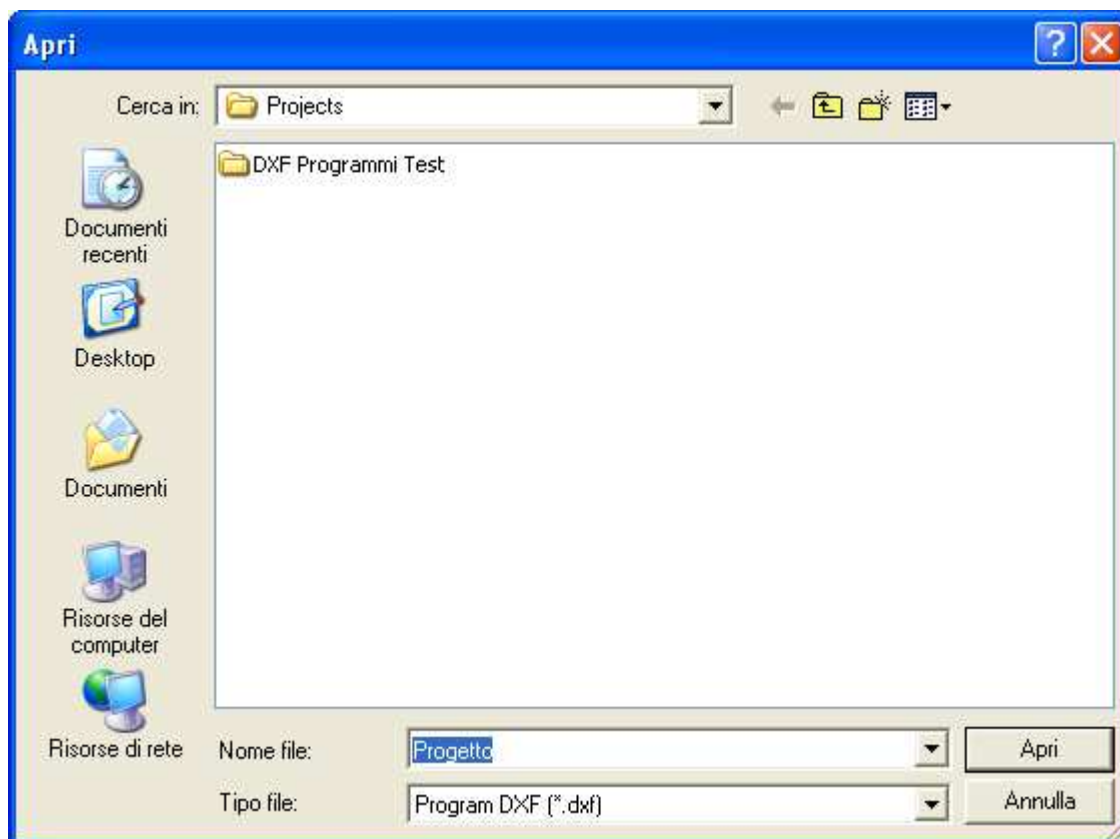
### 3.4.6 2D geometry imported from DXF

It is possible to import 2D geometry from an existing DXF file.  
To do this, there must be a project with at least one workpiece defined.

To import a DXF file use the “Import dxf” command in the “Home” “Import” menu.



After selecting the command, the dialogue box appears which allows you to select a “.dxf” file from the PC disk.  
The “DXF program (\*.dxf)” filter is already set in the “File type:” box.



Once you have selected the file, the menu shown here appears:

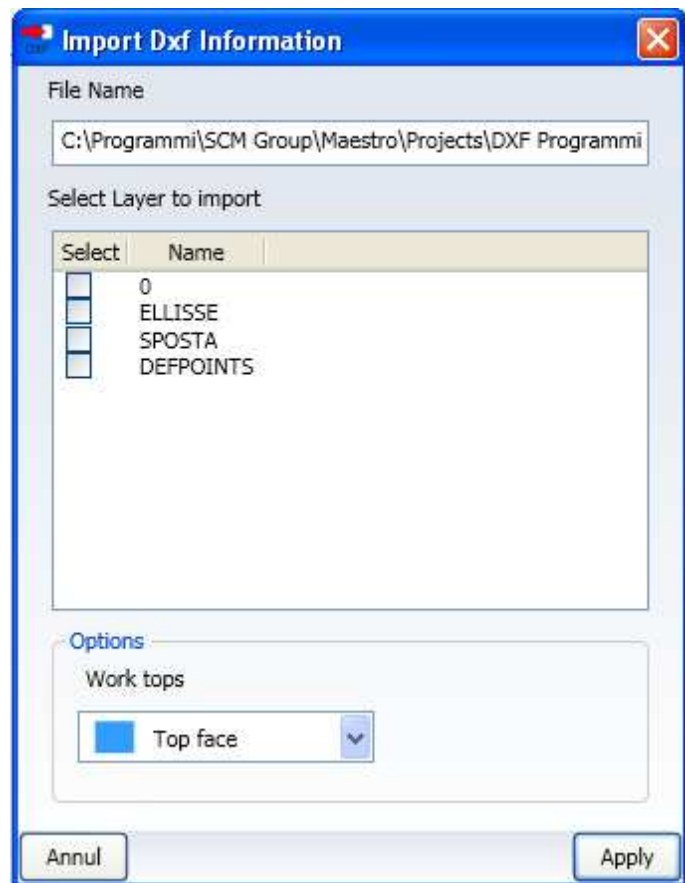
In the menu you can see the name of the dxf file which is being imported in the "File Name" field.

You can select:

- the Layers present in the dxf file
- the face on which you want to import the geometric entities present in the selected Layers

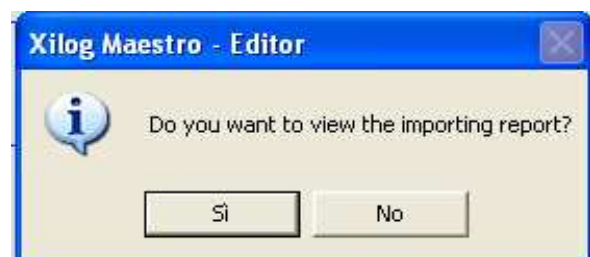
Use the "Cancel" key to interrupt the operation.

Use the "Apply" key to begin importing the dxf file.

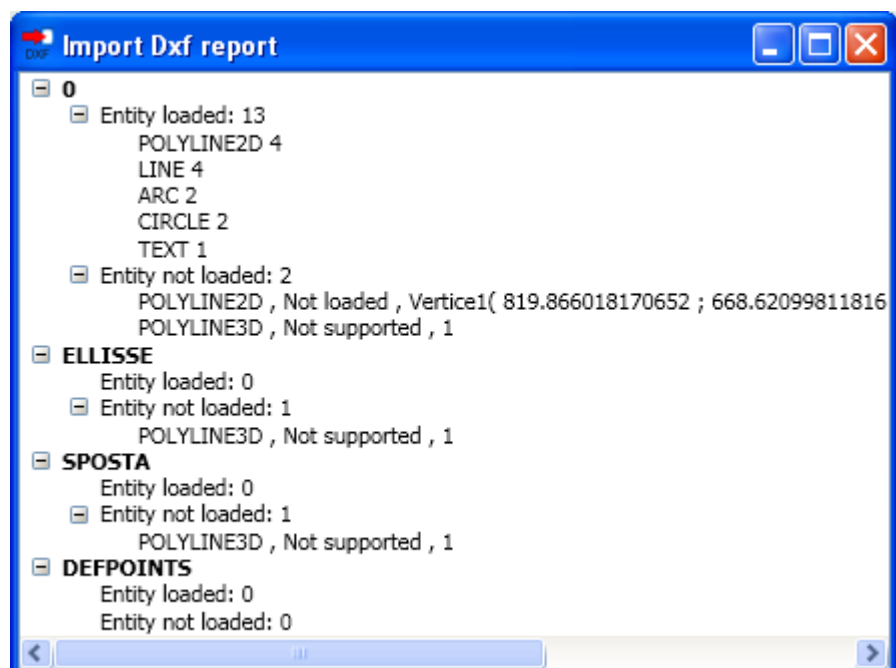


At the end of the import operation this window appears: (Do you want to see the import report?)

Select "No" to conclude the operation. Select "Yes" and another window opens, showing the dxf import operation report.



In the report, for each layer you can see how many and which entities were imported and how many and which entities were not imported and the reason why.





The report file is also saved in the “Maestro/Report/DXF” folder in XML format.

The name with which it is saved has the prefix “DxfImpor” followed by the date and time.  
Example: “DxfImport\_20110920\_085538.xml”.

The following Dxf versions are supported:  
12, 13, 14, 2000, 2004, 2007, 2010, 2013

With the restriction that in Maestro only the 2D geometries will be imported.

The geometries that can be imported are:

- DxfLine
- DxfCircle
- DxfArc
- DxfPolyline2D
- DxfLwPolyline
- DxfPoint
- DxfText
- DxfEllipse
- Spline 2D

Note: The following geometries cannot be imported:

- 1) SPLINE3D
- 2) POLYLINE3D

In Autocad 2004 ellipses can be exported as polylines using the “PELLIPSE” variable. This would allow it to be imported into the packet.

### 3.4.6.1 2D geometries advanced import from DXF

In order to associate machining operations to the geometries the CAD designer must structure the project and the drawing of the geometries in a certain way.

First of all the CAD designer must know the dimensions of the workpiece being produced and know which machinings must be applied to the geometries to draw. This information is needed to structure the project correctly.

The CAD project must be structured in layers, as follows:

1. Possess a layer dedicated to the definition of the piece;
2. Possess a layer to describe the surface, for each surface on which you wish to draw the geometries.
3. Possess a layer to describe processing, for each processing of geometries on a surface.

The following rules apply especially to create layers:

1. All layers that are to be interpreted and managed by "advanced import" must have their names beginning with the key: XLY.
2. The XLY key is followed by a set of key values. If the values are alphanumeric, they must start and end with the separator character \$.
3. The keys allowed in the layer name are:
  - **P**: work plane (E.g.: XLYP1 is the layer referred to the upper plane). The values allowed are: 1, 2, 3, 4, 5, 6 for standard planes (1=upper plane, 2=plane, 3=l plane, 4= front plane , 5= rear plane, 6=lower plane), and values from 7 upwards for the user planes.
  - **T**: technology and processing (E.G.: XLYP1T\$FORA1\$ geometries on the upper surface and technology application FORA1 to the geometries). The value of these keys is alphanumeric and represents the name of the technology (technology file name without an extension). The values must correspond to the technologies that will be used in Maestro.
  - **H**: piece height. Valid key only for the layer that defines the piece: it is only allowed if the surface is P1 and there are no associated technologies (E.G.: XLYP1H18). The value of this key cannot be alphanumeric.
  - **OX**: coordinate X of the plane origin. Key valid only for user planes and not for standard planes: all planes of greater index equal to 7 are allowed and no technologies must be associated (E.g.: XLYP7OX100). The value of this key cannot be alphanumeric.
  - **OY**: coordinate Y of the plane origin. Key valid only for user planes and not for standard planes: all planes of greater index equal to 7 are allowed and no technologies must be associated (E.g.: XLYP7OY100). The value of this key cannot be alphanumeric.
  - **OZ**: coordinate Z of the plane origin. Key valid only for user planes and not for standard planes: all planes of greater index equal to 7 are allowed and no technologies must be associated (E.g.: XLYP7OZ100). The value of this key cannot be alphanumeric.
  - **RX**: rotation angle expressed in degrees, of the reference system integral to the plane around its X axis. Key valid only for user planes and not for standard planes: all planes of greater index equal to 7 are allowed and no

technologies must be associated (E.g.: XLYP7RX45). The value of this key cannot be alphanumeric.

- **RZ**: rotation angle expressed in degrees, of the plane reference system around the Z axis of the absolute reference system. Key valid only for user planes and not for standard planes: all planes of greater index equal to 7 are allowed and no technologies must be associated (E.g.: XLYP7RZ45). The value of this key cannot be alphanumeric.
4. The layer name must not contain spaces, unless within the separator characters.
  5. The piece definition (dimensions DX, DY and DZ) is carried out by means of a layer that contains a rectangle whose dimensions define DX and DY. The thickness of the piece (DZ) is defined in the layer name by means of the H key followed by the corresponding value. Only one layer must contain this information.
  6. For correct positioning of the geometrical entity, you must have a reference for each surface used. Therefore, each surface containing one or more entities must have a layer name like  
XLYP<layer name>  
containing a rectangle having dimensions equivalent to those of the associated surface. The entity contained on it will then be positioned in the DXF project in relation to the said rectangle. If one or more layers refer to a surface, which has not been supplied with a layer description, processing contained therein will be rejected with an error message for the user. The layer associated with the upper surface must always exist since it is conventionally committed to describe the dimensions of the piece, therefore, it is the only layer that contains the "H" key. The geometry contained in this layer will describe the upper surface of the piece and, if necessary, the lower surface (should this not be described).
  7. The definition of the layer for a user plane follows different rules to the standard planes. The layer is defined by drawing the rectangle that describes the plane in any position of the DXF project. The geometries inside the rectangle must be drawn with local reference at the rectangle origin, that is in relation to the origin of the plane being created.
  8. The sequence used to define the technological layers (ES XLYP1T\$FORA1\$) defines the sequence with which the machining operations must be created inside the project. For example, in order to create 3 layers, one routing, one trimming and one drilling, to obtain the following type of machining:

- 1- Trimming
- 2- Drilling
- 3- Routing

the layers must be defined in the following order

XLYP1T\$Contornatura\$  
XLYP1T\$Foratura\$  
XLYP1T\$Fresatura\$

## VALID EXAMPLES:

XLYP1H18  
XLYP3  
XLYP1T\$FORA1\$  
XLYP3T\$FRESA LAMA\$  
XLYP2T\$FORA LAT5\$

## NON-VALID EXAMPLES

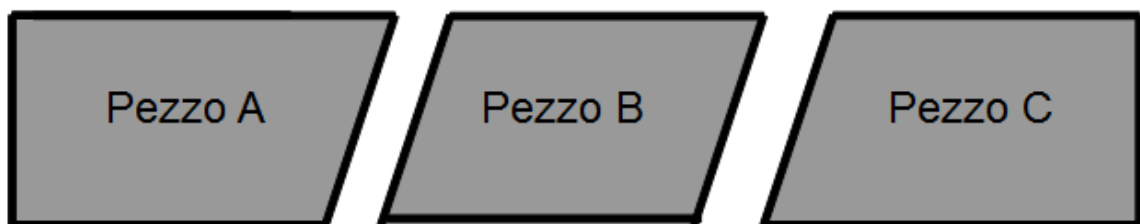
P1H18 (initial XLY key missing)  
XLYP1H18T\$FORA1\$ (H specified in combination with a technology)  
XLYP1 T\$FORA1\$ (presence of a space besides the separators \$)  
XLYP1TFORA1 (no separators \$ with alphanumeric type values FORA1)

Moreover, geometries must be made by keeping the characteristics of processing and the tool path in mind:

- If processing envisages the use of a tool compensation, for example the use of left correction, this restricts the CAD designer in drawing the geometries. For example, making closed geometries in a clockwise and anticlockwise direction; the first case of processing will be carried out externally from the geometry profile whilst the second will be inside it. Similarly, the designer will have to keep the direction in mind also for open geometries since processing an AB segment will be different from that of a segment covering the opposite direction (segment BA).

### 3.4.6.1.1 Import 2D geometries from DXF for machining

By way of an example, here is a project to make a plank of wood composed of three pieces. For each piece, it is necessary to define a .dxf file, which describes the associated geometries for each individual piece and its relative processing. Below is a representation of the plank and parts it is composed of.

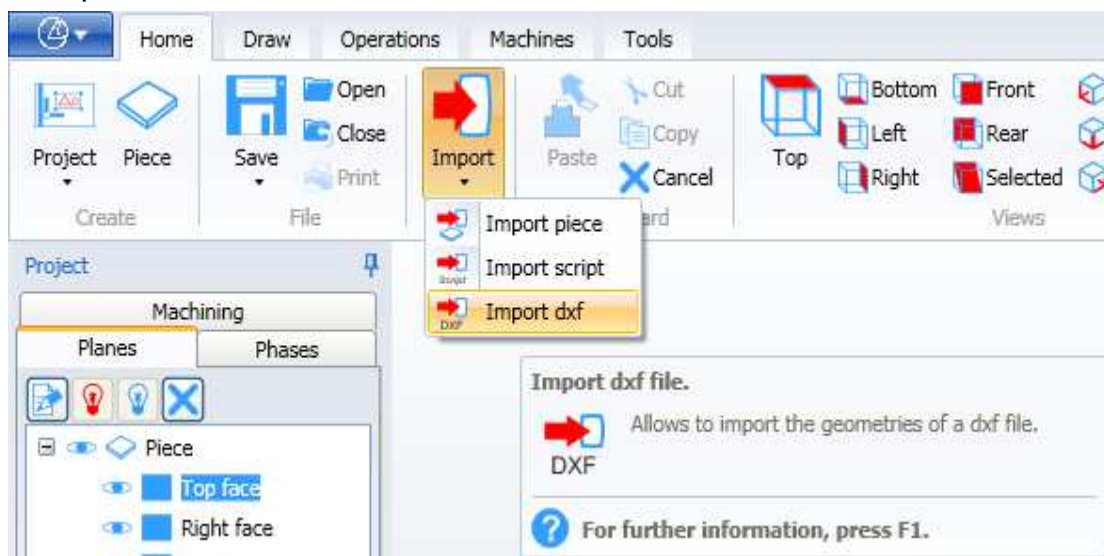


In the example, we assume that the geometries refer to the upper surface of all pieces taken into consideration and that each side of the geometry needs to be processed with a different technology than the others. In detail, for each piece, assuming we have to firstly execute contouring of all the geometries and then execute the required processing for each of their sides. The upper side of the geometry requires milling processing by means of a tool having a convex shape whilst the lower one requires a concave shape.

Instead, processing associated with the sides of the geometries enable the finish of the side, if this corresponds to an external side of the plank, or shaping it as a junction to the adjacent piece if this is the side of the junction.

For contouring operations, TECH0.tchx technology is assumed to be available, while milling with a convex tool is for TECH2.tchx technology and a concave tool is for TECH3.tchx technology. To make a junction, TECH1.tchx and TECH5.tchx technologies are assumed to be available, which correspond to milling the left and right side of the junction. The finish is instead obtained by means of TECH4.tchx technology. For all technologies taken into consideration, it is assumed that tool correction is on the left.

To import a DXF, you must use an "Importa dxf" control from the menu "Home" "Importa".



After having selected the control, a dialogue window will appear that enables you to select a ".dxf" file from the PC's disc.

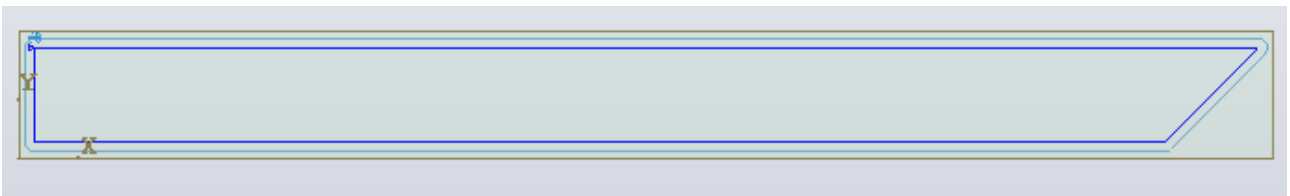
Below are the layers created for each piece with the relative geometries and associated processing.

**PIECE A:**

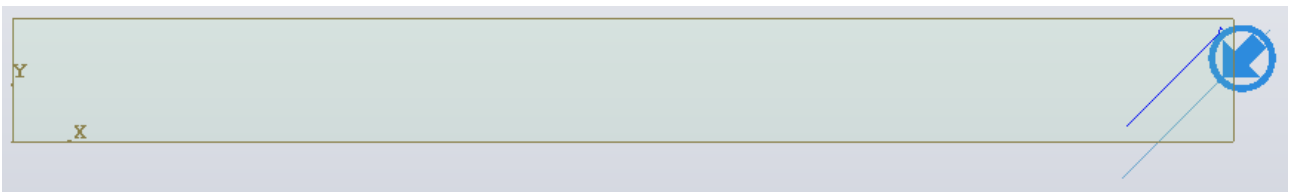
<i>Layer</i>	<i>Technology</i>	<i>Operation</i>
XLYP1H30	-	Piece definition and upper surface



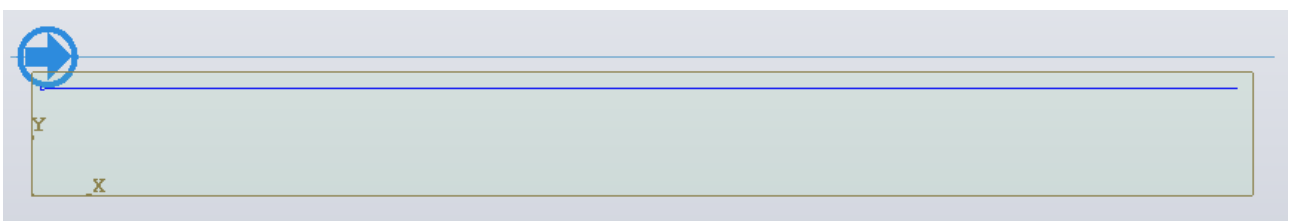
<i>Layer</i>	<i>Technology</i>	<i>Operation</i>
XLYP1T\$TECH0\$	TECH0.tchx	Contouring



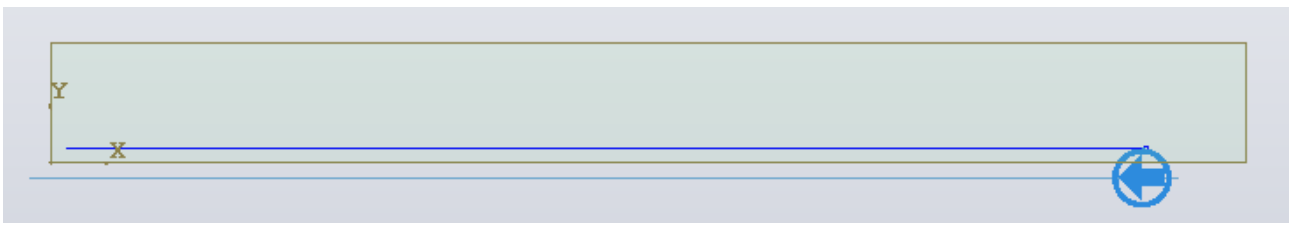
<i>Layer</i>	<i>Technology</i>	<i>Operation</i>
XLYP1T\$TECH1\$	TECH1.tchx	Left Junction



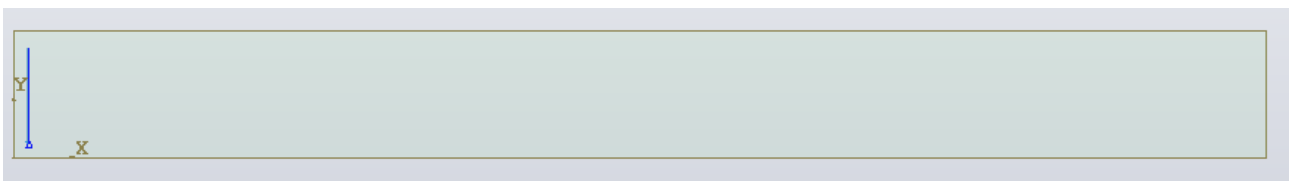
<i>Layer</i>	<i>Technology</i>	<i>Operation</i>
XLYP1T\$TECH2\$	TECH2.tchx	Convex milling tool



<b>Layer</b>	<b>Technology</b>	<b>Operation</b>
XLYP1T\$TECH3\$	TECH3.tchx	Concave milling tool

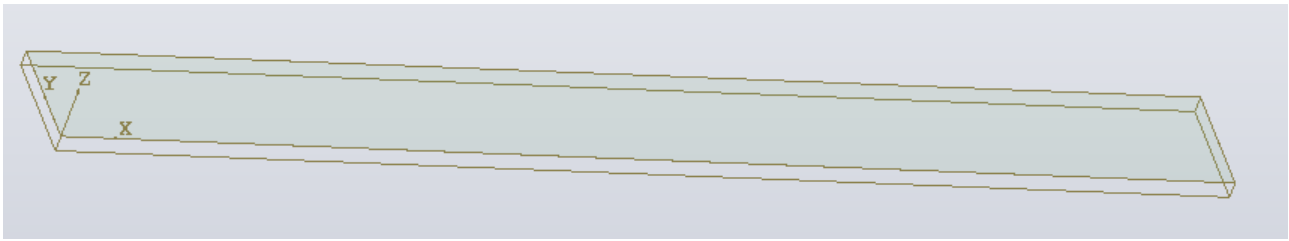


<b>Layer</b>	<b>Technology</b>	<b>Operation</b>
XLYP1T\$TECH4\$	TECH4.tchx	Finish

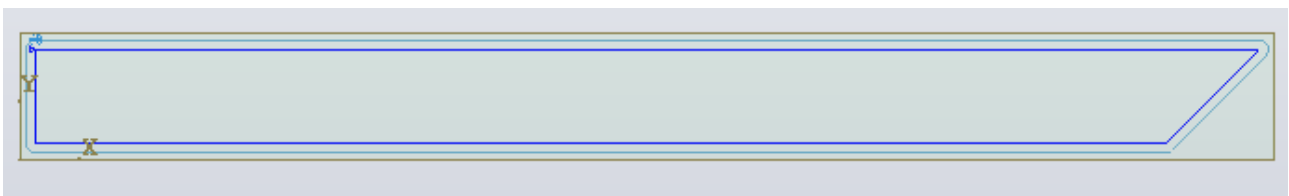


**PIECE B:**

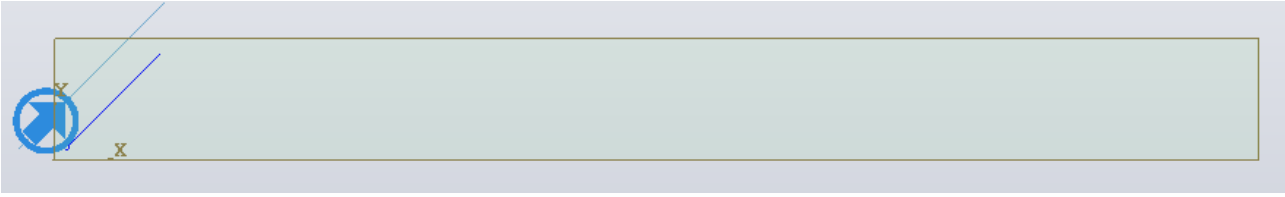
<b>Layer</b>	<b>Technology</b>	<b>Operation</b>
XLYP1H30	-	Piece definition and upper surface



<b>Layer</b>	<b>Technology</b>	<b>Operation</b>
XLYP1T\$TECH0\$	TECH0.tchx	Contouring

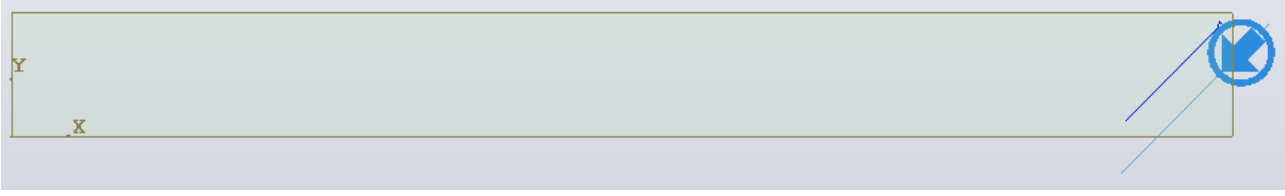


<b>Layer</b>	<b>Technology</b>	<b>Operation</b>
XLYP1T\$TECH5\$	TECH5.tchx	Right Junction



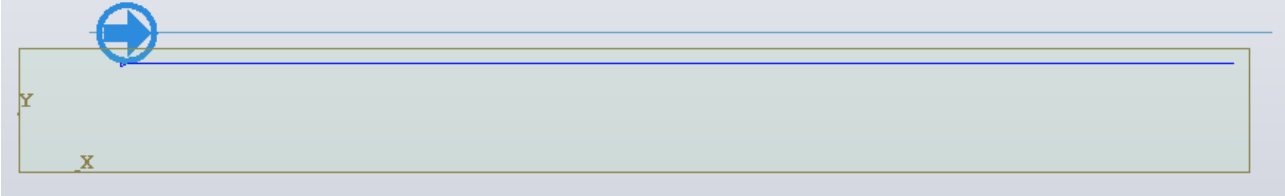
The diagram shows a rectangular workpiece with a coordinate system (X and Y axes) at the bottom-left corner. A blue arrow points from the left towards the right edge of the workpiece, indicating the direction of the Right Junction operation.

<b>Layer</b>	<b>Technology</b>	<b>Operation</b>
XLYP1T\$TECH1\$	TECH1.tchx	Left Junction



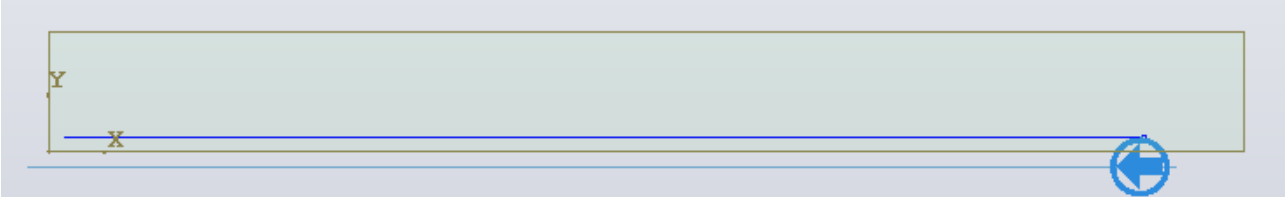
The diagram shows a rectangular workpiece with a coordinate system (X and Y axes) at the bottom-left corner. A blue arrow points from the right towards the left edge of the workpiece, indicating the direction of the Left Junction operation.

<b>Layer</b>	<b>Technology</b>	<b>Operation</b>
XLYP1T\$TECH2\$	TECH2.tchx	Convex milling tool



The diagram shows a rectangular workpiece with a coordinate system (X and Y axes) at the bottom-left corner. A blue arrow points from left to right along the top edge of the workpiece, indicating the direction of the Convex milling tool operation.

<b>Layer</b>	<b>Technology</b>	<b>Operation</b>
XLYP1T\$TECH3\$	TECH3.tchx	Concave milling tool




The diagram shows a rectangular workpiece with a coordinate system (X and Y axes) at the bottom-left corner. A blue arrow points from right to left along the bottom edge of the workpiece, indicating the direction of the Concave milling tool operation.

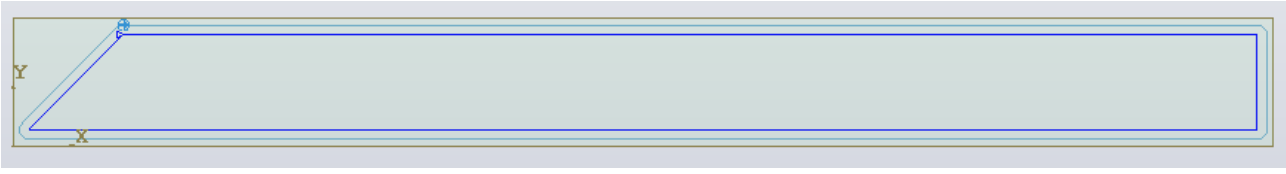


**PIECE C:**

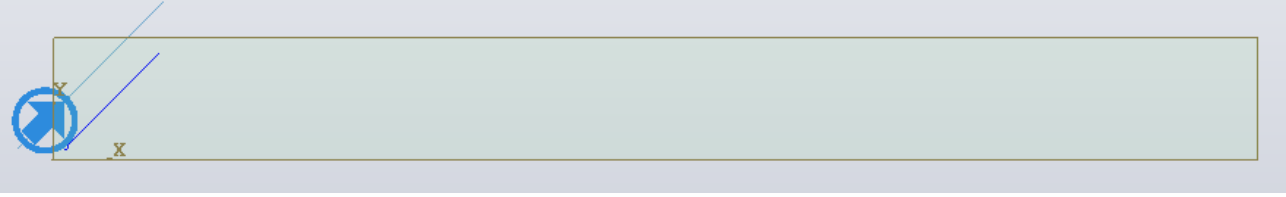
<i>Layer</i>	<i>Technology</i>	<i>Operation</i>
XLYP1H30	-	Piece definition and upper surface



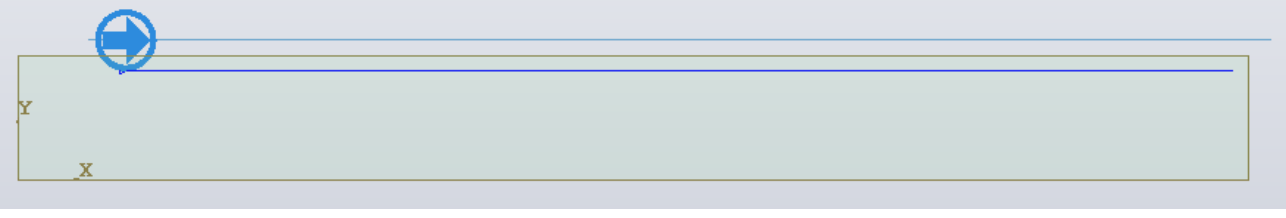
<i>Layer</i>	<i>Technology</i>	<i>Operation</i>
XLYP1T\$TECH0\$	TECH0.tchx	Contouring

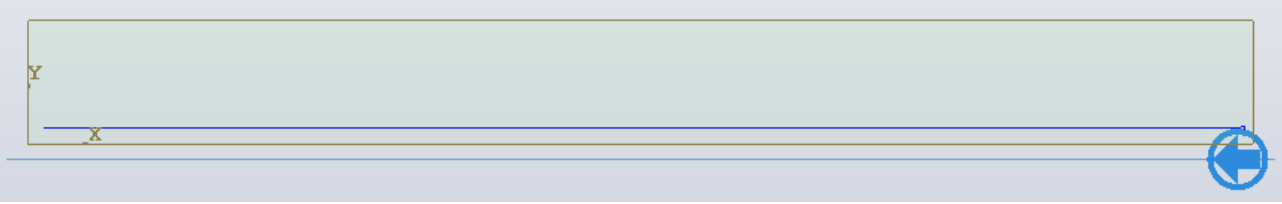


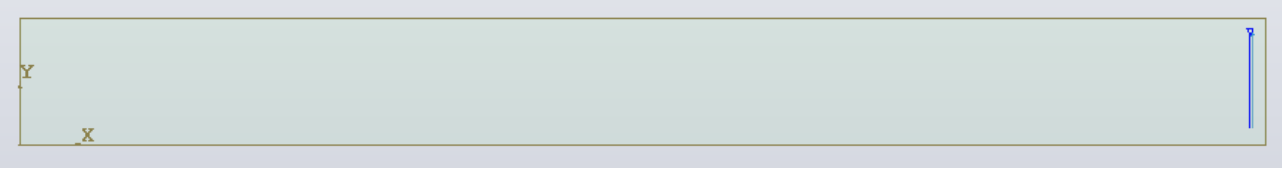
<i>Layer</i>	<i>Technology</i>	<i>Operation</i>
XLYP1T\$TECH5\$	TECH5.tchx	Right Junction



<i>Layer</i>	<i>Technology</i>	<i>Operation</i>
XLYP1T\$TECH2\$	TECH2.tchx	Convex milling tool



<b>Layer</b>	<b>Technology</b>	<b>Operation</b>
XLYP1T\$TECH3\$	TECH3.tchx	Concave milling tool
		

<b>Layer</b>	<b>Technology</b>	<b>Operation</b>
XLYP1T\$TECH4\$	TECH4.tchx	Finish
		

### 3.4.6.1.2 Import 2D geometries from DXF for edge banding

The edge banding process can be defined starting from a project structured with the specifications defined at the beginning of the chapter, introducing layers that identify the edge banding machining operations.

In particular a layer associated to an edge banding technology must be defined as specified in item 1, with the same structure and syntax (e.g. XLYP1T\$BORDATURA\$). The difference in this case is in the way the geometries to which the edge bandings are associated and the machining operations defined in the technological process are defined. As opposed to earlier where the entire technology is applied to each geometry, in this case one or more machining operations specified in the technology can be applied to each geometry section, with the start and end points being the ends of the geometry.

Independently of whether the geometry being machined is closed or open, to perform a specific machining on a section, the total geometry must not be drawn as a single curve (e.g. polyline) but as a set of more or less elementary curves (e.g. segments, arcs, polylines) and a colour associated to each section with the RGB code. This code must be the same colour code of the machining in the technology. In this way the system will create a single geometry as a combination of all the defined geometries and apply the associated machining operations to each section. The order in which the geometries are drawn is used to define the machining sequence of the edge banding process.

The drawing of the geometries must be produced so that the direction is always clockwise.

Figure 1 shows a practical example.

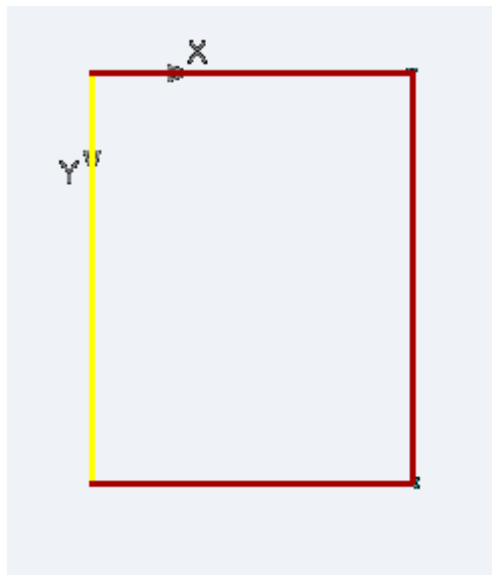


Figure 1.

The final geometry must be a rectangle, to which the edge banding process is applied.

The resulting sequence must be as follows:

- 1- Routing the whole rectangle
- 2- Edge banding upper side with edge B1
- 3- Upper side end trimming
- 4- Edge banding lower side with edge B1
- 5- Lower side end trimming
- 6- Edge banding right side with edge B1
- 7- Right side end trimming
- 8- Edge banding left side with edge B2
- 9- Left side end trimming
- 10- Trimming upper-right-lower side with trimmer RE1
- 11- Trimming right side with trimmer RE2
- 12- Scraping upper-right-lower side with scraper RA1
- 13- Scraping left side with scraper RA2
- 14- Shaving the 4 corners

To obtain the sequence the 4 lines are drawn to form a clockwise rectangle and a colour is associated to each line, as shown in figure 1.

The application of the technology associated to the layer, follows two fundamental rules:

- 1- Technological process
- 2- Order of the geometries and colour associated to the geometry

This means that during the application phase, if present in the technology, the routing is automatically applied to the whole geometry obtained by joining the 4 sections that must have matching start and end points. The routing is applied with start in the middle of the first side.

Immediately after the routing the edges present, that will be applied by following the order with which the geometries have been drawn, are analysed.

With the reference to the example, the geometries that must be edge banded must be drawn in the following order:

- 1- Red upper segment
- 2- Red lower segment
- 3- Red right segment
- 4- Yellow left segment

In this way, during the import phase, if in the technology a red edge banding and a yellow edge banding are found, the 4 edge bandings will be applied by following the sequence with which the geometries have been drawn and the single requested edge banding technologies.

The upper side will be edge banded first with edge B1, the lower side second with edge B1, the right side third with edge B1 and the left side fourth with edge B2.

Once the edge bandings have been inserted it checks if there is, for each section that has a colour, an end trimming technology with the same colour.

If there is the end trimming associated to the colour will be applied to each edge banding section.

The same applies for the trimming, for each geometry section with a colour it checks if there is the relative trimming technology, but in the application phase the trimmings with the same tool are joined together, if possible, in a single trimming operation. In the example, as the upper-lower-right side are red, they are given the same trimming, therefore a single trimming with trimmer RE1 is created. The yellow left side is trimmed with trimmer RE2.

The same applies for the edge scraping and glue scraping operations.

The shavings are managed automatically, during the application the applicability of the shaving for each edge banding is checked and where possible it is applied.

To obtain the rectangle edge banded with 2 edges, B1 on the upper-right-lower and B2 on the left, we have had to create the rectangle with 4 section in the correct sequence and associate to each section the edge banding colour present in the technology.

The example in figure 2 shows a case with sections not edge banded.

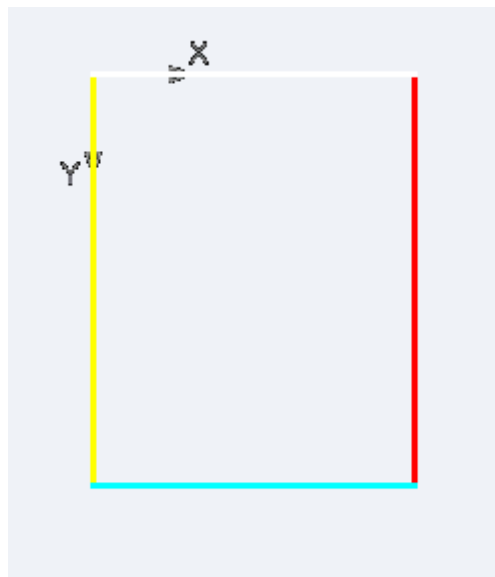


Figure 2.

In this case the upper section of the panel must not be edge banded. This is achieved by drawing the section, but the transparent RGB is associated as the colour.

As opposed to the example of figure 1 this achieves:

- 1- Routing the whole rectangle
- 2- Edge banding right side with edge B1
- 3- Right side end trimming
- 4- Edge banding left side with edge B2
- 5- Left side end trimming
- 6- Edge banding lower side with edge B3
- 7- Lower side end trimming
- 8- Trimming right side with trimmer RE1
- 9- Trimming left side with trimmer RE2
- 10- Trimming lower side with trimmer RE3
- 11- Scraping right side with scraper RA1
- 12- Scraping left side with scraper RA2
- 13- Scraping lower side with scraper RA3
- 14- Shaving the 4 corners

Similarly to example 1 the first machining applied is the routing of the whole panel, starting from the geometry start point.

Followed by the edge banding of the right with edge B1 (red), left with edge B2 (yellow), and lower with edge B3 (blue).

Followed by the end trimming associated to the single edge bandings.

Followed by the trimming of the right with trimmer RE1 (red), left with trimmer RE2 (yellow), and lower with trimmer RE3 (blue).

Followed by the scraping of the right with scraper RA1 (red), left with scraper RA2 (yellow), and lower with scraper RA3 (blue).

The shavings are applied automatically.

### 3.4.7 Geometry selection

Once all geometric entities or working processes have been inserted, it is often necessary to select them in order to carry out several operations

You can select a single entity by clicking with the left button of the mouse on the desired entity.

The selection is clearly visible because it is red-coloured.

In order to select more than one entity you can press - like for the single selection - "Shift".

In order to make a quicker multiple selection, this is a suitable method:

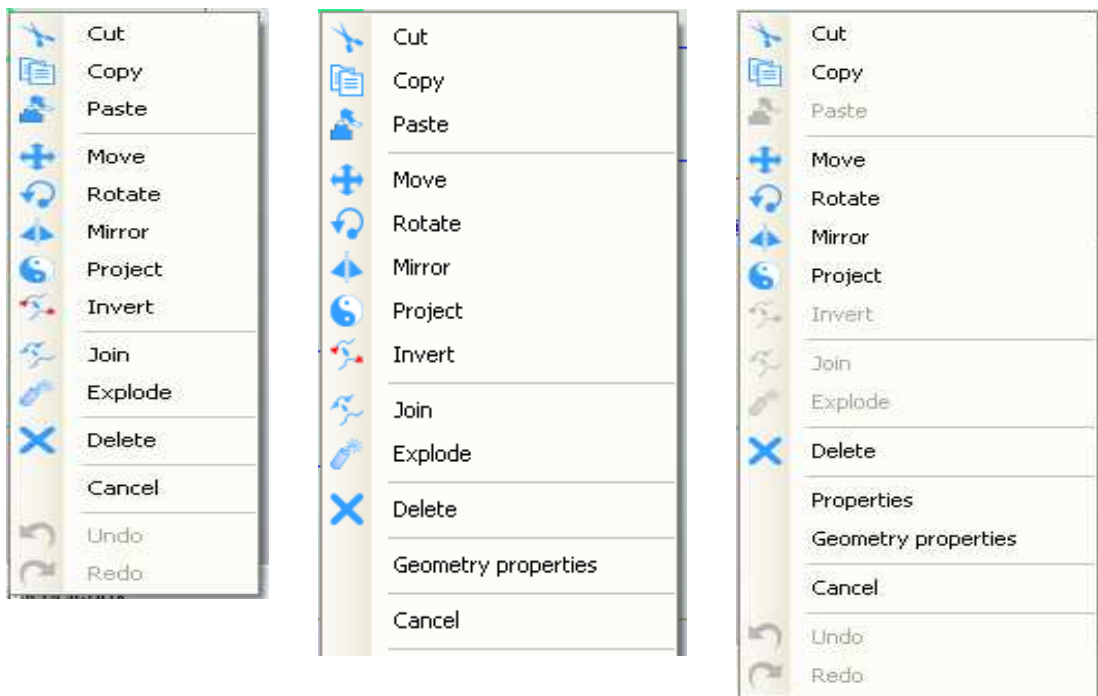
- press the left button of the mouse
- move the mouse (a rectangle is drawn)
- release the left button of the mouse

At this point all entities included in the rectangle will be selected and red-coloured.

### 3.4.8 Contextual Menu of graphic area

After selecting one or more geometric entities or working processes, if you click with the right button of the mouse, one of these two menus will appear with the commands related to the selected geometric elements.

The first menu will appear if you select two or more elements.  
The second one if you select one geometric element.  
The third if you select only one working process.



It is possible to display "Properties" only for one entity.

If there is no geometric entity among the selected elements, the commands "invert", "Join" and "Explode" are inhibited.

In case of one working process, it is possible to display the "Properties" of the working process or the "Properties" of the geometry concerning the working process.



### 3.4.9 Geometry removal

After selecting one or more geometric entities, you can delete:

- by clicking with the button "Canc"
- or clicking on the command "Remove" of the menu "Home" "Clipboard"
- or on the command "Remove" if the contextual Menu.

### 3.4.10 Geometry properties

Every geometric feature has several properties.

In order to display the properties of one element:

- Select the object (red-coloured)
- click on the command "Geometric properties of the contextual menu"
- The resulting table will show you the geometric data.

The tables displayed for every entity have been reported in the paragraphs related to the geometry inserting commands.

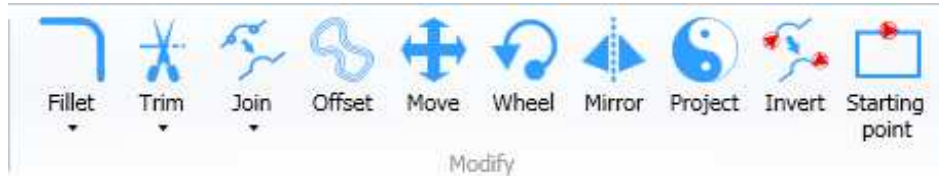
By marking the field "absolute Coordinates", the data will not refer to the faces on which the geometry has been built but to the three-dimensional "Zero" of the Project.

By changing the values in the fields of properties it is possible to modify the geometry.

This function cannot currently be applied to the 3D surfaces and curves.

### 3.4.11 Change existing geometries

Icons related to commands to modify basic geometries are to be found in the menu "Draw" "Modify".



All commands of geometry modification are interrupted by selecting a new command or pressing the button "Esc".

The "Cut", "Extend", "Start point", "Join", "Explode" commands cannot be applied to the 3D surfaces and curves. To change surfaces, see the relevant paragraph.

#### 3.4.11.1 Cut

It removes a workpiece of a geometric entity intersected by other entities. If an entity is not intersected by any line, it will be completely deleted.

In order to remove a workpiece of geometric entity:

- Click on the command "Cut"
- move the mouse (the entity workpiece to delete is green-coloured, in bold)
- click on the left button of the mouse

#### 3.4.11.2 Extend

It extends the geometry of a geometrical entity until it intersects with other entities.

To extend a geometrical entity:

- click on "Extend"
- select the entity to be extended
- select an entity that intersects the entity to be extended (the entity to be extended becomes blue and thick representing the new shape it will have, while moving the mouse on the many entities).



### 3.4.11.3 Invert

It inverts the initial point with the final one of any geometric entity.

The command is available in the menu "Draw" "Modify" as well as in the "Contextual menu).

In order to invert the direction of one or more geometries:

- Select one or more geometric entities
- click on the command "Invert"

The inversion is evident when the symbol linked with the geometric initial point has changed.

The "Invert" command, if applied to surfaces, inverts the normal of the surface.

### 3.4.11.4 Initial point

It changes the initial point of a closed geometry (circle or polyline).

In order to modify the initial point of a closed geometry:

- click on the command "Initial point"
- move the mouse
- click with the left button of the mouse (for a new initial point )

The representation of the initial point is performed with an arrow that detects the position and the direction.

### 3.4.11.5 Join

It creates a polyline by combining geometric entity with coinciding vertex.

The command is available in the menu "Draw" "Modify" as well as in the "Contextual menu".

In order to combine one or more geometries with coinciding vertex:

- Select one or more geometric entities
- click on the command "Join"

One or more polylines can be combined only with the entities that have coinciding vertex. In case of more entities with coinciding vertex, the entities inserted in the polyline will follow the selection order.

The direction of the new polylines is casual

### 3.4.11.6 Explode



A polyline explodes in all basic geometric elements in its structure. The command is available in the menu "Draw" "Modify" as well as in the "Contextual menu".

In order to explode one or more polylines:

- select one or more polylines
- click on the command "Explode"

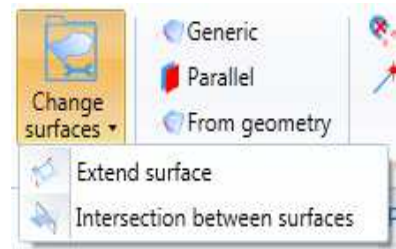
The direction of the single entities will be the one inside the polyline.

### 3.4.11.7 Change surfaces

To change a surface click on “Change surfaces”.

There are two commands to change the surfaces:

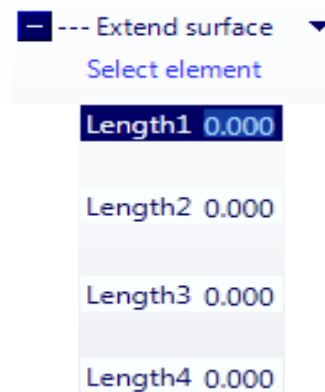
- Extend surface;
- Intersection between surfaces.



#### 3.4.11.7.1 Extend surfaces

To extend the surface proceed as follows:

- select the surface to extend;
- enter the extension length values.

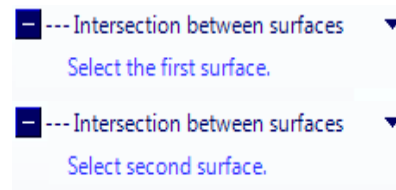


The surface extension occurs in tangency and it generates a new surface that replaces the original one.

#### 3.4.11.7.2 Intersection between surfaces

The surfaces intersection procedure is performed as follows:

- select the first surface;
- select the second surface.



### 3.4.11.7.3 Cut surface

The surface cutting procedure requires:

- selecting the surface to be cut;
- selecting one or more curves.

--- Cut surface along a curve  
Select the surface.

--- Cut surface along a curve  
Select one or more curves

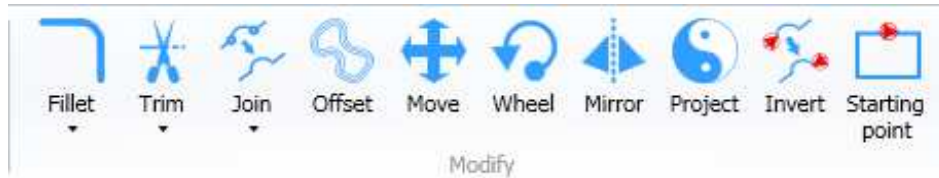
The permitted curves must be:

- Closed (if inside the surface)
- Open but with the initial and final points on the edge of the surface

The result will be to divide the original surface into two or more surfaces.

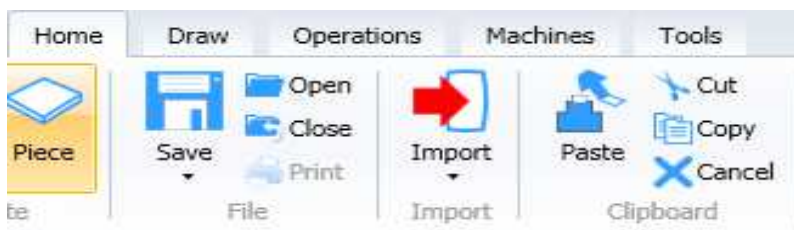
### 3.4.12 Move and/or copy the existing geometries

The icons on the commands to move, rotate, mirror or copy the existing geometries are to be found in the menu "Draw" "Modify"



All commands of geometry modification are interrupted by selecting a new command or pressing the button "Esc".

Further recurring commands are to be found in the menu "Home" "Clipboard".



The "Mirror" and "Oppose" commands are not valid for 3D surfaces and curve geometries.

### 3.4.12.1 Move

Move one or more entities from one point to another one.

The command is available in the menu "Draw" "Modify" as well as in the "Contextual menu".

In order to move one or more entities:

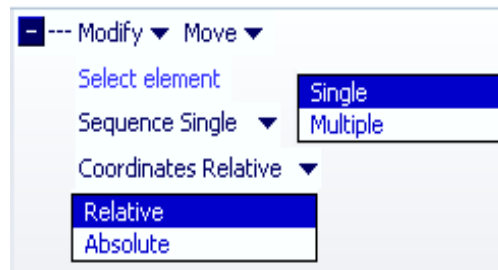
- select one or more entities
- click on the command "Move"
- type the reference point
- type the final point

Or:

- click on the command "Move"
- select one or more entities
- type the reference point
- type the final point

This command has three options:

- Create a copy
- Sequence (Single/Multiple)
- Coordinates (Relative/Absolute)



The option "Create a copy" allows you to make a copy out of the source text.

The option Multiple Sequence allows you to keep on moving the element by typing further final points.

The option "Absolute Coordinates" allows you to stop the select geometries at a certain level starting from a zero level of the work plane.



### 3.4.12.2 Rotate

Rotate one or more entities.

The command is available in the menu "Draw""Modify" as well as in the "Contextual menu".

In order to rotate one or more entities:

- select one or more entities
- click on the command "Rotate"

Or:

- click on the command "Rotate"
- select one or more entities

If you rotate two-dimensional elements:

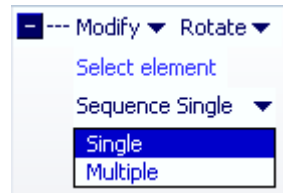
- type the reference point
- set an angle (typing the point or inserting the value)

If you rotate three-dimensional elements i :

- Select an axis or a vector or define a vector by entering two points
- set an angle (typing the point or inserting the value)

This command has two options at disposal:

- Single Sequence
- Multiple Sequence)



The Multiple Sequence option allows you to continue rotating the selected items by making a copy each time you set a new angle.

### 3.4.12.3 Mirror

It mirrors one or more entities compared to the axis of the centre line in the work face. The command is available in the menu "Draw" "Modify" as well as in the "Contextual menu".

This command works only in case of standard faces.

In order to mirror one or more entities:

- select one or more entities
- click on the command "Mirror"
- select the mirror mode in the menu
- click on the button "Apply"

This command has five options:

- Mirror in X
- Mirror in Y
- Mirror in X/Y
- Keep in the source form
- Invert the working process



The option "Mirror in X" mirrors the geometry compared to the Y-axis of the centre line in the active face.

The option "Mirror in Y" mirrors the geometry compared to the X-axis in the middle of the active face.

The option "Mirror X/Y" mirrors the geometry compared to the X and Y axis in the middle of the active face (this mode is like a rotation of 180°)

The option "keep the source geometry" allows you to make a copy out of the source geometry.

The option "Invert working process" allows you to invert the direction of the working process if the command "Mirror" is used to mirror a working process.

### 3.4.12.4 Oppose

It copies one or more entities on the standard face opposite to the active one. The command is available in the menu "Draw" "Modify" as well as in the "Contextual menu".

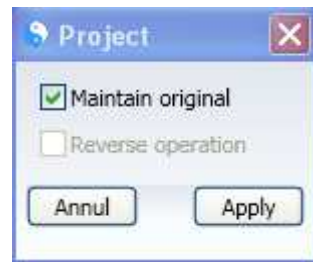
This command works only in case of standard faces.

In order to copy one or more entities on the opposite standard face:

- select one or more entities
- click on the command "Oppose"
- click on the button "Apply"

This command has two options at disposal:

- Keep in the source form
- Invert the working process



The option "keep the source geometry" allows you to make a copy out of the source geometry).

The option "Invert the working process" allows you to invert the working direction if the command "Oppose" is used for a working process.

### 3.4.12.5 Cut

It cuts one or more geometric elements or working processes in order to put them in the buffer for the option "paste".

The command is available in the menu "Home" "Clipboard" as well as the "Contextual menu"

and works only for elements of the active face.

In order to cut one or more entities from the active face.

- select one or more entities
- click on the command "Cut" or press "Ctrl-x"
- type one reference point.

As a result of the geometry of the selected working process it will disappear from the display.

### 3.4.12.6 Copy

It copies one or more geometric elements or working processes in order to put them in the buffer of the option "Paste)

The command is available in the menu "Home" "Clipboard" as well as the "Contextual menu"

and works only for the elements of the active face.

In order to copy one or more entities from the active face:

- select one or more entities
- click on the command "Copy" or press "Ctrl-c".
- type one reference point.

You do not notice any effect but the previously selected geometry will be available to be copied through the option "Paste"

### 3.4.12.7 Paste

It pastes one or more geometric elements or working processes that have previously been selected from the option "Cut" or "Copy"

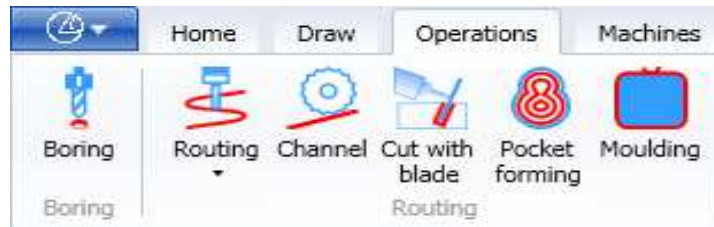
The command is available in the menu "Home" "Clipboard" as well as the "Contextual menu"

In order to paste one or more entities on the active face:

- click on the command "Paste" or press "Ctrl-v"
- type one reference point.

## 3.5 Working processes

The commands in order to insert the working processes are to be found in the menu "Operations"



Every working process must be carried out combining a tool with a 2D geometry. As a result, it is necessary to activate a Tool- Data-Base (see paragraph 10.8)

In order to set any working process, it is useful to keep this structure in mind:

- 1) Set the position in which you carry out the working process (Pos)
- 2) Set the Geometric Shape of the working process (Geo)
- 3) Set the Work mode (Mode)
- 4) Set the technology adopted (tool, speed and so on).

## 3.5.1 Boring

In order to bore, you can:

- click on the command "Boring"
- select one or more 2D elements

Or:

- select one or more 2D elements
- click on the command "Boring"

The selection of 2D elements can be single or multiple as the option for boring will make bores connecting them with the points or the circles.

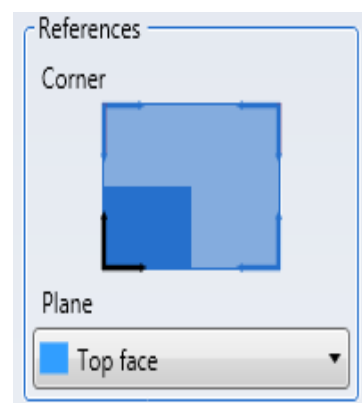
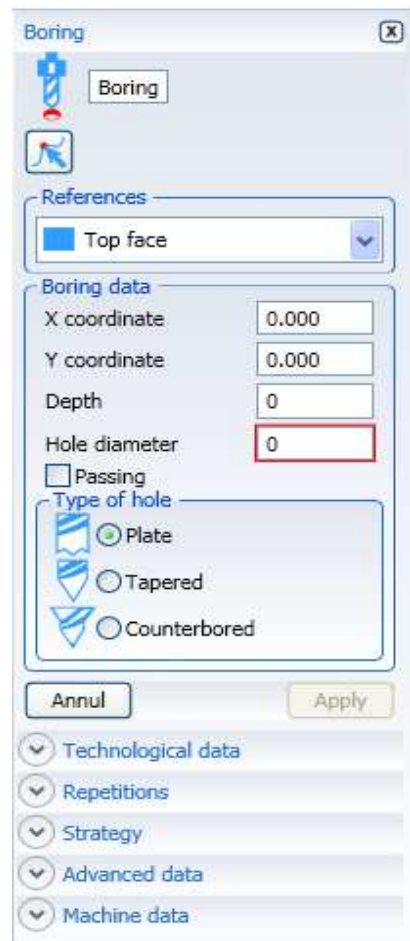
### 3.5.1.1 Bore position

If, among the selected elements, there are points and/or circles, the bore position is automatically set.

If no point or circle has been selected, you can set the position of a new point in this way:

- select the reference face
- select the reference edge on the face
- fill the field X Coordinate in
- fill the field Y Coordinate in

The reference edge on the face allows programming the hole with X and Y coordinates that are related to the user-selected reference point. In the case of PowerFlex machines, if you want to draw holes with coordinates relative to the mechanical zero of the machine, for the upper face holes you must select the left upper edge.



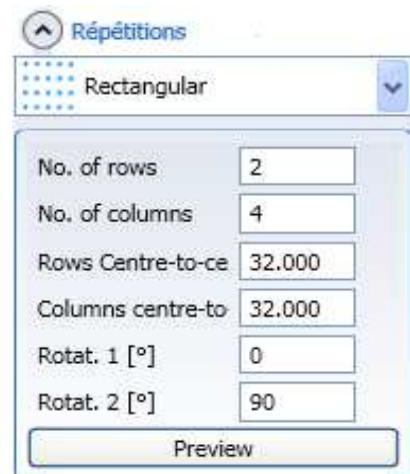
It is possible to set a set of matrix bores by using the sub-menu "Repetitions"

to this purpose, the nearby menu fields must be filled in.

Rotat. 1 represents the angle to set the lines

Rotat. 2 stands for the angle to set the columns compared to the lines

By clicking on the button "Preview", you obtain a representative table that displays the bore position as described above.



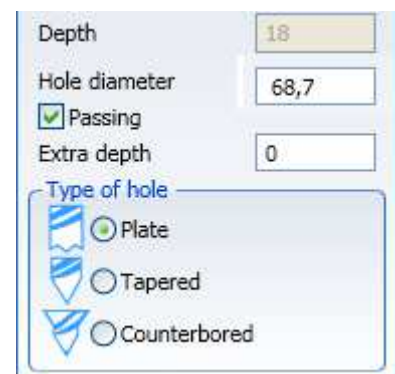
### 3.5.1.2 Bore structure

In order to set the bore structure, Diameter, Depth and bore Type must be set in the menu.

In field "Depth" you will insert the value corresponding to the thickness of the workpiece in case the option "Passer-by" is selected.

If the "boring" operation has been performed by selecting a circle rather than a point, the field "Bore diameter" will be filled with the value of the circle diameter.

In this case, if you wish to change the diameter, you can unmark the heading "Keep circle diameter" and set a new diameter.



The field "Extra depth" is filled with the extent through which the bore must go beyond the thickness of such workpiece (this heading will appear only if the option "Passer-by" has been selected")

The type of bore must be specified when it is not linked to a drilling tool and in this case, in order to choose the tool, you need this information.

The hole types are as follows:

- Hole with flat bottom
- Hole with tapered bottom
- Counterbored hole in top part and with tapered bottom

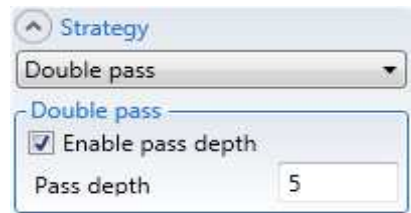
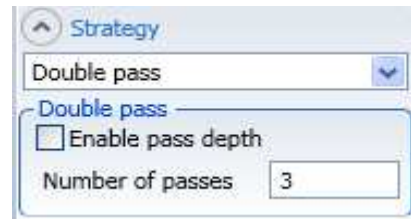
Select the types from the "Type of hole" field

### 3.5.1.3 Boring mode

In order to set how to structure the bore, you use the sub-menu "Strategy"

The bore can be performed in a single or a multiple rub.  
The following can be set with multi-pass:

- the "Number of passes" required to make the hole.
- The "Pass depth" visible when selecting "Enable pass depth" field.



### 3.5.1.4 Boring technology

In this step, you set all technical aspects related to tools, spindles and machines at disposal.

See the following chapters:

3.5.13 Technological data (shared by the machining operations)

3.5.15 Advanced data (common to machining)

3.5.16 Machine data (shared by the machining operations)

### 3.5.1.5 Drilling deduced from the surface

If a cylindrical surface is selected which refers to a hole, all the geometric parameters of the dialog boxes will be automatically filled.

In the case of multiple selection of coaxial surfaces of suitable shape and position, the system can recognize flared or counterbored holes.

The user can still modify the precompiled fields.

If a through hole is recognized, if possible a lance point will be used and a depth determined to guarantee the through hole.

The condition in which the hole is made is that the hole axis is orthogonal to one of the six planes of the part or to a previously created user plane.



## 3.5.2 Inclined drilling

To insert one or more holes we can:

- click on the "Drilling" command
- select one or more 2D elements

Or:

- select one or more 2D elements
- click on the "Drilling" command

The 2D elements selected can be single or multiple because the "Inclined drilling" function will produce holes by associating them only with the points or circles.

### 3.5.2.1 Position of the inclined hole

If the selected elements have points and / or circles, the position of the holes is determined automatically.

If no point or circle has been selected, the position of a new point can be defined by entering the coordinates X, Y, Z

### 3.5.2.2 Shape of the inclined hole

To define the shape of the hole, the Diameter, Depth, Angles A and B Type of hole must be set in the menu.

If drilling was done by selecting a circle instead of a point, the "Hole diameter" field will represent the value of the circle diameter.

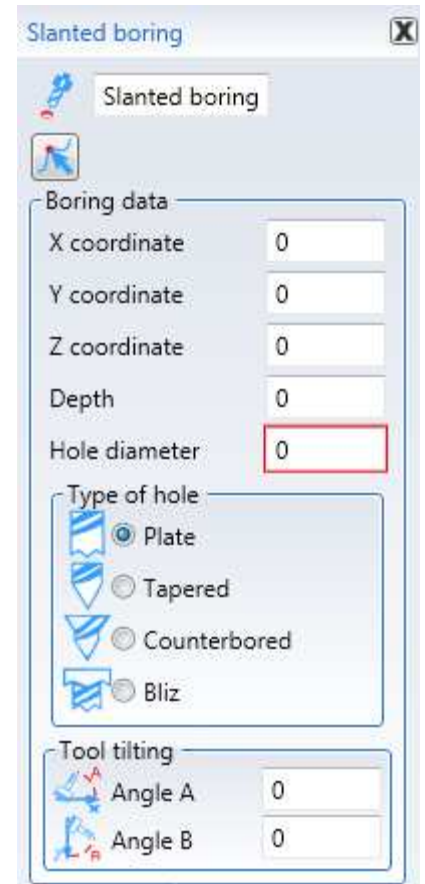
The Type of hole must be specified when no tool is associated with the drilling, and in this case, this information is needed to make the automatic tool selection.

The types of hole are the following:

- Hole with flat bottom
- Hole with conical bottom
- Flared hole in the upper part and with a conical bottom
- Counterbored hole (Blitz)

The types can be selected from the "Hole type" field

The angles A and B are used to determine hole inclination.



### 3.5.2.3 Inclined drilling mode

The same applies to drilling

### 3.5.2.4 Inclined drilling technology

The same applies to drilling

### 3.5.2.5 Drilling deduced from the surface

If a cylindrical surface is selected which refers to a hole, all the geometric parameters of the dialog boxes will be automatically filled.

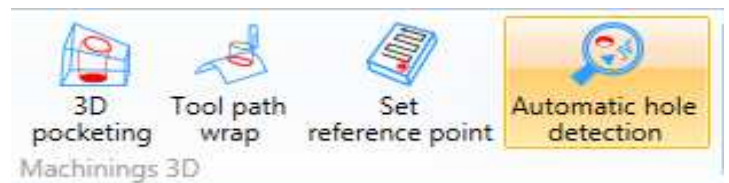
In the case of multiple selection of coaxial surfaces of suitable shape and position, the system can recognize flared or counterbored holes.

The user can still modify the precompiled fields.

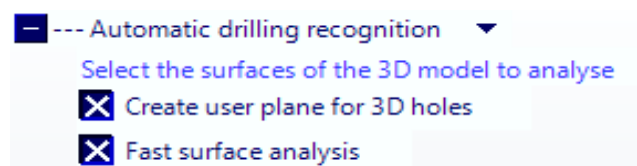
## 3.5.3 Automatic drilling deduced from the surfaces

This function recognizes all the inclined holes and holes present in the selected surfaces.

To use this function, use the "Automatic hole recognition" command in the "Operations-3D Machining" menu



The operating mode is similar to that described in drilling and inclined drilling with the following additional functions:



"Create User Plan for 3D Drilling" Creates a plane for each recognized inclined hole.

"Fast surface analysis" will only recognize cylindrical holes and involves a low processing time. By removing the check to this parameter, also countersunk and counterbored holes will be recognized at the cost of longer processing time.

Once the surfaces to be analysed have been selected, the maximum diameter below which the surfaces are recognized as holes must be set.

The generated holes will start from the work plane closer to the hole geometry. It is advisable to check the correct assignment of the tools which will perform inclined drilling.

If more than one hole has been generated on the same user plane, it is advisable to check that they can be implemented by switching to the safety level from the user's plane, otherwise the programming must be corrected.

## 3.5.4 Milling

In order to insert one or more milling operations you can:

- click on the command "Milling"
- select one or more 2D elements

Or:

- select one or more 2D elements
- click on the command "Milling"

The selection of 2D elements can be single or multiple because the function "Milling" will perform milling operations connecting them with lines, polylines, arcs and circles.

### 3.5.4.1 Milling position

If there is any line and/or polyline and/or arc and/or circle among the selected elements, the milling position is automatically set.

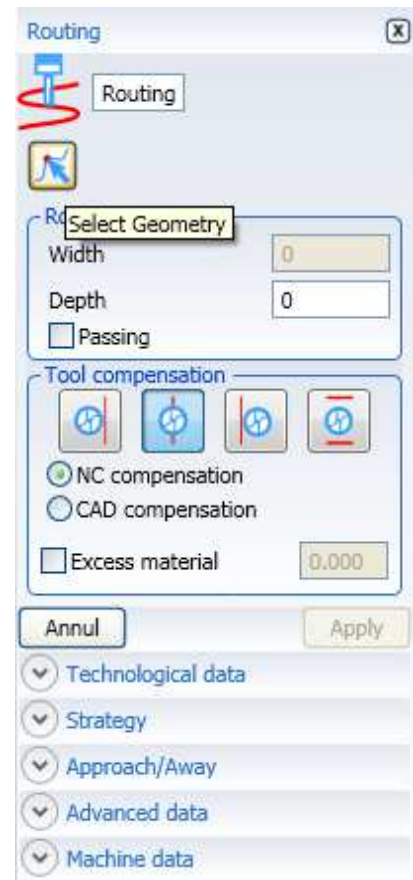
The real position of every milling is affected by the used tool correction and the super-material.

In the following tables it is possible to understand how the different combination of options work.

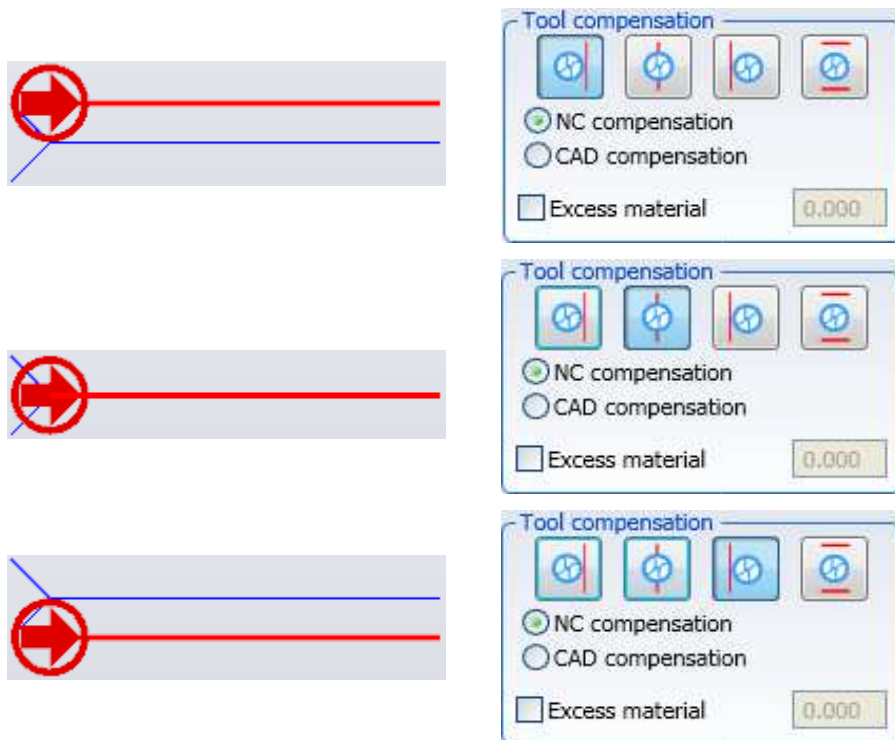
The light blue line stands for the "2D geometry" and the red line represents the tool path in the tool centre.

By marking "Correction C.N.", it implies that the coordinates of the 2D geometry are passed and the effective path (red line) will be calculated by the CN.

If you mark "Cad correction", the coordinates of the red line are directly transferred to the CN and this latest will not make any correction.



Behaviours with left, central and right corrections without any correction in length and without any super-material.



Behaviours with left, central and right corrections in length and without any super-material.



Behaviours with left, central and right correction without any correction in length and with super-material.



Behaviours with left, central and right corrections in length and with super-material.



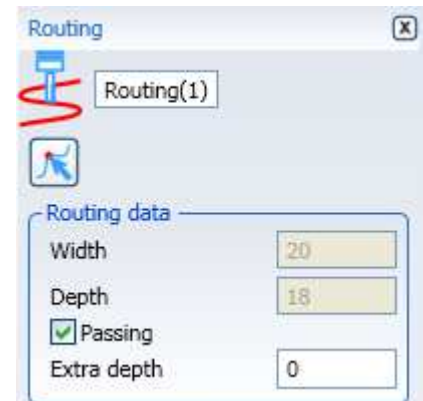
### 3.5.4.2 Milling form

In order to set the milling form you have to set Length and Depth in the menu.

The field Width is automatically filled when a tool is connected with the milling.

In field "Depth" you will insert the value corresponding to the thickness of the workpiece in case the option "Passer-by" is selected.

The field "Extra depth" sets the extent of the milling to cross the workpiece. (This heading will appear only the option "Passer-by" has been selected)



The field "Indivisible processing" (currently used only for Cyflex machines) ensures that such machining is not performed in one or more stages for those automatically generated when using Cyflex

If an error is generated during the optimisation phase, the operator must replace the machining operation with the selection with two or more machining operations that produce the same operation.

### 3.5.4.3 Milling modes

In order to set how to perform the milling, you use the sub-menus "Strategy" and "Approach/Removal".

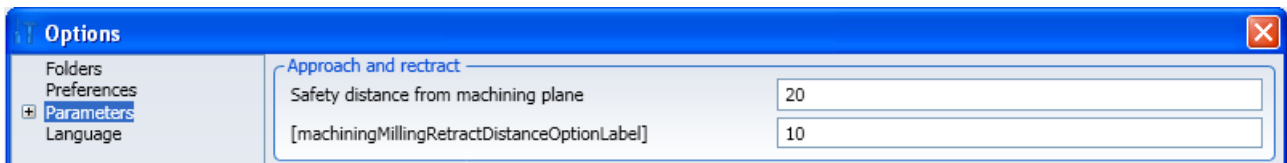
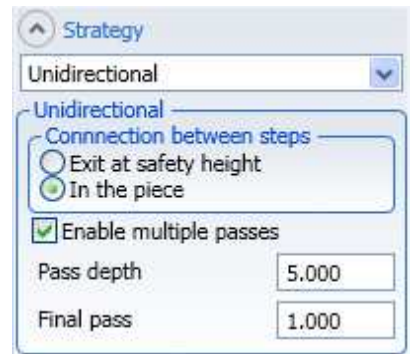
The milling can be performed only in one or more rubs.

In case of "multi-rub", you must choose between One-way or two- ways strategy.

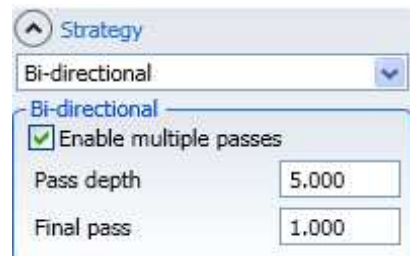
In the "one-way strategy", you have to establish if the rise between two rubs must occur:

- at a safety level on the workpiece
- or in the workpiece

Use the "Tools" "Options" "Parameters" "Machining" menus to modify these two parameters.



For the one-way strategy as well as for the two ways strategy, the heading "Enable multi-rub" must be marked and the values "Rub depth" and depth "Last rub."



Another modification to the milling mode can be set by the menu "Advanced data" through the parameter "Invert".(See 3.5.15 Advanced data (common to machining))

By marking this parameter, the working process can be performed starting from the final point of the selected 2D geometry.(See 3.5.14 Approach/Retraction (shared by the machining operations))

### 3.5.4.4 Milling technology

See the following chapters:

3.5.13 Technological data (shared by the machining operations)

3.5.15 Advanced data (common to machining)

3.5.16 Machine data (shared by the machining operations)

## 3.5.5 Slanted routing

To enter one or more tilting routing proceed as follows:

- click on “Tilting routing”
- select one or more 2D elements

Or:

- select one or more 2D elements
- click on “Slanted routing”

The 2D elements selection can be single or multiple.

### 3.5.5.1 Tilting routing position

The position of the tilting routing is determined automatically by the geometry selected. The tool correction is available only if the tool is perpendicular to the right or left and the Angle A is equal to 0.

To use the tool correction see chapter 3.5.4.1 Routing position.

### 3.5.5.2 Tilting routing shape

To define the tilting routing shape set the parameters in the menu:

The “Width” is set automatically based on the diameter of the tool selected (cannot be edited).

The “Depth” is the position Z in relation to the plane selected.

The “Angle A” indicates the tool rotation in relation to axis X

The “Angle B” indicates the tool rotation in relation to axis Z.

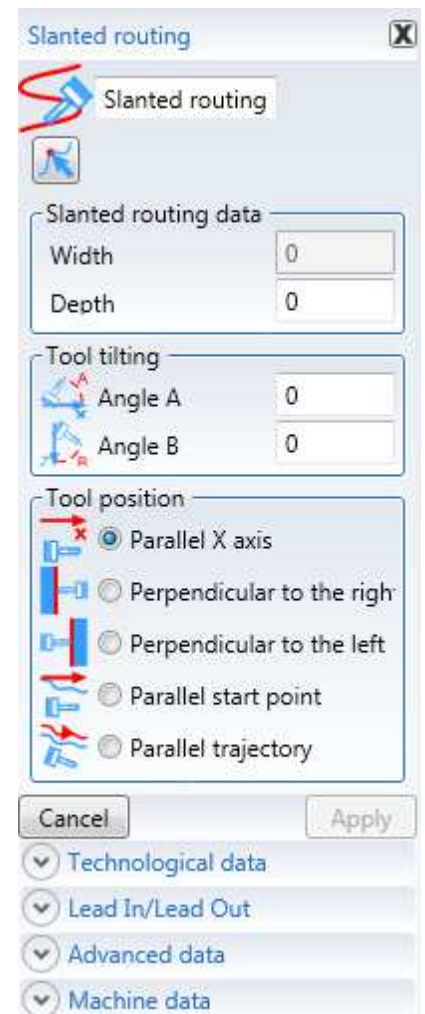
The “Tools position” can be set in the 5 modes visible in the menu. Bear in mind that the positions are considered before the setting of the “Angle A” parameter.

By selecting “Parallel axis X” the tool axis will be parallel to axis X, angle B will rotate the tool in the X-Z plane.

After having selected this mode, setting Angle A the tool axis will no longer be parallel to axis x.

During the whole tilting routing in this mode the tool will always maintain the same setup.

Selecting “Perpendicular to the right” the tool will enter from the





right in relation to the first geometrical entity of the trajectory.  
The setup will be maintained for the whole machining operation.  
Selecting “Perpendicular to the left” the tool will enter from the left in relation to the first geometrical entity of the trajectory.  
The setup will be maintained for the whole machining operation.

Selecting “Parallel start point” the tool will position itself with an inclination of Angle B set in the plane of the first geometrical entity of the trajectory.  
The setup will be maintained for the whole machining operation.

Selecting “Parallel trajectory” the tool will position itself with an inclination of Angle B set in the plane of the first geometrical entity of the trajectory.  
The setup will be recalculated in the same for each subsequent geometrical entity of the whole machining operation.

### **3.5.5.3 Tilting routing mode**

To define how the tilting routing is performed use the “Approach/Retraction” submenu.

See the use of the parameters of the “Approach/Retraction” menu in paragraph 3.5.14  
Approach/Retraction (shared by the machining operations)  
The arc approach and retraction are disabled.

### **3.5.5.4 Tilting routing technology**

See the following chapters:

- 3.5.13 Technological data (shared by the machining operations)
- 3.5.15 Advanced data (common to machining)
- 3.5.16 Machine data (shared by the machining operations)

## 3.5.6 Chamfering

To enter one or more chamfering proceed as follows:

- click on “Chamfering”
- select one or more 2D elements

Or:

- select one or more 2D elements
- click on “Chamfering”

The 2D elements selection can be single or multiple.

### 3.5.6.1 Chamfering position

The position of the chamfering is determined automatically by the geometry selected.

### 3.5.6.2 Slotting shape

To define the slotting shape set:

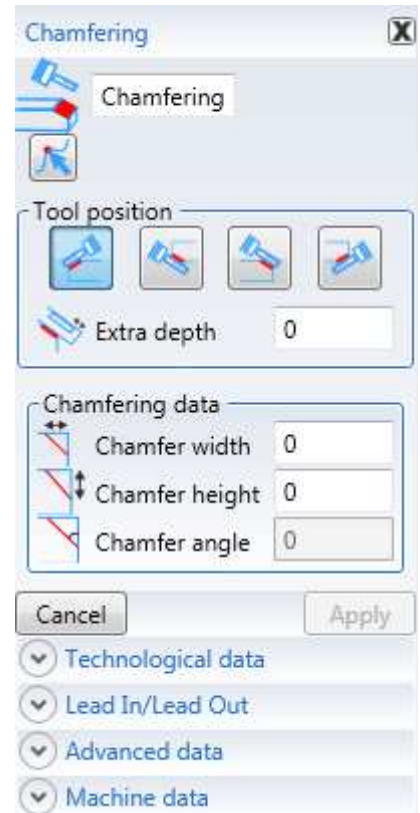
The position of the tool in relation to the geometry (see “Tool position” icons).

The chamfer width quota measured on the plane selected.

The chamfer height (quota measured at right angles to the plane selected).

The chamfer angle is determined automatically after setting the width and height.

The “Extra depth” field is used to move the tool beyond the height set.



### **3.5.6.3 Chamfering mode**

To define how the chamfering is performed use the “Approach/Retraction” submenu.

See the use of the parameters of the “Approach/Retraction” menu in paragraph 3.5.14  
Approach/Retraction (shared by the machining operations)

### **3.5.6.4 Chamfering technology**

See the following chapters:

3.5.13 Technological data (shared by the machining operations)

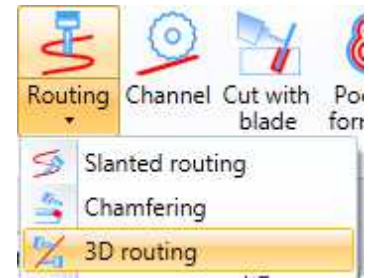
3.5.15 Advanced data (common to machining)

3.5.16 Machine data (shared by the machining operations)

### 3.5.7 3D routing

To enter a 3D routing click on “3D routing” in the “Routing” menu.

The geometry cannot be selected, but entered by points.  
The tool inclination can be associated to each programmed point.



#### 3.5.7.1 3D routing position

The 3D routing position is determined by all the points entered during the machining definition.

The tool correction is not available.

#### 3.5.7.2 3D routing shape

The 3D routing shape can be a segment or a broken line of various segments.

To define the 3D routing shape set the coordinates of the various points of the broken line.

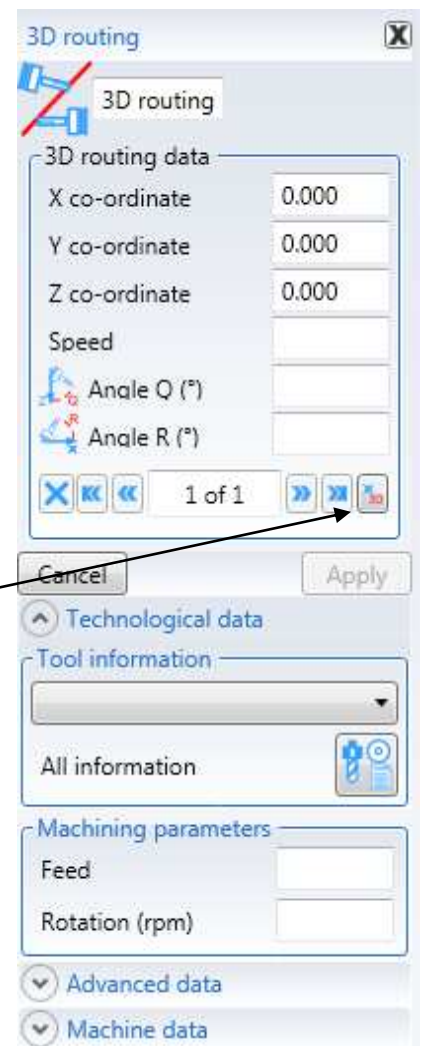
The “Angle Q” indicates the tool rotation in relation to axis Z.  
The “Angle R” indicates the tool rotation in relation to axis X

The tool will maintain the same setup for the whole machining of the defined section.

To enter other sections use the “Enter point” key.

To see the points entered use the right and left arrows to navigate in the broken line.

To delete a point use "Z".



#### 3.5.7.3 3D routing mode

The “Approach/Retraction” mode cannot be used.

#### 3.5.7.4 3D routing technology

See the following chapters:

3.5.13 Technological data (shared by the machining operations)

3.5.15 Advanced data (common to machining)

3.5.16 Machine data (shared by the machining operations)

## 3.5.8 Channel

The channel is a milling process with one limit and some added-values compared to the general milling process.

The limit is due to the fact that the channel can be applied only for a line.  
The added-values will be showed in the menu explanation.

In order to insert one or more channels, we can:

- Click on the command "Channel"
- select one or more 2D elements

Or:

- select one or more 2D elements
- Click on the command "Channel"

The selection of 2D elements can be single or multiple because the function "Channel" will perform milling cutters associated with the lines.

### 3.5.8.1 Channel position

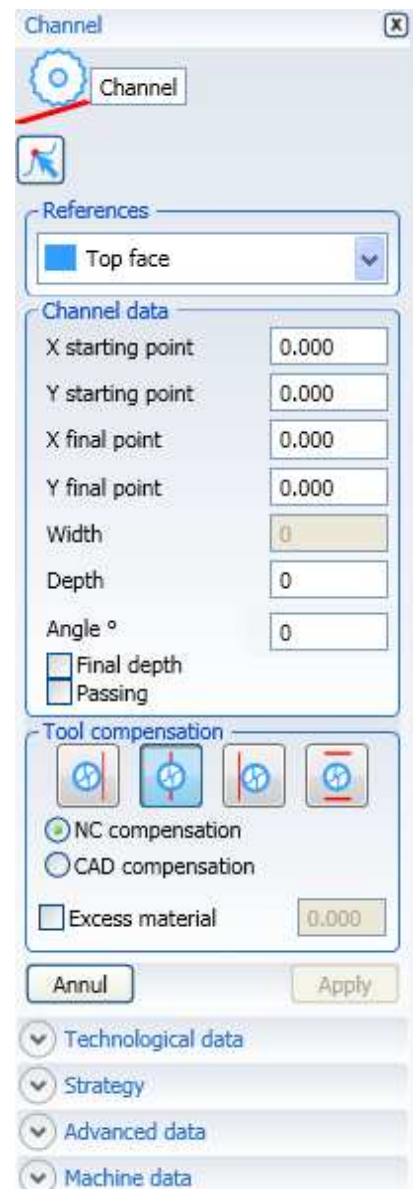
if there is any line among the selected elements, the channel position is automatically set.

If no line has been selected, the position of a new line can be set in this way:

- select the reference face
- fill the X field initial point
- fill the Y field initial point
- fill the X field final point
- fill the Y field final point

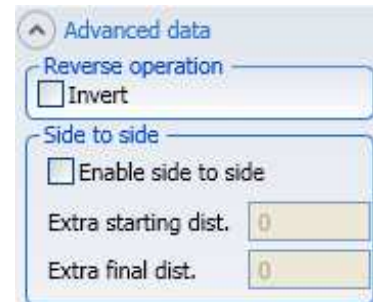
The real position of every milling is affected by the used tool correction and the super-material.

In the tables in paragraph 3.5.4.1(milling position) it is possible to understand how the different combinations of options work.



We can also change the channel position by using the parameter "Enable edge" in the sub-menu "Advanced data". By using this function, the selected segment will be stretched until it crosses the panel edges.

You can also have a channel extension if you activate the parameters "Extra initial distance" and "Extra final distance".



### 3.5.8.2 Channel structure

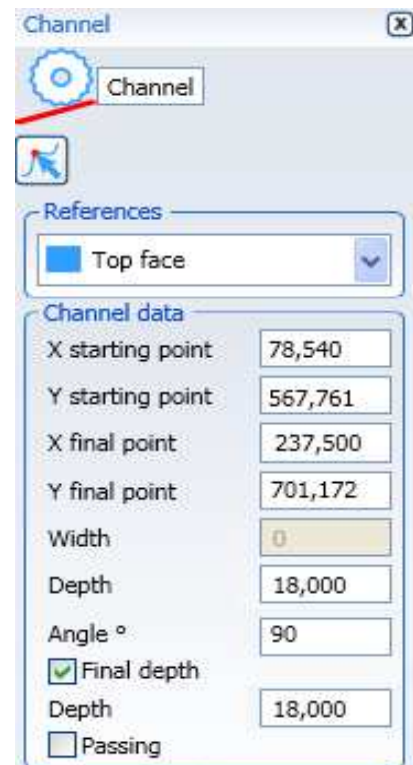
In order to set the channel structure, Length and Depth must be set in the menu.

The field Width is automatically filled when a tool is connected with the channel.

In field "Depth" you will insert the value corresponding to the thickness of the workpiece in case the option "Passer-by" is selected.

It is possible to set a "Final Depth (associated with the final point of the geometry) different from the one associated with the final point).

Enter the channel's inclination value, varying from 0° to 180° (not included) in the "Angle °" field. (90° = perpendicular channel to face)



### 3.5.8.3 Channel execution mode

The channel execution mode looks like the milling mode (see paragraph 3.5.4.3).

### 3.5.8.4 Channel technology

See the following chapters:

3.5.13 Technological data (shared by the machining operations)

3.5.15 Advanced data (common to machining)

3.5.16 Machine data (shared by the machining operations)

### 3.5.8.5 Channel deduced from surface

If a flat surface is selected, all the geometric parameters of the dialog boxes will be automatically filled.

The user can modify the precompiled fields and must insert the blade to be used.

### 3.5.9 Blade cut

In order to insert one or more blade cuts you can:

- click on the command "Blade cut"
- select one or more 2D elements

Or:

- select one or more 2D elements
- click on the command "Blade cut"

The selection of 2D elements can be single or multiple because the option "Blade cut" will perform cuts associated with the lines.

### 3.5.9.1 Cut position

Blade cuts can be performed only on lines on the upper face.

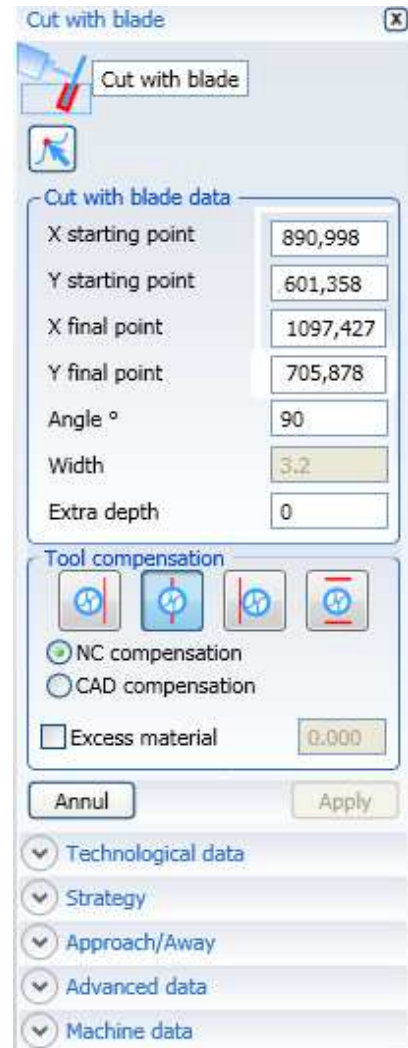
If there are lines among the selected elements, the cuts position is automatically set.

If no line has been selected, the position for a new line can be set in this way:

- Fill the X field initial point
- Fill the Y field initial point
- Fill the X field final point
- Fill the Y field final point
- fill the Inclination field

The real position of every cut is affected by the used tool correction and by the super-material.

In the tables in paragraph 3.5.4.1 (milling position) it is possible to understand how the different combinations of options work.



We can modify the cut position using the sub-menu "Advanced data"

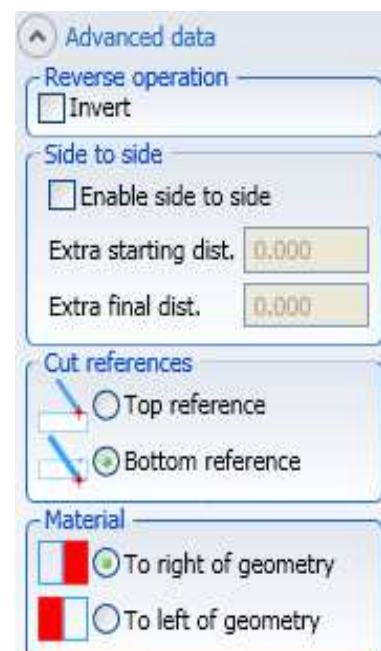
The parameter "Enable edge" extends the selected segment until the cross of the panel edges.

You can extend it using the parameters "Extra initial dist." and "Extra final dist."

By using the parameter "Lower reference" the selected geometry will lie on the lower face.

This working process generates also a work plane on one of the two surfaces caused by the cut.

The surface will be created on the new external face of the workpiece, so you must specify the material placement after the cut using the parameters "To the right of the geometry" "To the left of the geometry".





### 3.5.9.2 Cut structure

In order to set the structure of the blade cut, Length and Extra Depth must be set in the menu.

The field Width is automatically filled when the blade is associated with the cut.

The Depth (whose field is not present) will correspond to the thickness of the workpiece.

The field "Extra depth" indicates the extent of the cut overcoming the thickness of the workpiece.



### 3.5.9.3 Cut mode

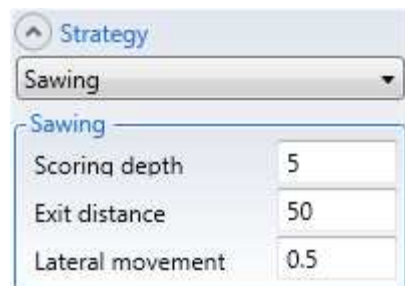
Use the following as cutting mode:

- The "invert" function
- The "Cutting" Strategy

See "Advanced Data" menu in paragraph 3.5.4.3 (Milling mode) for use of "Invert" parameter

The "Cutting" strategy foresees incision for equal length to the selected geometry, without cutting the workpiece at "Incision depth", increased by the "Outlet distance".

The blade descends at programmed depth at end of first incision, simultaneously moving outside the cut equally to "Side movement" height. The cut will be completed by returning to initial point.



Note: Side movement accepts values from -0.5 to +0.5m included.

### 3.5.9.4 Cut technology

See the following chapters:

3.5.13 Technological data (shared by the machining operations)

3.5.15 Advanced data (common to machining)

3.5.16 Machine data (shared by the machining operations)

### 3.5.9.5 Cut deduced from the surface

If a flat surface is selected, all the geometric parameters of the dialog boxes will be automatically filled.

The user can modify the precompiled fields and must insert the blade to be used.

## 3.5.10 Contouring

The contouring is a milling that can apply the external contour of the workpiece or the geometries.

In order to perform the contouring to the external workpiece:

- click on the command "Contouring"

The resulting menu will show you the parameter of the selected "workpiece".

By inserting the workpiece contouring, a closed polyline will be created on the upper face, that overlaps with the external profile of the workpiece.

In order to insert one or more geometric contouring, you can:

- click on the command "Contouring"
- click on the parameter "Geometry"
- select one or more 2D elements

Or:

- select one or more 2D elements
- click on the command "Contouring"

The selection of 2D elements can be single or multiple because the contouring will perform millings associated only with closed polylines or with circles.



### 3.5.10.1 Contouring position

The contouring position is automatically set by the selected geometry.

If the contouring is set to the geometry, then it is possible to choose the "External" "Internal" "Side" from the menu where you can set it.

### 3.5.10.2 Contouring structure

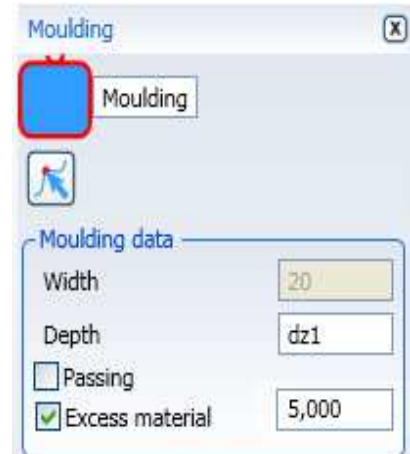
In order to set the contouring structure, you must set Length and Depth in the menu.

The field Length is automatically filled when you associate a tool to the contouring.

In field "Depth" you will insert the value corresponding to the thickness of the workpiece in case the option "Passer-by" is selected.

The field "Extra depth" sets the contouring extent to go beyond the workpiece (This heading will appear only if the option "Passer-by" has been selected)

The contouring leaves extra material for subsequent machining thanks to the "Excess-material" field.



### 3.5.10.3 Contouring modes

See the use of the parameter "Invert" of the menu Advanced data in the paragraph 3.5.15  
Advanced data (common to machining)

See the use of menu parameters "Approach/Removal" in paragraph 3.5.14  
Approach/Retraction (shared by the machining operations)

### 3.5.10.4 Contouring technology

See the following chapters:

3.5.13 Technological data (shared by the machining operations)

3.5.15 Advanced data (common to machining)

3.5.16 Machine data (shared by the machining operations)

## 3.5.11 Emptying

In order to insert one or more empty workpieces you can:

- click on the command "Emptying"
- select one or more 2D elements

Or:

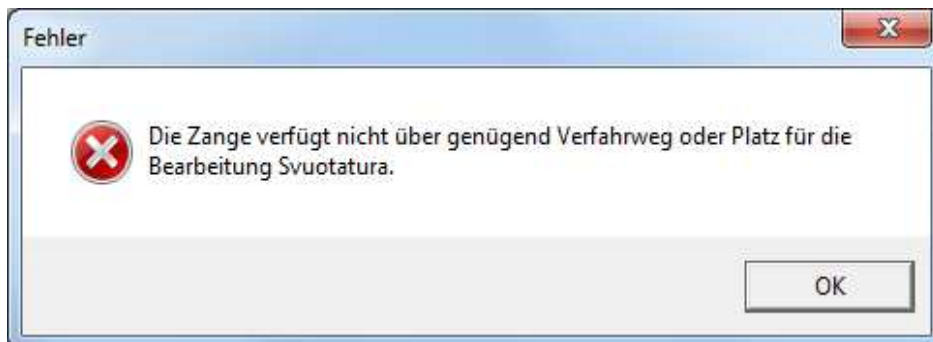
- select one or more 2D elements
- click on the command "Emptying"

The selection of 2D elements can be single or multiple since the option "Emptying" will carry out working processes associated with closed polylines or with circles.



For "gripper" machines (CyflexS and CyflexHP), embroidery work is carried out within the same stapling phase and is not automatically broken.

So if an emptying that is X in size X above the stroke X of the CyflexS clamp or the pitch between CyflexHP clamps is programmed, Master does not optimize the program by communicating this error:



In this case it is up to the operator to break the pockets in multiple overlapping pockets and reduced X size

### 3.5.11.1 Emptying position

The emptying position is automatically set by the selected geometry.

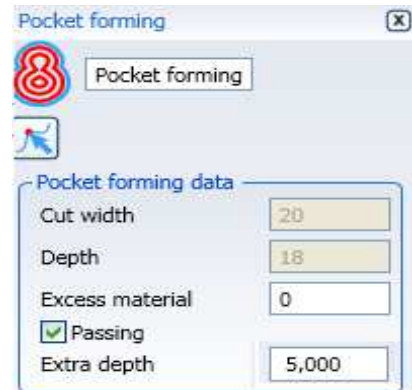
### 3.5.11.2 Emptying structure

In order to set the emptying structure, you have to set Length Cut and Depth in the menu.

The field Length cut is automatically filled when you associate a tool with the emptying.

In field "Depth" you will insert the value corresponding to the thickness of the workpiece in case the option "Passer-by" is selected.

The field "Extra depth" indicates the extent of the emptiness on the workpiece ( This heading will appear only if the option "Passer-by" has been selected).

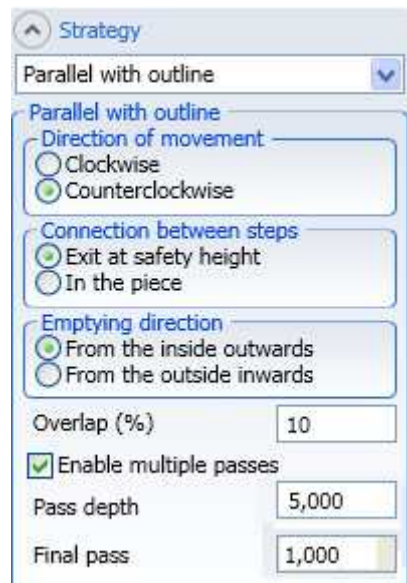


### 3.5.11.3 Emptying mode

In order to set how to perform the milling, you use the sub-menus "Strategy" and "Approach/Removal".

The strategy "Parallel to the contour" aims at making a set of polylines to a certain offset from the selected polyline. The space among polylines is calculated according to the tool-diameter and the parameter Overload. (%).

For a multi-rub management, see paragraph 3.5.4.3 (milling modes)



See the use of menu parameters "Approach/Removal" in paragraph 3.5.14 Approach/Retraction (shared by the machining operations)

### 3.5.11.4 Emptying technology

See the following chapters:

3.5.13 Technological data (shared by the machining operations)

3.5.15 Advanced data (common to machining)

3.5.16 Machine data (shared by the machining operations)

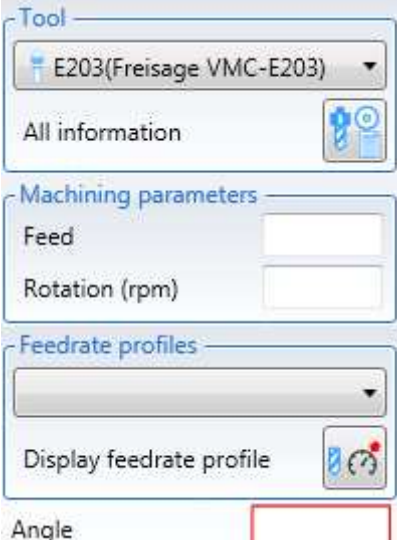
### 3.5.12 Work on the lower face

You must have a Weldon head to perform work on the lower part of the workpiece. The head should be configured correctly following the instructions in the chapter 10.9.1.7.1.

Using the Weldon head requires you to program the machining exclusively on the lower face of the workpiece.

When working with the Weldon head, an "Angle" field will appear, indicating the orientation (around the Z axis) that will have to be headed during machining.

This angle will be calculated beforehand by Master who will achieve the approach to machining from the side closer to the machining itself.



The screenshot shows a software interface with several sections:

- Tool:** A dropdown menu showing "E203(Freisage VMC-E203)". Below it is an "All information" button with a gear icon.
- Machining parameters:** Two input fields labeled "Feed" and "Rotation (rpm)".
- Feedrate profiles:** A dropdown menu and a "Display feedrate profile" button with a refresh icon.
- Angle:** A text label followed by an empty input field with a red border.

### 3.5.13 Technological data (shared by the machining operations)

This chapter describes those parameters common to most machinings in order to group the description.

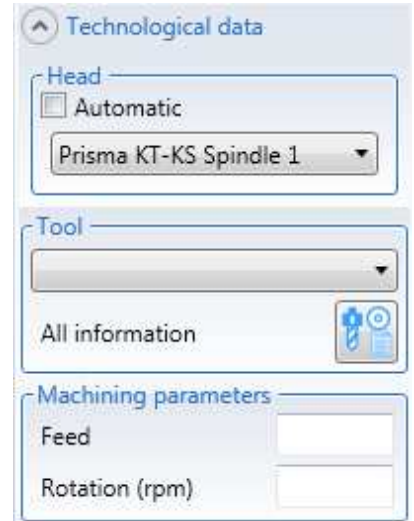
The "Technological Data" session is used to:

Select the head for the machining.  
Selecting "Automatic" the system will determine which head to use for the machining.

Select the tool for the machining.

Set the feed speed during the machining.

Set the number of spindle RPMs.



For PowerFlex machines in the "Technological Data" section you can:

Select the head for the machining. Selecting "Automatic" the system will determine which head to use for the machining.

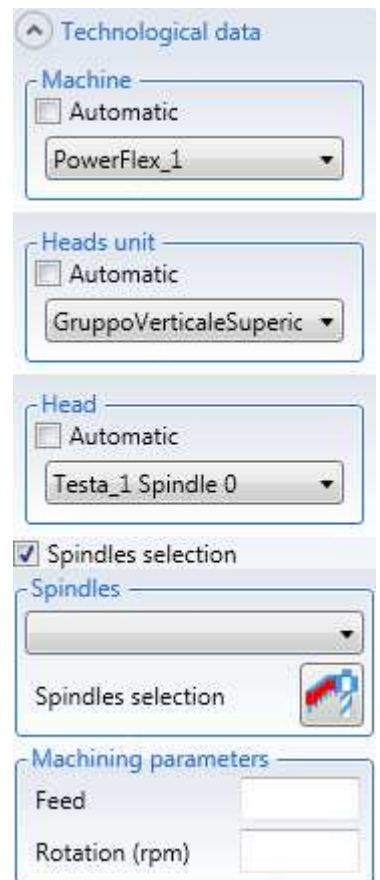
Select the Head Group you want to use to do the work. Selecting "Automatic" the system will determine which head to use for the machining. To select a heads group, you must select the machine with which to perform the machining.

Select the head for the machining. Selecting "Automatic" the system will determine which head to use for the machining. To select a head, select the group of heads with which to work.

Select the Spindle with which to do the work. To select a spindle, select a head and activate the "Spindle Selection"

Set the feed speed during the machining. Please note that the actual speed used in the machine may differ from the one shown here, depending on the machining couplings operated by the optimizer by the technology profiles associated with the machine.

Set the number of spindle RPMs.



### 3.5.14 Approach/Retraction (shared by the machining operations)

In the menu "Approach/Removal" you set the in-and-out mode from the milling path.

For the entry as well as for the exit, you can choose the line or arc geometry.

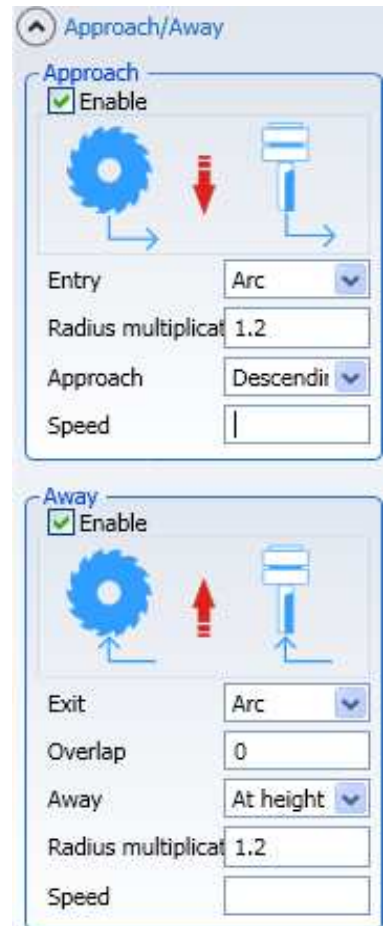
The length of these paths is set by "Radius multipl." that, in case of a milling cutter of 10 radius and a multiplier of 1.2, a length line of 12 will be added to our path or a radius ray of 12 with a width of 90° and tangent to the initial/final point or the 2D geometry.

These additional properties can be performed by the machine:

- in case of security level (on level)
- while the point of approach is reached (in fall)
- while the point of removal is reached (in rise)

The surmount is the workpiece worked along the first object of the polyline.

if this value is higher than the length of the first polyline element, the working process would end in the last workpiece of the element.





### 3.5.15 Advanced data (common to machinings)

This chapter describes those parameters common to most machinings in order to group the description.

The "Advanced Data" menu is used to:  
Invert the machining, starting from the final point of the geometry selected.

Enter a condition that determines whether the machining must or must not be executed.

Set the start point automatically at:  
- In the middle of the first section  
- In the middle of the longest section

Enter a comment.

Enter a "Safety quota" for the tool before executing the machining.

Set the execution priority.

This parameter allows you to group project machining while maintaining the technological sequence during optimization.

Optimization is done in these cases:

1. When you plan to machine 2 or more workpieces on the surfaces
2. When processing a workpiece on Cyflex machines

When machining multiple workpieces on the surface, the Master Trainer Optimizer may change the technology sequence.

To maintain the desired technological sequence, you must define the operational priority for all the jobs which need to remain related by inserting positive and increasing values. If a Priority value is lower than or equal to the previous one, the batch will be interrupted.

When working on the Cyflex machine, machining associated with increasing Priority values will be performed at the same stage.

Such method is due to accuracy problems with Cyflex machines when the piece is moved from one stage to the other.

Machinings consisting of more tool paths: for example, a counterboard hole made with two different tools might be imprecise if machining consists of two stages.

"Execution priority" mode requires the insertion of a growing value in all machinings that need to be performed in the same stage.

When the value is equal to zero or to another value that is lower than the last one entered, the series of machinings to be performed in the same stage is interrupted.

Advanced data

Reverse operation  
 Invert

Condition  
True

Automatic start point  
None

Comment

Safety quota 20

Execution priority 0

Example of pgmx

- Hole Execution priority = 1
- Hole Execution priority = 2
- Hole Execution priority = 5
- Hole Execution priority = 3
- Hole Execution priority = 4
- Hole Execution priority = 6

When this program is optimised, holes 1,2,5 shall be executed in the same stage, and holes 3,4,6 shall be executed in the same stage too although this is different from the first one.

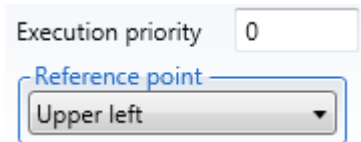
You want to prevent hole 1 from being in a different stage compared to holes 2 and 5.

For PowerFlex machines in the "Advanced Data" menu, you can:

Set the Processing Priority of the Processing with reference to the following rules:

- Priority 0: Workmanship is not priority over other workmanship, its execution can take place before, after, or at the same time as other workings, regardless of the priority value of the latter.
- Priorities 1 to N: Work must be done after all jobs have a priority value other than zero and lower than the one set for machining, but before all those with a value other than zero and higher than working priority value. If two jobs have the same priority value, this does not imply their execution at the same time, but only the use of the same sorting rule for both.

Set the "reference point" of the machining. This parameter allows you to specify the reference point that will be considered for machining machining machining offset after panel measurement. For each machining you can specify the reference point by choosing between the four edges of the workpiece or the center point of the workpiece.



If the user does not specify any reference, during the optimization phase the machining will be assigned the default reference for the work face. Below is the table of default references, used for PowerFlex machines with upper left stop of the workpiece:

Work face	Default reference
Upper	Upper left
Left	Lower left
Right	Lower right
Front	Lower left
Rear	Lower right
Lower	Lower left

### 3.5.16 Machine data (shared by the machining operations)

This chapter describes all the "Machine functions" that can be found in most machining operations to group the description.

The machine functions available depend on the machine configuration, therefore instead of describing them together they will be described individually.

This session will probably have a certain amount of dynamism, as it manages devices in the machine that in the future may be removed or new ones may be added.

The machine functions currently available are:

- 1) Jerk
- 2) Jerk3D
- 3) Main hood
- 4) Additional hood
- 5) Rotating axes brakes
- 6) 5-axes head unrolling
- 7) Electronic feeler
- 8) NC speed adjustment
- 9) Tool blower
- 10) Mechanical hold-down

#### 3.5.16.1 Jerk

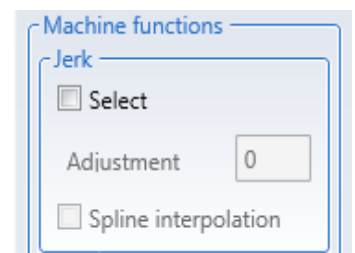
This parameter affects the machine behaviour during the bi-dimensional machining operations.

The "Adjustment" parameter is set with a value that is significant from -2 to +5.

With the -2 value the machine will be slow but the machined pieces will have a good finish.

With the 5 value the machine will be fast but the machined pieces will not have a good finish.

Selecting the "Spline interpolation" parameter the finish will improve when the machining has been described with some "Splines" as the edge will be rounded.



### 3.5.16.2 Jerk3D

This parameter affects the machine behaviour during the three-dimensional machining operations.

One of the items from the drop down menu at the side can be selected.

Selecting "Roughing" the finish will be less accurate

Selecting one of the 3 Finishes the finish will improve.

Generally, as various factors can affect the finish, it is advisable to perform tests in the machine until the desired finish is achieved.

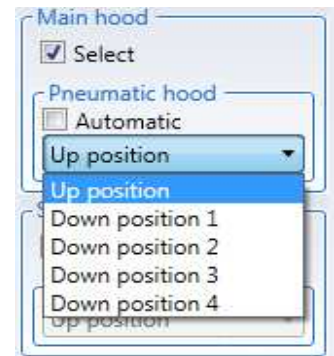


### 3.5.16.3 Main hood

This parameter is used to position the main hood of the machining unit in 5 positions that can be selected from the drop down menu at the side.

Selecting the "Automatic" option the hood will position itself so that it skims the surface of the piece.

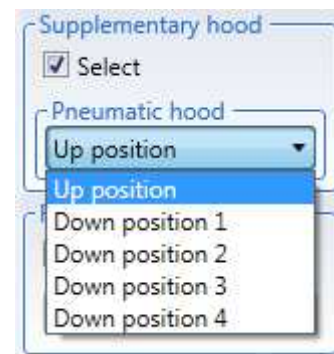
It will position itself at a value which is the tool length minus the useful length.



### 3.5.16.4 Additional hood

This parameter is used to position the additional hood of the machining unit in 5 or 8 positions that can be selected from the drop down menu at the side.

The number of positions displayed depends on the machine configuration.

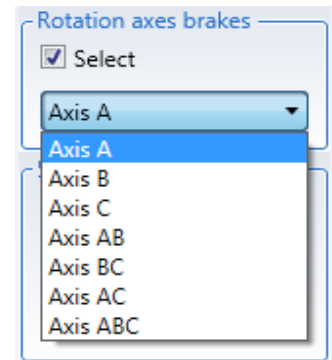


### 3.5.16.5 Rotating axes brakes

The braking of the rotating axes is used in order to improve the finish of the machined piece.

Braking the rotating axes reduces the vibrations due to the fact that the motor maintains the rotating axis constantly in position.

The axes that can be braked can be selected from the drop down menu and depend on the machine configuration.

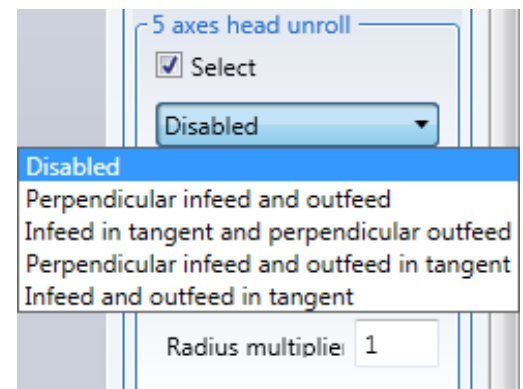


### 3.5.16.6 5-axes head unrolling

This function is used when wanting to continue machining rotating always clockwise or counterclockwise with the machining unit.

The machine, in order to execute the machining requested once it reaches the axis limit switch, must reposition the axis on the opposite limit switch. to do this:

- the tool must be removed from the piece
- the head rises in Z
- the axis goes to the opposite limit switch (in jargon it "unrolls")
- the head drops back down in Z
- the tool must return to the machining.



The mode in which the tool will restart the machining can be selected in the drop down menu.

For entries in "Tangency" there will be an arc entry whose radius will be equal to the tool radius multiplied by the value set in the "Multiplier" field.

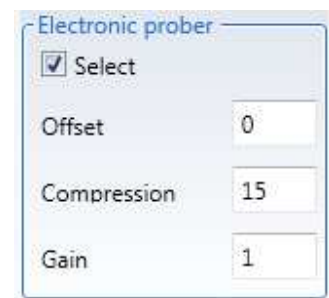
### 3.5.16.7 Electronic feeler

The electronic feeler is a device that changes the tool Z position based on the irregularities of the panel being machined.

The management is performed with the following parameters:

- Offset is the distance between tool and feeler
- Compression refers to the compression of the feeler springs
- Gain is a factor that combined with the feed speed delays the Z movements of the tool in relation to when the feeler has found a change in the Z position of the piece.

The compression and gain values can be set after carrying a few tests on the pieces being machined.

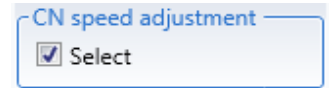


### 3.5.16.8 NC speed adjustment

The NC speed adjustment, also referred to as "Adaptive cut" allows the NC to change the feed speed when an excessive cutting stress is detected.

This method could result in non-homogeneous finishes on the piece.

To enable this function click the "Select" field.



### 3.5.16.9 Tool blower

The following types of blowing options can be enabled:

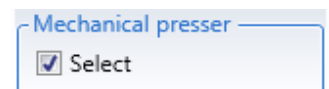
- Air (air is blown on the tool)
- Air and oil (air and oil are blown on the tool)
- Head (a pneumatic option is enabled on the head)
- Air tank (air is blown on the tool from the lubricant/coolant)
- Air and oil tank (air and oil are blown on the tool from the lubricant/coolant)



### 3.5.16.10 Mechanical hold-down

The mechanical hold-down presses the panel on the machine table to guarantee a precise machining operation.

To enable this function click the "Select" field.



### 3.5.17 Edge banding process

Edge banding a panel is a process that includes the following operations:

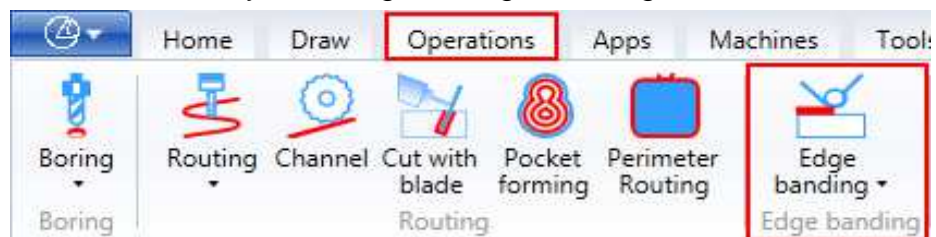
- Edge banding
- End trimming
- Trimming
- Edge scraping
- Glue scraping
- Shaving

The edge banding process consists of applying the edge to a panel, whilst the end trimming, trimming, scraping (edge/glue) and shaving operations all complete the process, but are only found with the edge banding process.

#### 3.5.17.1 Edge banding

The edge banding operation applies an edge to the panel contour.

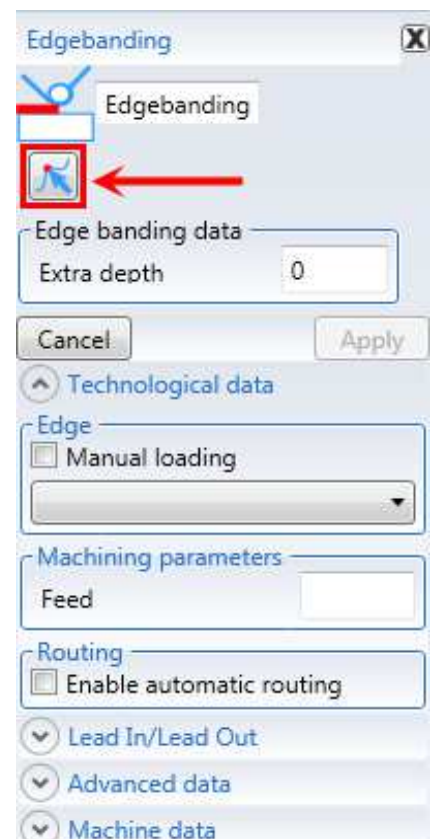
The edge banding process is created by selecting the Edge banding button in the "Operations" menu.



Clicking on the button opens the Edge banding window:

Press the Select Geometry button in the window to select a geometry on which to apply the machining. Please note that the geometries must lie on the upper plane of the panel and the edge banding cannot be applied to a point.

Pressing the select geometry button also activates the "Select Machining Start Point and End Point" command, which is used to select the start and end point of the edge banding. You can also specify the routing parameters that can be applied before the edge banding automatically.

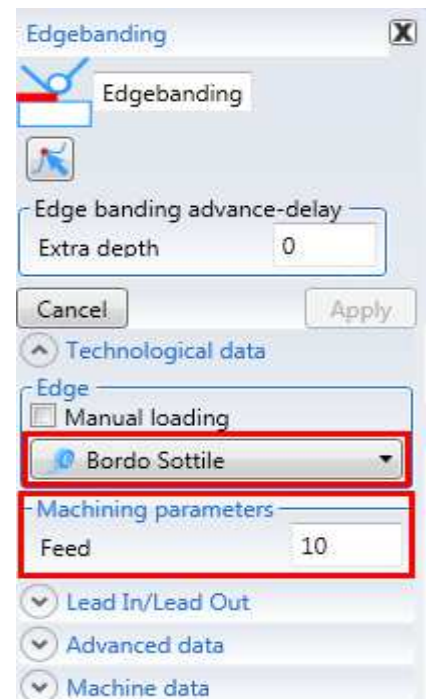




### 3.5.17.1.1 Technological Data

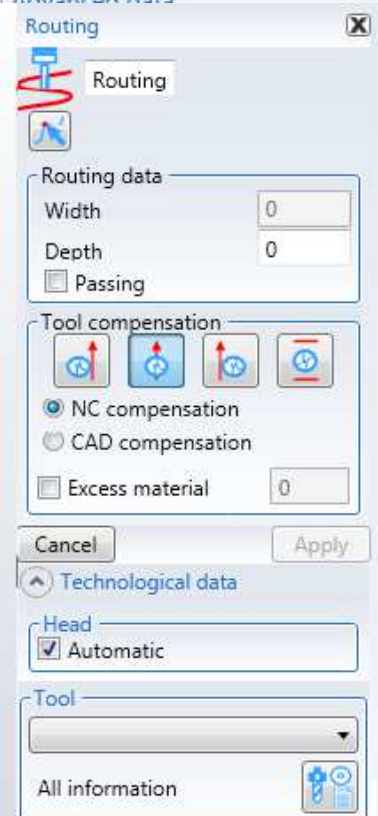
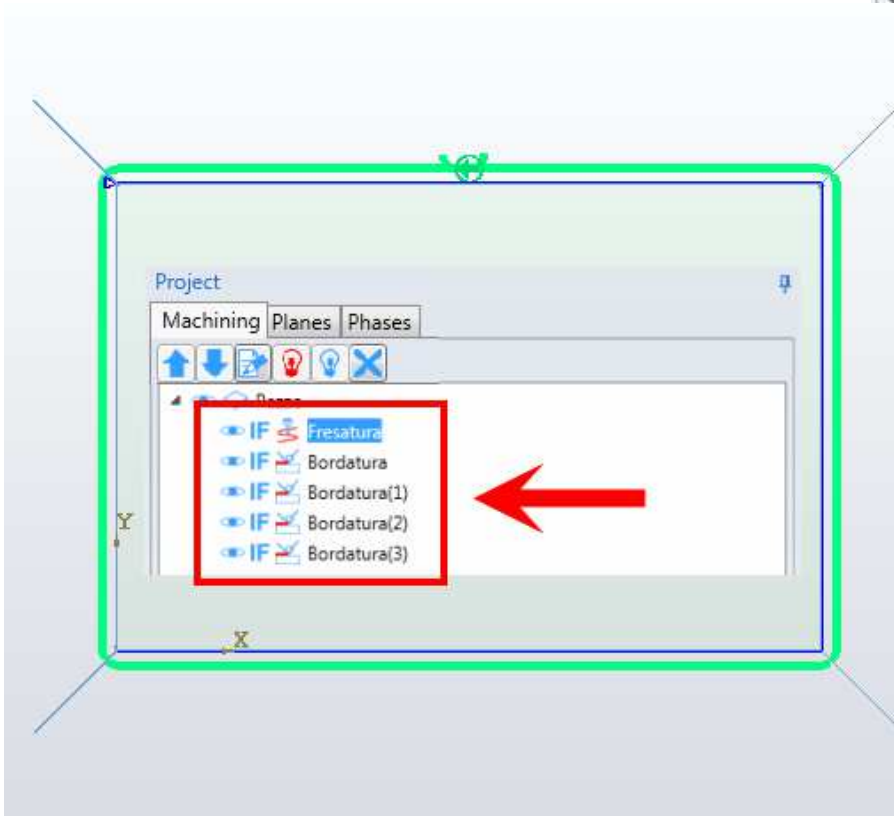
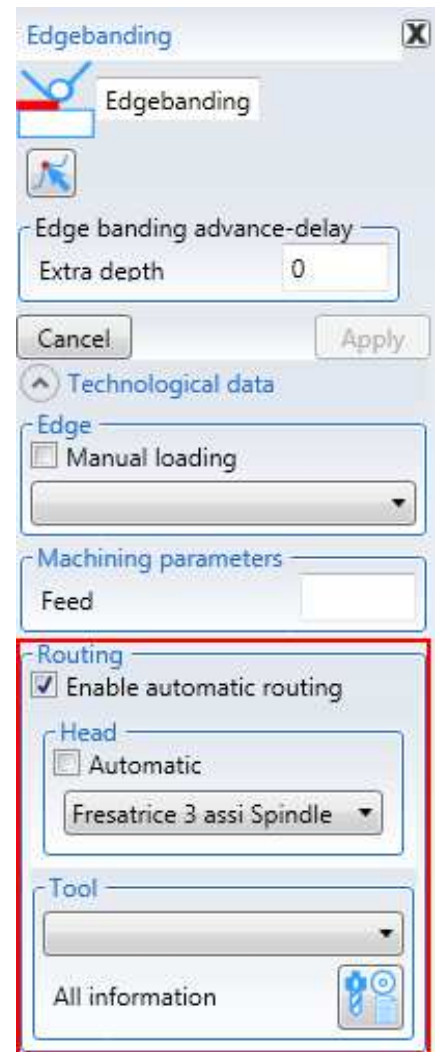
The edge banding technological data define the following:

- the type of edge banding; whether the edge is loaded automatically or manually
- the type of edge to apply; when an edge is selected, its machining parameters and the feed speed in particular, will be displayed automatically.

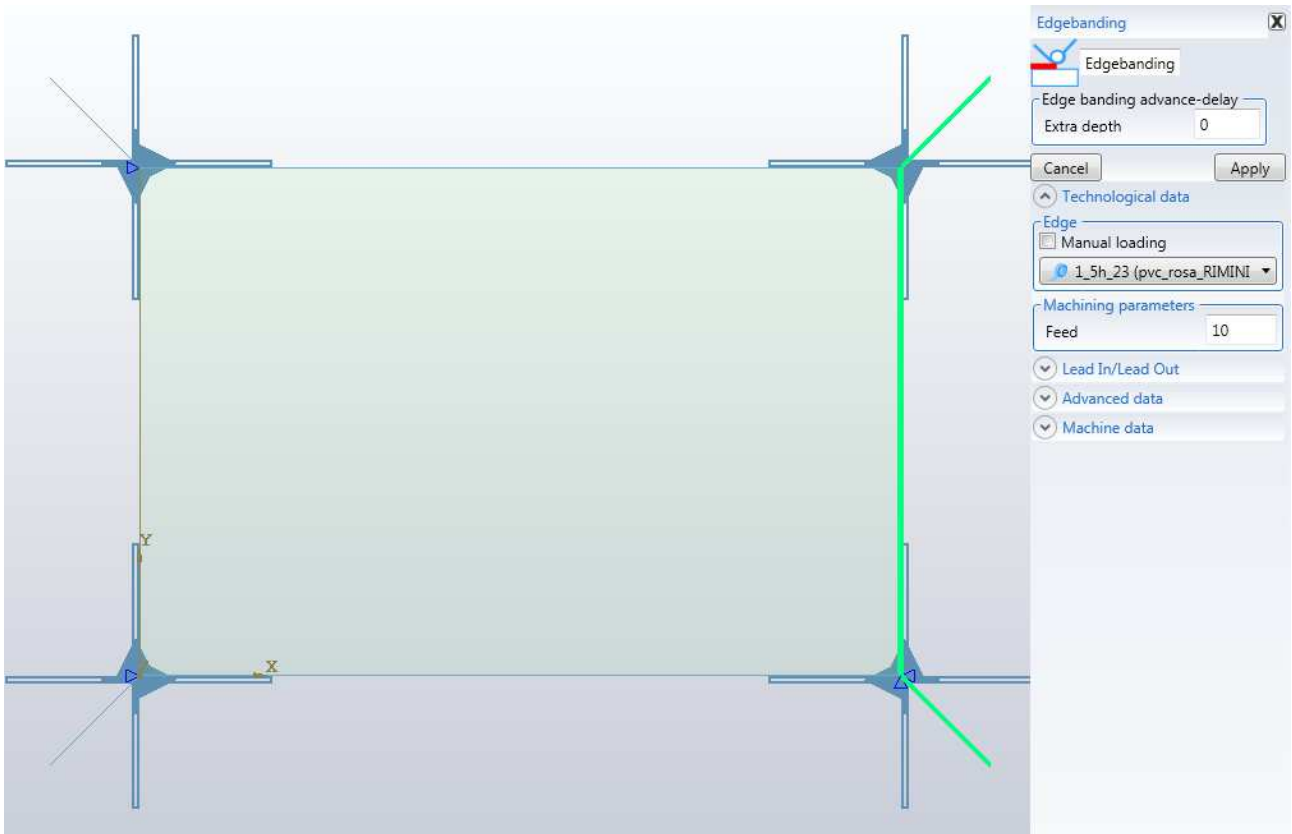




Enabling the routing in automatic with the relative router parameters, two types of machining operations will be generated: one routing and one edge banding, totally separate so that they can easily be changed in the future.



Selecting an edge banding operation in the machining tree creates the panel displaying the machining data.



### 3.5.17.1.2 Approach/Retraction Data

The approach data are:

- Speed from magazine to first approach point
- Speed along approach section
- Speed in last approach section, when the head enters with an arc on the trajectory.

Downstroke point:

- Coordinate polars of down point: distance and angle.
- Edge banding head inclination in the down section

Approach point:

- Coordinate polars of entry point: distance and angle.
- Edge banding head inclination in the entry section

The retraction data are:

- Speed along retraction section
- Stop time: any stop time in the last retraction section.

Closure point:

- Coordinate polars of exit point: distance and angle.
- Edge banding head inclination in the exit section

Lead In/Lead Out	
Lead In	Lead Out
Speed	20
Magazine approach speed	20
Stop time	0
Downstroke point	
Length	0
Angle	0
Rotation (degrees)	-40
Approach point	
Length	100
Angle	45
Rotation (degrees)	-40

Lead In/Lead Out	
Lead In	Lead Out
Speed	10
Stop time	0
Closure point	
Length	100
Angle	45
Rotation (degrees)	-45

### 3.5.17.1.3 Edge banding advanced data

This data is found in the technical specifications of the edge assigned to the edge banding.

#### Edge Dispensing

- Infeed edge: quantity of edge to insert in infeed to the profile.
- Total Edge: extra quantity of edge to insert in relation to the profile calculation
- Safety edge: quantity of edge dispensed to avoid having the edge in tension during the unwinding

#### Closure data

- Cut correction: quantity of edge in excess in relation to the trajectory being edge banded.

#### End trimmer

- if selected it enables the edge cut by the edge banded end trimmer blade (always active for closed edge banding)

#### Multi edge banding

- if selected it assigns the multi edge banding state to the edge banding (with a previous edge banding operation)

#### Lamps

Flat that enables the edge heating lamps in the approach section

Advanced data

Edging dispensing

In infeed	0
On profile	0
Safety	0

Closure

Cut correction 0

End trimmer

Enable

Multi edgebanding

Enable

Condition

True

Comment

### 3.5.17.1.4 Machine Data

#### Machine functions:

- Autostart: if enabled it can be Active or the Stand-by function after machining.

#### Head rotation direction, which can be:

- Optimized
- Counterclockwise
- Clockwise

The selection must be made based on the type of machining being carried out, the type of approach and the type of geometry being edge banded.

Machine data

Machine functions

AutoStart

On

Head rotation

Select

Optimised

Optimised

Counterclockwise

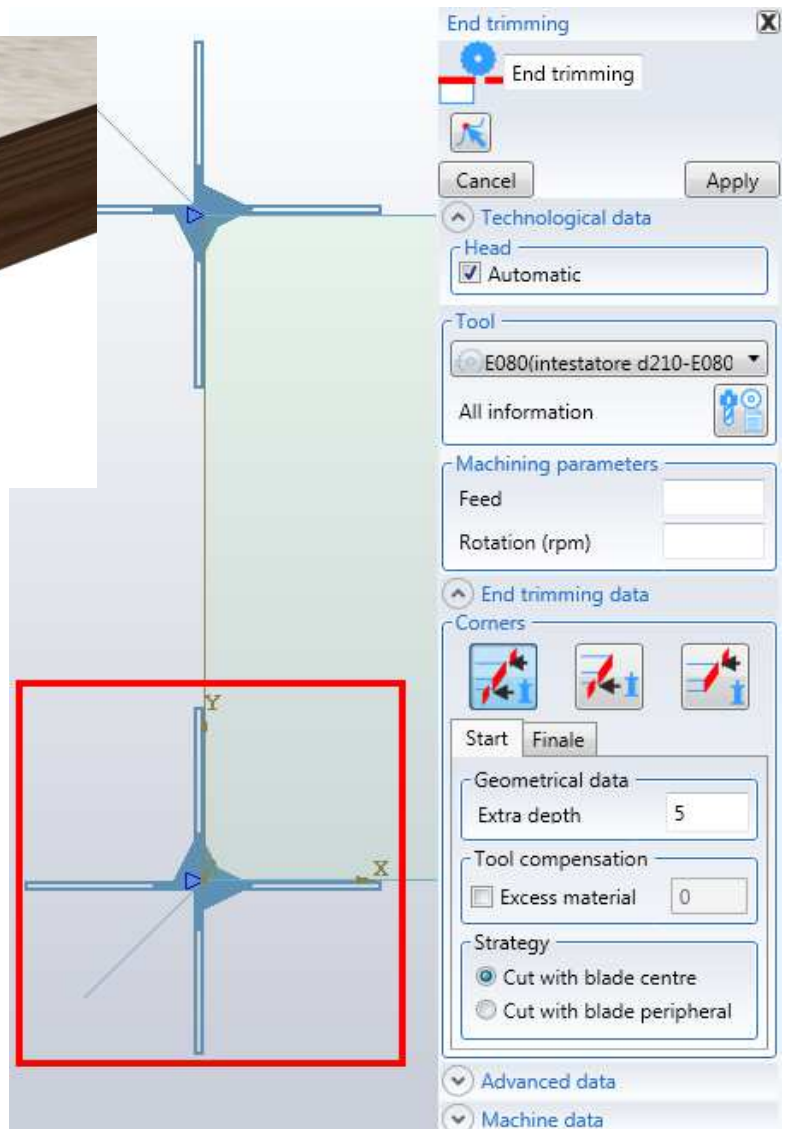
Clockwise

### 3.5.17.2 End trimming

The end trimming operation is linked to the edge banding operation on open profiles. End trimming is not available on other operations.

End trimming is a cut with a blade performed with a special tool called end trimmer, which removes the excess edge from the panel.

Selecting an end trimming operation in the machining tree creates the panel displaying the machining data.



#### End trimming data

- X, Y coordinates of the end trimming point
- Extra depth of the machining
- Angle: end trimmer blade angle

#### Tool correction

- Machining allowance: machining movement in relation to the point given by the end trimming coordinates.

Strategy: this is the machining mode

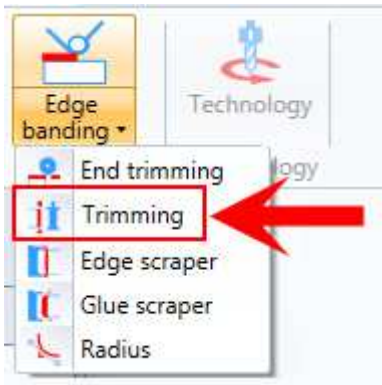
- Cut with Blade Centre
- Cut with Blade Periphery

- Advanced Data

- Safety quota: machining safety quota.

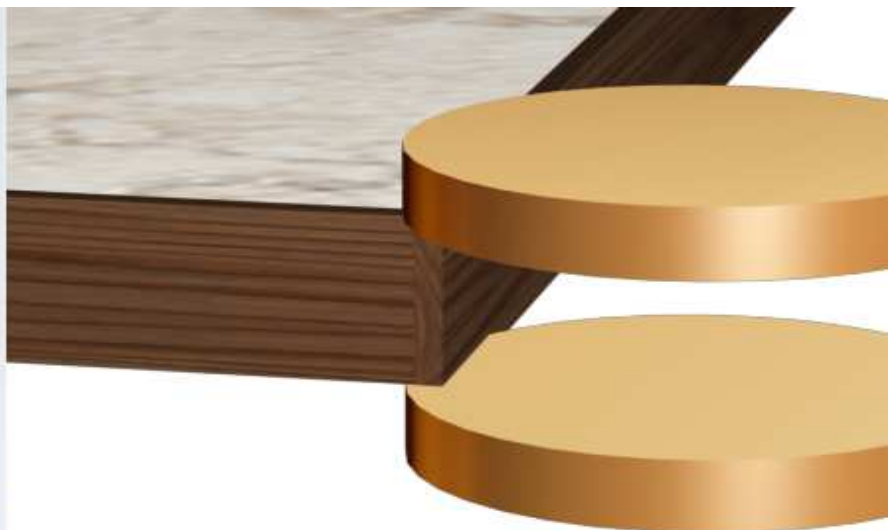
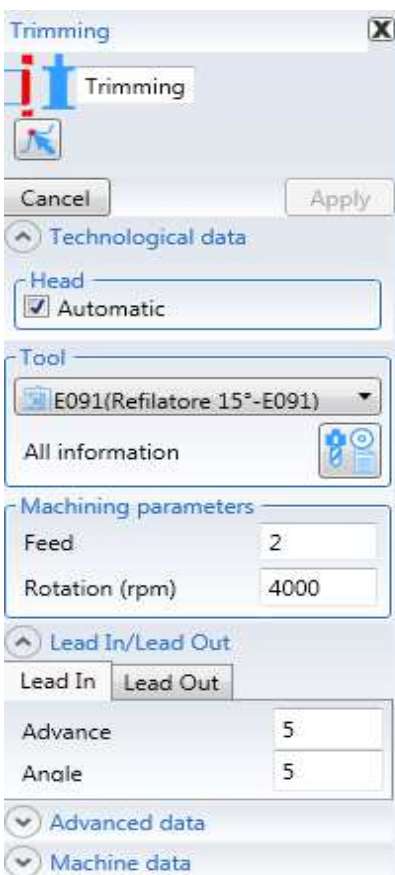


### 3.5.17.3 Trimming

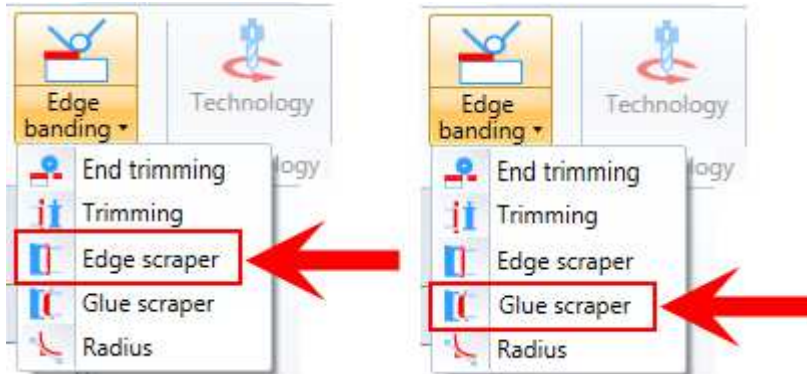


The trimming, performed with the trimming tool, removes the excess edge from the profile. The trimming operation is linked to the edge banding operation. Trimming is not available on other operations.

Selecting a trimming operation in the machining tree creates the panel displaying the machining data.



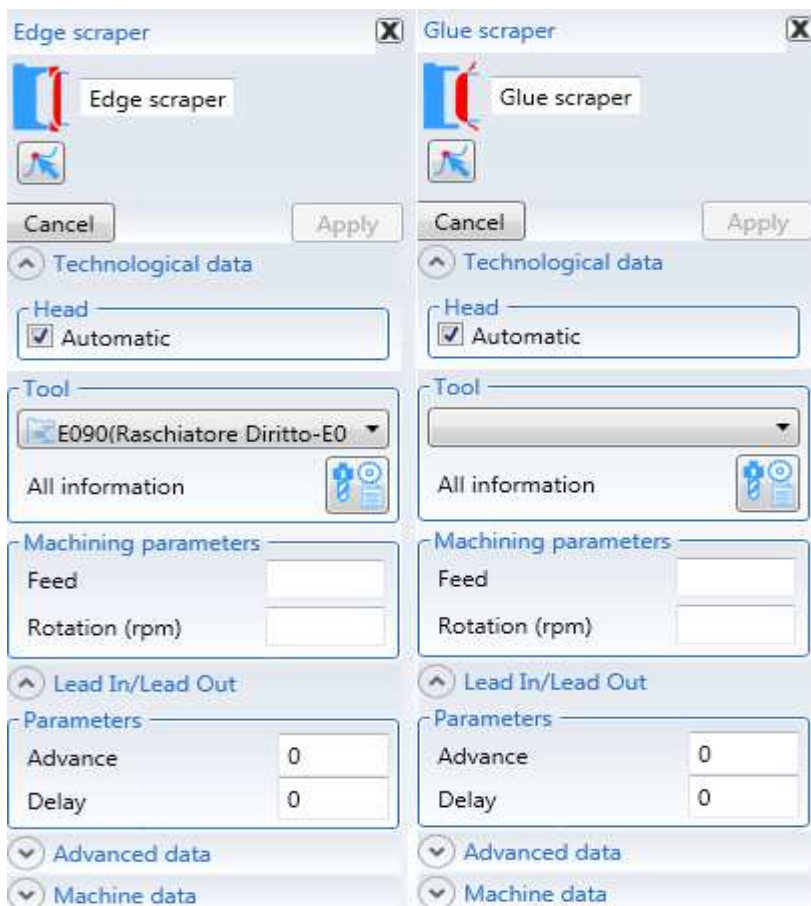
### 3.5.17.4 Scraping



Scraping, performed with the scraping tool, scrapes the excess glue or edge from the profile after the trimming operation.

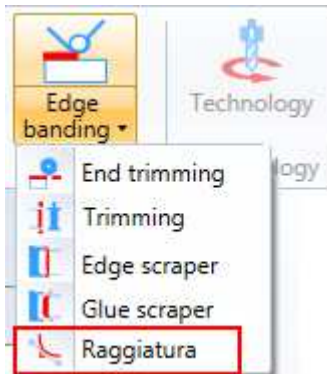
The scraping operation is linked to the edge banding operation. Scraping is not available on other operations.

Selecting a scraping operation in the machining tree creates the panel displaying the machining data.

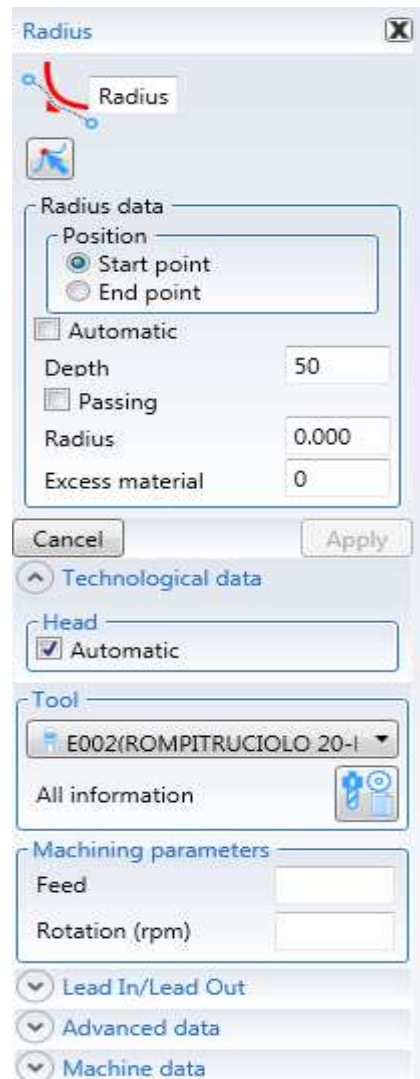
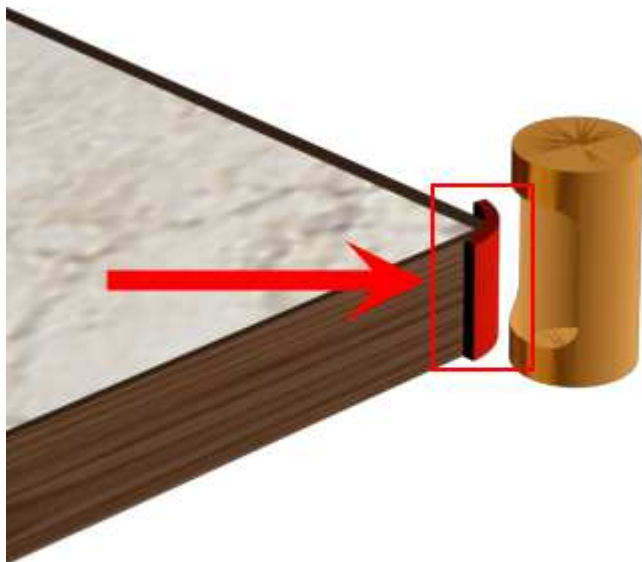




### 3.5.17.5 Shaving

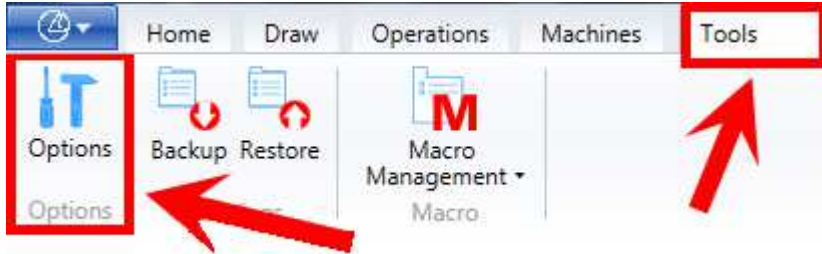


Shaving is linked to the edge banding operation and it removes any irregularities from an edge at the external angles of an edge banded panel. It is performed with a double diameter cutter and it is in short a routing operation and the technological information of this machining operation are the same as those for general routing.

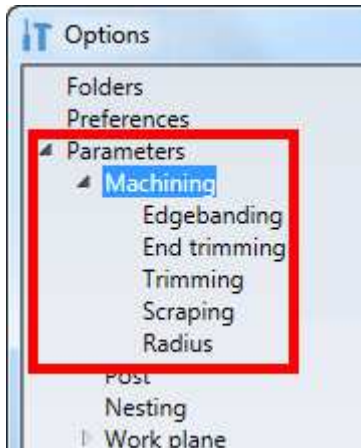


### 3.5.17.6 Edge Banding default settings

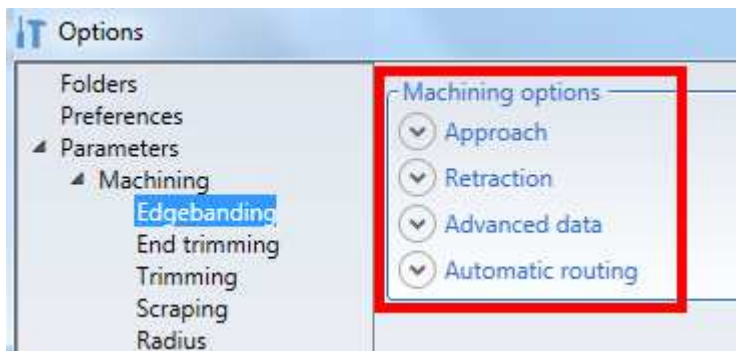
The default settings can be set for all the edge banding operations. Go to: tools, options.



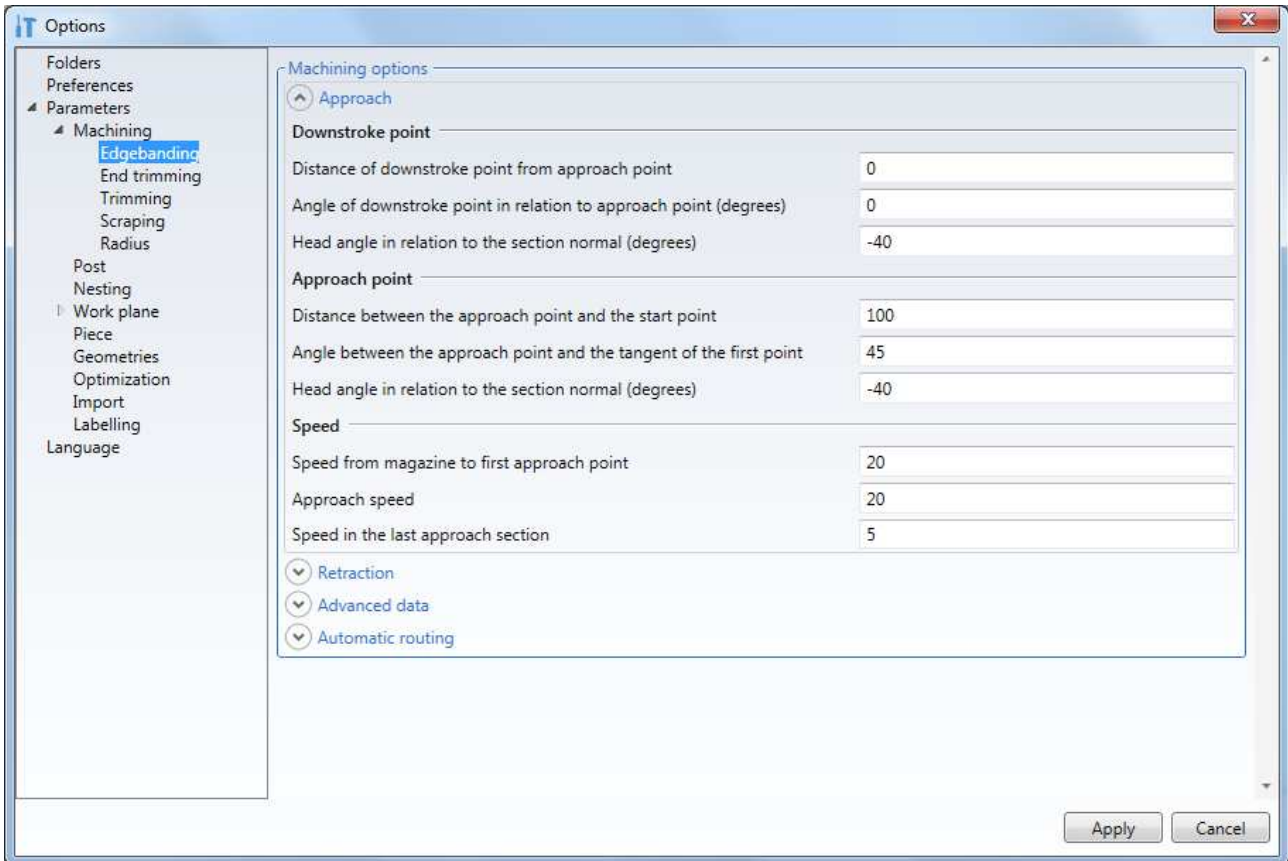
Select parameters, machining to display all the edge banding machining operations and the properties for each operation that can be entered as default and that will be displayed automatically during the edge banding machining.



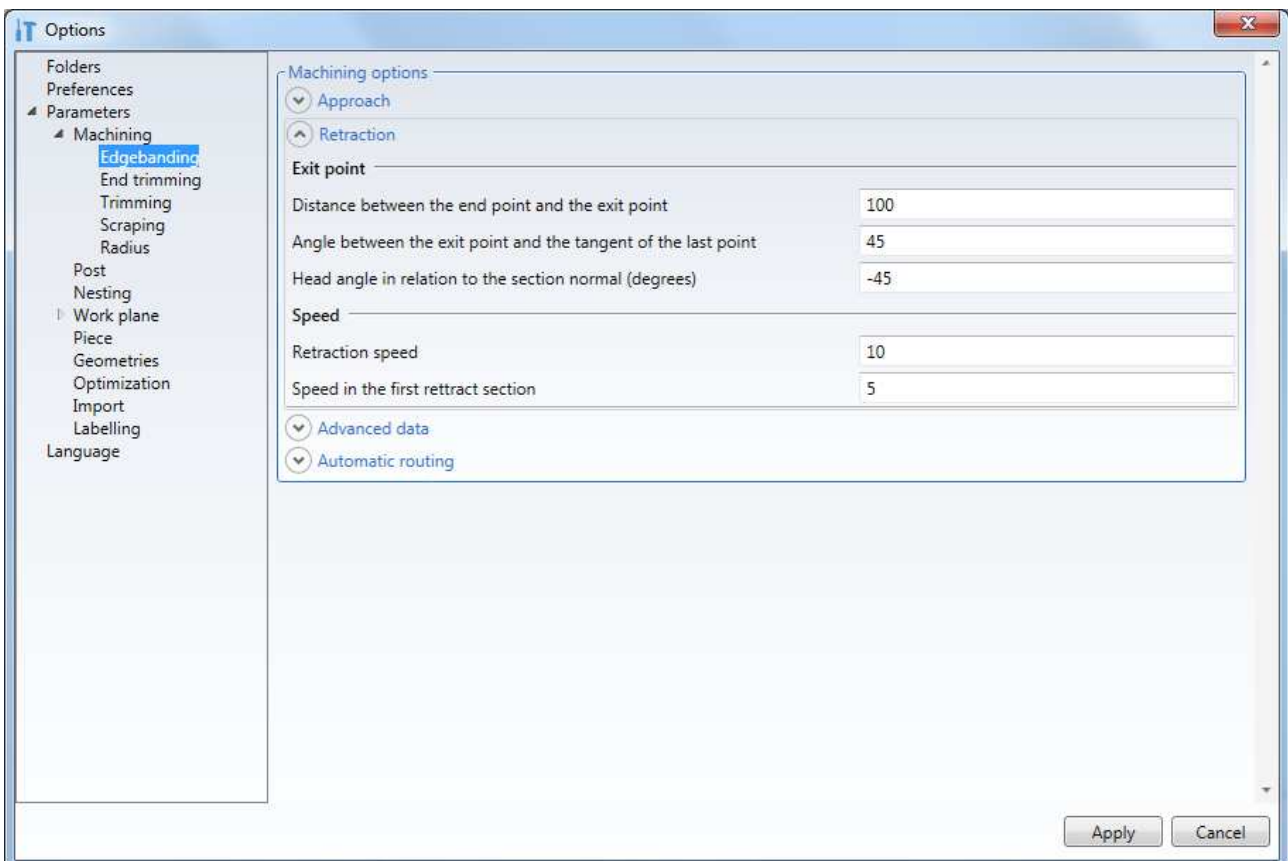
#### 3.5.17.6.1 Edge banding Default



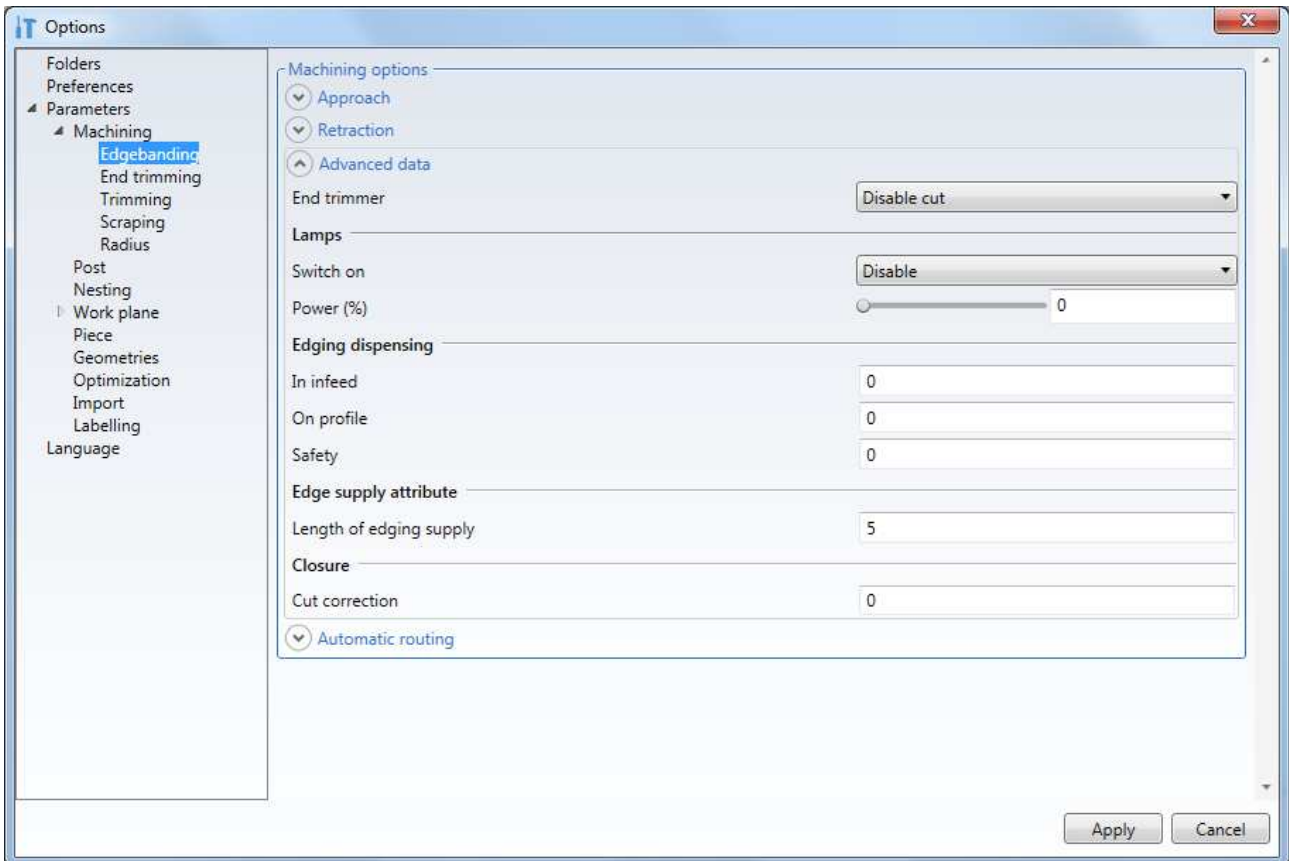
## Edge banding approach



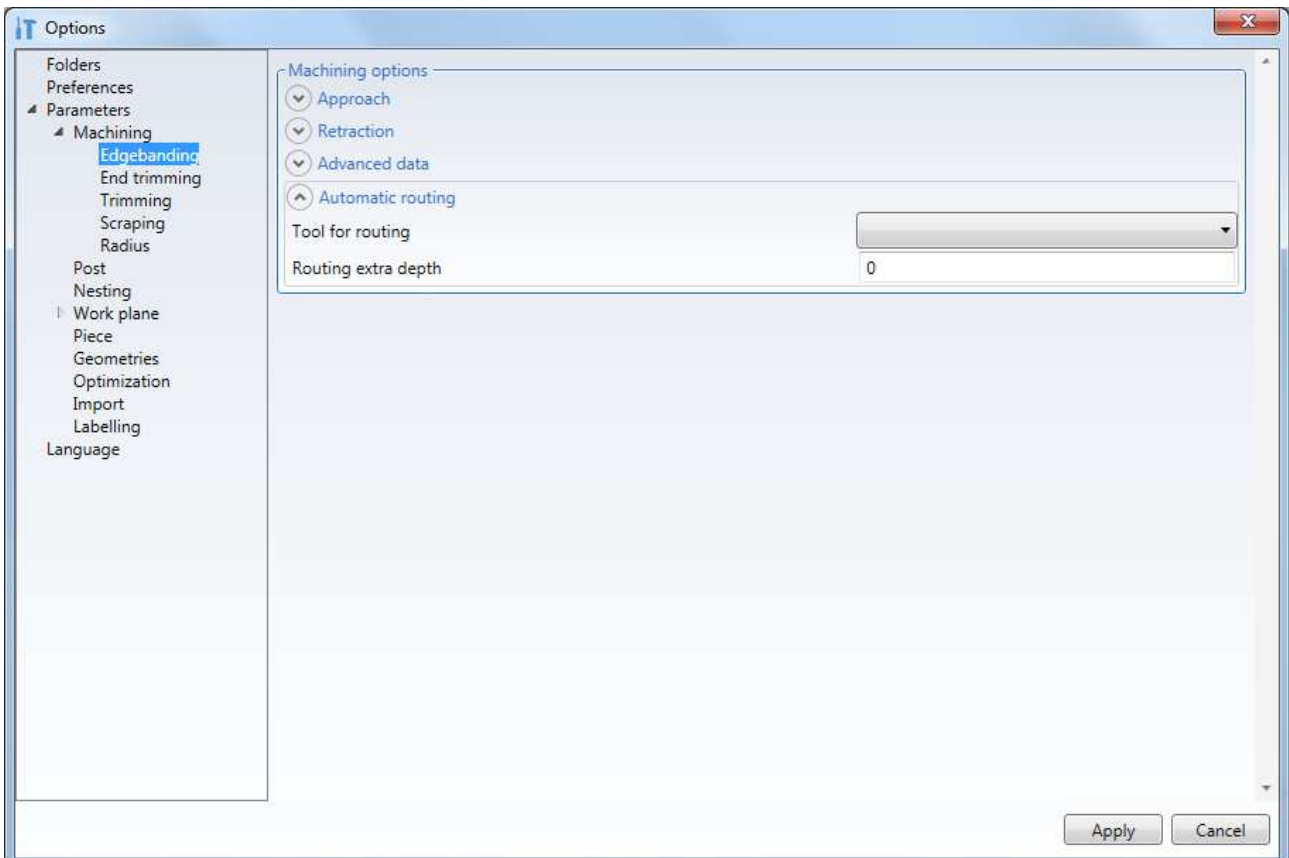
## Edge banding retraction



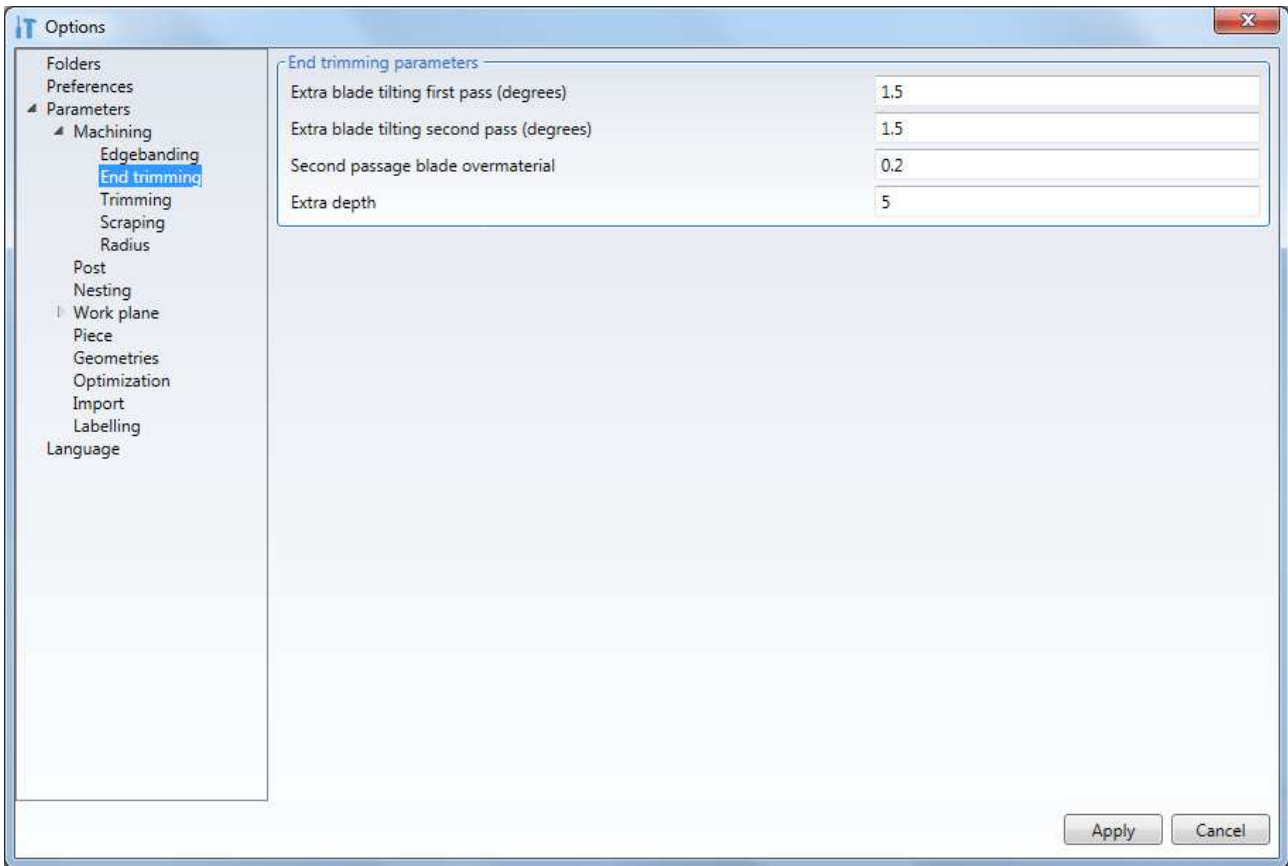
## Edge banding advanced data



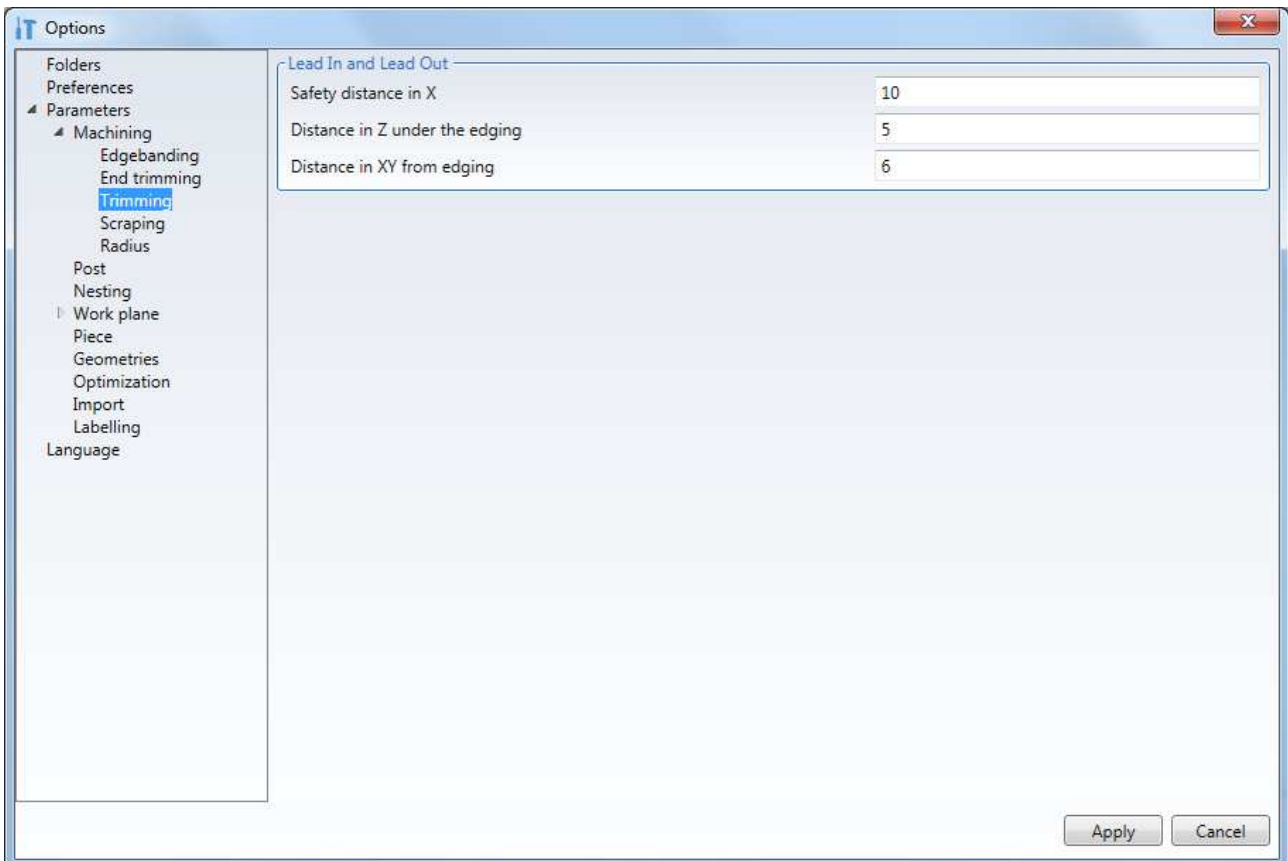
## Edge banding automatic routing



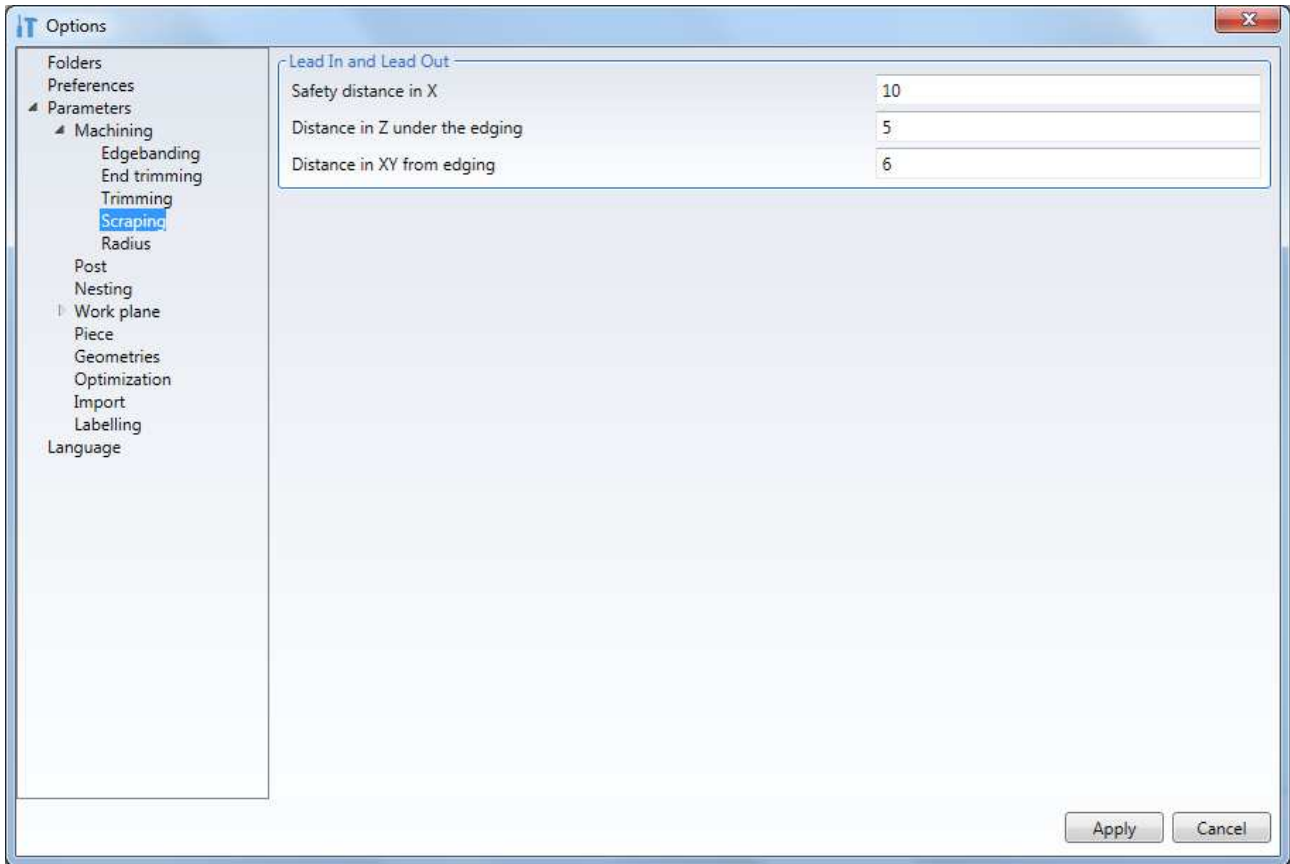
### 3.5.17.6.2 End trimming default



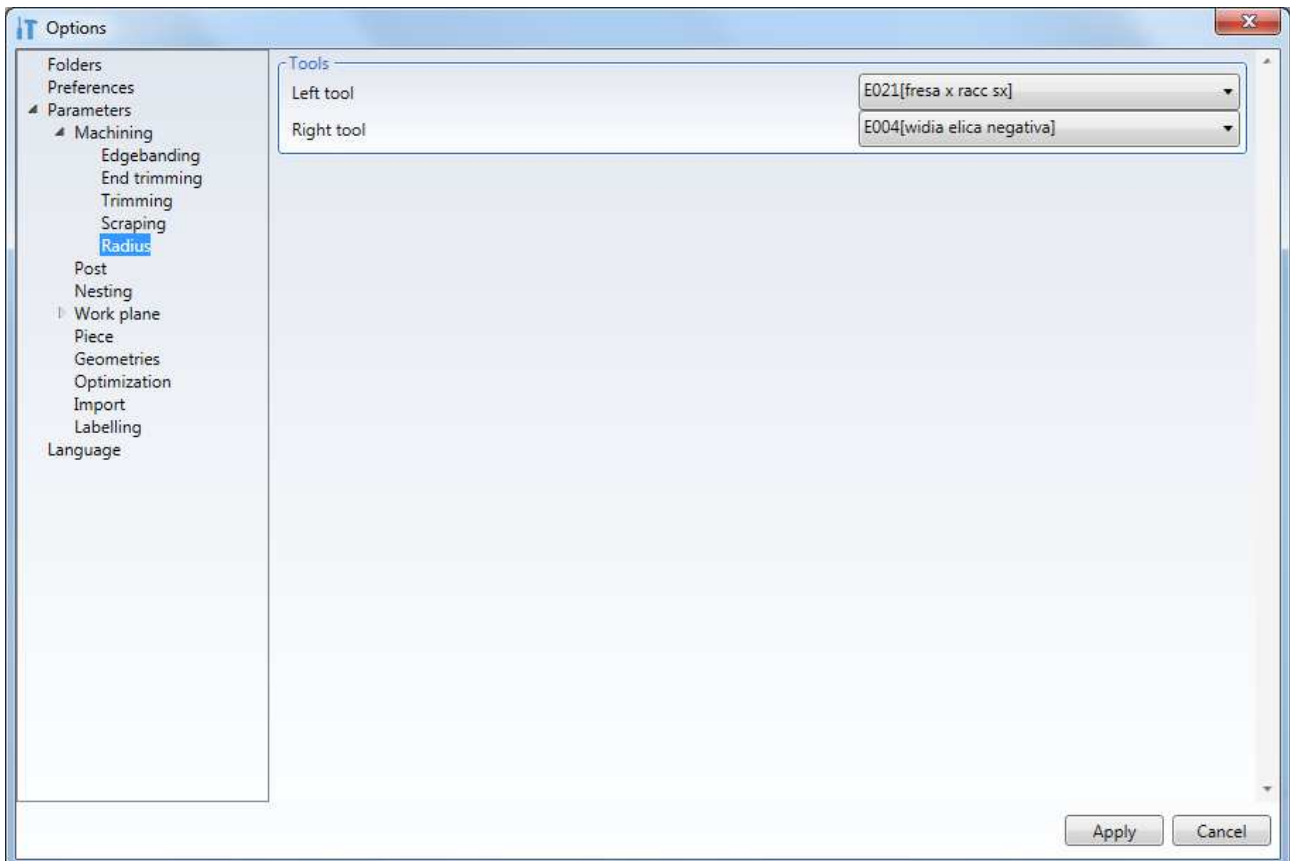
### 3.5.17.6.3 Trimming default



### 3.5.17.6.4 Scraping default



### 3.5.17.6.5 Shaving default



### 3.5.18 Attributes

Various attributes can be assigned to the machining operation created. The attributes are assigned in the menu of the same name:



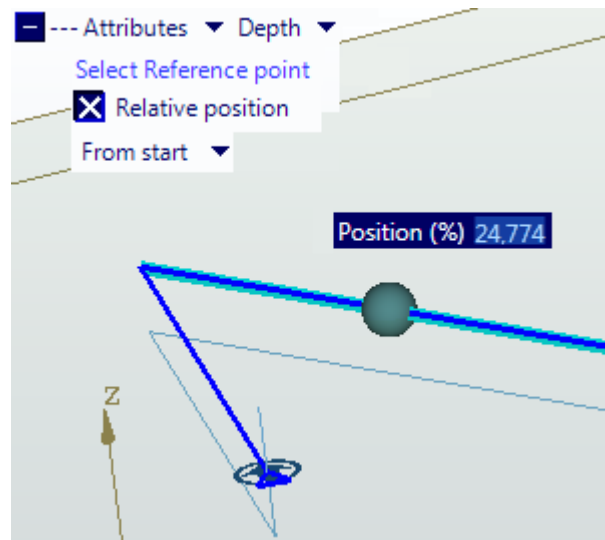
#### 3.5.18.1 Depth

The depth is used to change the machining depth along a section of machining.

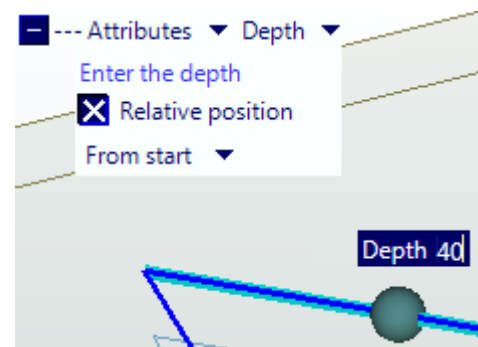


Select the Depth icon and then select the machining geometry for which the machining depth must be changed. For composite lines it is important to select the exact geometry section where the depth attribute must be entered.

Then select the point along the section selected. The positioning can be done by moving the mouse or by entering the section percentage.

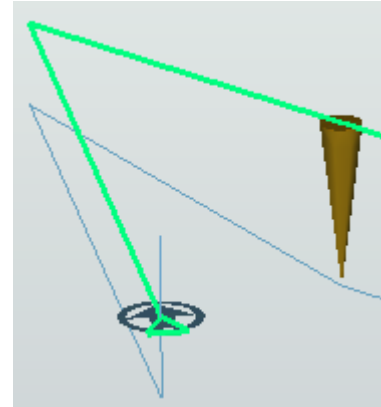


Then enter the depth value.

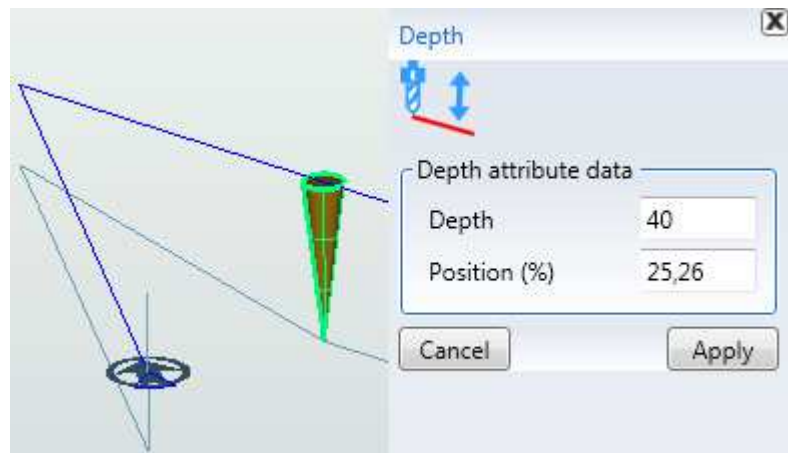


A cone symbol, that represents a machining depth change, will be displayed.

For composite lines the depth will remain constant with the machining depth for all the geometry sections before the section containing the attribute. Then the depth will change until the point with the attribute and then it will remain constant until the end of the composite line or until the next depth attribute.

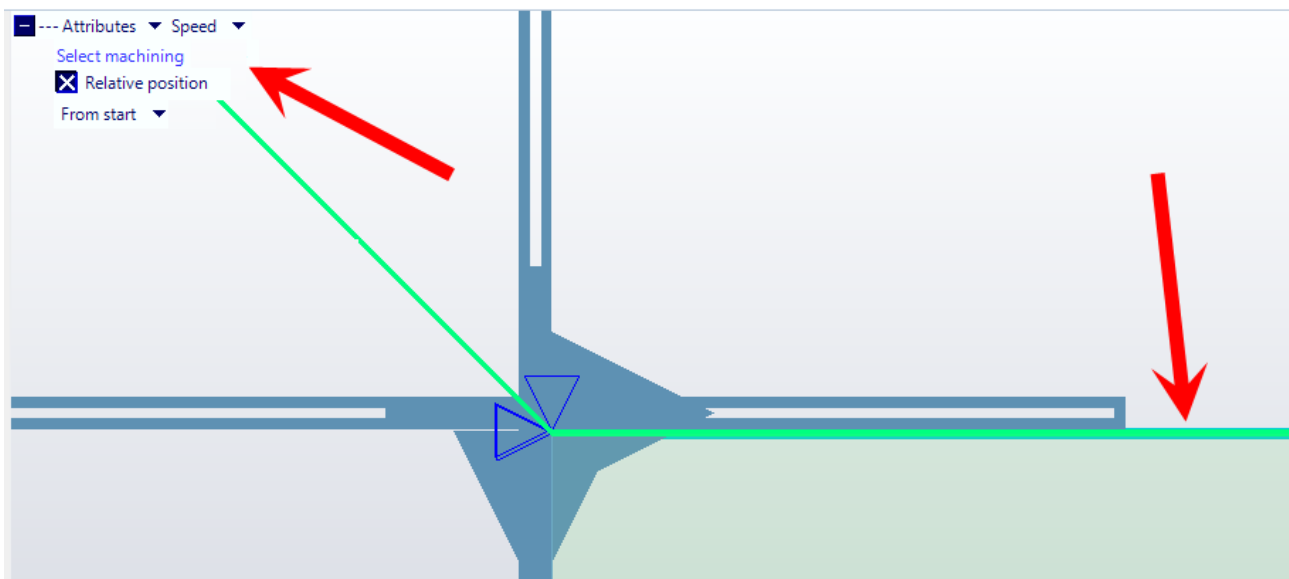


To change the depth attribute parameters again simply select the cone symbol and change the Depth or Position values that will be displayed in the window.



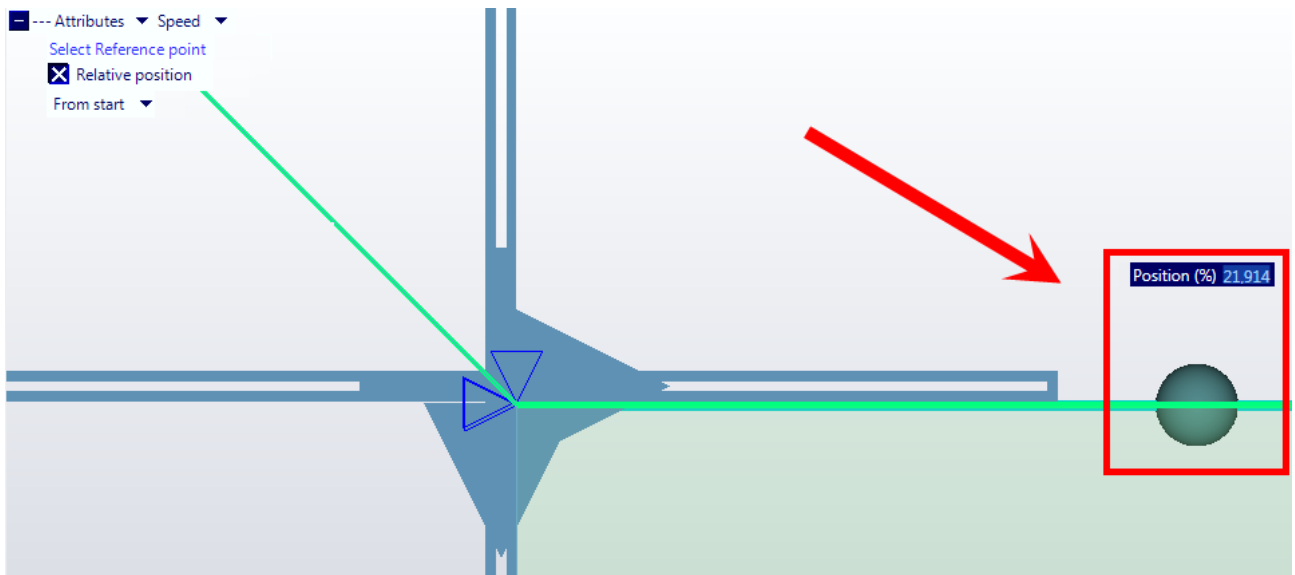
### 3.5.18.2 Speed

Select the Speed icon and then select the machining geometry for which the machining speed must be changed for just a section or completely.

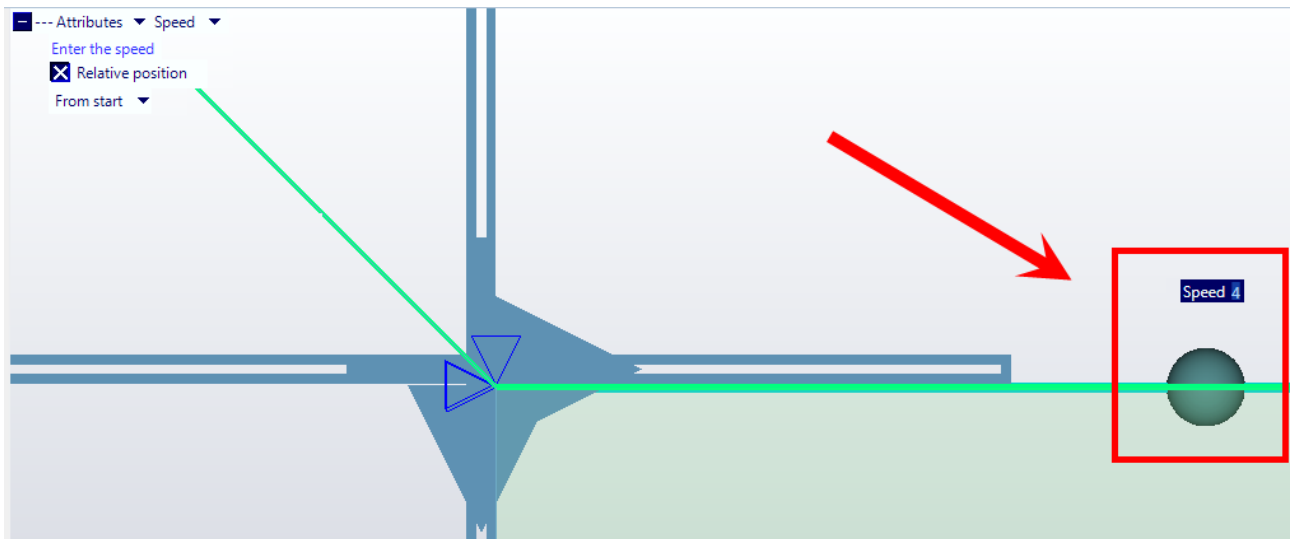




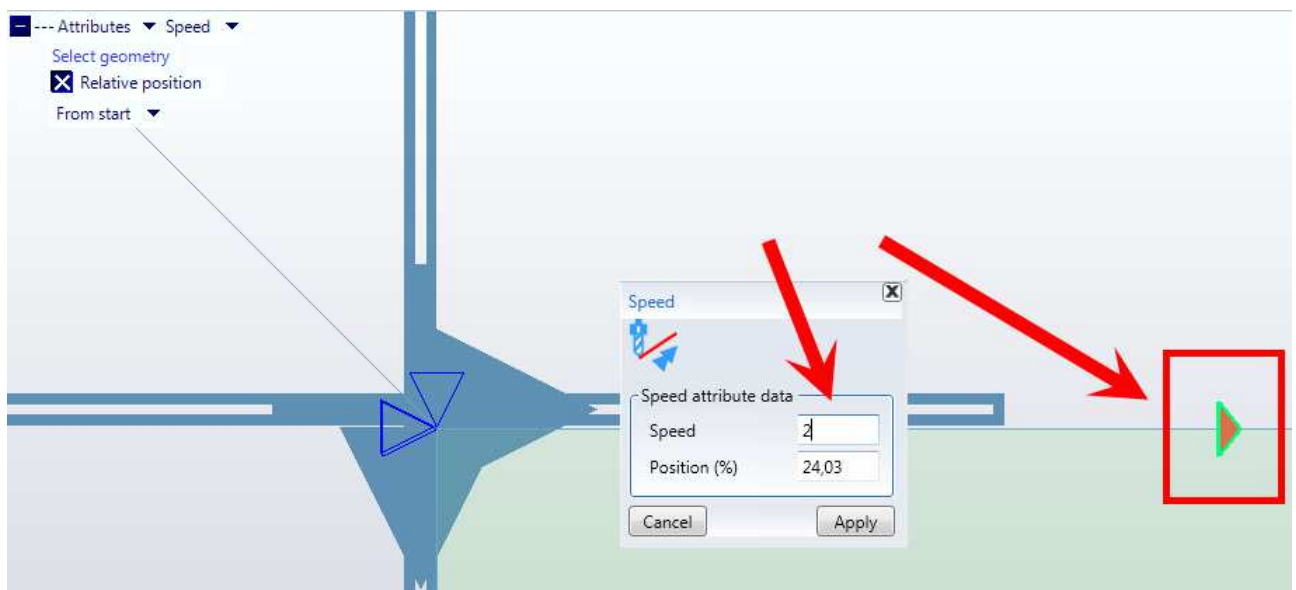
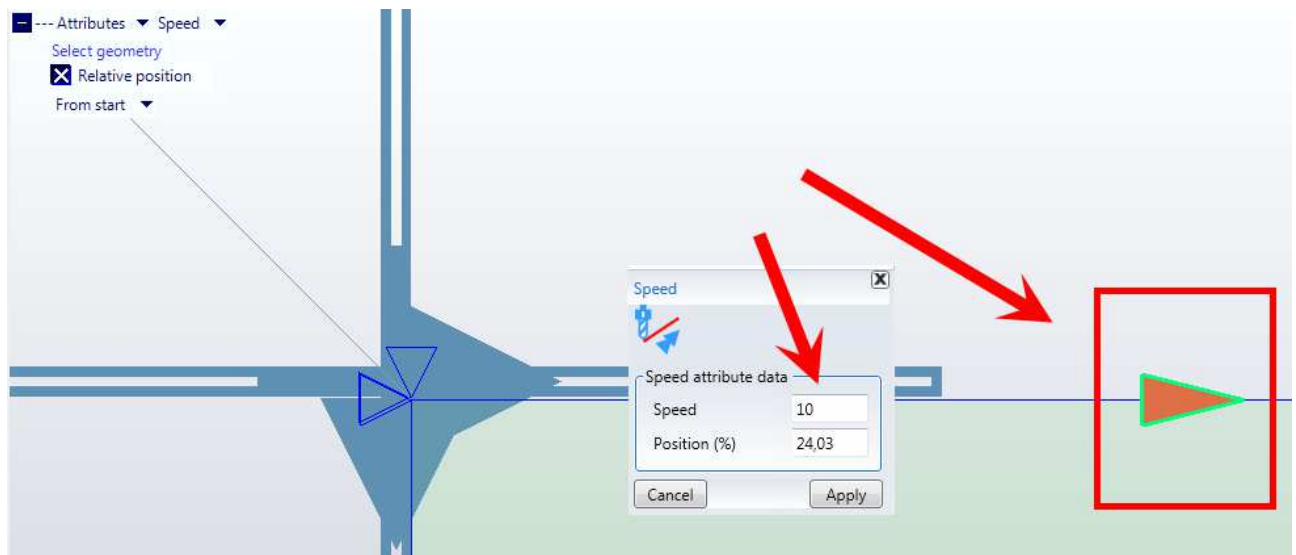
Once selected a marker will be displayed highlighting the percentage of the section where the speed will be changed.



After the section percentage has been entered, left click on the mouse to enter the new speed.



Once completed, a triangle-shaped marker, proportional to the speed, will be displayed to highlight the machining speed change.

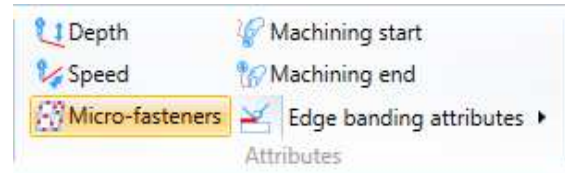


### 3.5.18.3 Microjoints

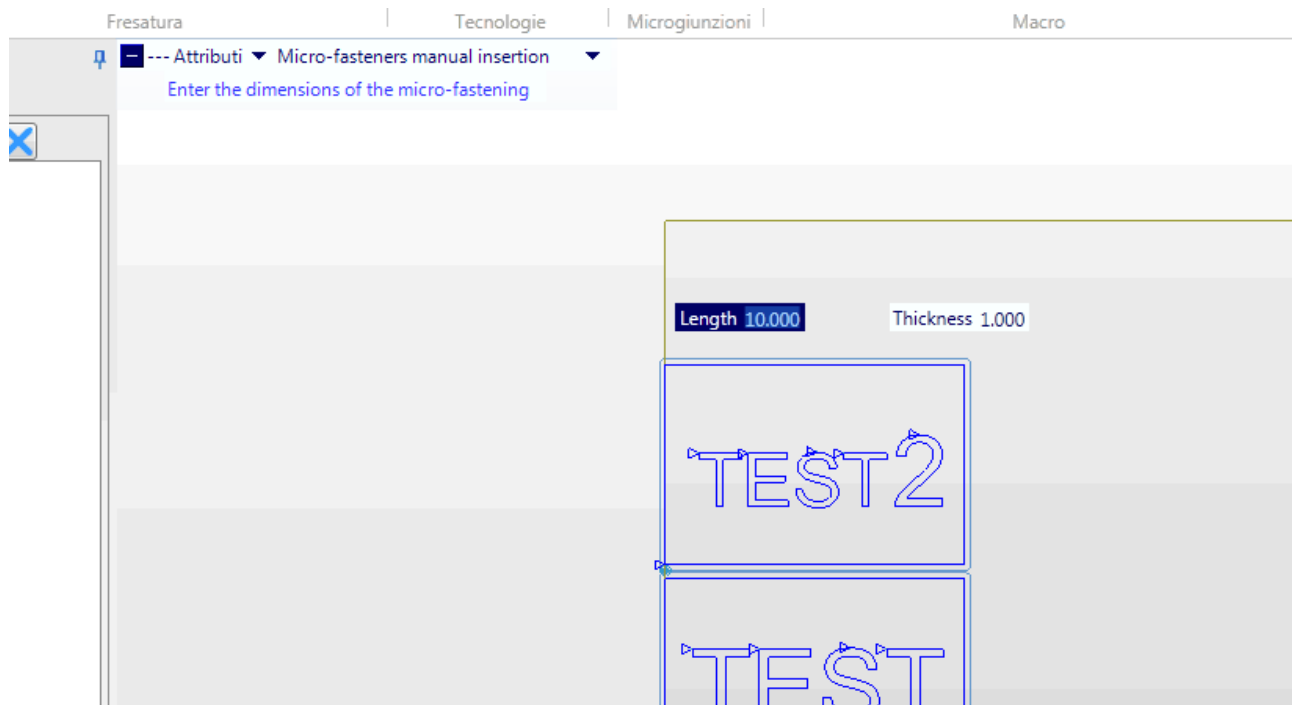
Cut pieces can be inserted between the different microjoints' contouring to keep the contoured workpieces linked, avoiding movements.

**Passing contouring is the necessary condition for inserting a microjoint.**

The Microjoints button is in the Operations-Attributes menu.

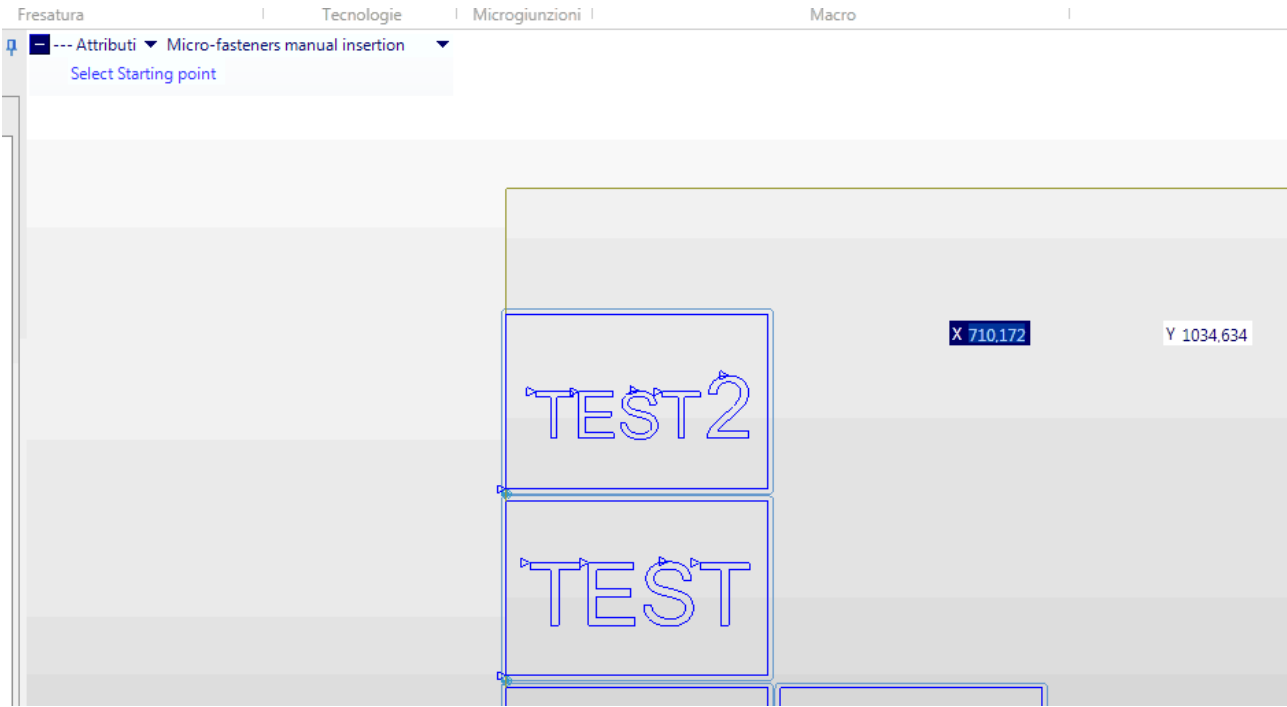


Click on it to display an initial phase for entering the microjoints dimensions:



Particularly length and thickness  
Length = microjoint dimension (bridge)  
Thickness = bridge height

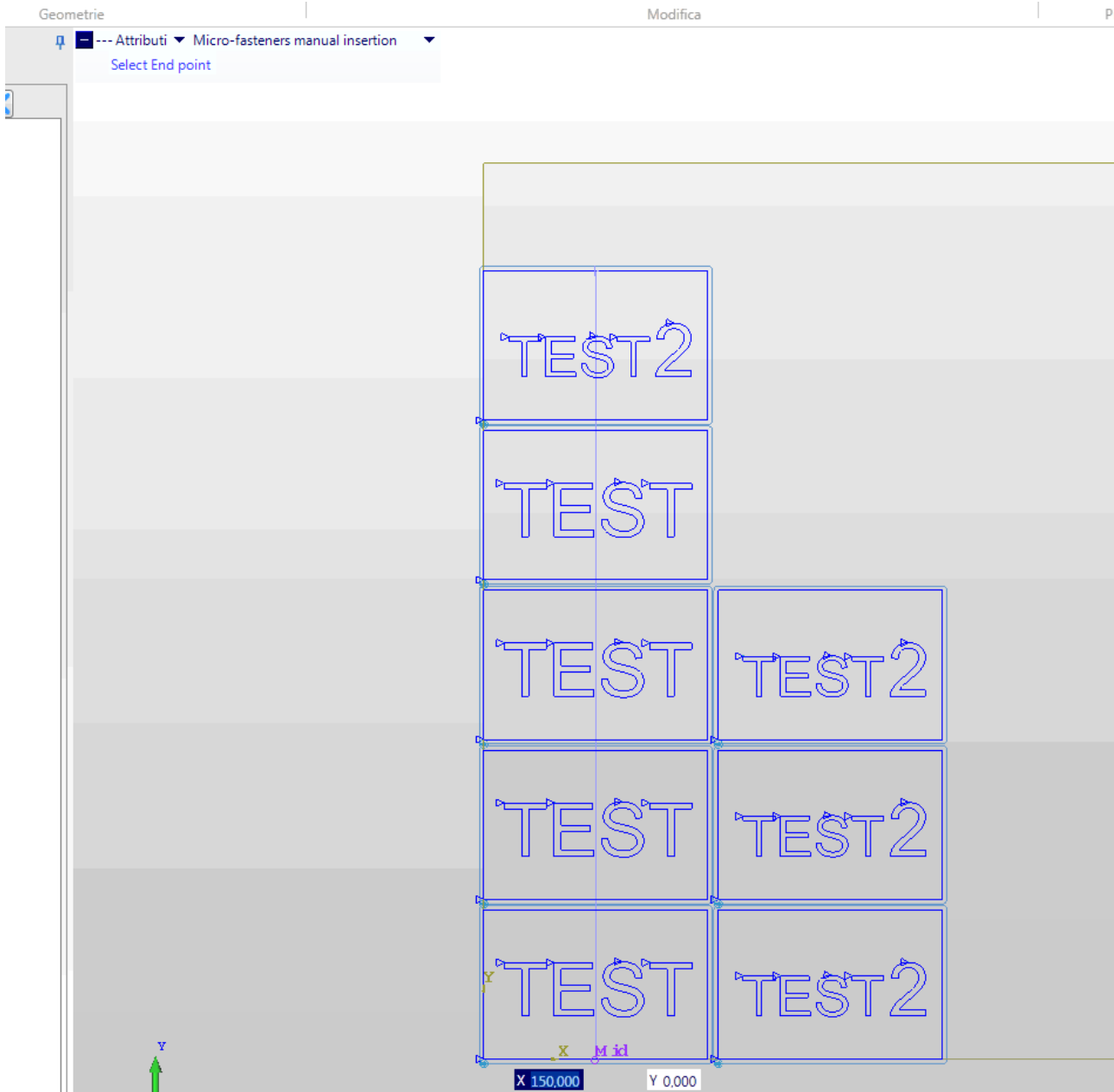
Insert the initial point (use the snap function) once the dimensions are confirmed through the send key.



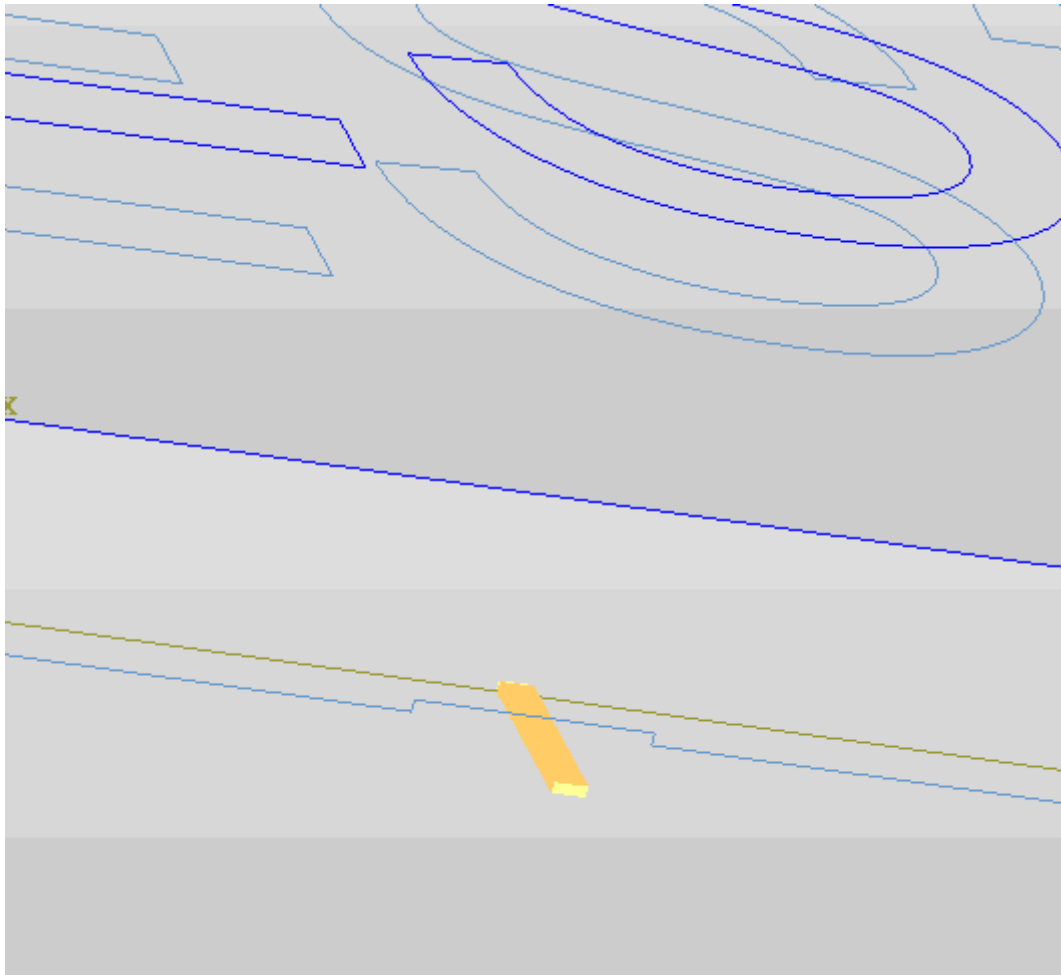
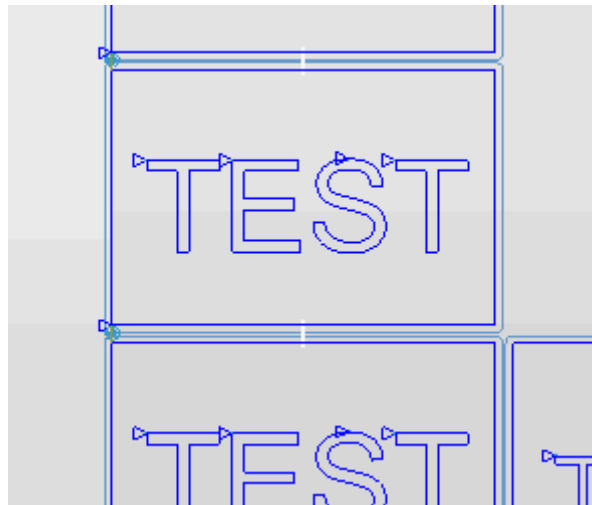
Insert the final point at the end.

The figure shows an example of microjoints made in the medium point of rectangular figures' longer side.

System intercepts all passing contouring machining upon final point confirmation, in this case inserting 10x1 mm microjoints in all detected intersection points.

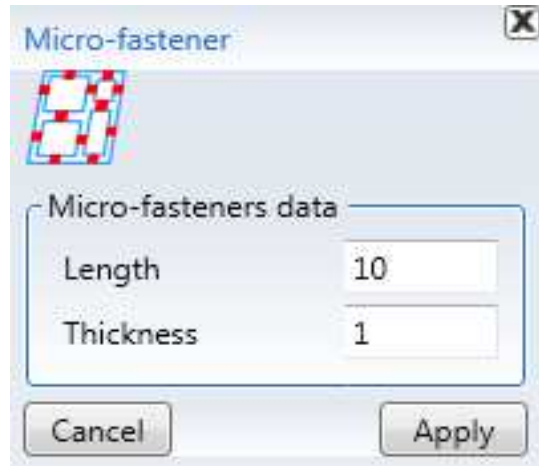


Yellow parallelepipeds graphically represent the microjoints.



The passing machining part shown in figure is interrupted by the 1 mm re-ascension for a 10 mm section, to then return to being passing.

The microjoints' dimensions can be individually modified. The specific dimensions of the selected microjoint appear on right by clicking on the microjoint.

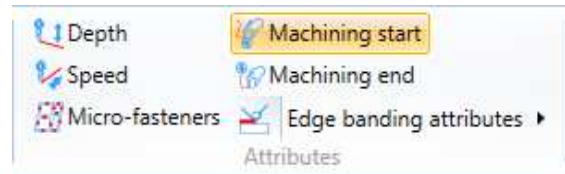


The dimensions can be modified and confirmed with the Apply key.

N.B. Microjoints are not only restricted to nesting function but are also managed in the individual program.

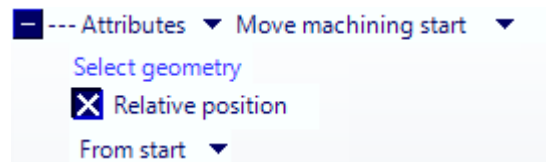
### 3.5.18.4 Start work

This attribute is used to set the machining start point.

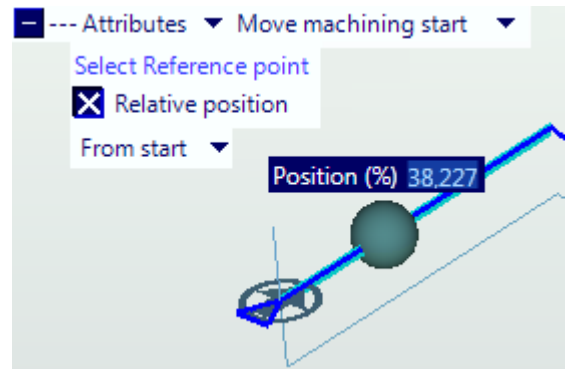


Select the "Start work" icon and then select the machining geometry for which the start point must be changed.

For composite lines it is important to select the exact geometry section where the new start point must be entered.

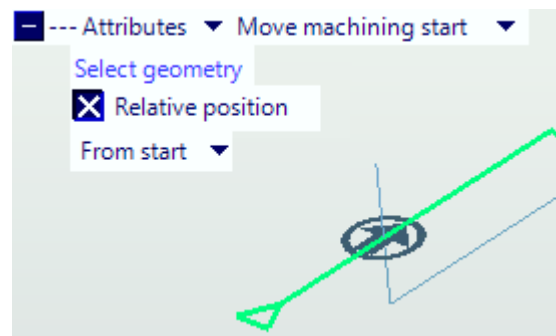


Then select the point along the section selected. The positioning can be done by moving the mouse or by entering the section percentage.



The machining start will be moved to the requested point.

For composite lines any geometry sections before the section where the new machining start has been entered, will be ignored.





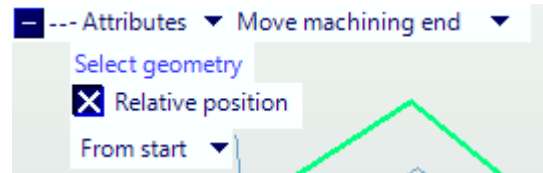
### 3.5.18.5 End work

This attribute is used to set the machining end point.



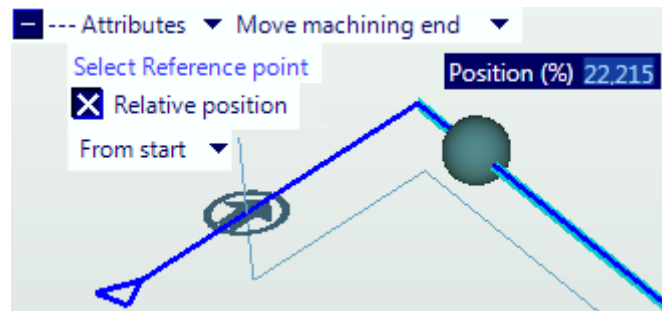
Select the “End work” icon and then select the machining geometry for which the end point must be changed.

For composite lines it is important to select the exact geometry section where the machining must end.



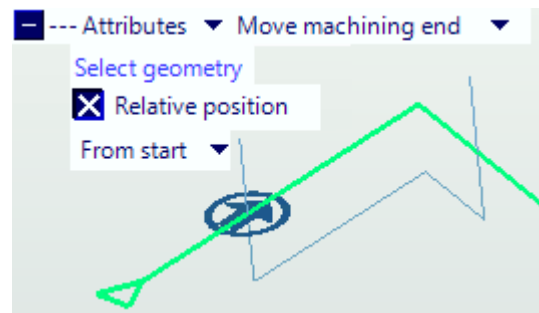
Then select the point along the section selected.

The positioning can be done by moving the mouse or by entering the section percentage.



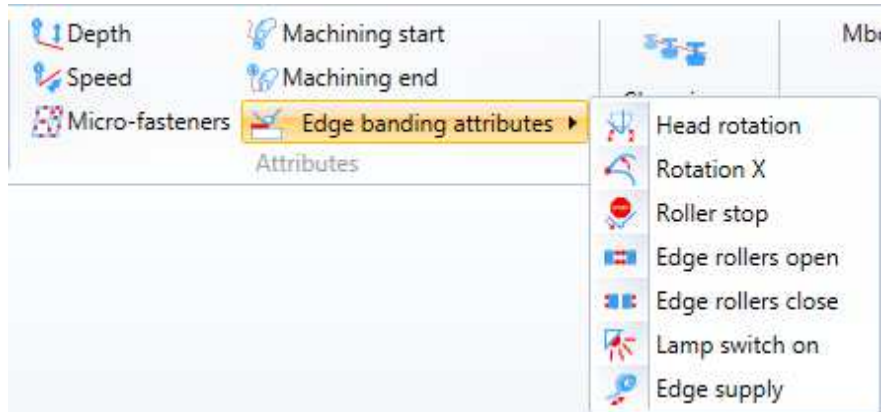
The machining end will be moved to the requested point.

For composite lines any geometry sections after the section where the new machining end has been entered, will be ignored.



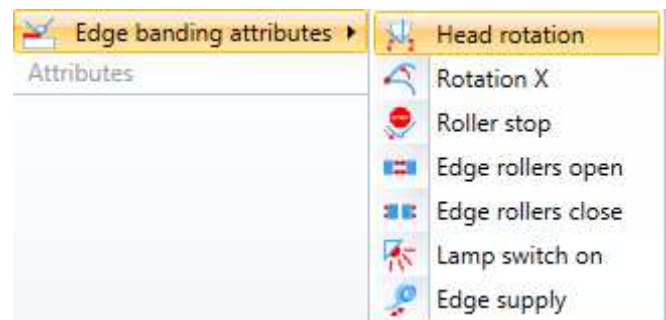
### 3.5.18.6 Edge banding attributes

Due to the special nature of the machining and the considerable number of attributes necessary for the edge banding we have created a group of attributes used solely for edge banding.



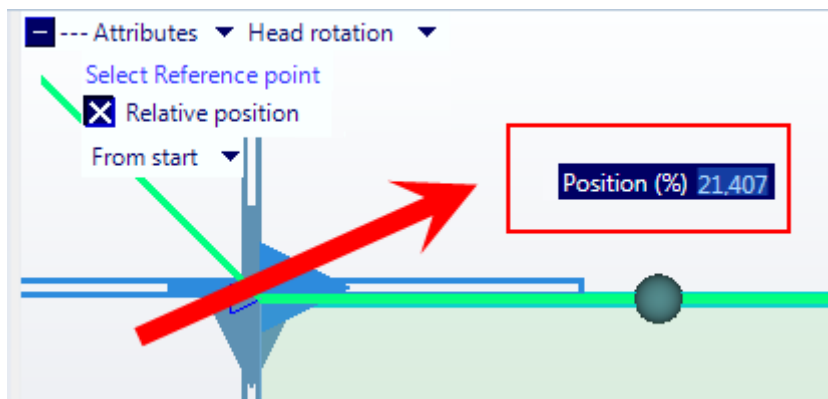
#### 3.5.18.6.1 Head rotation

The edge banding head rotation is used to tilt the edge banding head as necessary.

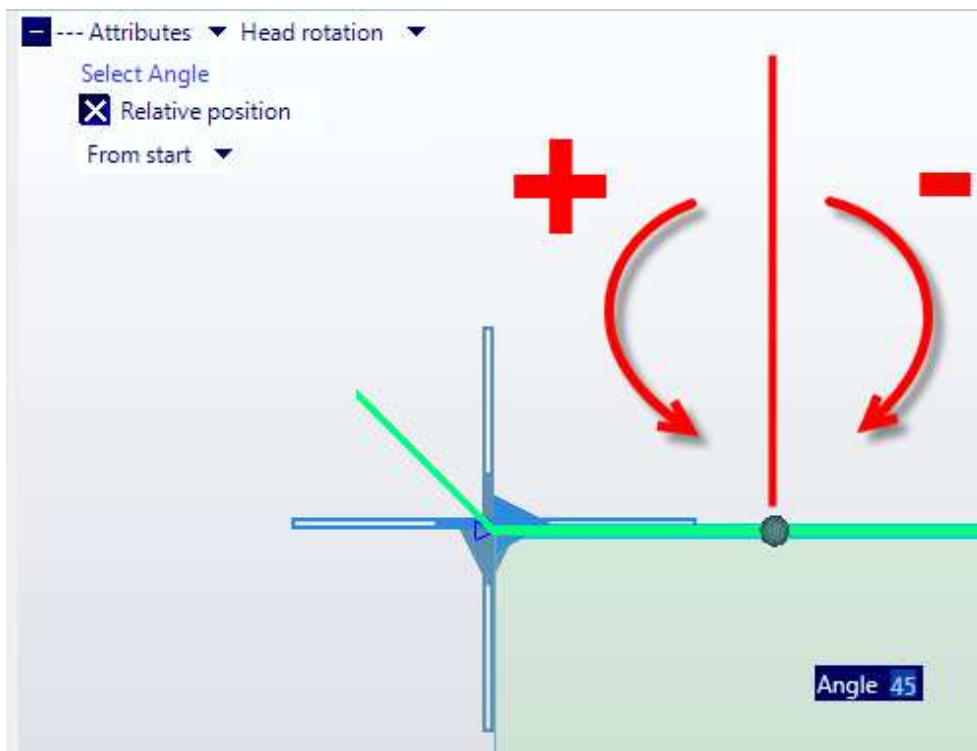
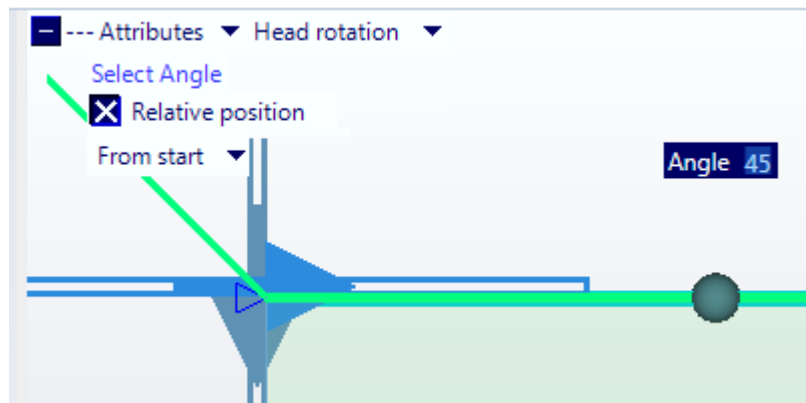


The operation is similar to the speed change in a specific point.

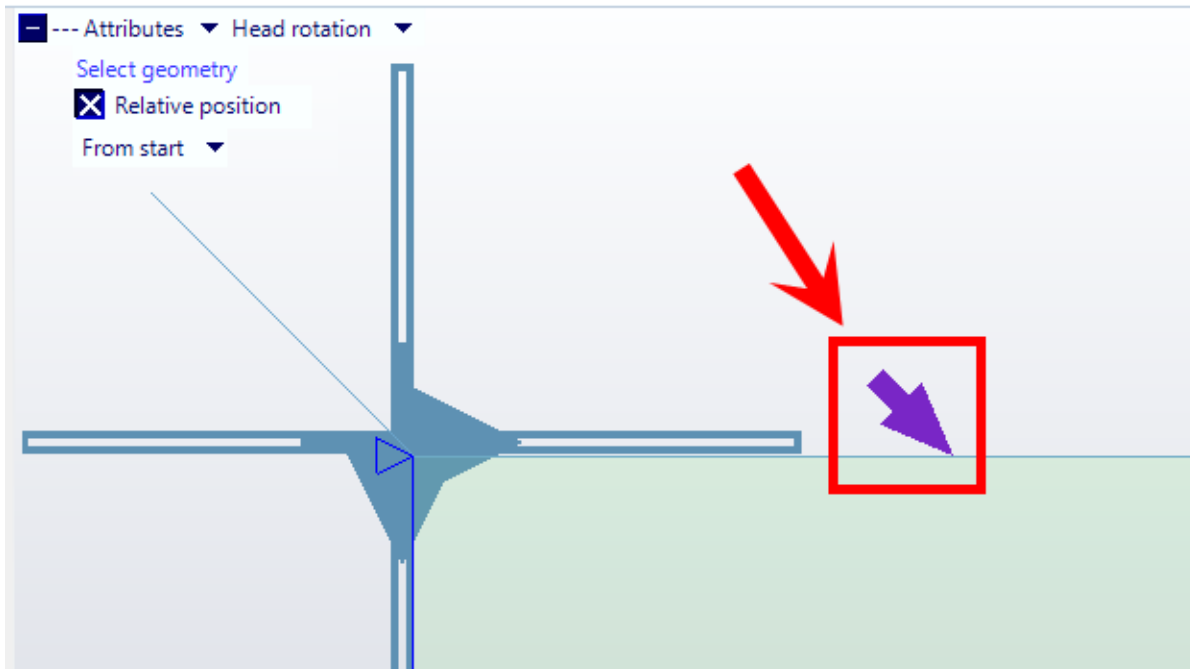
Select a machining operation where the rotation is to be applied and enter the percentage where the change must take place.



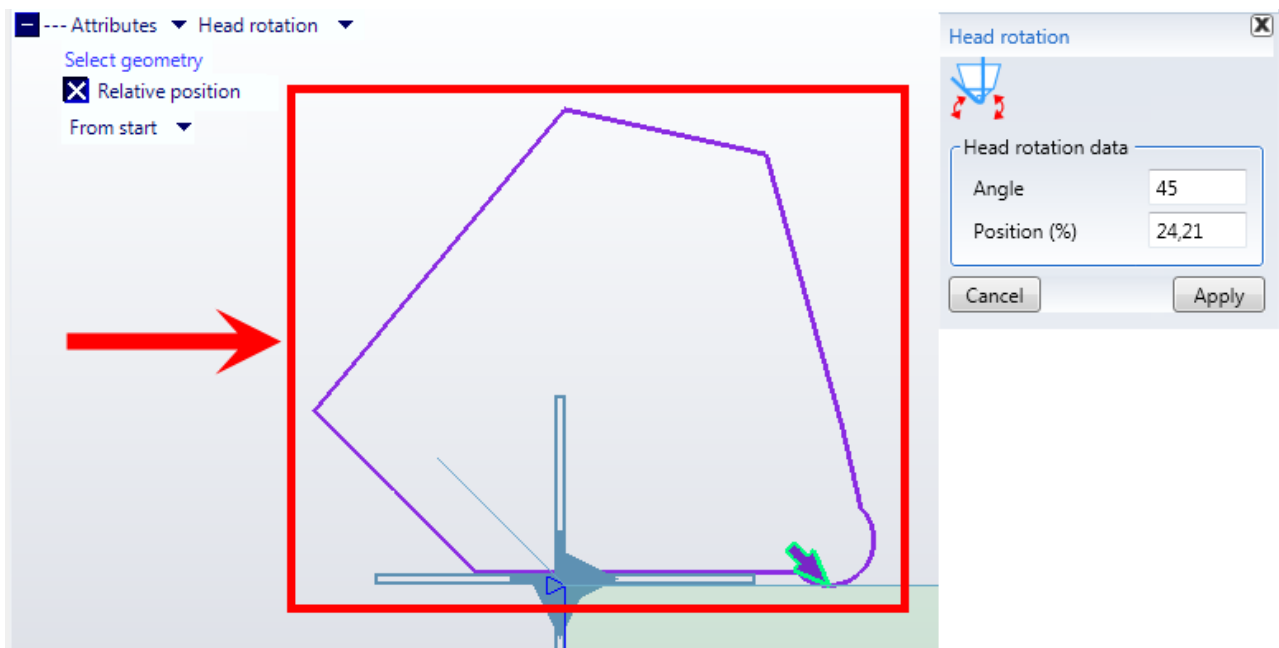
Enter the rotation angle in relation to the vertical line.



An arrow will be displayed indicating the rotation change.

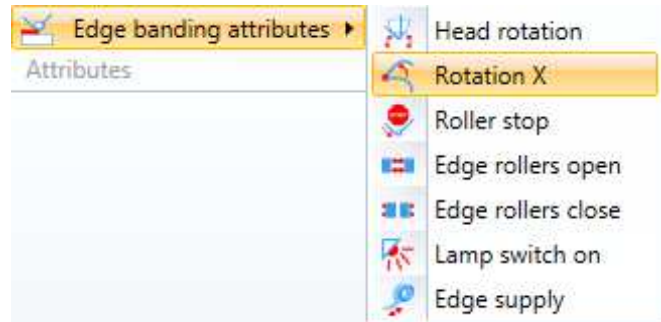


If selected the edge bander head will be displayed tilted by the angle entered previously.

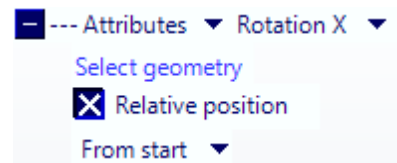


### 3.5.18.6.2 Rotation X

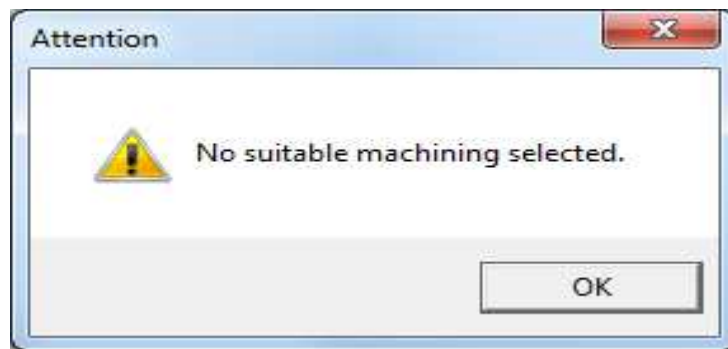
The “Rotation X” attribute is used to change the tilting of the trimmer in a specific point of the trimming operation.



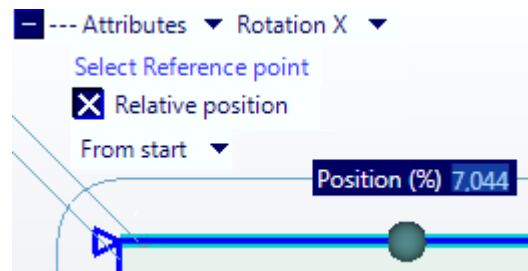
Select the “Rotation X” icon and then select the geometry of a trimming operation.



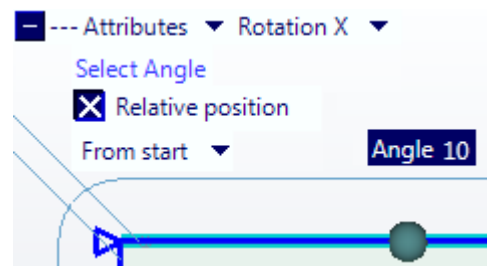
If the machining operation selected is not trimming, the following message will be displayed:



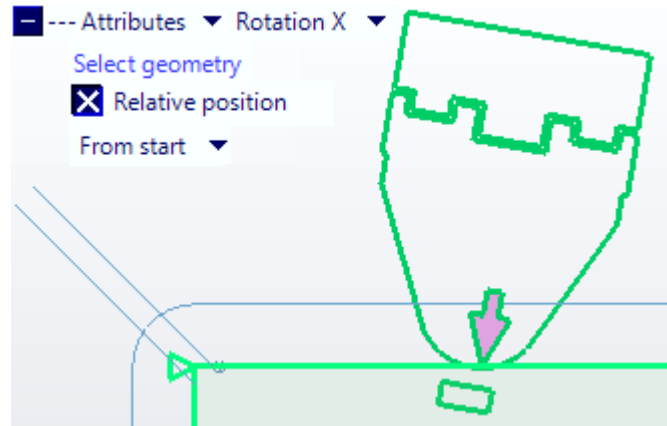
Then select the point along the section selected. The positioning can be done by moving the mouse or by entering the section percentage.



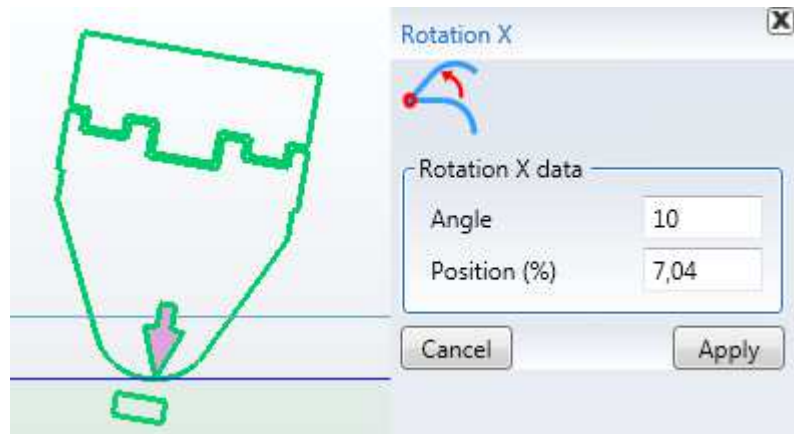
Then enter the rotation angle of the trimmer, bearing in mind that 0° refers to the trimmer at right angles to the geometry and that the value entered can be positive or negative.



An arrow symbol will be displayed tilted at the angle set.  
If the trimmer dxf has been entered in the fixturing rows, the trimmer image will appear on the profile.

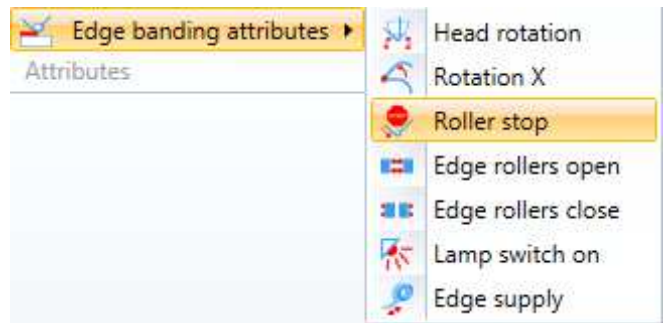


To change the Rotation X attribute parameters again simply select the arrow symbol and change the Angle or Position values that will be displayed in the window.

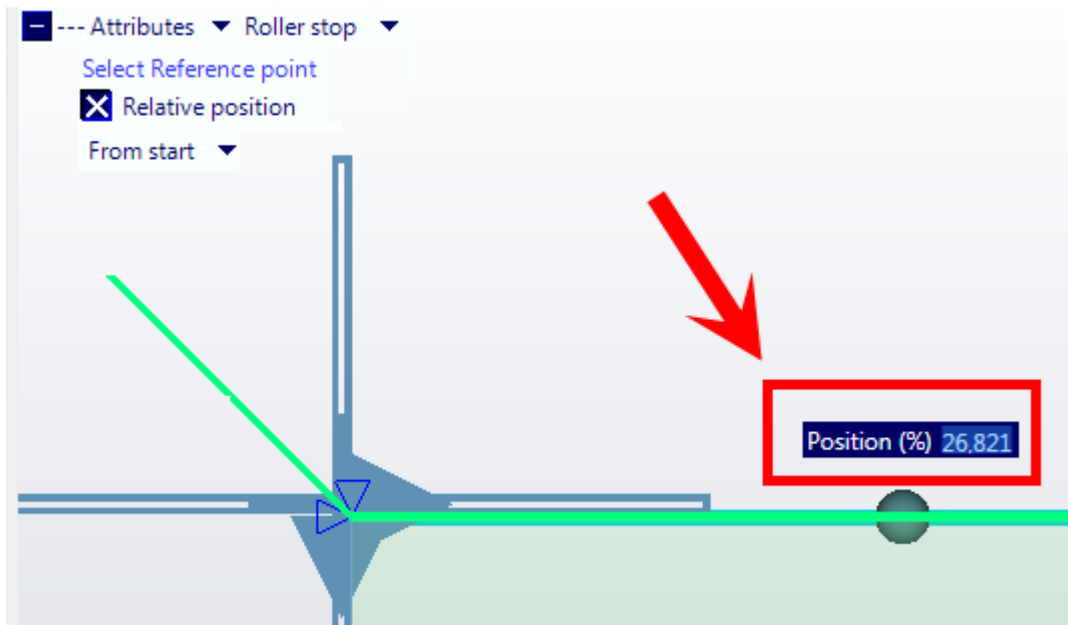


### 3.5.18.6.3 Ruller stop

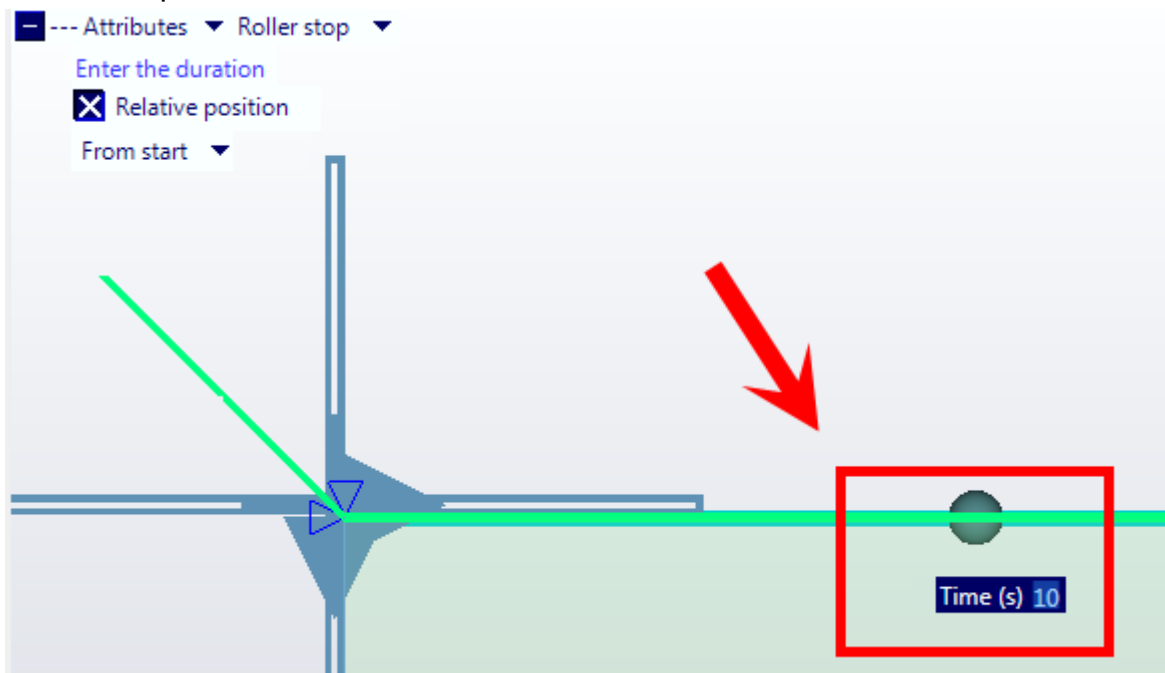
The edge banding head roller stop is used to stop the roller as needed to help the gluing of the edge to the panel in specific conditions in terms of shape and type of edge used.



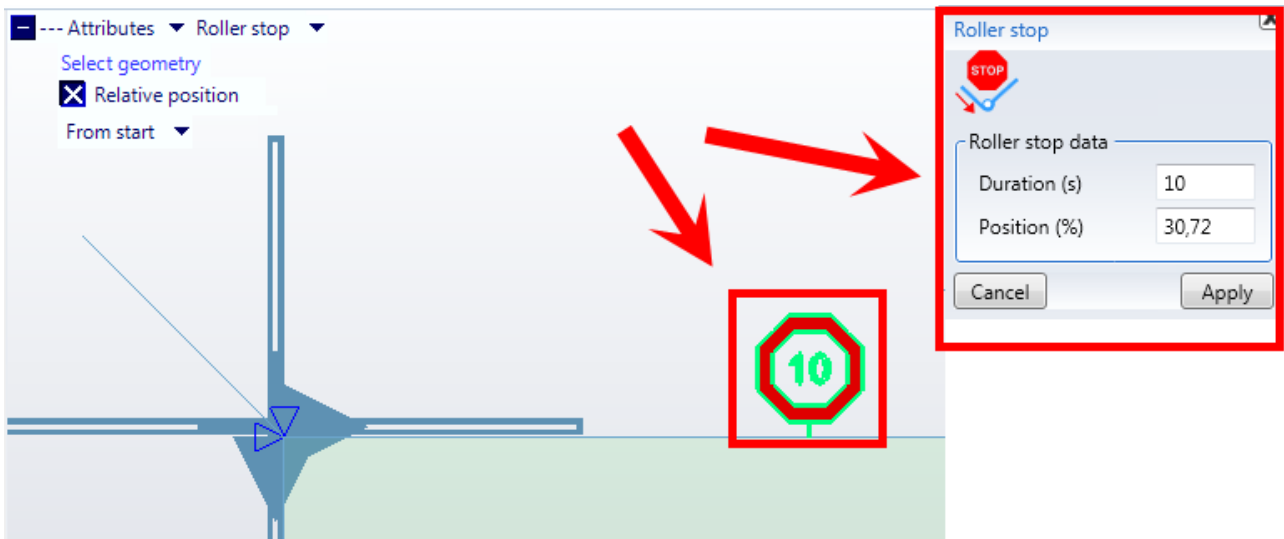
The operation is similar to the speed change in a specific point. Select a machining operation where the roller stop is to be applied and enter the percentage where the change must take place.



Enter the time, expressed in seconds.



A symbol will be displayed indicating the time change.

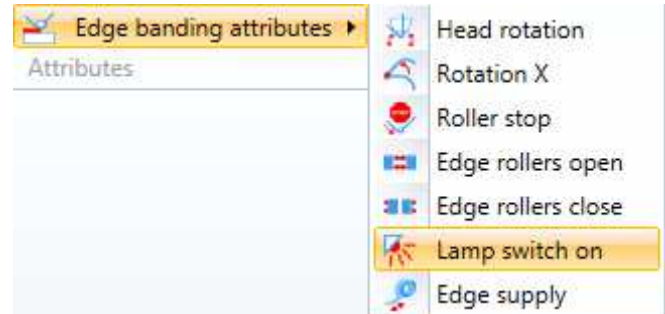


### 3.5.18.6.4 Edge rollers opening

### 3.5.18.6.5 Edge rollers closure

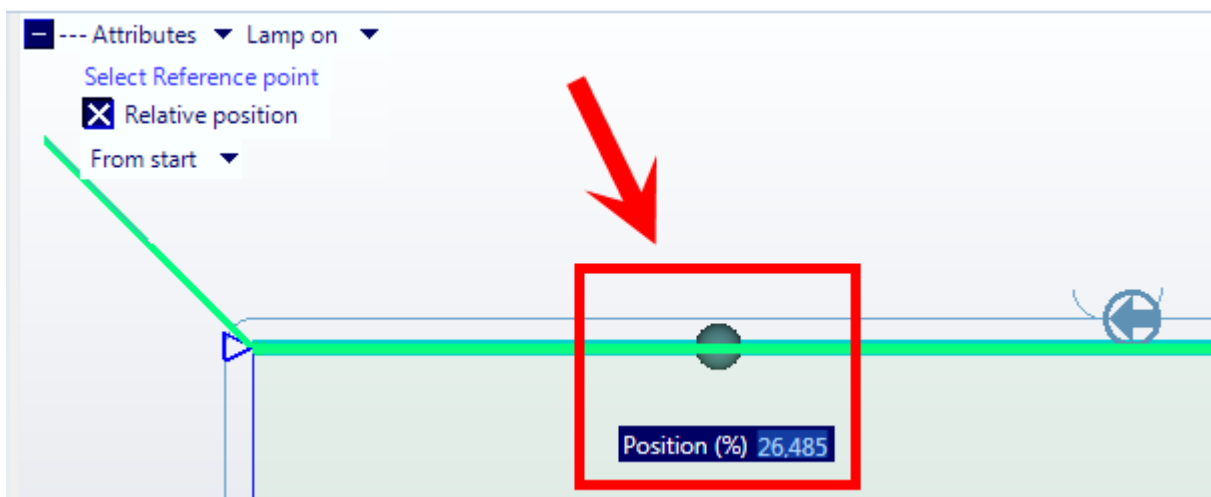
### 3.5.18.6.6 Lamp on

This attribute is used to switch on the edge heating lamps at a specific point to heat the edge and increase its flexibility so that it follows the geometry better.



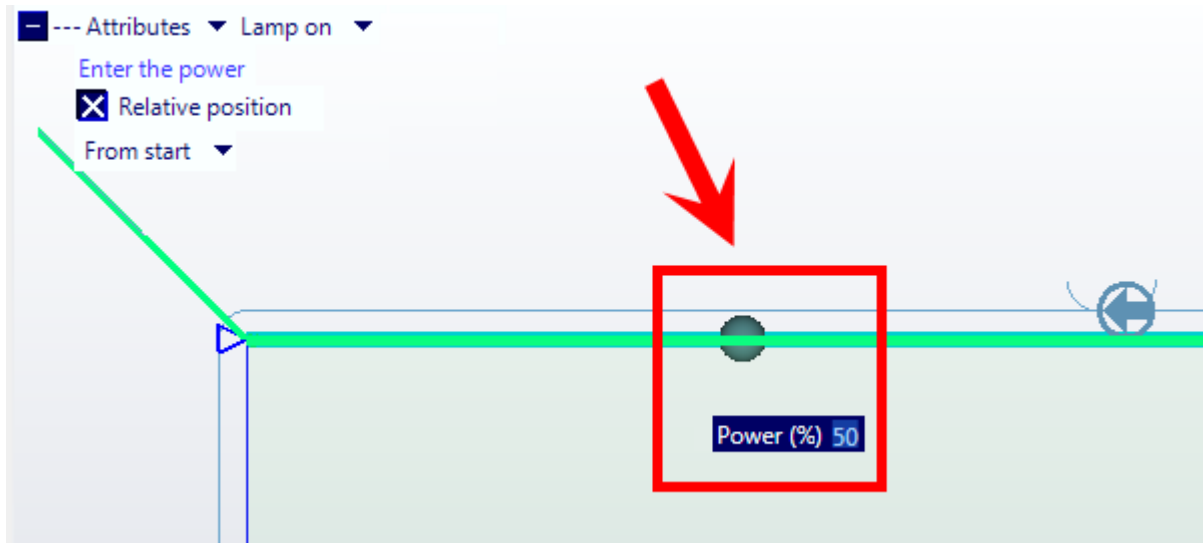
The operation is similar to the speed change in a specific point.

Select a machining operation where the lamp switch on is to be applied and enter the percentage where the change must take place.

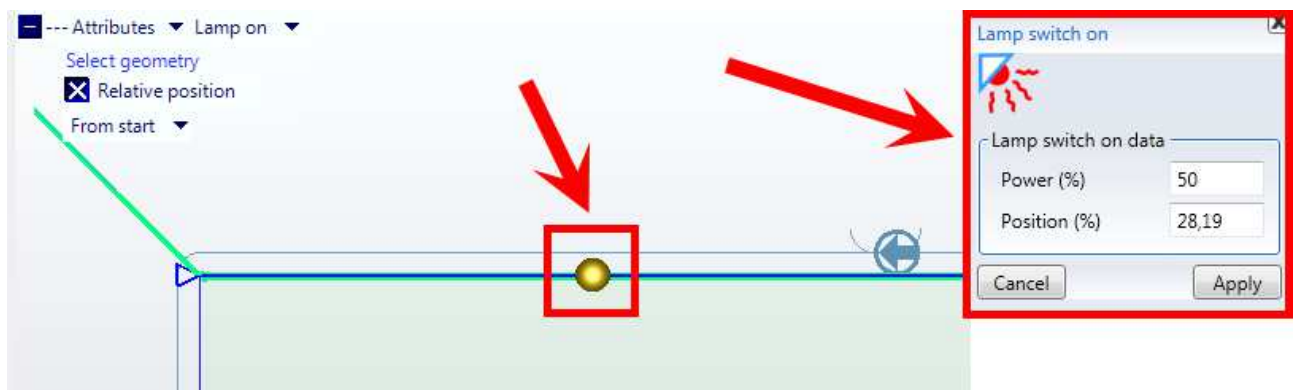




Enter the lamps power percentage.

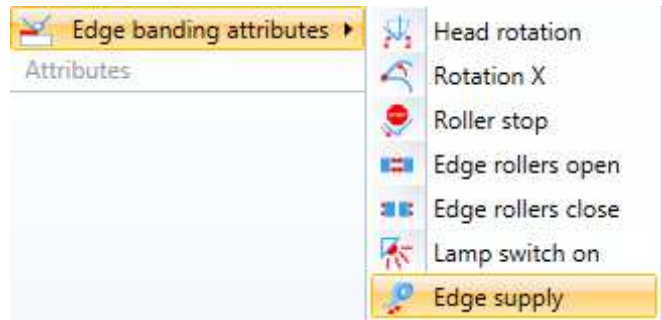


A symbol will be displayed indicating the lamp switch on change.

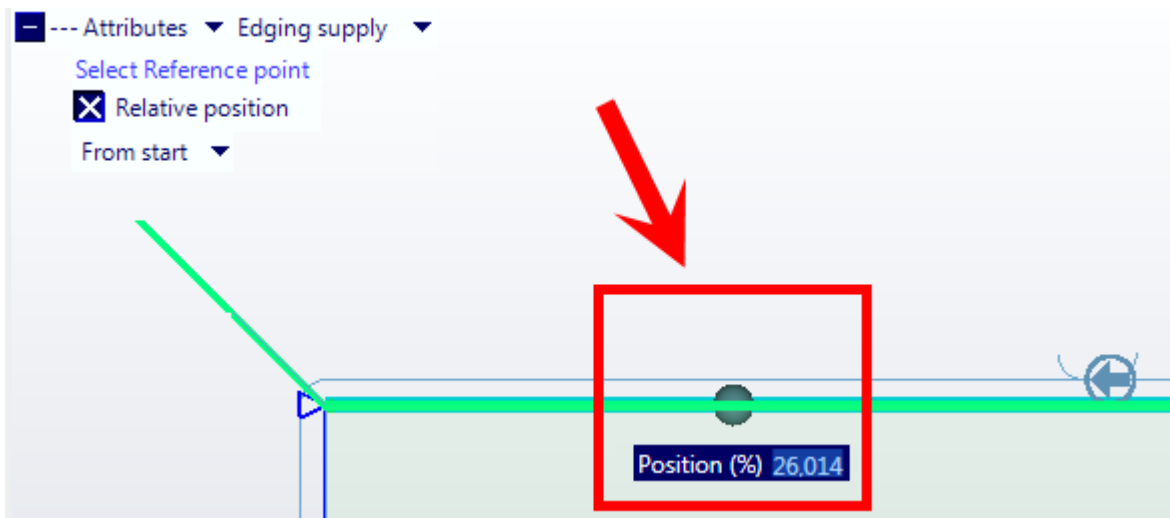


### 3.5.18.6.7 Edge supply

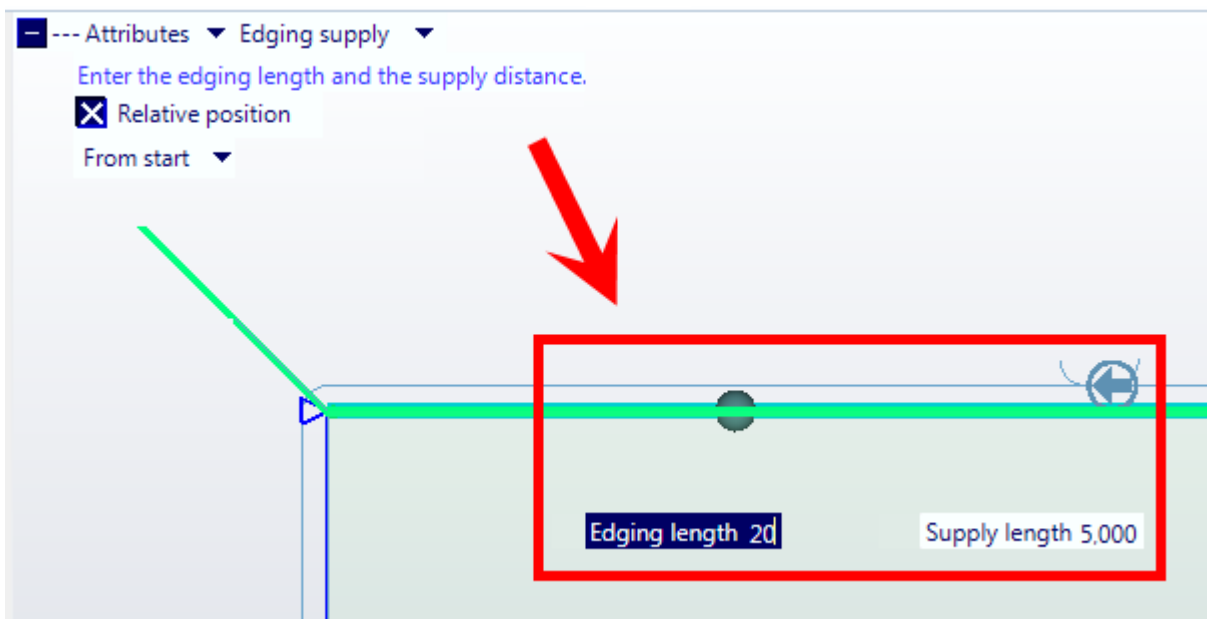
This attribute is used to increase the length of edge during the edge banding to avoid unusual bends and improve the application on the geometry being followed.



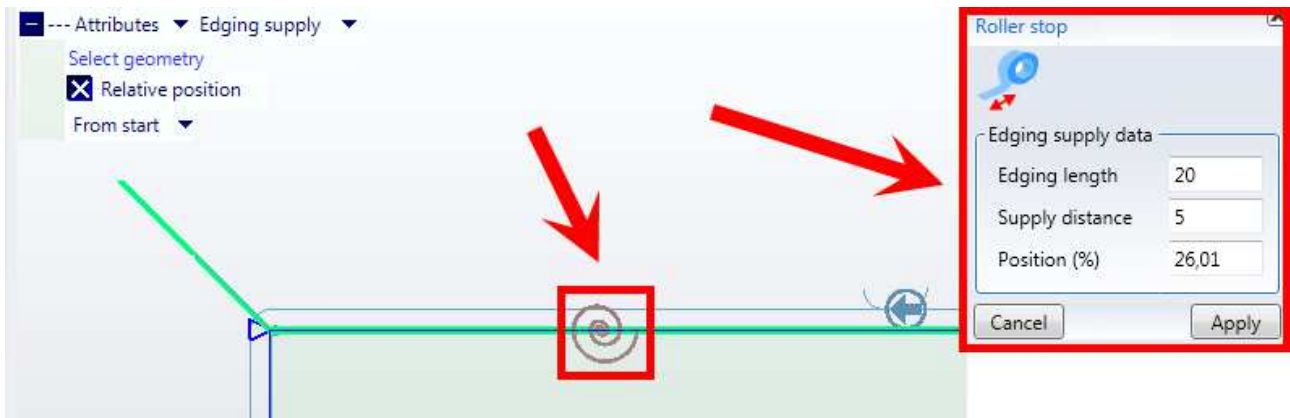
The operation is similar to the speed change in a specific point. Select a machining operation where the extra length is to be applied and enter the percentage where the change must take place.



Enter the measurement in millimetres of extra edge to be available at the head side.



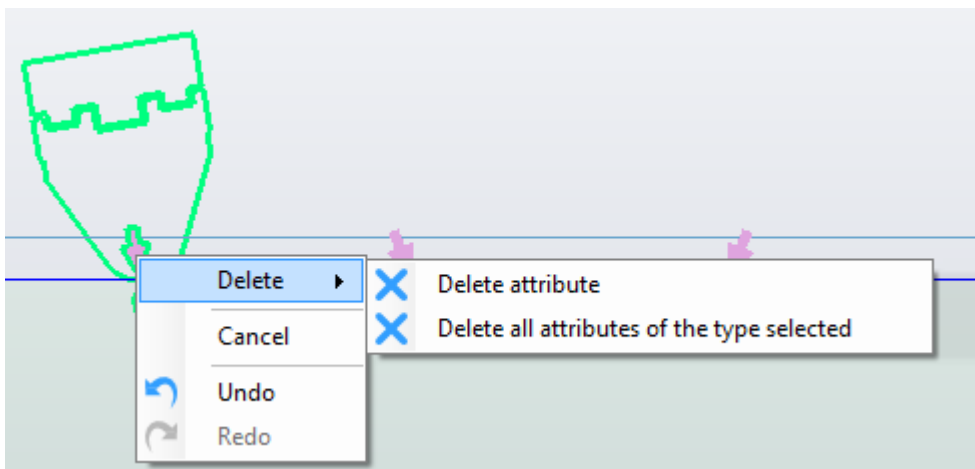
A symbol will be displayed indicating the extra edge length change.



### 3.5.18.7 Delete attributes

To delete any type of attribute simply select the attribute to delete and right click on the mouse.

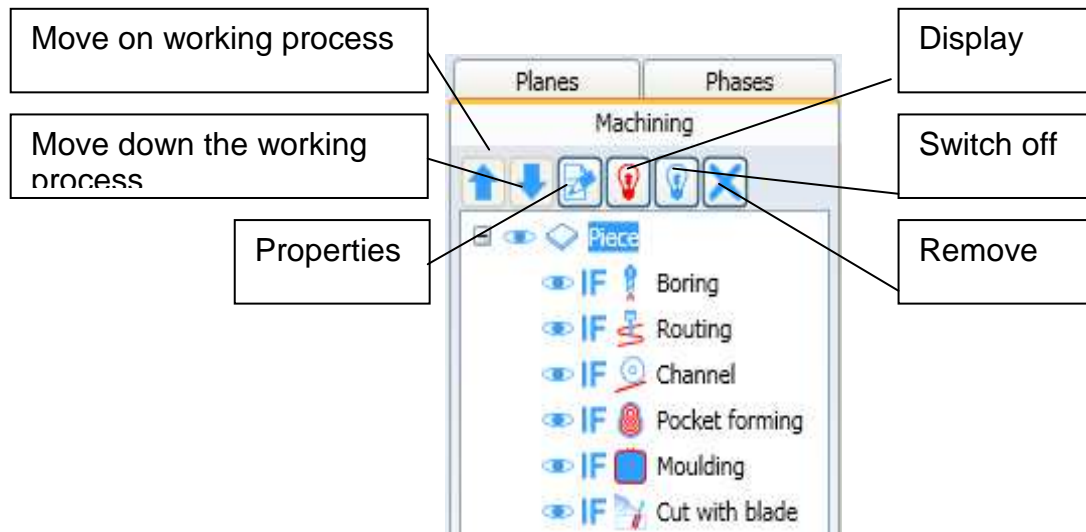
The following menu will be displayed:



Only the attribute selected, or all the attributes of the same type on the same geometry, can be deleted.

### 3.5.19 Working process tree

The working process tree represents the workpiece and all working processes set in a certain project.



#### 3.5.19.1 Object selection

You can perform the object selection on the tree by clicking with the left button of the mouse on any displayed element.

It is possible to select the workpiece or a working workpiece and it is not possible to select more than one object.

When an object is selected, the name is written in white on a blue background.

The selected object in the tree will be marked also in the graphic area.

#### 3.5.19.2 Working process ordering

The working process tree shows us the working processes in our project as well as their sequence. so you could benefit from ordering the working process sequence.

In order to move a working process in the tree:

- select the working process
- use the command "Move on the working process, arrow upwards"
- or "Arrow downwards"

This option is inhibited by the working process tree when the project has more than one step (see paragraph 3.9.6 )

### 3.5.19.3 Working process display

On the working process tree you can display or not the different working processes. To the left of every working process name, you can see the icon that represents the type of working process and to the left of the icon there is the symbol that stands for an eye. By clicking with the left button on the eye, you change its status from open to closed. The open eye means that the working process will be displayed. The closed eye means that the working process will not be displayed.

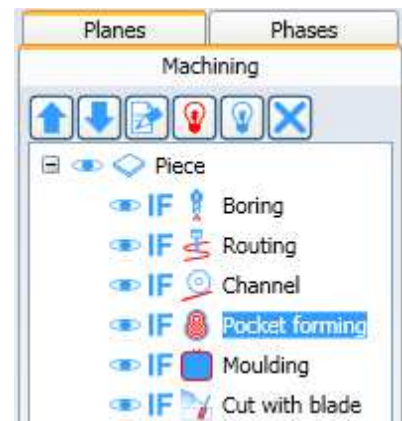
In this example you can see a project where the Emptying has been selected and the Milling and the Countouring have been switched off.

If you wish to switch all working processes off you can:

- 1) close all eyes to the left of the working processes.
- 2) close the eye to the left of the workpiece
- 3) or click on the light bulb that is off.

If you wish to display all working processes you can:

- 1) open all eyes to the left of the working processes.
- 2) open the eye to the left of the workpiece
- 3) or click on the light bulb that is on.

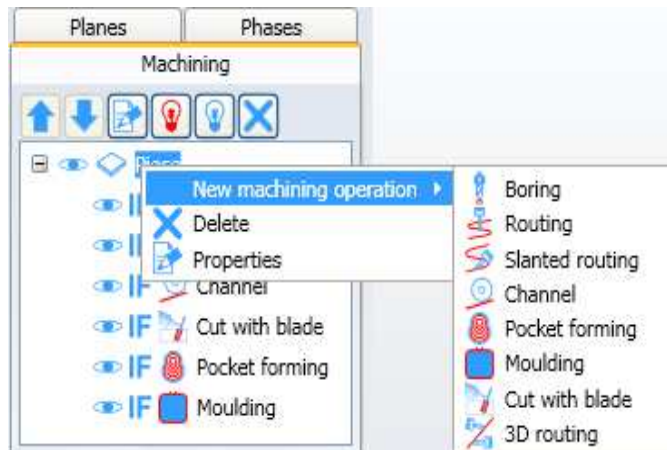


The selection on the tree has an immediate impact on the graphic area.

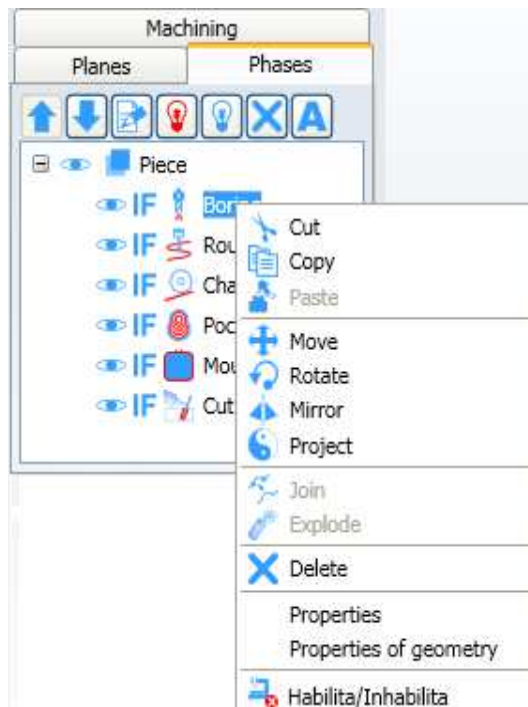
### 3.5.19.4 Contextual menu

After selecting any object of the tree, by clicking with the right button of the mouse, a contextual menu will show us what kind of operations we can carry out with the selected object:

Contextual Menu workpiece



Contextual Menu Working process



### 3.5.20 Working process selection

The working process selection in the graphic area occurs in the same way as the geometry selection (see paragraph 3.4.7)  
you can also select a single working process from the working tree (see paragraph 3.5.19.1)

### 3.5.21 Contextual menu for working processes

The contextual menu for working processes is available:

- In the graphic area (see paragraph 3.4.8 )
- and in the working process tree (see paragraph 3.5.19.4 )

### 3.5.22 Working process removal

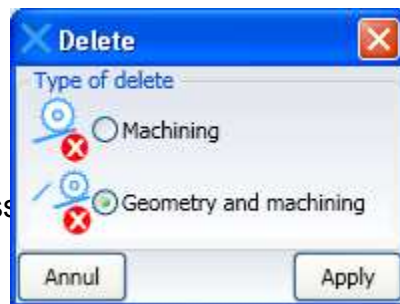
After selecting one or more working process you can remove them:

- by clicking with the button "Canc"
- or clicking on the command "Remove" of the menu "Home" "Clipboard"
- or on the command "Remove" if the contextual Menu.

This menu will appear:

You can choose if you wish to:

- remove only the working process
- remove Geometry and Working process



### 3.5.23 Working process properties

If you wish to know an existing working process properties, the dialogue menu during the working process inserting will be showed.

### **3.5.24 Working process modification**

After selecting one working process (on the screen or in the tree) it is possible to modify the parameter values in the dialogue menu.

### **3.5.25 Move and/or copy existing working processes**

The options used to move or modify working processes are:

- Move
- Rotate
- Mirror
- Oppose
- Cut
- Copy
- Paste

these same options are also used for the geometries (see paragraph 3.4.12)



## 3.5.26 Machining technologies

The technologies differ from the machining in that they do not have an associated geometry.

As the technological information is often much more than the geometrical information it is useful to save them in order to apply them to other geometries.

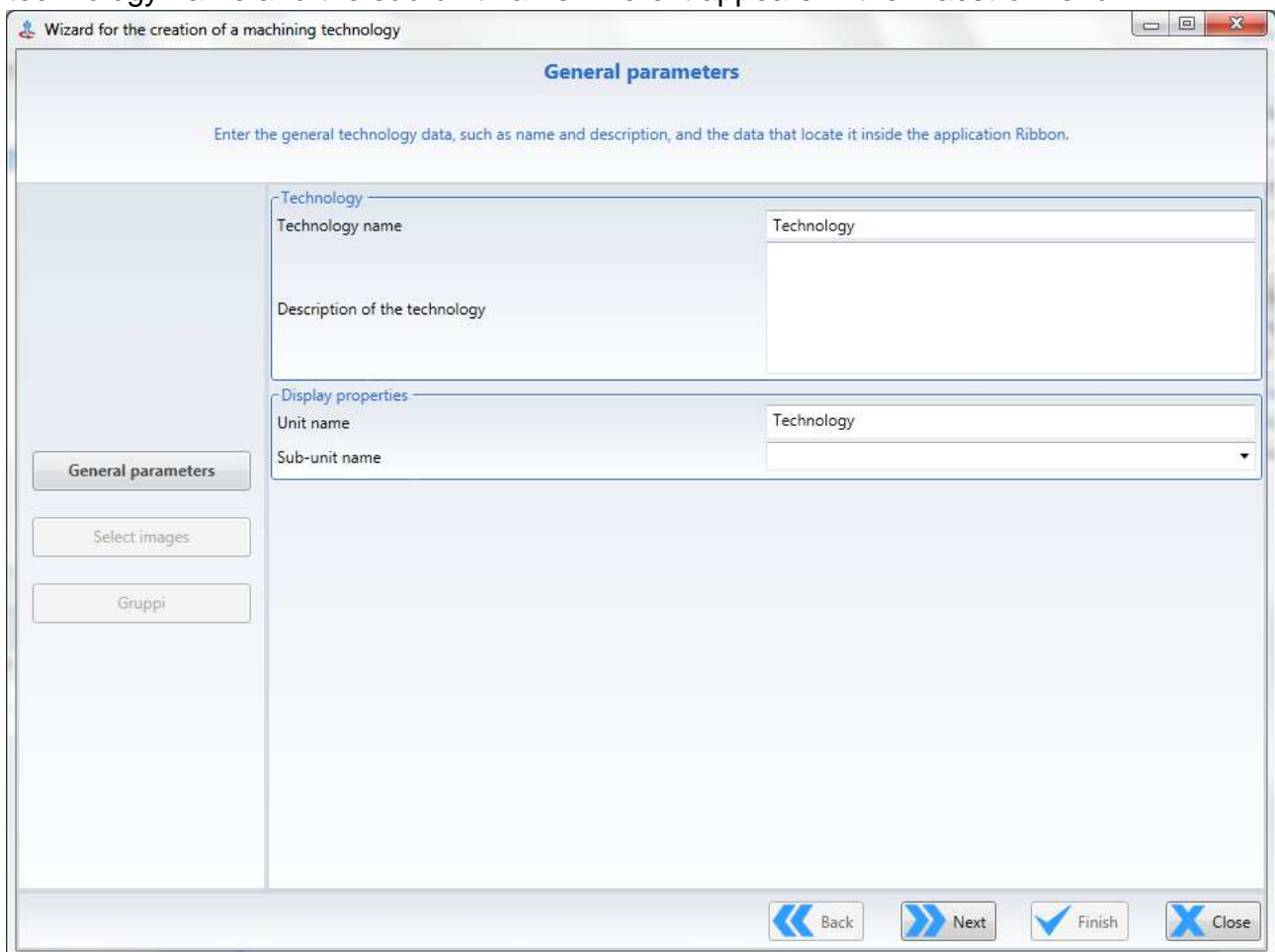
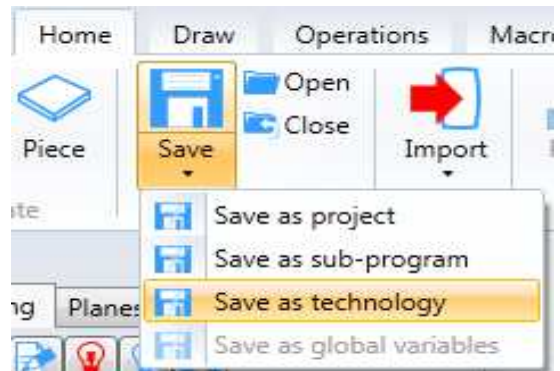
### 3.5.26.1 Create Technology

To create a technology do a project with a single machining and save it as technology.

You will be asked to save the technology file with the .tchx extension.

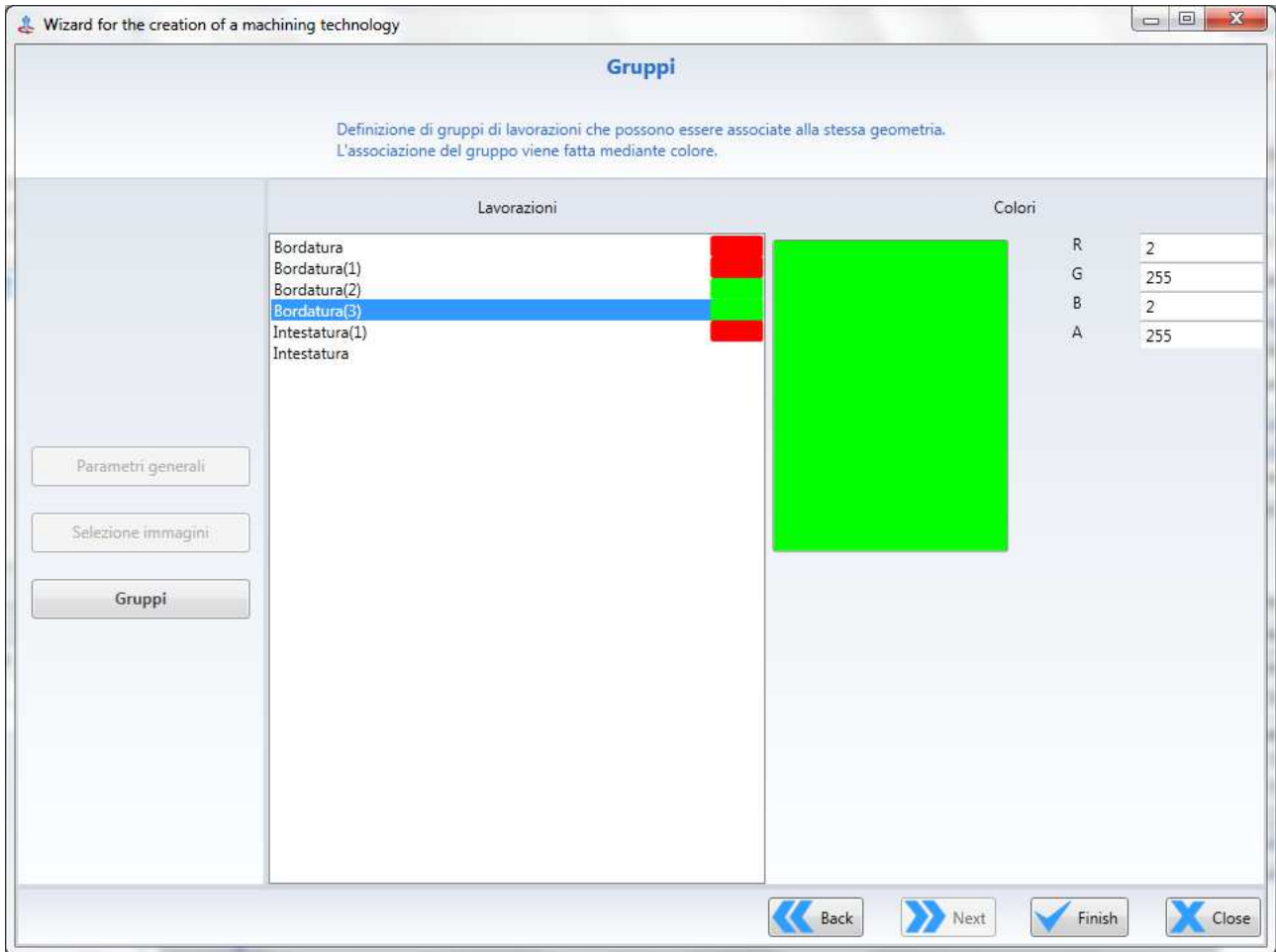
To reuse these files they must be located in the "Technologies" sub-folder of the Maestro installation folder.

The screen below will appear, to enter the technology name and the sub-unit name where it appears in the Maestro menu.

A screenshot of a wizard dialog box titled 'Wizard for the creation of a machining technology'. The main section is 'General parameters' with the instruction: 'Enter the general technology data, such as name and description, and the data that locate it inside the application Ribbon.' The dialog is divided into two sections: 'Technology' and 'Display properties'. In the 'Technology' section, there is a text box for 'Technology name' containing the word 'Technology' and a larger text area for 'Description of the technology'. In the 'Display properties' section, there is a dropdown for 'Unit name' containing 'Technology' and a dropdown for 'Sub-unit name'. On the left side, there are three buttons: 'General parameters', 'Select images', and 'Gruppi'. At the bottom, there are four navigation buttons: 'Back', 'Next', 'Finish', and 'Close'.

After having selected “Next”, the second screen where the bitmap can be entered, is displayed.

If the technology being saved is an edge banding process, the machining can be grouped with colours (as shown in the figure).



These groupings do not affect the interactive use, but they apply during the DXF import phase.

The segments with the same colours in the technology will be associated to the relative machinings. (see chapter **Errore. L'origine riferimento non è stata trovata.**)

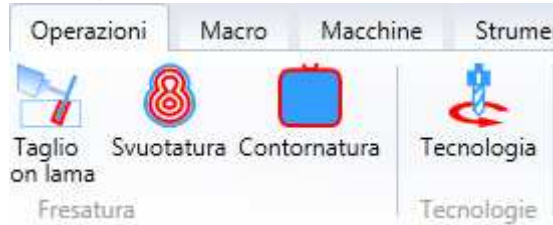
The technology is saved by pressing “End”.

### 3.5.26.2 Copy technology

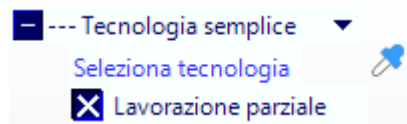
As well as imply copying the machining, you can copy all the technological information of a machining and apply them to new 2D geometries.

(Equivalent to copying the whole machining, except for the geometry).

To do this use the “Technology” command in the “Operations” “Technologies” menu.



The following menu is displayed after clicking on Technology:



The dropper shape cursor reminds you that you are selecting an existing technology. To select a technology to copy on other 2D geometries, simply select an existing machining in the project.

As soon as a machining is selected the cursor becomes a syringe.

And the “Select geometry” item appears in the menu.

The previously selected technology will be applied to all the geometries selected.



To apply the technology to just one part of the geometry selected, check the “Partial machining” item in the menu.

In this case, after having selected the geometry the menu asks to “select start point” and then “select end point”, the technology will be applied only to the part of the geometry from the start point to the end point.

### 3.5.26.3 Apply technology

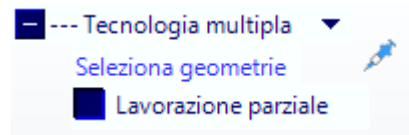
A previously saved existing technology can be applied (see chapter 1.2.11.1 Create Technology) to one or more geometries.

In fact all the technologies created will appear in the “Technologies” menu:



To apply a technology to one or more geometries select the desired technology from the technologies menu.

The geometry selection menu is displayed and the cursor changes to the syringe symbol.



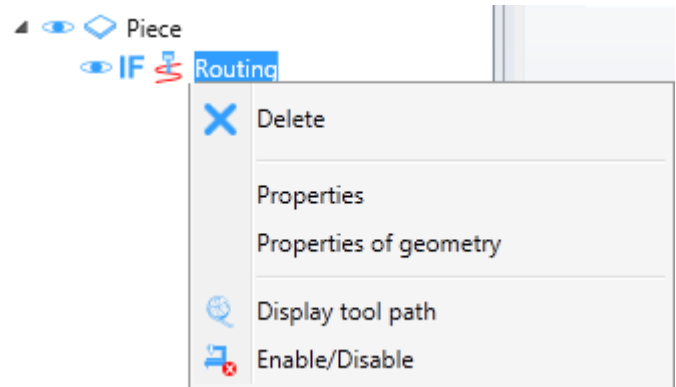
Select a geometry to apply the previously selected geometry.

To apply the technology to just one part of the geometry selected, check the “Partial machining” item in the menu.

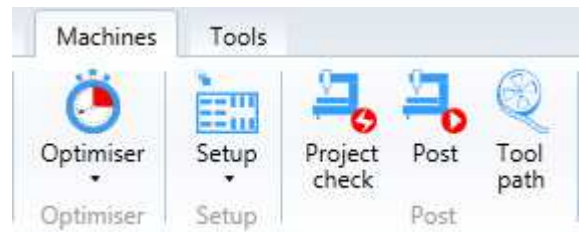
### 3.5.27 Display tool path

The tool path of any machining operation can be displayed with the "Display tool path" command.

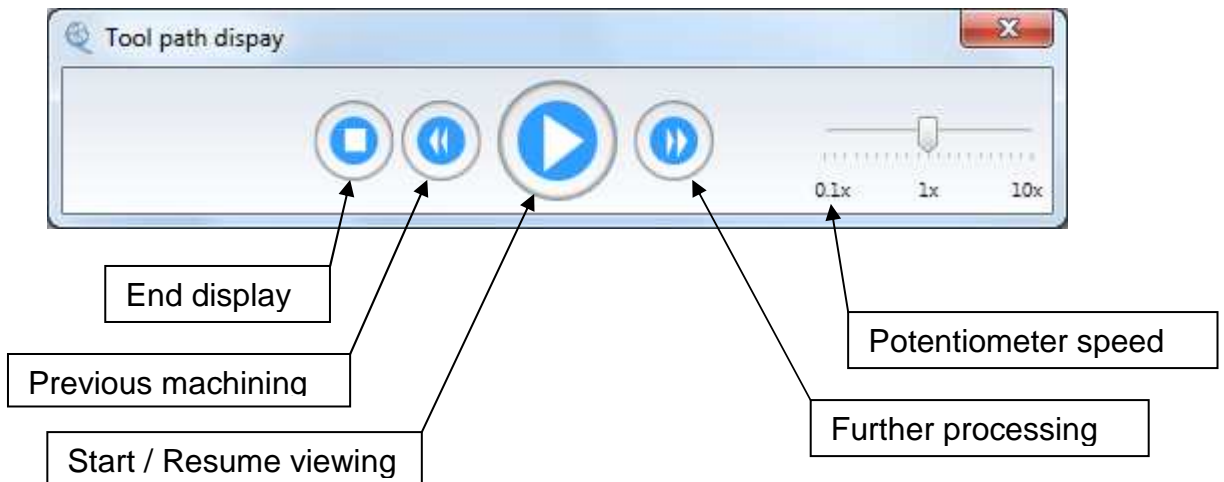
The command can be launched from the context menu of the machining operation selected.



or with the "Tool path" icon in the Machines menu.



After having selected the command the following player is displayed:

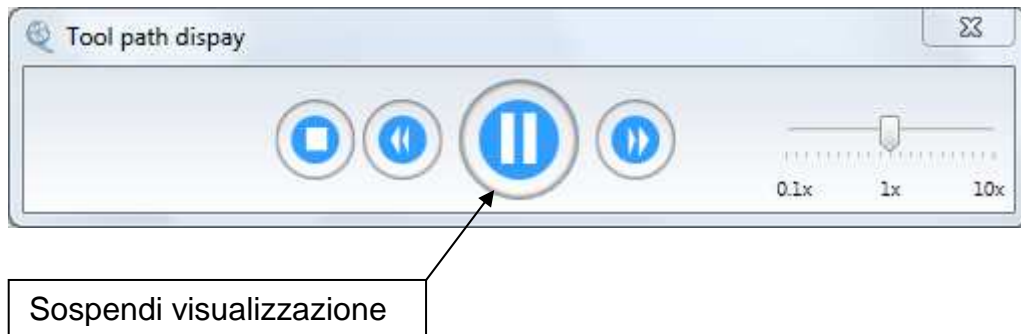


The "Previous machining" and "Next machining" buttons are not available when the command is launched from the context menu of a single machining operation.

Click on "Start/Resume display" to display the tool path, scrolling the tool with the geometry described in the .tlgx file.

The machining speed can be changed with the "Speed potentiometer".

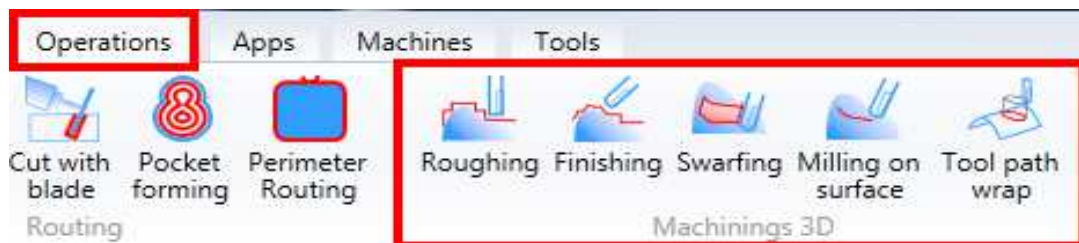
When the tool path is displayed the menu changes as follows:



“Interrupt display” suspends the display and the “Start/Resume display” button will be available.

### 3.6 3D machining

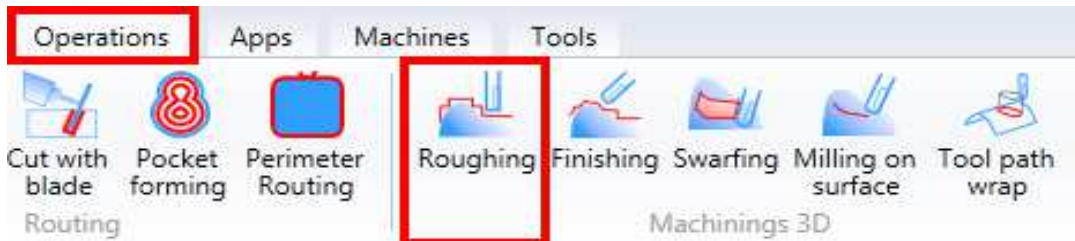
This function is used to create 3D machining operations of surfaces and curve machining on surfaces.



These operations are executed only if the surfaces are within the workpiece volume.

### 3.6.1 Roughing

This command is used to rough a surface and it can be applied to one or more surfaces at the same time with a single tool path to remove as much material as possible from the unfinished workpiece.

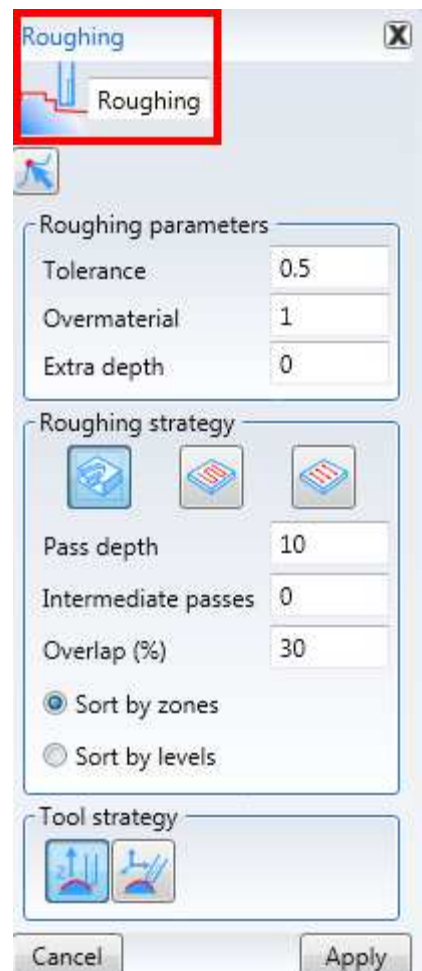


to perform the surface roughing, proceed as follows:

- click on "Roughing"

The specifications of the 3D roughing will be displayed on the right.

The roughing parameters have the same properties of any other machining operation (tolerance, machining allowance and extra depth).



The main properties are the machining strategies:

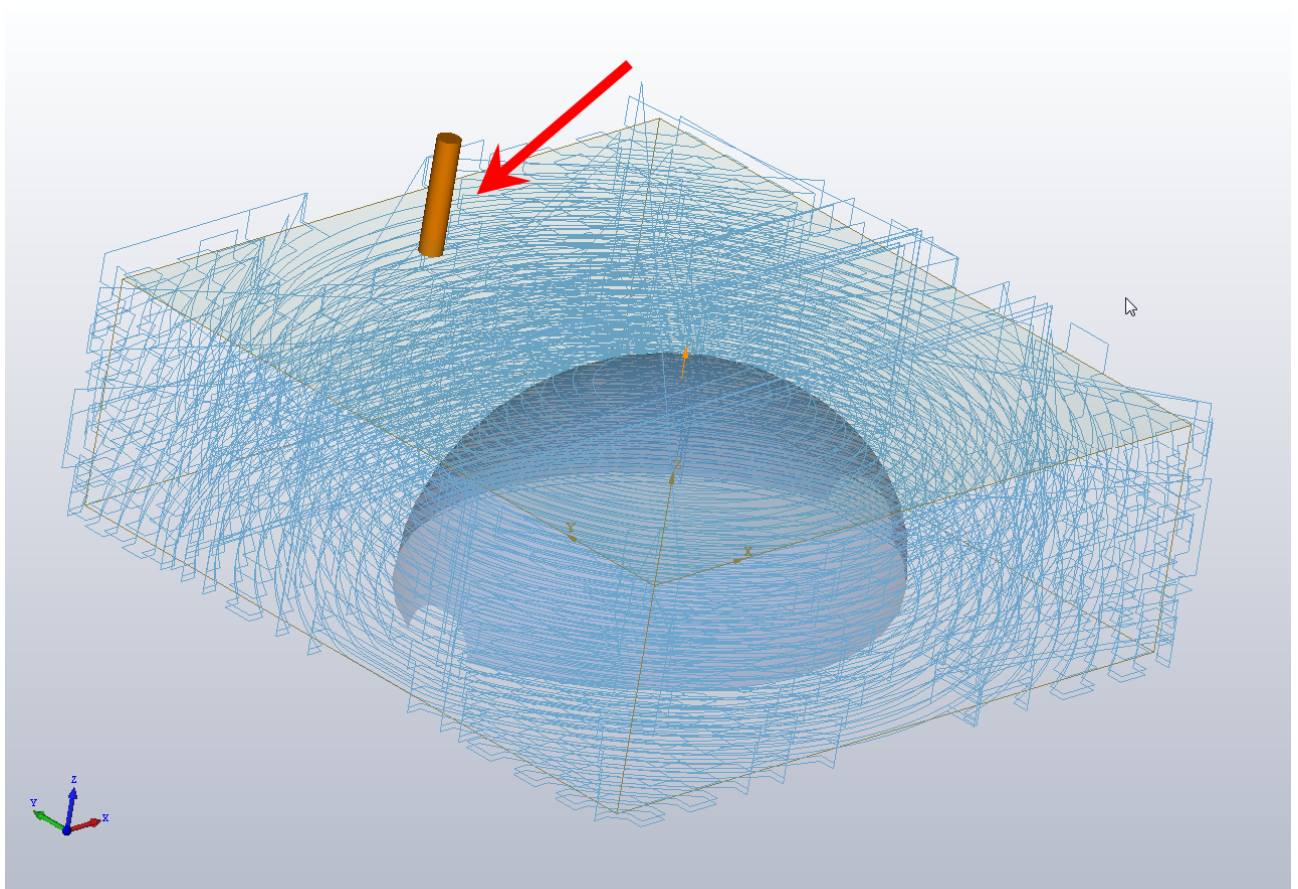
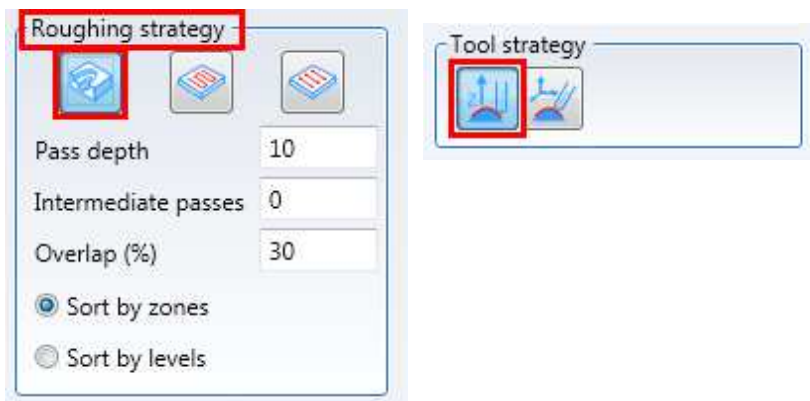
- terraces
- bidirectional
- unidirectional

The tool strategy defines the tool movements in relation to the surface:

- 3-axes management with vertical tool
- 3-axes management with tilting tool

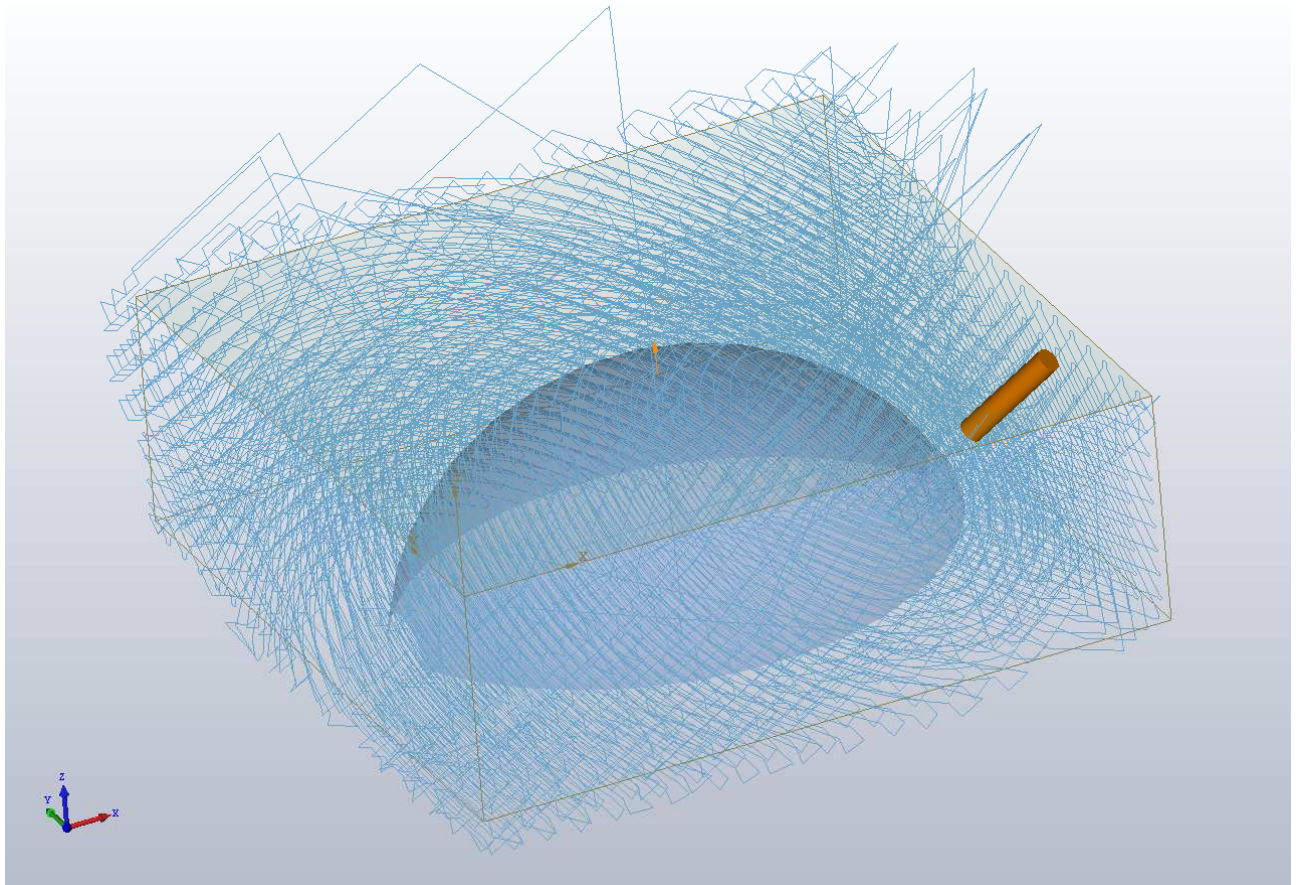
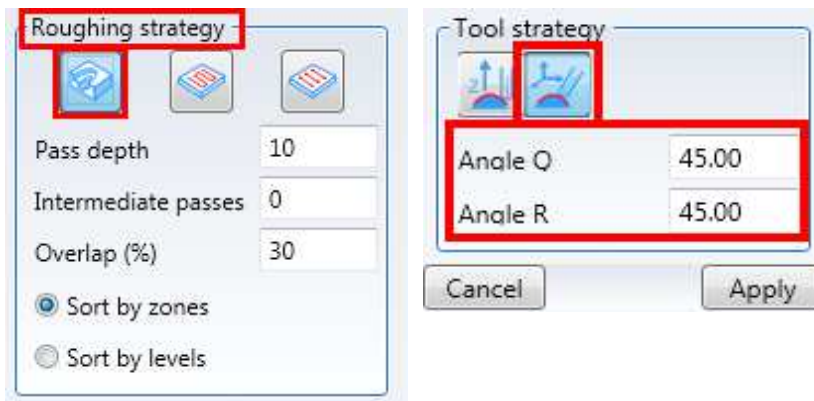
By combining the various strategies you can achieve roughing operations on different surfaces:

- terraces roughing, 3-axes vertical tool






- terraces roughing, 3-axes tilting tool



- bidirectional, 3-axes vertical tool

Roughing strategy



Pass depth 10

Intermediate pass 0

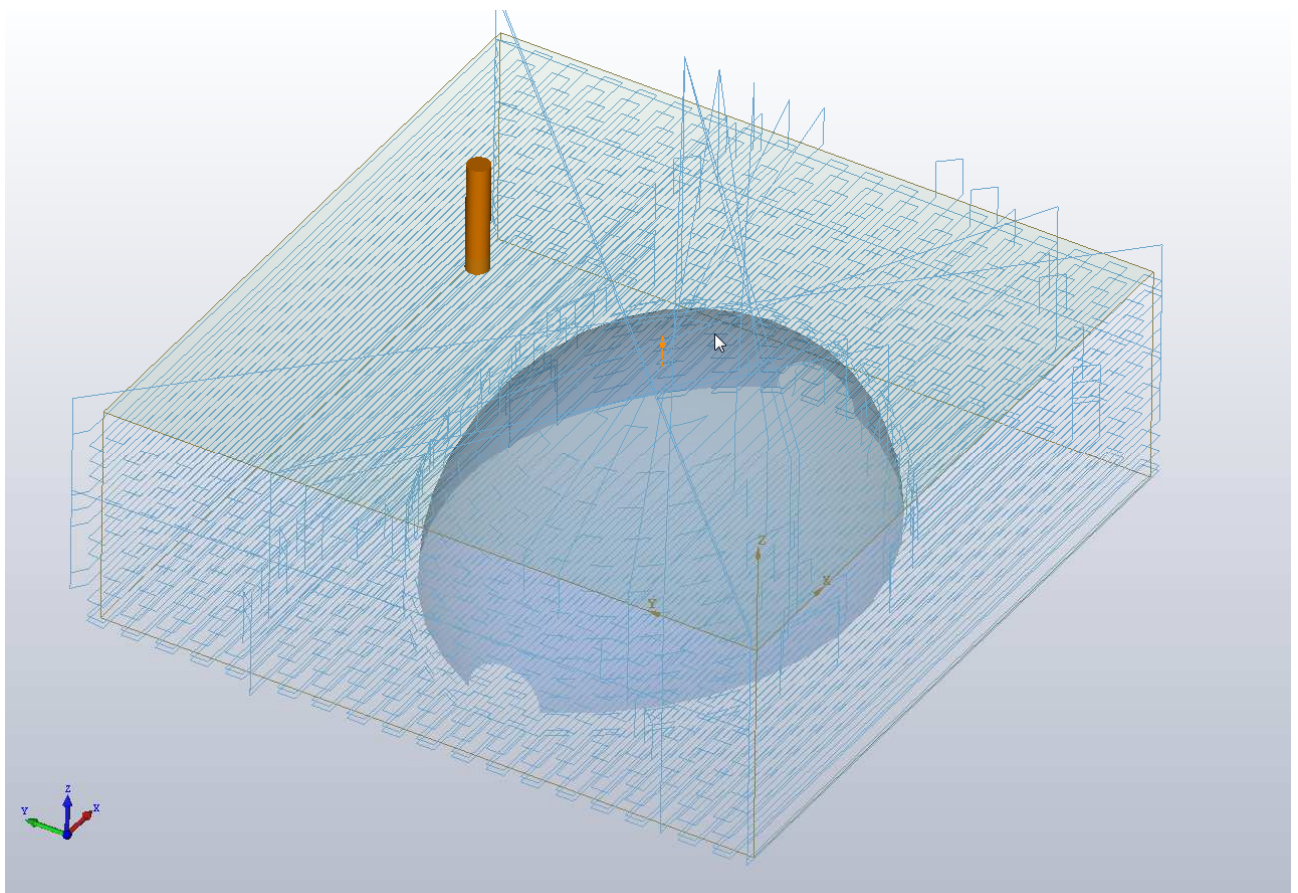
Overlap (%) 30

Pass direction 0

Sort by zones


Sort by levels

Tool strategy



- bidirectional, 3-axes tilting tool

Roughing strategy



Pass depth 10

Intermediate pass 0


Overlap (%) 30

Pass direction 0

Sort by zones

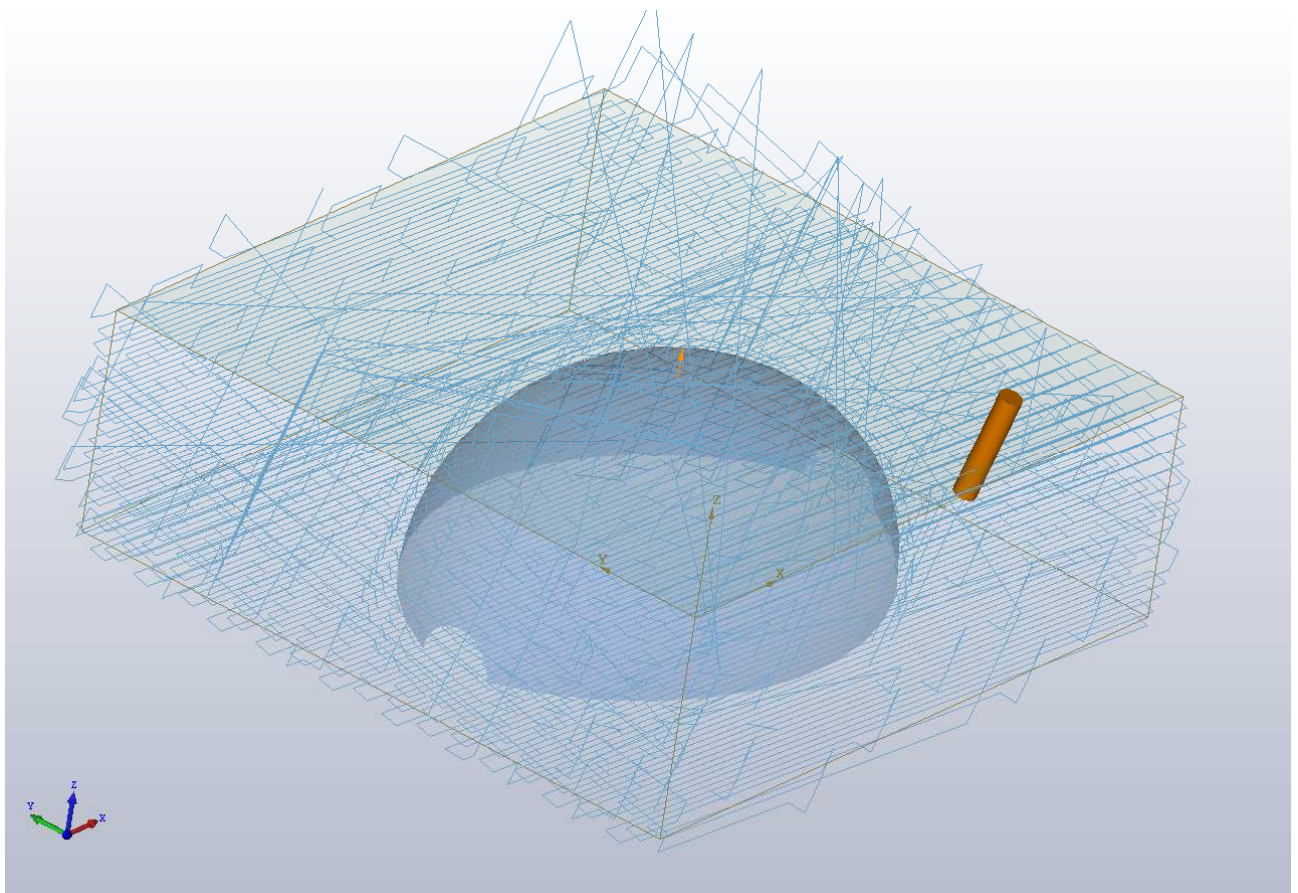
Sort by levels

Tool strategy



Anale Q 15.00

Anale R 15.00



- unidirectional, 3-axes vertical tool

Roughing strategy

Pass depth: 10

Intermediate pass: 0

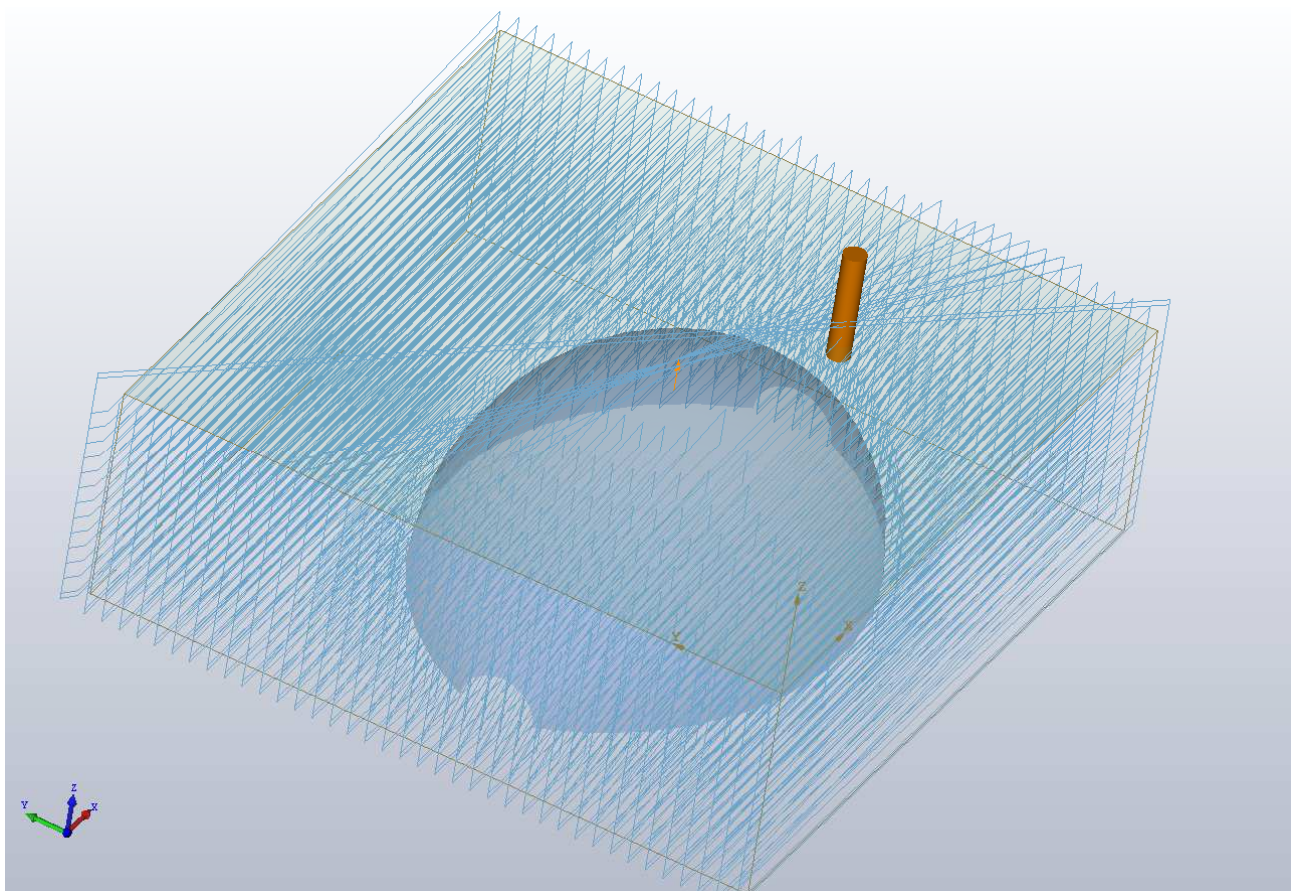
Overlap (%): 30

Pass direction: 0

Sort by zones

Sort by levels

Tool strategy



- unidirectional, 3-axes tilting tool

**Roughing strategy**

Pass depth: 10

Intermediate pass: 0

Overlap (%): 30

Pass direction: 0

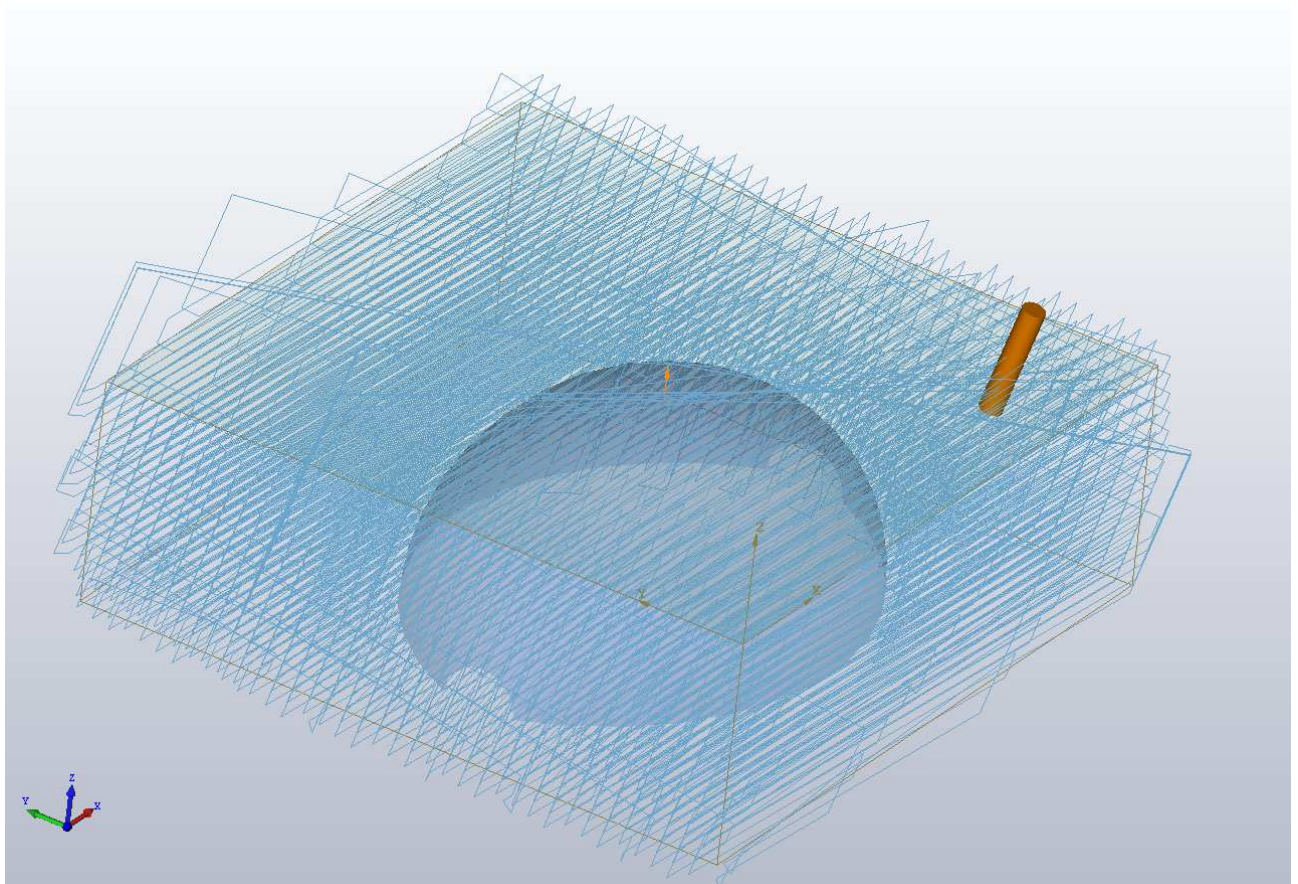
Sort by zones

Sort by levels

**Tool strategy**

Anale Q: 15.00

Anale R: 15.00



## 3.6.2 Finishing

The “Finishing” command is used to machine one or more surfaces. It is located in the “Operations” “3D machining” menu.



After having selected the command the menu at the side will be displayed.

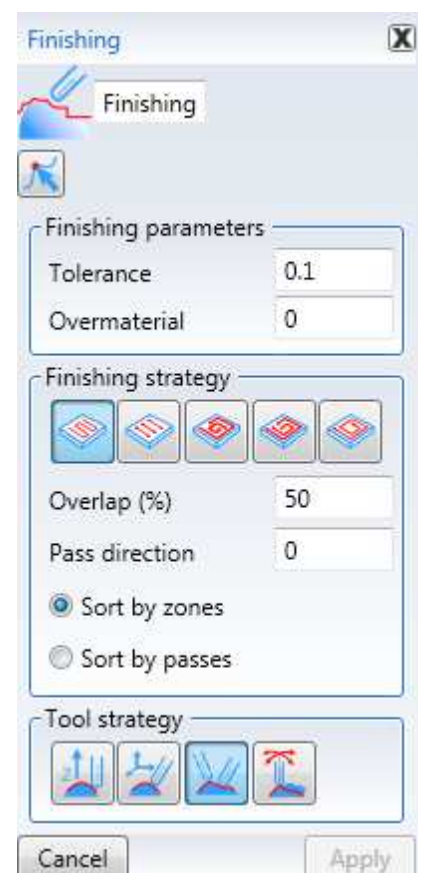
The surface can be selected after having clicked on “Select geometry”. A tolerance and a machining allowance in relation to the surface can be set.

There are 5 finishing strategies that will be illustrated with 6 examples.

The Superimposition represents the percentage with which the tool passes again on the previous machining.

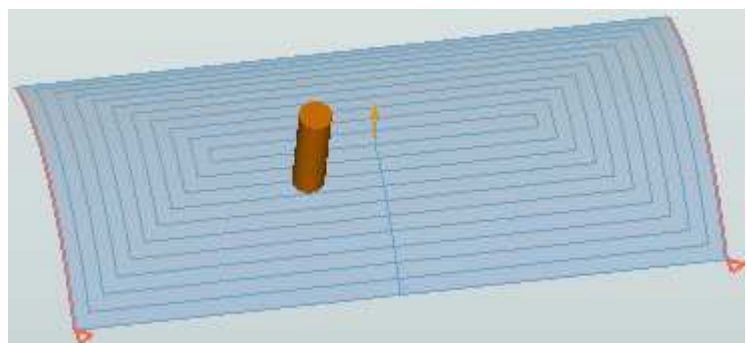
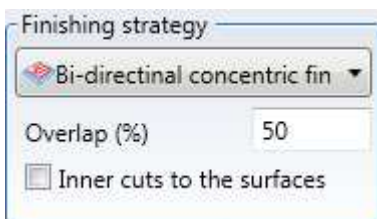
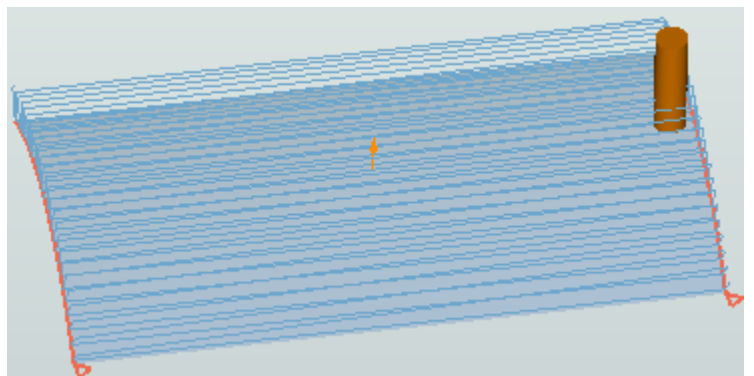
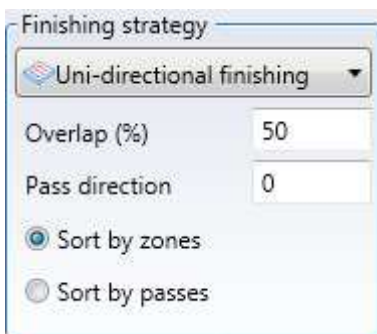
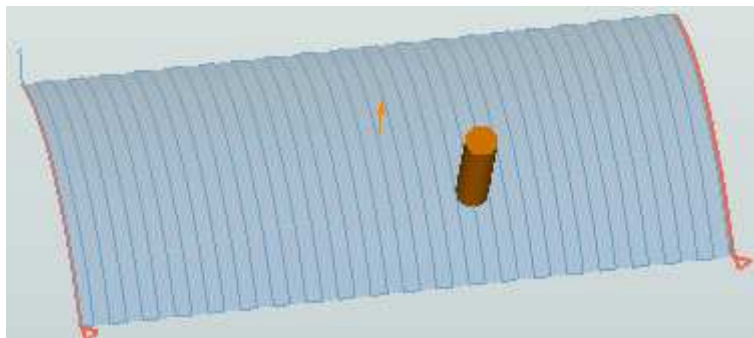
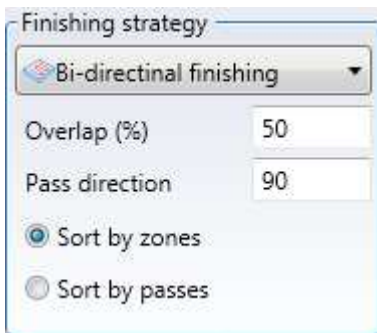
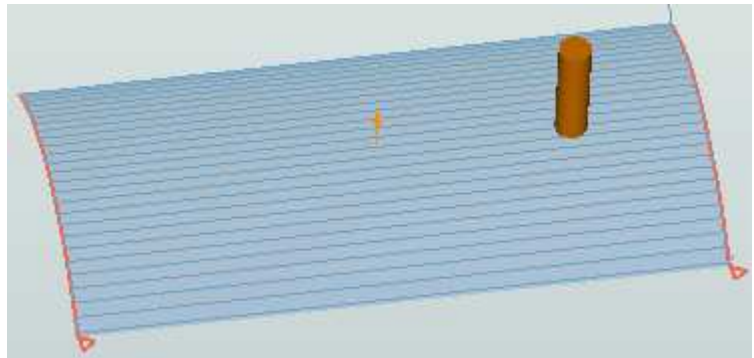
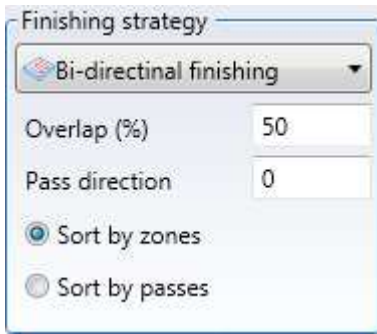
The pass direction indicates an angle in relation to the X axis (0° along X – 90° along Y).

There are 4 tool strategies that will be illustrated with 4 examples.



### 3.6.2.1 Finishing strategies

Six examples will be illustrated for the five finishing strategies available. The first two will have the same strategy with two right angle pass directions. All the examples use the “normal to surface” tool strategy and the tool strategies will be described below.

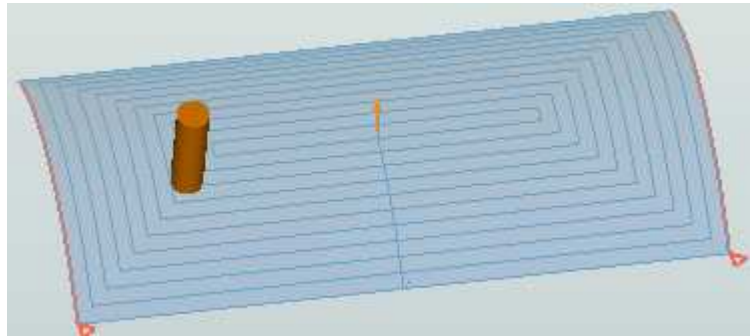


Finishing strategy

Uni-directional concentric

Overlap (%) 50

Inner cuts to the surfaces

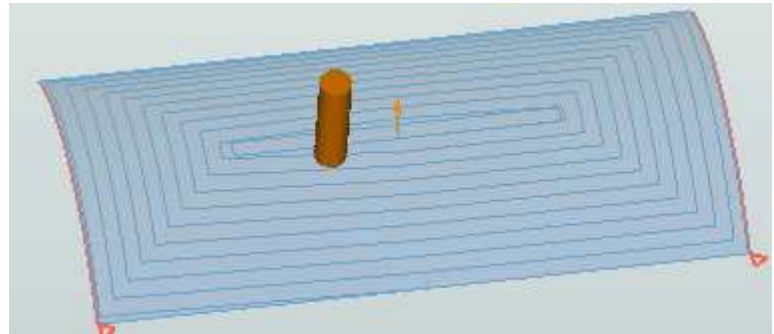


Finishing strategy

Spiral finishing

Overlap (%) 50

Inner cuts to the surfaces



Finishing strategy

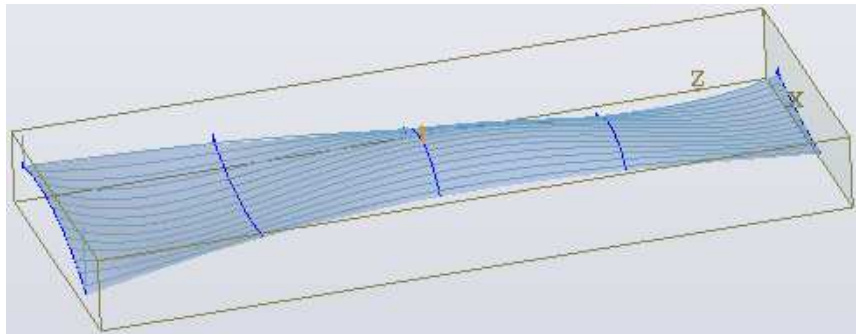
Finish along isoparametric

Overlap (%) 50

Inner cuts to the surfaces

Main direction

Crossways direction



Finishing strategy

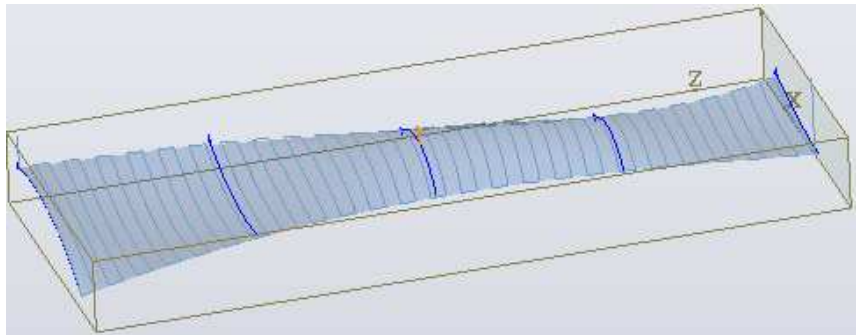
Finish along isoparametric

Overlap (%) 50

Inner cuts to the surfaces

Main direction

Crossways direction



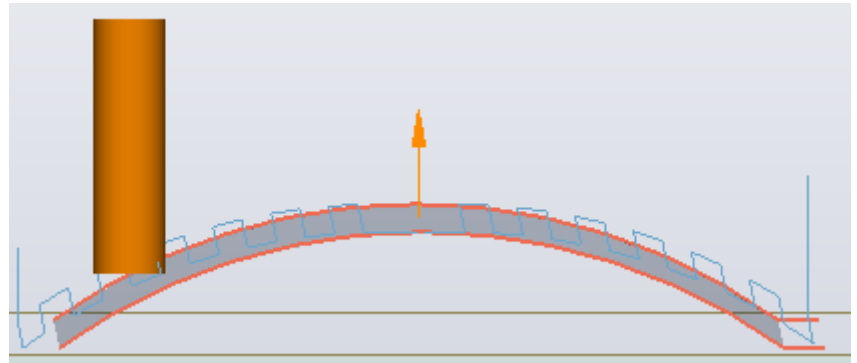


### 3.6.2.2 Tool strategies


Tool strategy



Operation with 3 axis with vertical tool

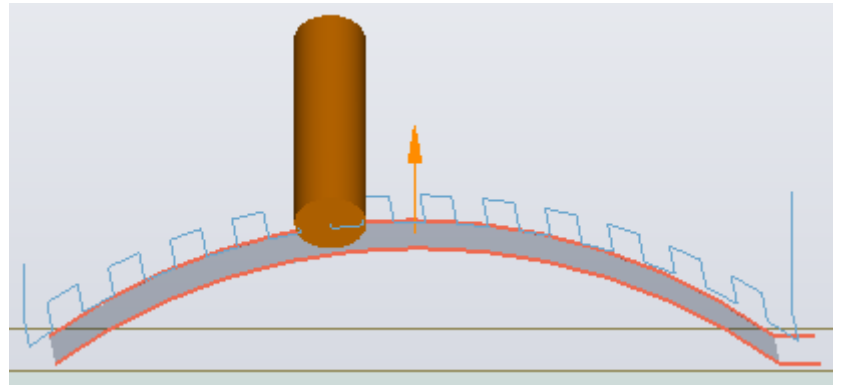


Tool strategy




Operation with 3 axis with tilted tool

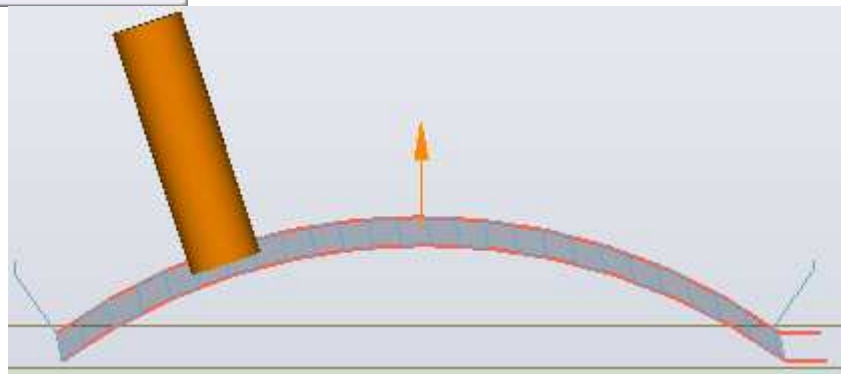
Angle Q	45.00
Angle R	90.00




Tool strategy



Operation with 5 axis with tool always normal to surface

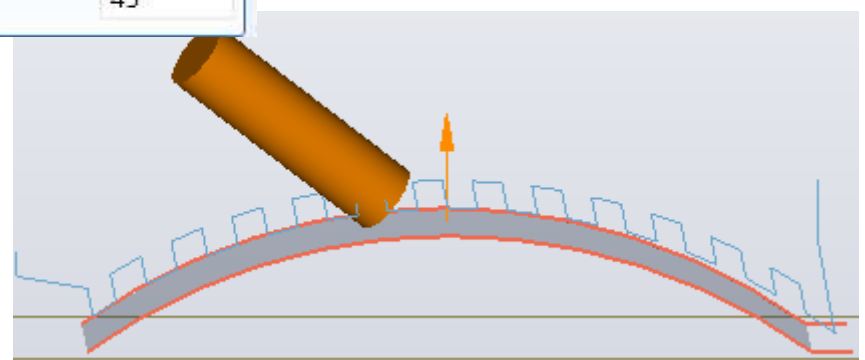


Tool strategy



Operation with 5 axis with tool tilted in relation to tool path

Angle in advance	30
Side angle	45



### 3.6.3 Swarfing

Command used to create surface machining with tool side. It can be applied to two curves, an upper one and a lower one, or directly to one or more surfaces.

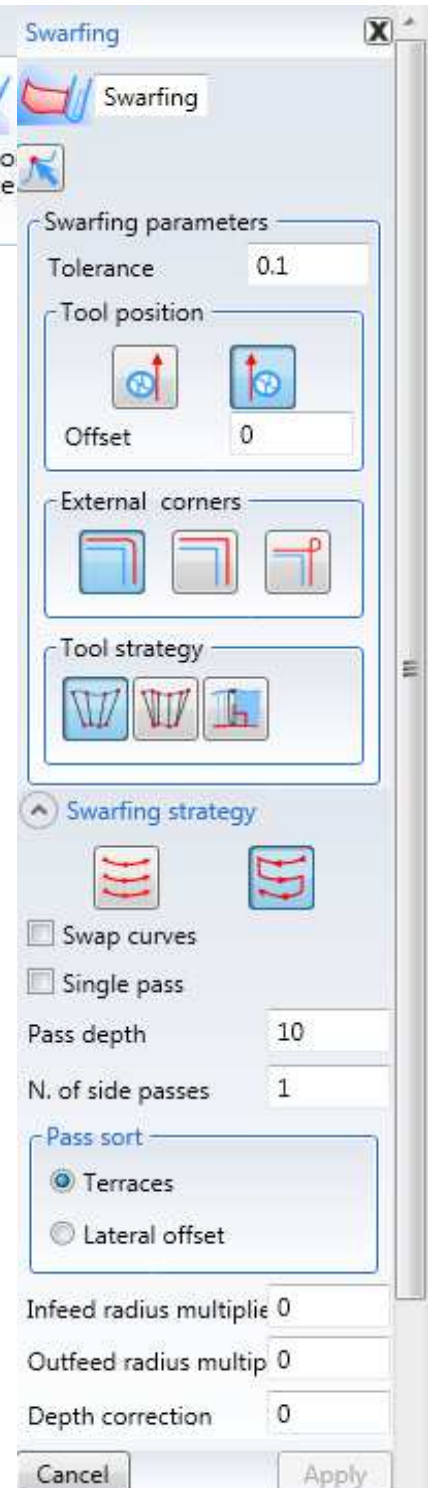
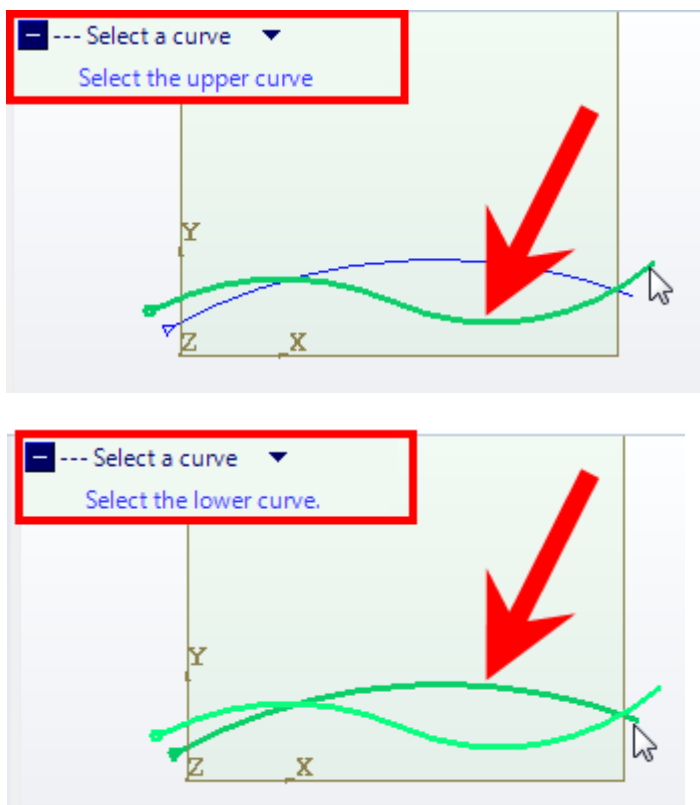
N.B.

To perform the machining directly on the surface, firstly select the surface and then click on “Swarfing”. Now the surface is recognised automatically and you can proceed with the direct creation of the machining operation.

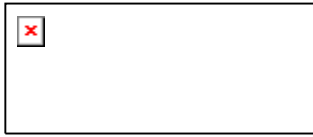


Once the “Swarfing” 3D machining is open the machining menu will be displayed, to select the various options.

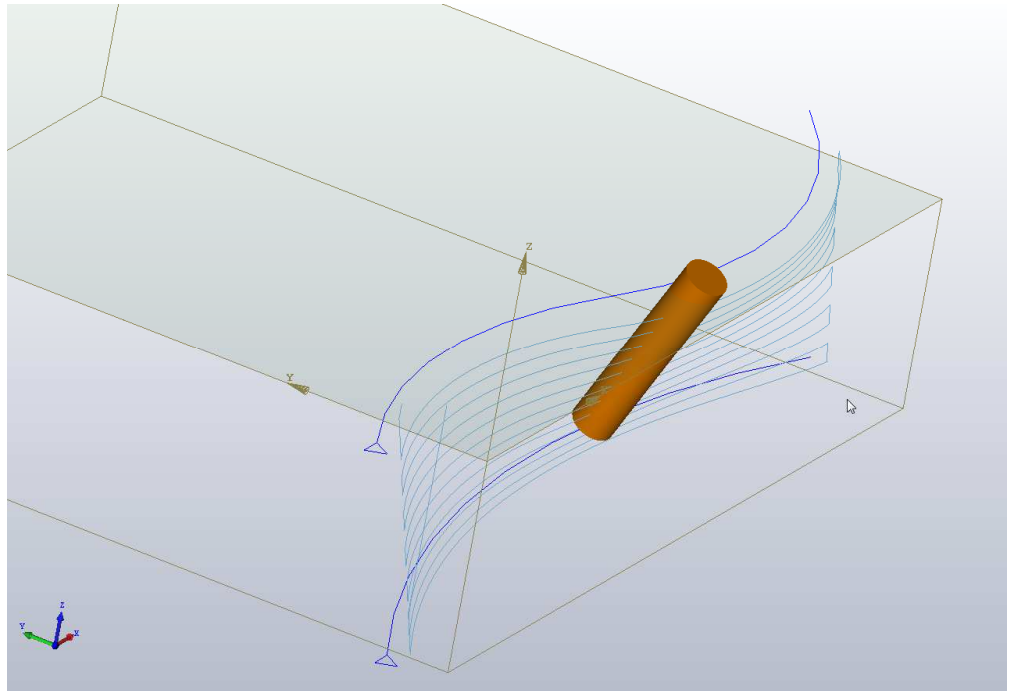
You will also be asked to select the upper curve and then the lower curve to create the surface in which the machining must be generated.



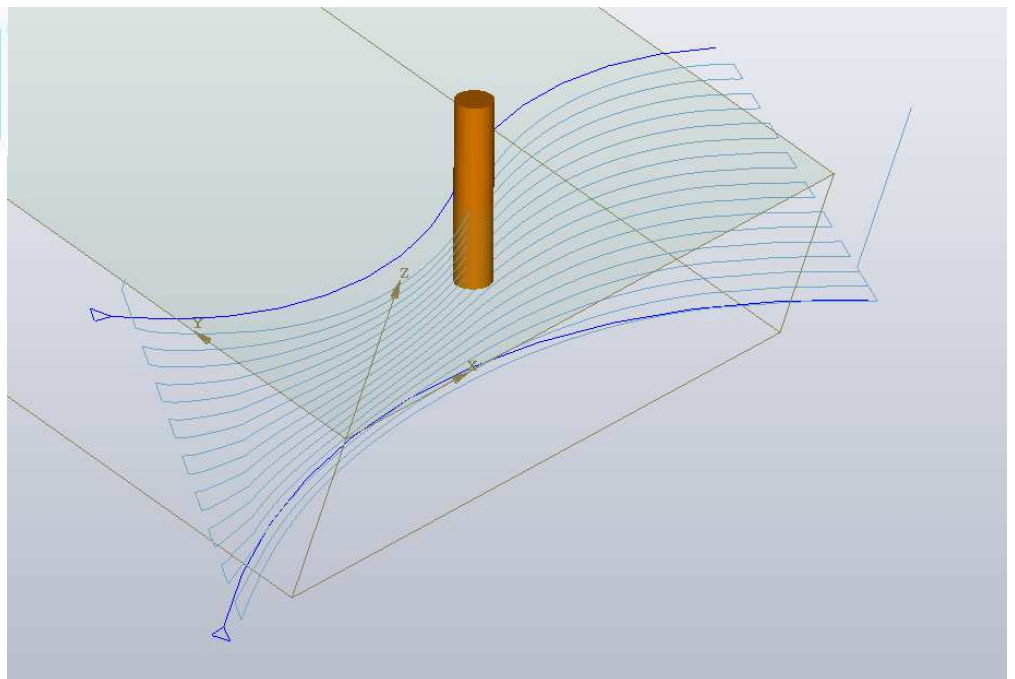
Also in this case by combining the various strategies you can achieve machining operations with different results.



Tool strategy  
Synchronise curves

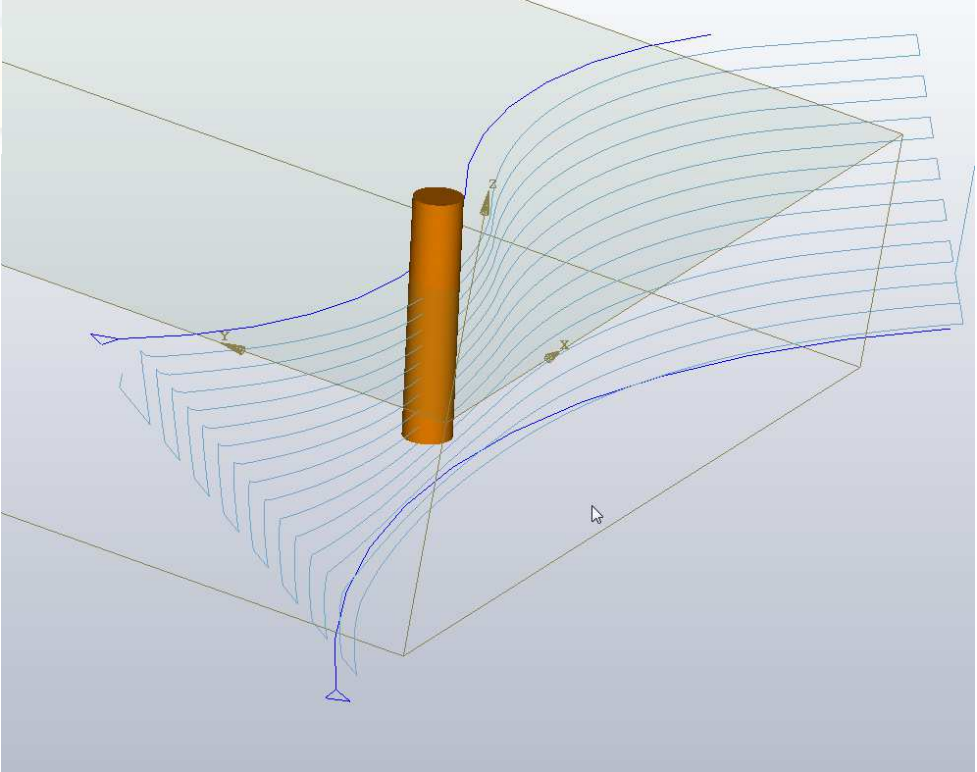


Tools strategy  
Minimum distance





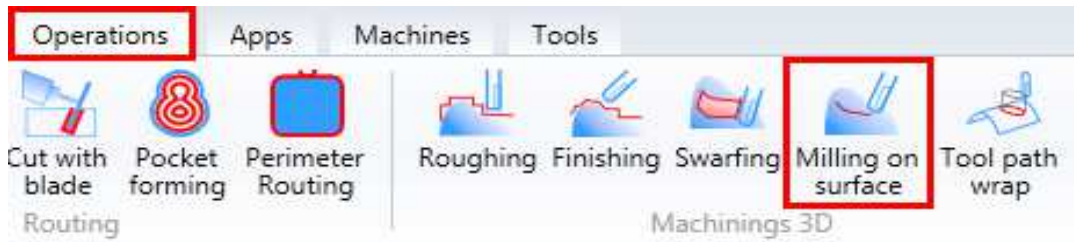
Tool strategy  
Perpendicular to the  
lower curve



### 3.6.4 Routing on surface

The “Routing on surface” command is used to perform the machining of a 3D curve that lies on a surface.

It is located in the “Operations” “3D machining” menu.

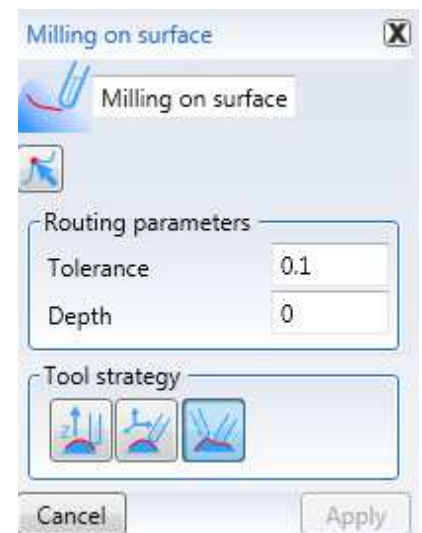


After having selected the command the menu at the side will be displayed.

A precision tolerance can also be set.

A depth can be set; with “0” the tool will skim the surface.

The tool strategies are three of the five described for the “Finishing”.

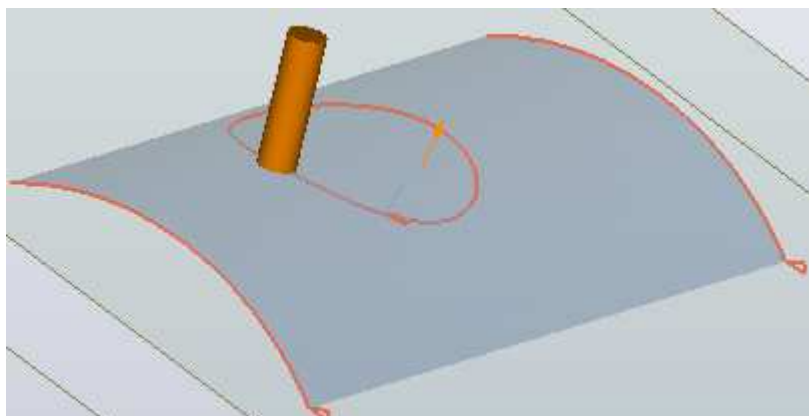


To proceed with the machining select a 3D curve and then select a surface.

The following commands will be displayed:



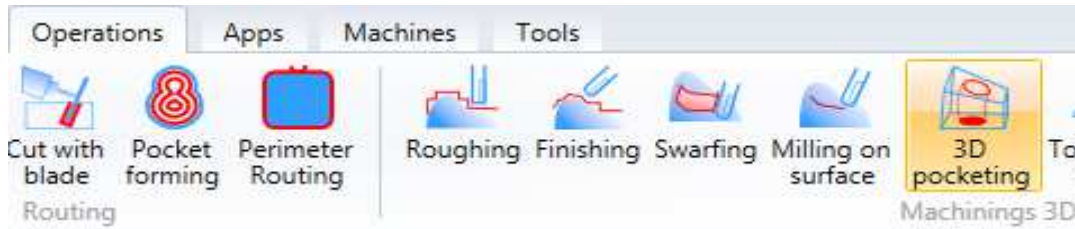
Example:



### 3.6.5 3D pocketing

The "Emptying 3D" command allows you to create a pocket by selecting the surfaces that delimit it.

It is located in the "Operations" "3D machining" menu.



To perform the processing, select the following surfaces:

- Surface ▼  
Select the lower surface of the cavity
- Surface ▼  
Select any boundary surfaces
- Surface ▼  
Select any higher reference surface

The bottom surface is mandatory, the side ones are optional because if you select only the bottom one would be completely cleaned.

Any upper surface will only be used when the bottom surface is not parallel to the upper level, in this case one of the following can be specified

emptying strategies which are (from left to right):

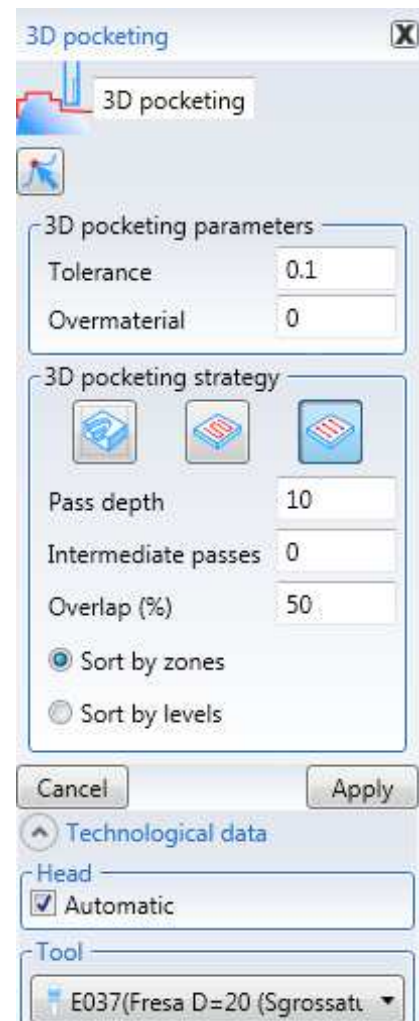
- Offset from bottom of the pocketing
- Adaptive offset from the start to the bottom of the pocketing
- Offset from the start of the pocketing

The Offset from the bottom runs parallel passes to the bottom of the pocket.

The Offset from the beginning of the pocketing executes passes parallel to the upper surface.

The Adaptive Offset performs variable shape passes between the bottom of the pocket and the upper surface.

The remaining parameters are analogous to those present in roughing machining.

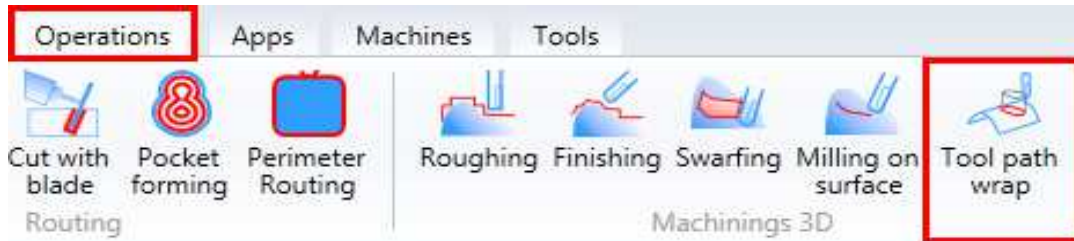


### 3.6.6 Tool path wrap

The “Tool path wrap” command is used to apply a tool path of a drilling, routing or pocketing on a surface.

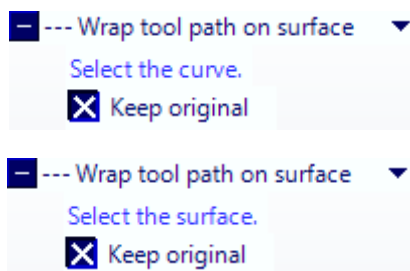
The original tool path will be wrapped on a surface and the tool orientation will be perpendicular to the surface.

It is located in the “Operations” “3D machining” menu.



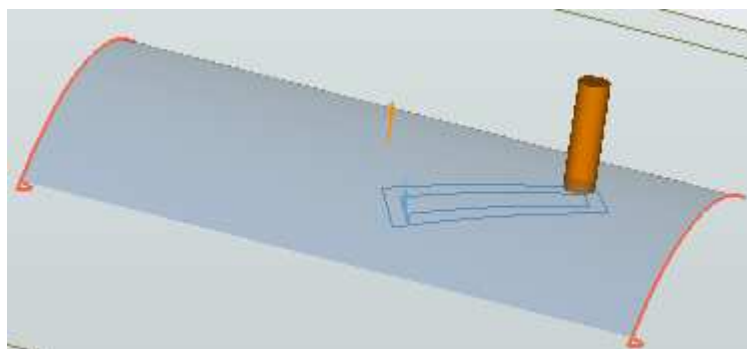
To proceed with the machining select a machining tool path and then select a surface.

The following commands will be displayed:



The original machining will be transformed in a 3D routing with the same specifications.

Example of a pocketing transported to the surface:



## 3.7 Macro and Subprograms

### 3.7.1 Creating machining-macros (.xsp)

The “.XSP” Macro are a sequence of instructions used to add geometries or machining operations to a project.

The macros can be launched more than once in the same project and the names associated to them will be transcribed in the workpiece machining tree, as if they were elementary machining operations.

The Maestro installation package contains various “.XSP” macros that can be used as outline to create new macros.

The macro creation procedure can be described in two phases:

1. Creation of the macro basic structure ( wizard);
2. Implementation of the functions in the macro.

#### 3.7.1.1 Creazione della struttura base della macro ( wizard);

During the planning phase the user must decide which parameters to supply as input variables for the macro. The workpiece dimension parameters are passed to the macro with the wizard, clearly for the user and therefore do not have to be parameterized. The remaining parameters must be entered manually by the user, with the wizard as described in the subsequent procedures.

To start the macro creation wizard select *Tools* in the Maestro menu bar, and press *Macro Management* and select *Create macro*.



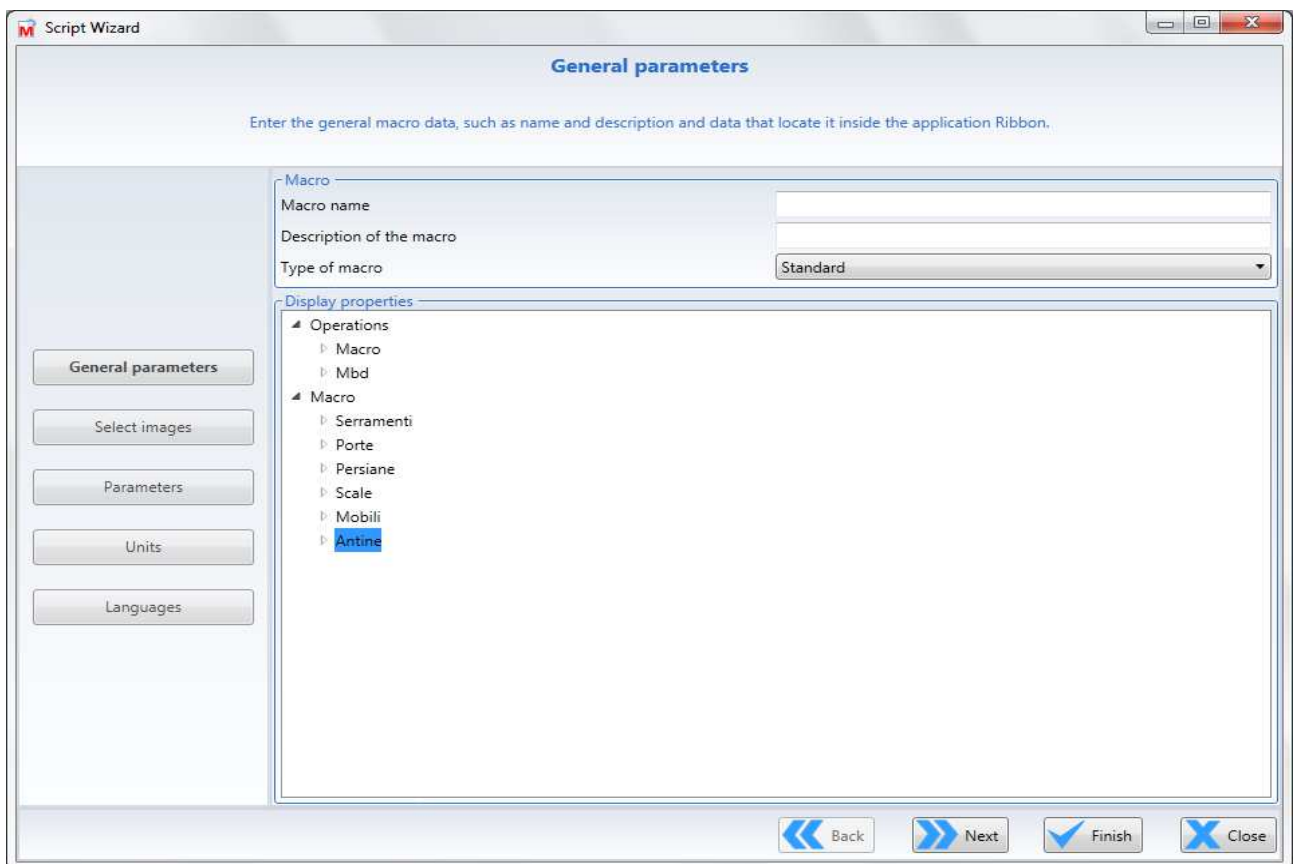


The wizard window is displayed on the screen, used by the user to access four different areas: general parameters, image selection, parameters, units and languages.

### 3.7.1.1.1 General parameters

In this section, in the first field the user must associated an identification name to the macro, whilst in the remaining fields a short function description, a category, unit and sub-unit type identifier can be entered.

These last fields can be used to organize the macro properties hierarchically, in order to retrieve them more easily when required, from the Maestro menu. In particular there are two options for the category: "Operations" and "Macro". The first category enters the macros in the "Operations" tab , whilst the second in the "Macro" tab, in the Maestro menu. If there is no Macro tab, it is created the first time a macro of this category is created. Based on the category chosen groupings and sub-groupings can be created with the following fields

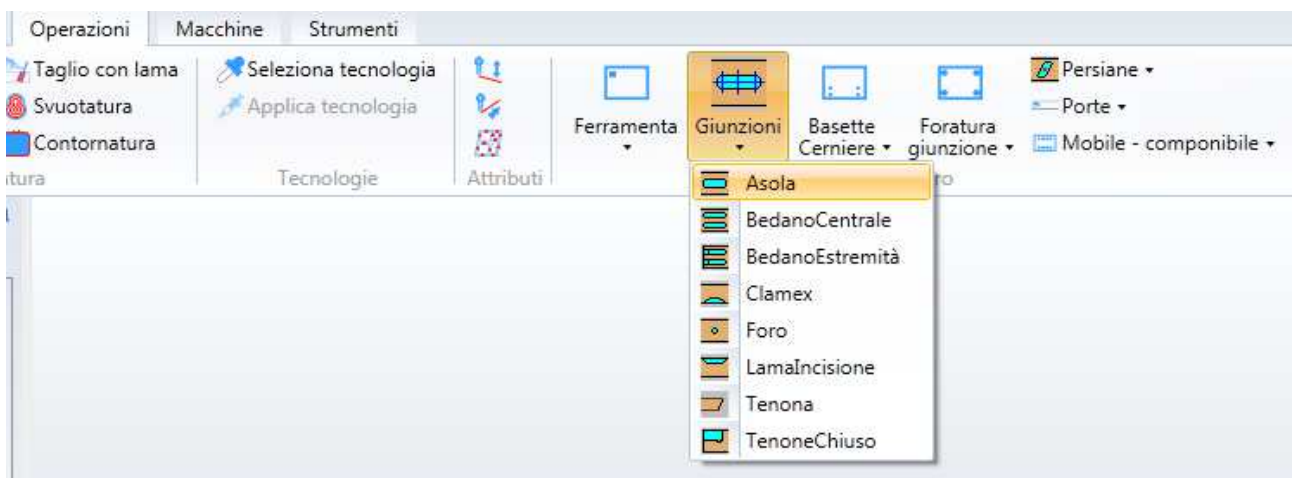


For example the general parameters for the Slot macro are:

<b>Macro</b>	
Nome della macro	Asola
Descrizione della macro	
Tipo della macro	Standard
<b>Proprietà di visualizzazione</b>	
Nome della categoria	Operazioni
Nome del Gruppo	Macro
Nome del Sottogruppo	Giunzioni

These parameters define the Slot macro as a “Standard” macro, of the “Operations” category, “Macro” unit and “Joints” sub-unit.

The user can check that the macro is placed in Maestro, in the specified sub-unit.

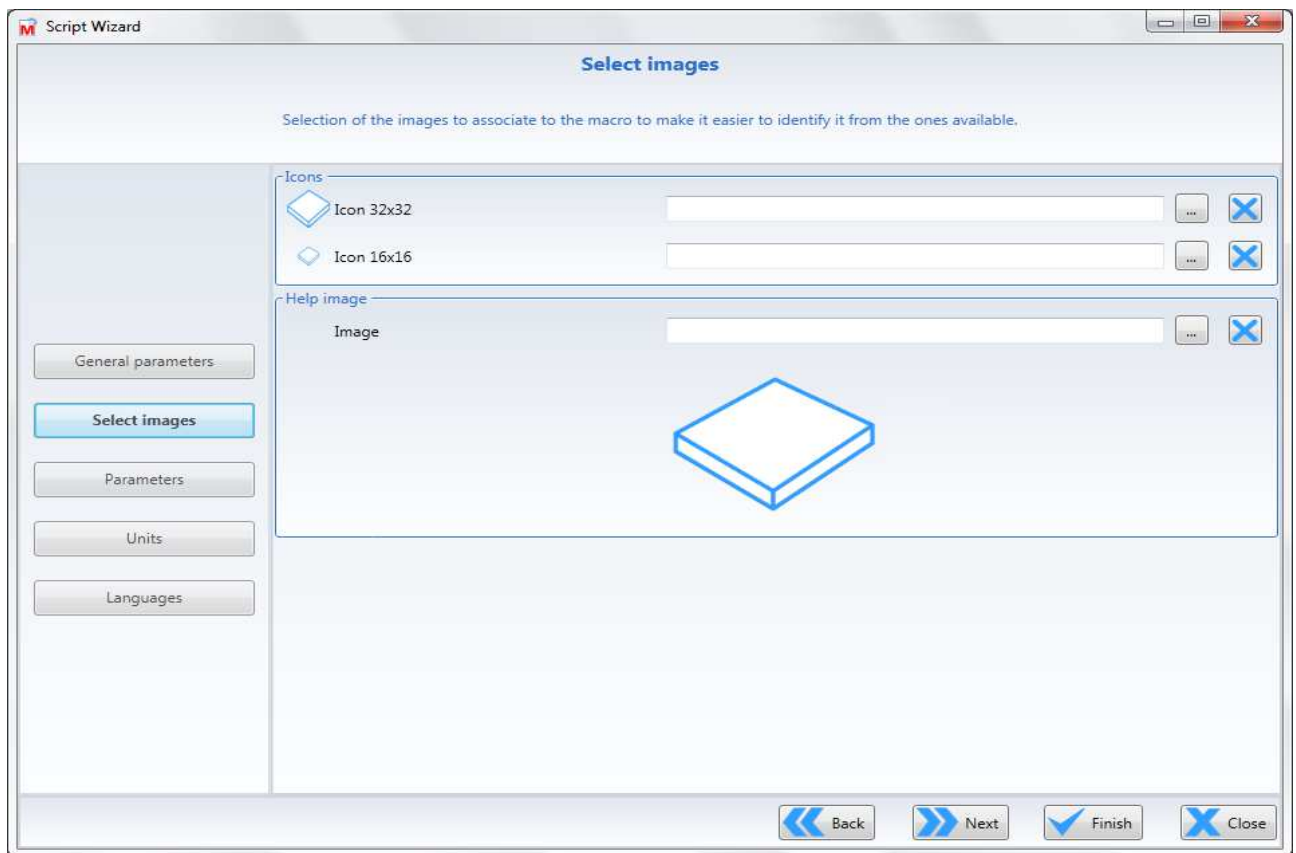


### 3.7.1.1.2 Image selection

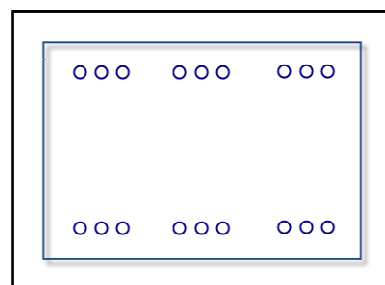
This section is used to associate different types of images to the macro:

- 16x16 pixel image: visible in the Maestro menu on the left of the macro (used when small bitmaps are required);
- 32x32 pixel image: visible in the Maestro menu on the left of the macro (used when large bitmaps are required);
- 200x154 pixel image: visible with the macro parameters.

The following formats are supported: .bmp or .jpg or .png



In general it is good practice for the 16x16 and 32x32 images to provide a general idea of the macro function, whilst the 200x154 image must provide the detailed meaning of the macro input parameters. See the example below:

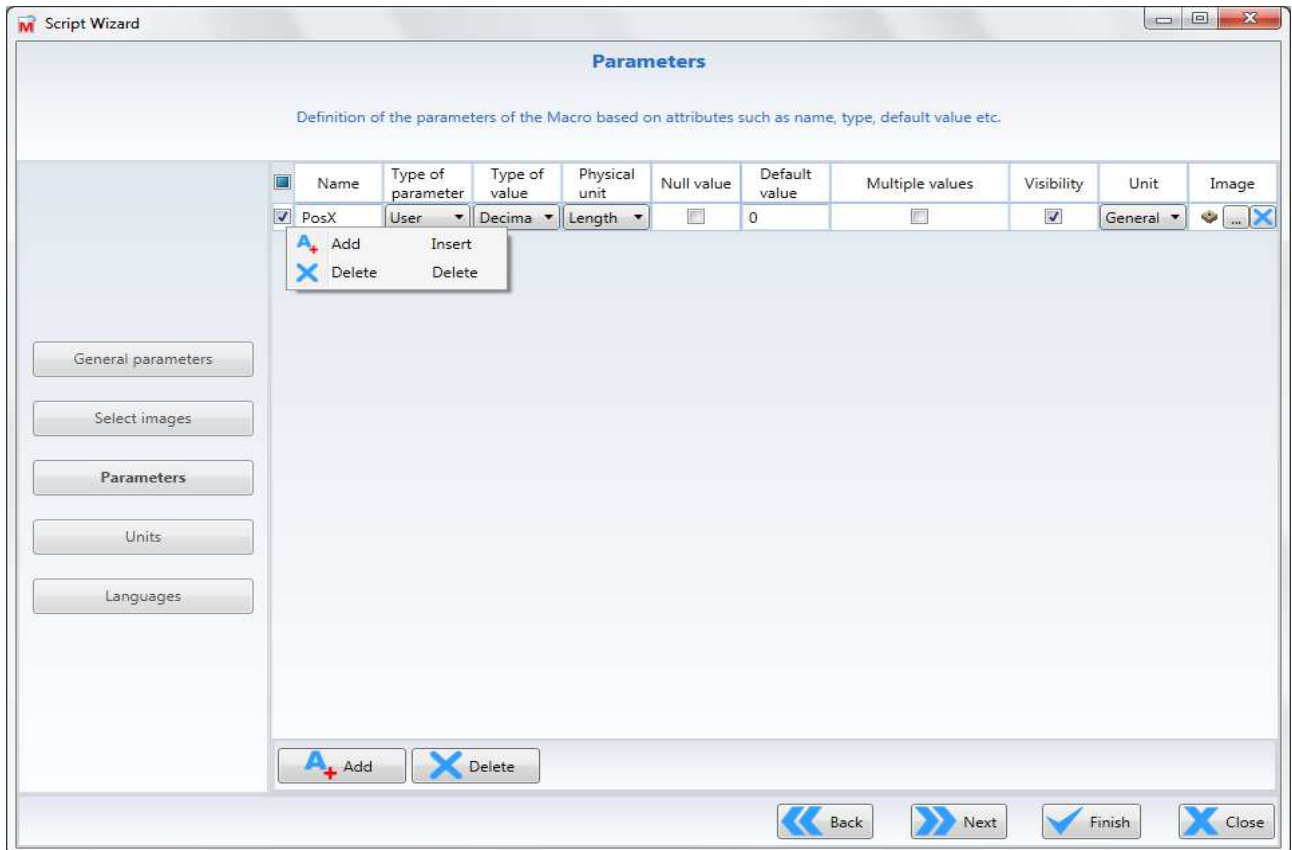


### 3.7.1.1.3 Parameters

This section is used by the user to describe the macro input parameters. Usually the input parameters for a macro are divided in:

1. geometric parameters;
2. technological parameters (tool, strategy, etc...).

To add an input parameter, right click on the mouse and select *Add* in the menu.



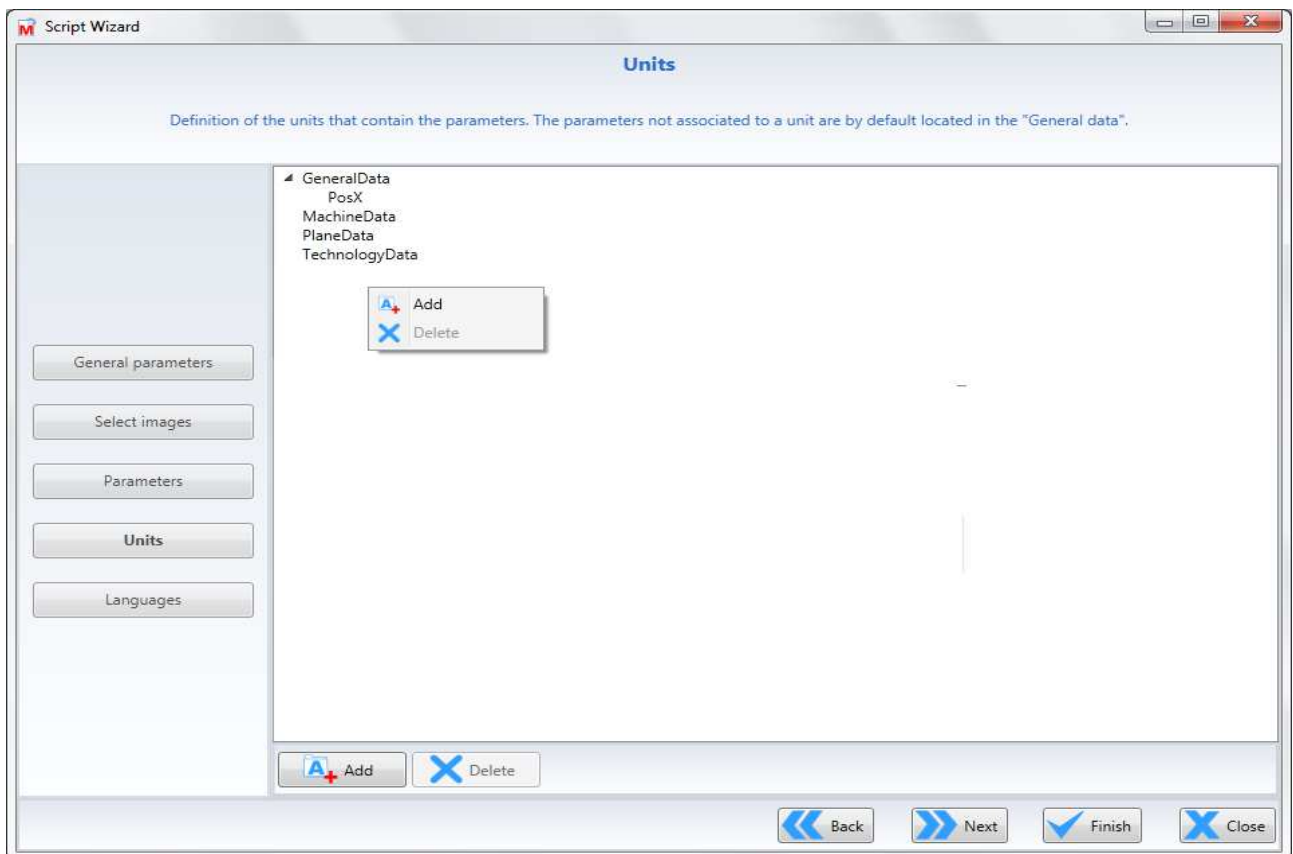
The following fields can be defined for each parameter in the table:

- Name parameter identification name (name of the variable used for the macro input parameter);
- Type of parameter: defines the type of parameter (selected from: machining head, tool, user and worktable);
- Type of value: defines the type of value in the parameter (selected from: Boolean, decimal, integer, string);
- Physical unit: defines the unit of measurement associated to the parameter value (length, speed or dimensional size);
- Null value: specifies if a null value can be assigned to the parameter;
- Default value: used to set a default value for the parameter;
- Multiple values: defines a restricted group of elements and ties the parameter value to this group (if activated the group elements must be defined in terms of key/value);

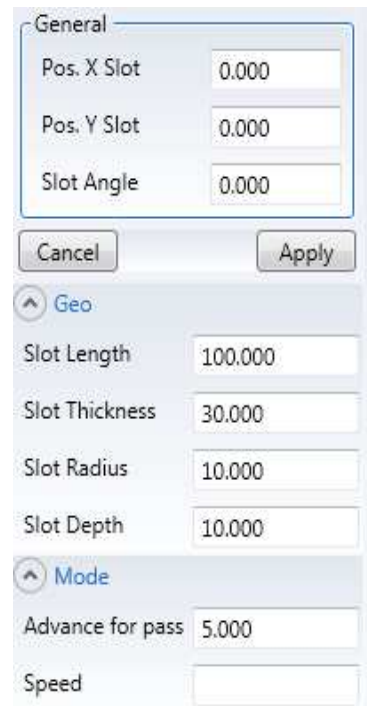
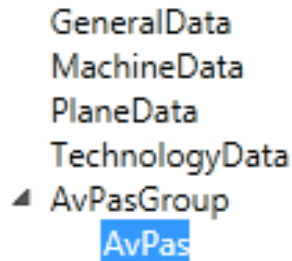
- Visibility: allows the visibility of the parameter in the interactive use of the macro (in the Maestro expander);
- Group: defines the group of the parameter;
- Image: used to associate a descriptive image to the parameter.

### 3.7.1.1.4 Groups

In this section the user can decide how to group together the parameters entered in the previous section, so that they appear in the desired group in the macro configuration menu.

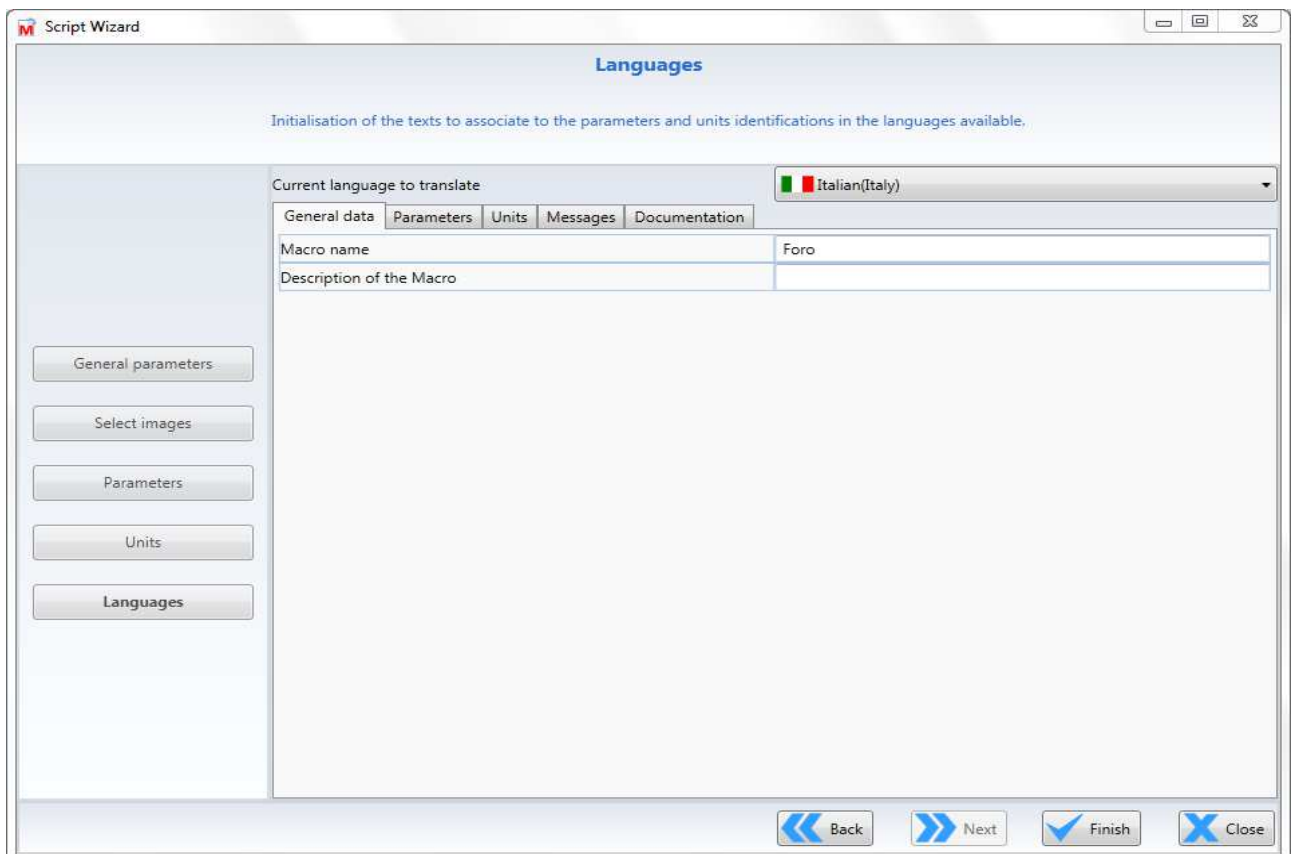


For example for the Slot macro, the pass feed parameter (AvPas) is defined in the AvPasGroup group and corresponds in the Maestro expander to the “Strategy” category of the image on the right



### 3.7.1.1.5 Languages

In this section the user can translate the descriptive information of the macros in the desired languages. In particular, in the Parameters tab, the user must associate to every previously entered parameter, the name that will be associated to the parameter and that will be displayed in the Maestro editor. Or he can also enter a brief description of the parameter use. These operations must be repeated by the user for every language that the macro will be available in.



Similarly in the Groups tab the group ID created can be translated and a brief description in the relative language can be associated.

In the Messages tab various error message can be translated, according to the key/message layout. To recall the desired message when there is the corresponding error condition, use the `Print` scripting instruction. This function uses as input parameter the key associated to the message, preceded by the special character `@` (the `@` character is used to identify and interpret the string that follows as a message key).

In the Documentation tab, any documents in the relevant language can be associated to be available in the macro.

The flag can be enabled to help in the compilation and to copy the instructions from another language, in order not to have to start from zero for each new language introduced.

Once the wizard procedure is finished (End key), the macro is created and saved in the Macros folder of Maestro, that is usually found with this path *C:\Program Files\Scm Group\Maestro\Macros*.

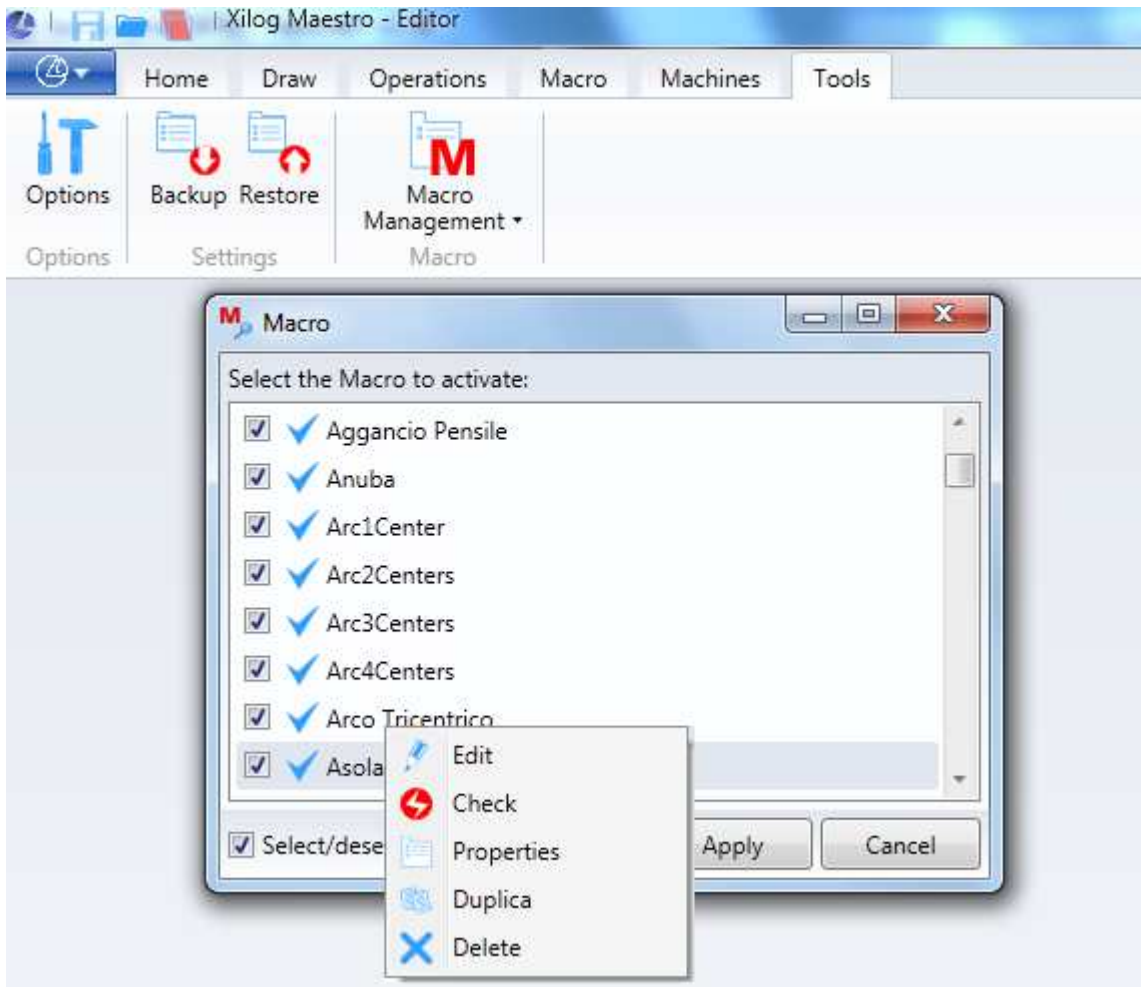
With the completion of this first phase the basic structure of the macro is created, but no function has been implemented inside it. The wizard has in fact created in the `.xsp` file of the macro a source file with extension `.cs`, with the parameters and values supplied by the user, but the user has not described the behaviour in the execution of the file.

At this point the user must define the macro behaviour with the MSL scripting language. Maestro must be restarted to proceed with this second phase (implementation of a macro). Now this file can be accessed and the desired specifications can be implemented and edit the macro created as the described in the next section.

### 3.7.1.1.6 Macro Management

To access the macro management function select *Tools* in the Maestro menu bar, and press *Macro Management* and select *Display macro*.

A window is displayed containing all the macros of Maestro and the ones created by the user. Move on the desired macro and right click on the mouse to access the menu.



The flag in the list, on the left of each macro, allows the macro to be viewed or blocked from the Maestro interactive menu. This does not mean that the macro will be deleted, but just that it will be visible from the menu. At any time the user can access the complete list of all the macros and change the display status of the ones that he is interested in.



Select *Edit* to change the .cs file of the macro.

As the ideal solution to editing the macro is using Visual Studio 2010 or a later version, we recommend using this tool, where possible. If the user does not have this tool, a free version called Express can be downloaded at the website: [www.microsoft.com/visualstudio/](http://www.microsoft.com/visualstudio/).

See the MSL manual for the structure and content of the .cs file.

Select *Check*: to check if the macro is correct. In this case the macro is checked for any file and parameters creation errors. The result of the check will be displayed on the screen.

Select *Properties*: to open the wizard graphic interface again. At this point the basic structure of the macro can be changed by adding parameters or changing existing parameters.

Select *Delete*: to delete the macro. Note that deleting a macro also affects the programs that use the macro deleted. As the macro no longer exists, these programs can no longer be changed.

Select *Edit*: To open the source code (extension. Cs) associated with the selected macro.

### **3.7.1.2 Implementation of the functions in the macro.**

See the MSL manual.

## **3.7.2 Create Subprograms (.xsp)**

The subprograms are used to enter recurring 2D geometries or machining operations in the project.

The subprogram is created as follows:

- 1) Create project
- 2) Define the parameters
- 3) Save the subprogram

### **3.7.2.1 Create project**

The creation of a subprogram starts by defining any project.

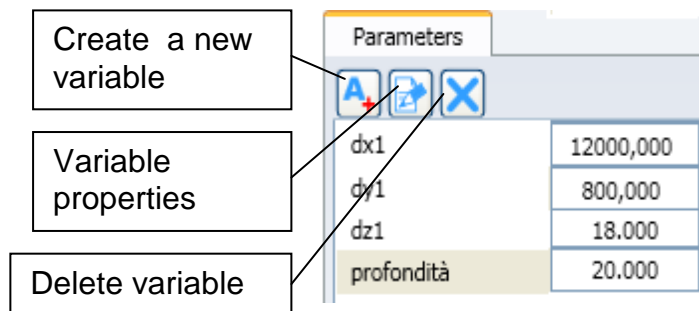
### 3.7.2.2 Define the parameters

If you insert a workpiece in the project, parameters dx1, dy1, dz1 will be available in the workpiece dimensions.

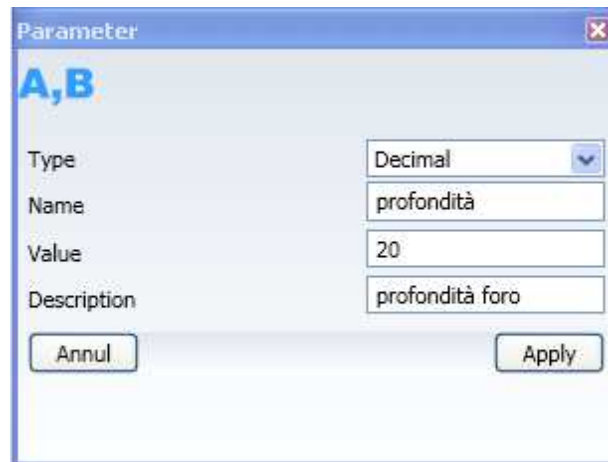
By inserting a second workpiece in the project, also dx2, dy2, dz2 will be also included.

The list of parameters available in the project is to be found in the interface down on the left.

By using the command "Create a new variable", it is possible to insert new parameters.



By using the command "Variable Properties", the same window used for the creation will appear:



By using the command "Delete variable", it will be possible to delete the variables created by the user but not those which have been automatically created (dx1, dy1, dz1).

All parameters available in the project can be used in every field that accepts a number in order to create 2D geometries as well as to set working processes.

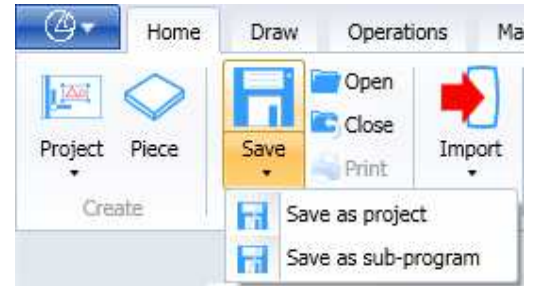
In order to use a parameter, it is sufficient to write it in place of such number, or you can also write the formula with more parameters and mathematic operators.

Parameters can also be exported with the sub-program as input data (see paragraph 3.7.2.3 "Saving the sub-program").

### 3.7.2.3 Save the sub-program

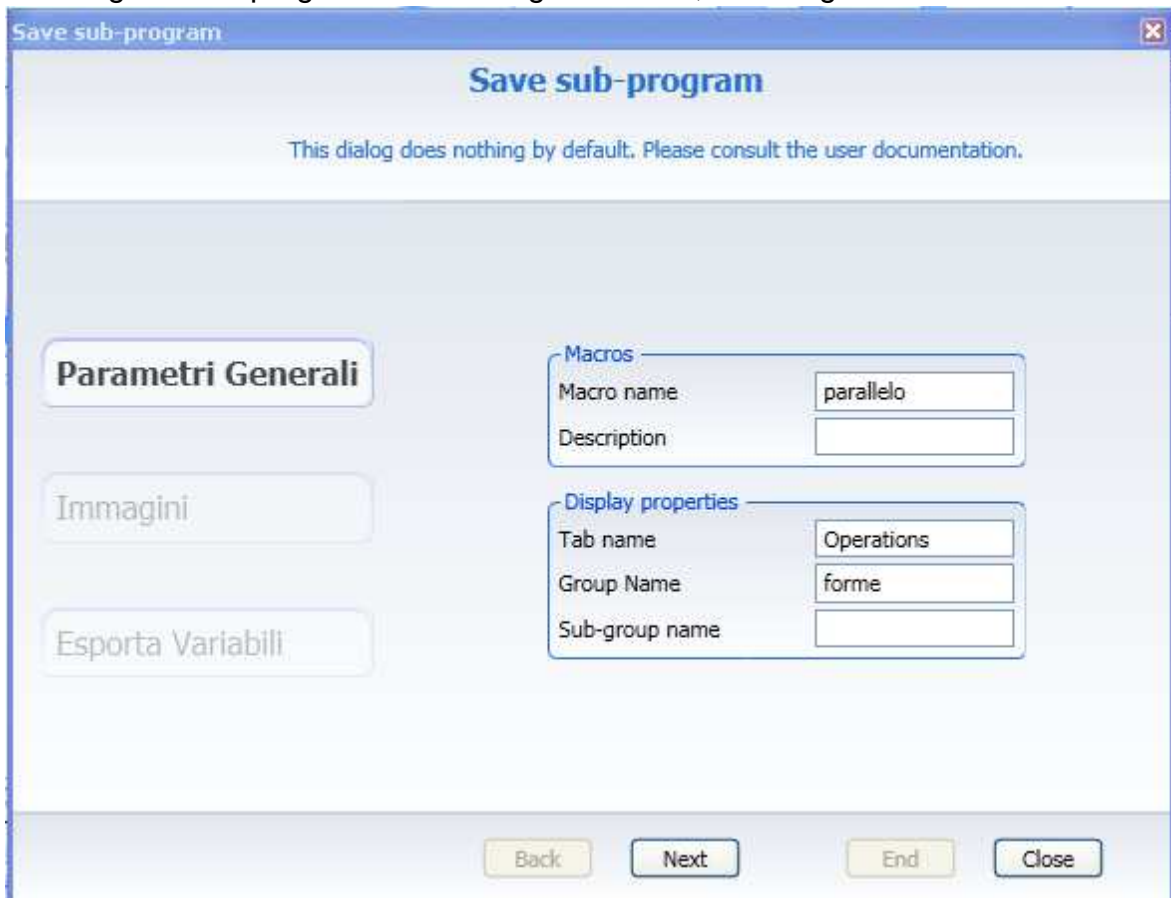
After the project setting, you can save the 2D geometries and the working processes as sub-program in order to reuse them for further projects. Subprograms can be performed only for projects with one workpiece.

In order to save a sub-program, use the command "Save as sub-program" in the menu "Home" "Save".



The resulting window will allow you to save the .xsp subprogram. The default folder for saving programs is the Maestro set up sub-folder "Macros". You can also save sub-programs in another folder but if you wish to use them, you must save them also in the default folder.

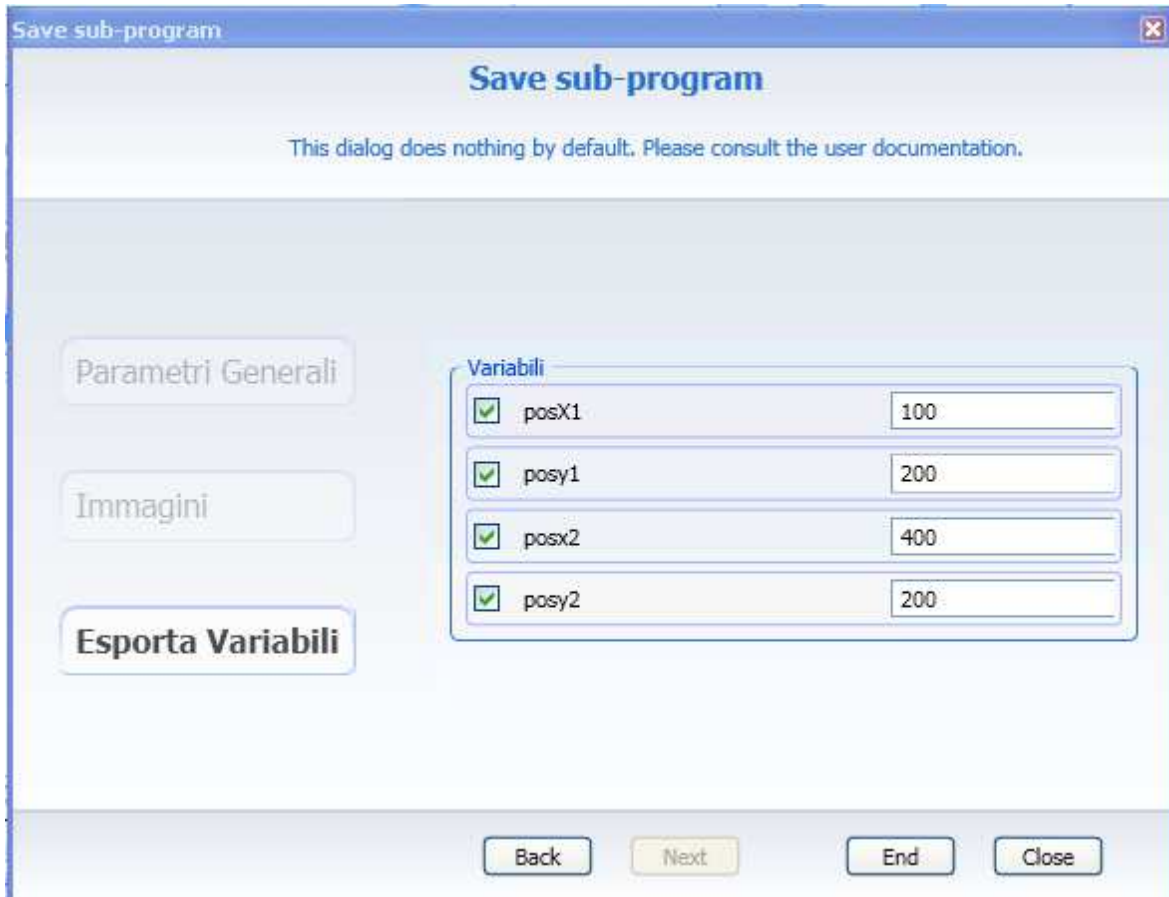
After naming the sub-program and clicking on "Save", following window will be showed:

A screenshot of the 'Save sub-program' dialog box. The dialog has a title bar 'Save sub-program' and a subtitle 'Save sub-program'. Below the subtitle is a message: 'This dialog does nothing by default. Please consult the user documentation.' The dialog is divided into two main sections: 'Parametri Generali' on the left and 'Macros' on the right. The 'Macros' section has a sub-section 'Display properties' with the following fields: 'Macro name' (parallelo), 'Description' (empty), 'Tab name' (Operations), 'Group Name' (forme), and 'Sub-group name' (empty). At the bottom of the dialog are four buttons: 'Back', 'Next', 'End', and 'Close'.

Fields included in "Display properties" serve as indicators for the menu where the sub-program will be saved.

If you click on "Next", on the resulting window - almost like the previous one- you will be asked to list the names of the bitmap to link with the subprogram. (Optional)

If you click on "Next", the resulting window will allow you to establish those variables to be exported.



If you click on "End", the saving process will end.

You can check the saving process :

- in the menu "Operations" (written in the field "Tab Name")
- since the sub-menu "Forme" has been created (written in the field "Group Name").
- and it contains the sub-program "esagono" (field " Macro Name")



### **3.7.2.4 Open an existing sub-program**

If you wish to open an existing sub-program, click on the icon "Open".  
The resulting window shows you the list where you may select a pgmx file.  
Select "File type" "Sub-program (\*.xsp)".  
Select the folder "Macros".  
Select a xsp file and then click on "Open".  
Or click twice on the xsp file.

For a project that has already been started, following option would be automatically performed:

“Closing project”

### 3.7.3 Use of .XSP macros and subprograms

The .XSP macros can be used when a project is already open in the Maestro editor, or when creating a workpiece program with the MSL scripting language (see next chapter).

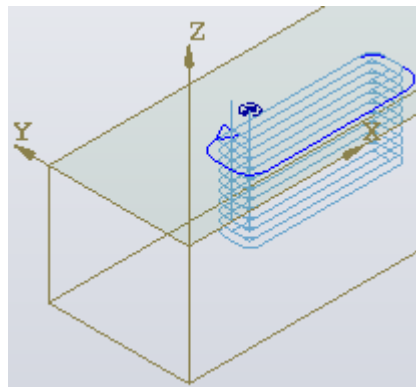
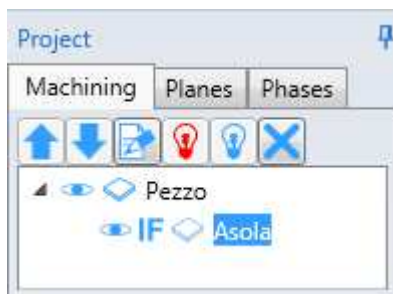
To use the Slot macro in a project, it is in:

Operations→Macro→Joints→Slot



After having clicked on “Slot” the various parameters can be changed, and the macro applied by clicking “Apply”

The effect of the macro can be seen in the machining tree and in the Maestro graphic area.



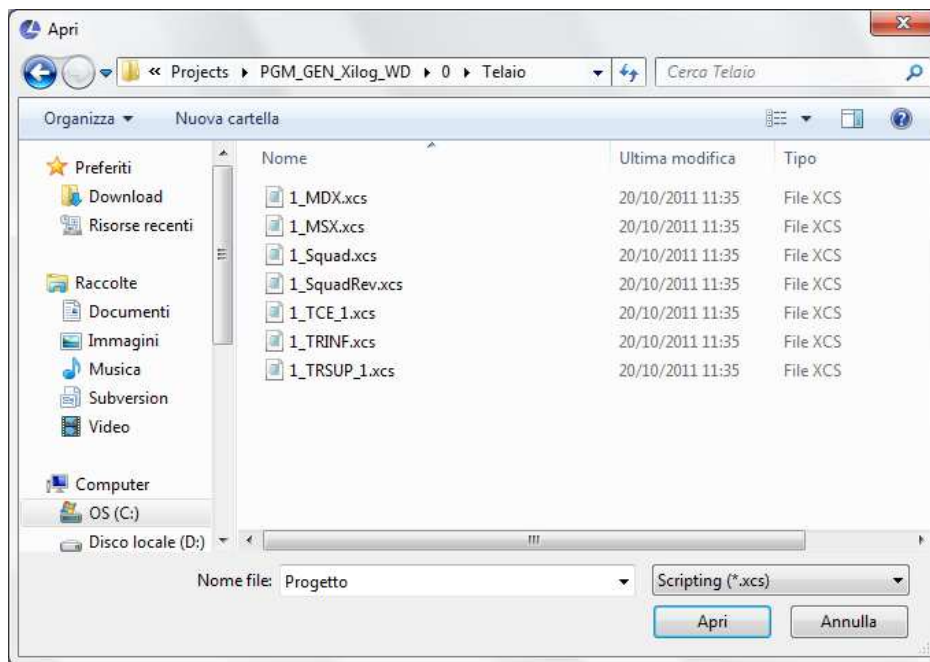
In the tree the name of the macro (Slot) can be seen, and in the graphic area the geometry and the machining described in the Slot.cs function.

### 3.7.4 Import Script (.xcs)

To use a workpiece program in .XCS format in the Maestro interactive environment, use the “Import script” command in the submenu “Import” of the “Home” menu.



The following will be displayed where a .xcs. file to import can be selected.



After having confirmed the selection of the .xcs file with “Open” the script is interpreted and the workpiece will appear and any machining described inside it.

Bear in mind that a script file (.xcs) can contain all the workpiece information and the relative machining, including the names of the tools associated to the machining, but it does not contain the “.tlgx” tools database file.

To achieve the desired workpiece correctly when the script was produced, before executing the “Import script” command, the “.tlgx” file, containing all the tools used in the script functions, must be prepared and rendered active.

To activate the .tlgx see the “Activating the tools database” chapter of the Maestro manual.

## 3.8 NC functions

Used to define and make the machine perform operations different from the machining operations.

The following NC functions can be entered in a program:

1. Enter an ISO instruction: CreateIso
2. Machine parking: CreatePark
3. Null operation: CreateNullOperation
4. Workpiece probing: CreateWorkPieceProbing
5. Display an operator message CreateMessage
6. Table cleaning
7. DY check

The functions are in the Machines – NC functions menu.



The icons of the available CN functions are visualised according to the machine set-up, therefore they are all visible at all times.



### 3.8.1 Enter an ISO instruction

To enter an ISO instruction in the machining tree select the “ISO” command in the “NC functions” menu.

The following dialogue window is displayed:

Maestro will not check the syntax of the instructions entered by the user.

Any syntax error in the ISO instruction will be performed on the Machine Panel.

The Flag on the Xiso option lets you report the Iso instruction written.

If the Xiso flag is not selected in the post phase the “ISO” prefix is added to the instruction.



Examples:

Machining Tree	Dialogue window	Post generated
		<pre> ***** , .OP1 ;GO X0 Y0 ***** , GO X0 Y0 </pre>
Machining Tree	Dialogue window	Post generated
		<pre> ***** , .OP1 ;GO X0 Y0 ***** , ISO "GO" X0 Y0 </pre>

### 3.8.2 Machine parking

The machine parking functions moves the upright in the parking zone to access the workpieces positioned in the machine in the unload phase.

To enter a parking instruction in the machining tree select the “Parking” command in the “NC functions” menu.

The parking is available in the following modes:

- No stop
- Stop with start wait
- Stop with release and start wait

The modes can be selected from the “Stop” drop down menu.

Example of parking with start wait

Machining Tree	Dialogue window	Post generated
		<pre>.***** , . OP1 ;Parking .***** , PARK S=1</pre>

### 3.8.3 Null operation

The “Null operation” function does not perform machining operations on the workpiece but it can be used to park the machine in a desired point and perform a tool change.

To enter a null operation instruction in the machining tree select the “Null operation” command in the “NC functions” menu.

The following parameters can be entered in the dialogue menu:

X final (position X to position the machine)

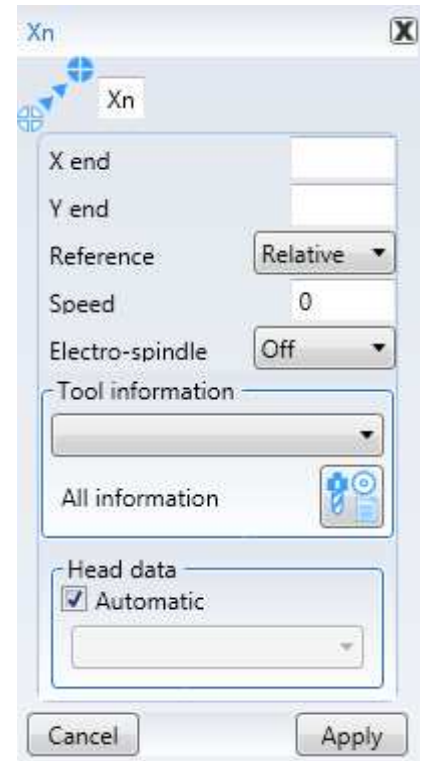
Y final (position Y to position the machine)

Reference (Relative or Absolute)

Speed (movement speed)

Electrospindle (Off or On)

The “Tools information” and “Heads data” fields must be filled in only if you want to change a tool.



## 3.8.4 Workpiece probing

The workpiece probing is an operation that is used for precise machining operations, even on workpieces that are irregular or positioned incorrectly.

There are three workpiece probing modes:

- 1) Probing with side position correction
- 2) Probing with tool length correction
- 3) Probing with workpiece roto-translation

### 3.8.4.1 Probing with side position correction

The probing with face position correction is performed with a probing tool.

The effect generated after the probing is to redefine the position of the working face so that all the machining operations on that face are precise.

The function is disabled each time the face is changed.

To enter a face position correction probing instruction in the machining tree select the “Probing” command in the “NC functions” menu.

The following parameters can be entered in the dialogue menu:

References (field used to select the work face).

X (X probing position)

Y (Y probing position)

Head return (can be performed to three different positions):

- Parking quota
- Probing quota
- Skimming quota

The “Tools information” field is used to select the probing tool.

The “Heads data” field is used to select the head used for the probing.



### 3.8.4.2 Probing with tool length correction

The probing with tool length correction is performed with a pneumatic probe fitted on the spindle.

The effect generated after the probing is to redefine the tool length so that all the machining operations performed with that tool on that face are precise.

On long workpieces, even when working on the same face with the same tool, it is preferable to perform various probing operations in different positions in case the workpiece is warped.

The function is disabled each time the tool or the face are changed.

To enter a tool length correction probing instruction in the machining tree select the “Special probing” command from the “Probing” menu in the “NC functions” menu.



The following parameters can be entered in the dialogue menu:

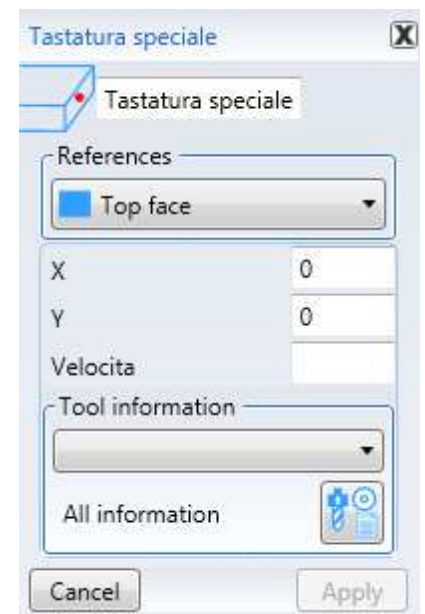
References (field used to select the work face).

X (X probing position)

Y (Y probing position)

Speed (probing speed)

The “Tool information” field is used to select the tool whose length will be corrected and that will be used for the subsequent machining operations.



### 3.8.4.3 Probing with workpiece roto-translation

The probing with workpiece roto-translation is performed with a probing tool.

The effect generated after the probing is to redefine the position and angular orientation of the workpiece so that all the machining operations performed with that tool on that face are precise.

The function remains active for the whole workpiece program.

To enter a probing with workpiece roto-translation instruction in the machining tree select the “Workpiece rotation translation probing” command from the “Probing” menu in the “NC functions” menu.



The following parameters can be entered in the dialogue menu:

First quota side X (first probing position X)

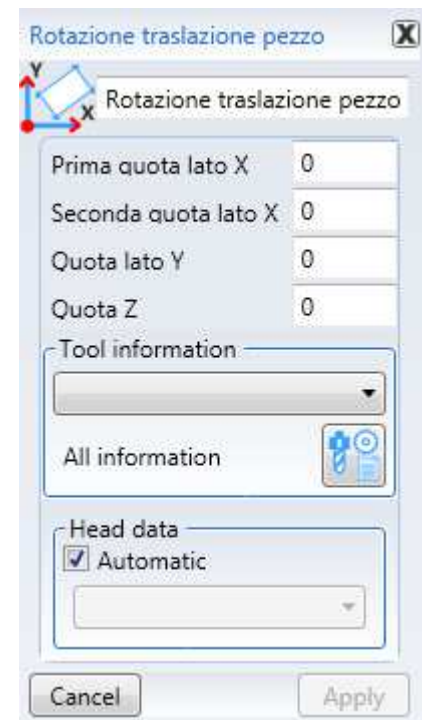
Second quota side X (second probing position X)

Side Y quota (third probing position Y)

Quota Z (position Z of all the probings)

The “Tools information” field is used to select the probing tool.

The “Heads data” field is used to select the head used for the probing.



### 3.8.5 Display an operator message

The display an operator message function is similar to the parking functions and allows the operator to view a message and restart the program without moving the machine.

To enter a message for the operator in the machining tree select the “Print message” command in the “NC functions” menu.

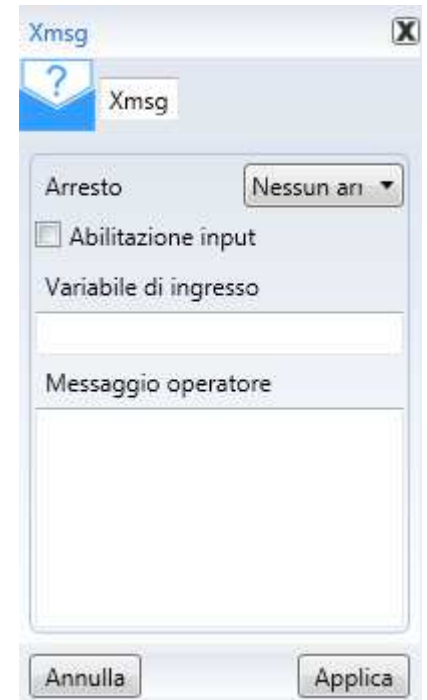
The message can appear in the following ways:

- No stop
- Stop with start wait
- Stop with release and start wait

The modes can be selected from the “Stop” drop down menu.

The “Input enable” and “Input variable” variables are not managed.

The “Operator message” field can be filled with the message that will be displayed for the operator during the program execution.



### 3.8.6 Table cleaning

The table cleaning NC function enables a cleaning cycle of the machine table with the extraction hood.

To enter a message for the operator in the machining tree select the “Table cleaning” command in the “NC functions” menu.

Hood enable: determines the extraction hood enable or disable.

- Hood position:
  - o for the motor—driven hood the value indicates a position in millimetres
  - o for the pneumatic hood the value indicates one of the possible positions
- Cleaning speed: speed for the cleaning cycle: 25 metres per minute, default
- Cleaning start X quota: position along the X axis where the cleaning cycle starts in relation to the workpiece origin (optional parameter, necessary for Ergon type machines)
- Cleaning end X quota: position along the X axis where the cleaning cycle ends in relation to the workpiece origin (optional parameter, necessary for Ergon type machines)



In the machining tree it will be displayed as follows:



Examples:

- 1) All the programmed parameters and the motor-driven hood

Dialogue window



Post Generated

```

*****
;
.OP1 ;Table cleaning
*****
XHOODPLANE a=1 s=25.000 Q=100.000 l=50.000 L=600.000
    
```



2) All the programmed parameters and pneumatic hood in position 1

Dialogue window



Post Generated

```
.*****  
,  
.OP1 ;Table cleaning  
.*****  
,  
XHOODPLANE a=0 s=25.000 Q=1.000 l=50.000 L=600.000
```

3) no programmed parameter

Dialogue window



Post Generated

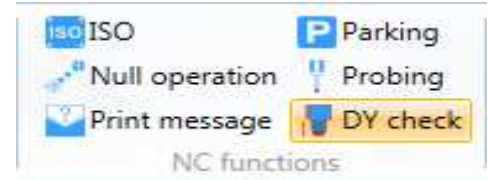
```
.*****  
,  
.OP1 ;Table cleaning  
.*****  
,  
XHOODPLANE a=0 s=25.000
```

### 3.8.7 DY check

CN function "DY check" is meant for CYflex machines and allows measuring the piece in Y right after the loading stage.

If the dimensions of the piece in Y are not consistent, the machine shall signal an error.

In order to set the DY check of a panel, select control "DY check" from the "C.N. functions" menu.

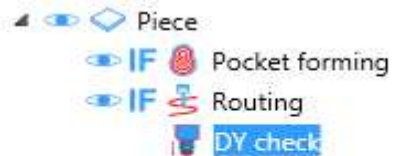


CN function "DY check" only has the parameter "Distance" that represents the distance from the right edge of the piece whose dimension needs to be checked.

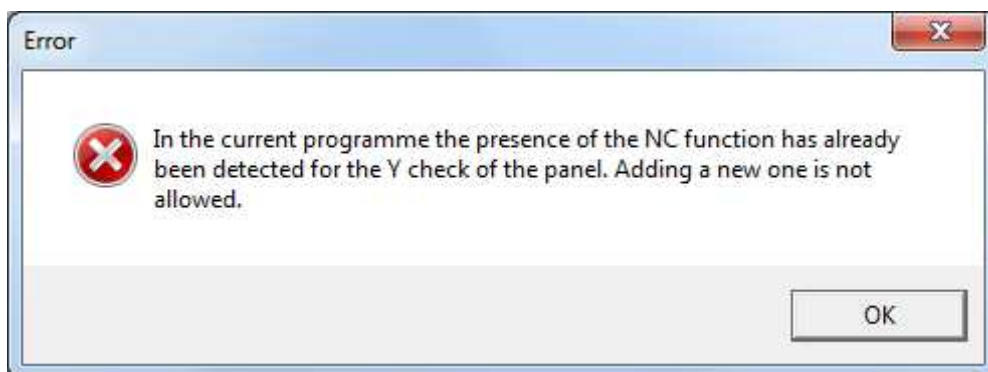


By clicking on the function "DY check" allows adding such function to the machinings tree.

Its position is not relevant in that during the "Optimisation" stage this will always comes first.



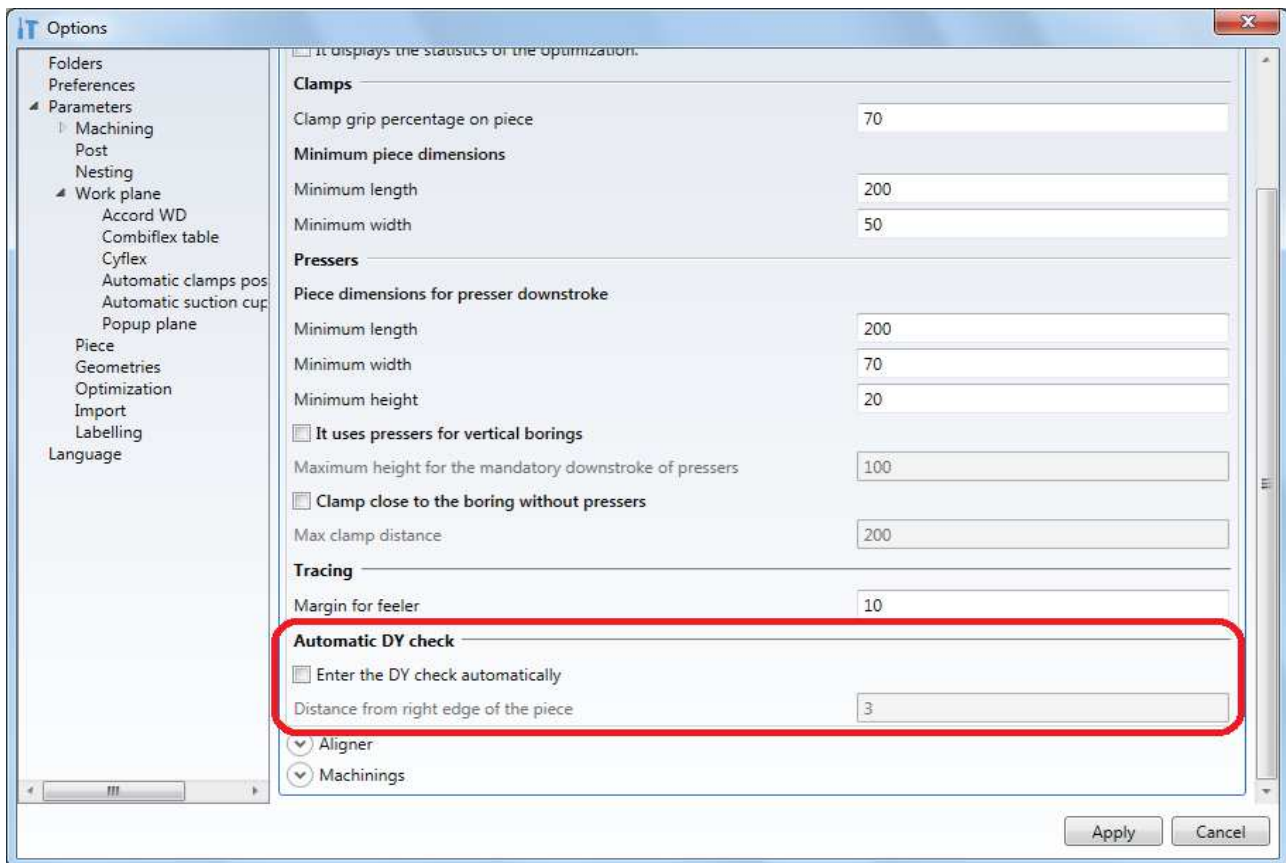
If you try to set another "DY check", the following message will appear:



### 3.8.7.1 Automatic DY control

In the Cyflex S, the Y dimension control of the panel is fundamental, because in these types of machines there is the aligner. If a piece is inserted with a different DY from the one specified in the program, there is a risk of collision between the machining operations and the aligner.

Maestro has been fitted with an automatic Y dimension control of the panel, with a specific parameter in the options:



This option is displayed only when Cyflex S is selected as the type of machine and is disabled by default, to save the current behaviour.

When the function is activated, the possibility of specifying the distance from the right edge of the panel is enabled. This distance is the one that will be used in all the NC functions to control the dimension of the piece entered automatically.

During the optimisation of a program, check if the option is active: if it is, if the pgmx does not already contain the piece dimension control NC function, it will be entered automatically with the distance specified in the options. If the user has already entered his personal control, it will be saved so as not to change the distance value already entered.

With other types of machines, such as the Cyflex HP, the option, even if enabled, has not effect.

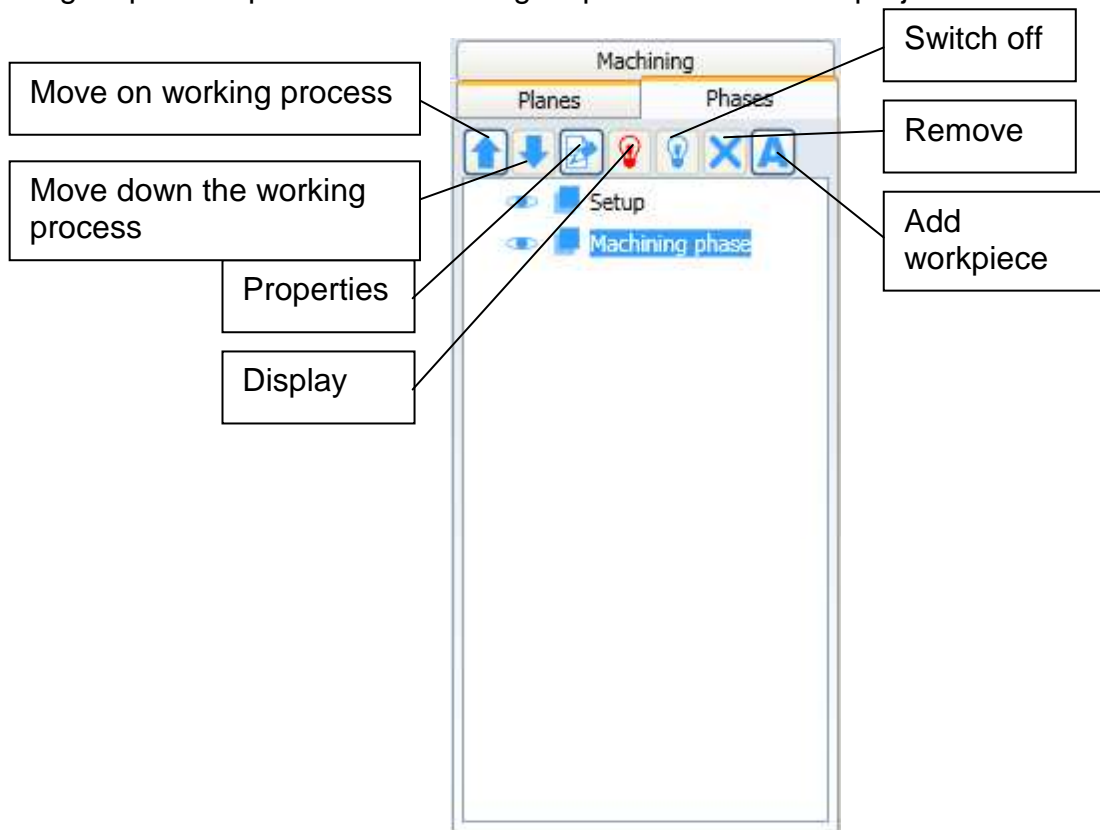
## 3.9 Working steps

A working step describes the positioning of one or more workpieces and which working processes must be performed if they are in the positions described.

When you create a new project, the step "Setup" is automatically created.

### 3.9.1 Working step tree

The working step tree represents all working steps set for a certain project.



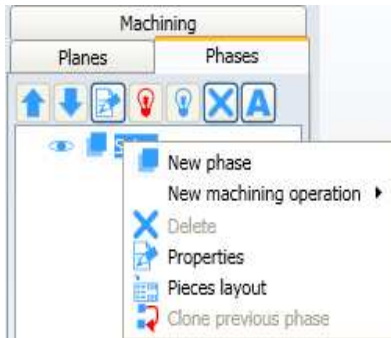
Through this tree it is possible to:

- Insert a working step
- Remove a working step
- See the step properties
- Add working processes
- Move the working processes from one step to another one.
- Display or switch off the working processes
- Add the workpiece name to the working process name.

### 3.9.2 Inserting a working step

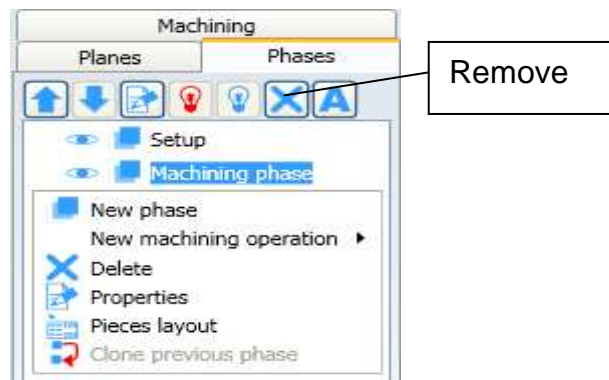
In order to insert a new working step, you use the command "New step" in the contextual menu of the step tree.

After selecting the command, the menu to the right will show you where you can insert the name of the "Working step".



### 3.9.3 Removal of a working step

In order to remove a working step, you use the button with the red cross or the command "Remove" in the contextual menu of the step tree. The initial "Setup" cannot be removed.



### 3.9.4 Working step properties

In order to display a working step, you use the button "Properties" or the command "Property" in the step contextual menu.

After selecting such command, the menu to the right will show you the name of the "Working step" and the workpiece positioning in that step.



### 3.9.5 Modifying the workpiece position in one step

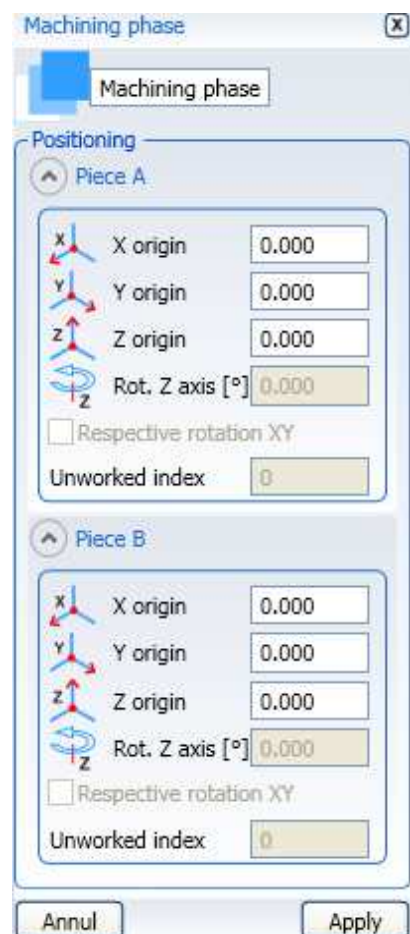
It is possible to modify the workpiece position in a certain step by using the command "Properties" in the contextual menu of the step tree.

By clicking on the arrows to the left of the workpiece name, the menu will show the current position of the machine workpiece.

You can change the workpiece position in the selected step by modifying the values to the right of the parameters.

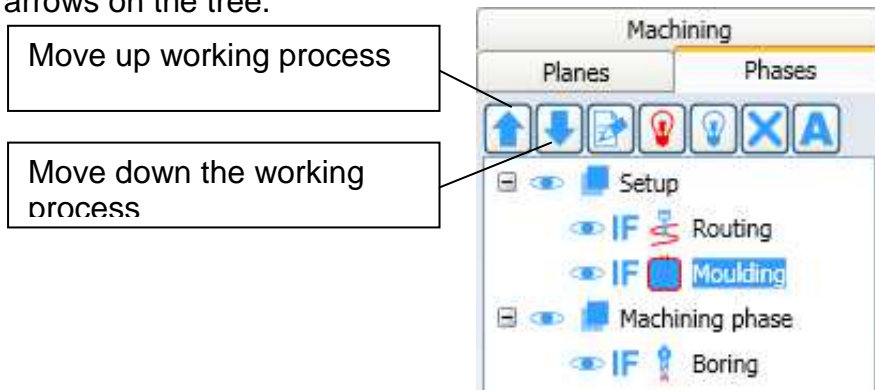
- X origin
- Y origin
- Z origin

By clicking on the button "Apply"

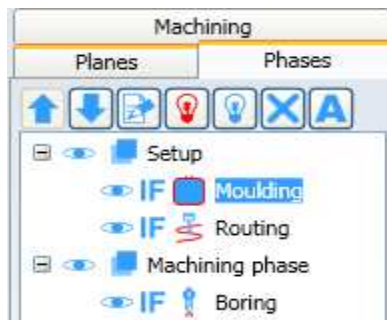


### 3.9.6 Moving working processes from one step to another one

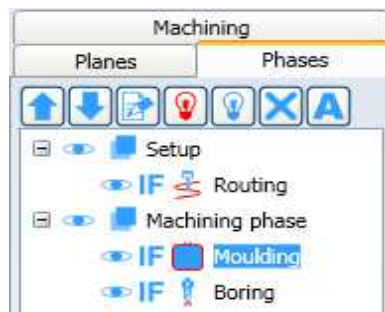
In order to move the working processes from one step to another one, you use the buttons with the blue arrows on the tree.



In the image above, if you click on the button with the arrow downwards, the Contouring will be shifted on the Milling but it will remain the same during the step "Setup".



By using the arrow downwards, the Counterling will always occur before the boring, but will shift from "Setup" to "Working step"

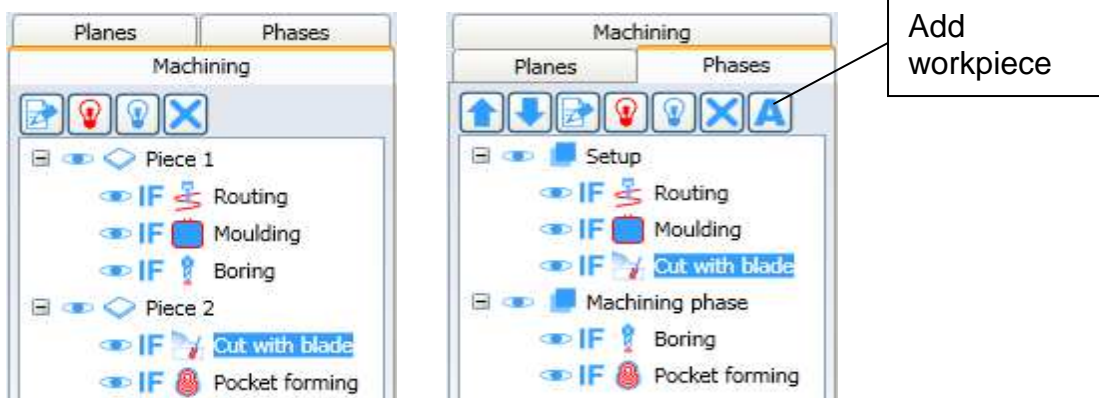


### 3.9.7 Add workpiece name to working process name

If you look at the step tree, we see what kind of working processes are performed in every step but we do not know to which workpiece they belong.

In order to obtain this information, you use the button "Add workpiece name".

Project example with 2 workpieces and 2 steps:



As you see in the step tree, you cannot understand to which workpieces the working processes belong. By using the command "Add workpiece name", the step tree will appear in this way:

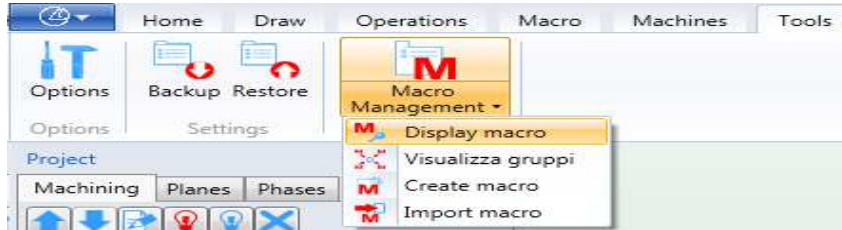


The working processes are not modified, but you see only a temporary representation that can be restored by clicking on "Add workpiece name".

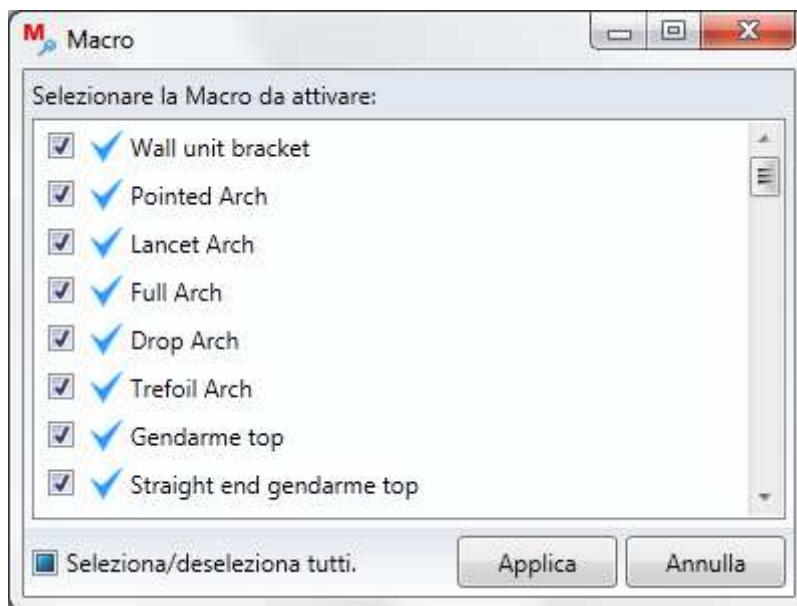


### 3.9.8 Sub-programs Activation/Deactivation

Use the "Tools – Sub-programs management" control to activate or deactivate the "Macros" folder Sub-programs.



The following window appears once the "Sub-programs management" control is clicked:



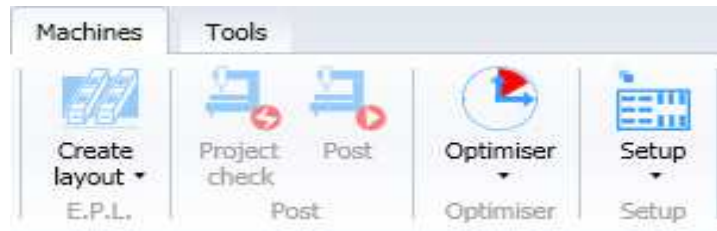
From where the available sub-programs can be activated or deactivated.

### 3.10 Workpieces layout in machine

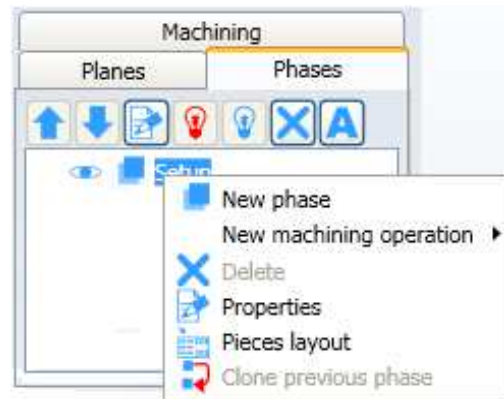
Arrange the workpieces in machine based on its work top, blocking devices and workpieces to be machined.

"Setup" is a detached session from the project environment. Use the following to access "Setup":

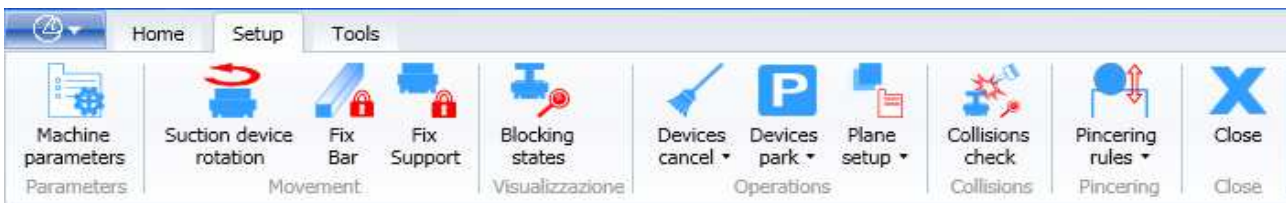
The "Setup" control in the "Machines" menu.



Or the "Workpieces layout" in the context menu of the phases' shaft. The menu appears when right-clicking the mouse on a phase name.



"Setup" shows the following menu:



Some controls will not be active due to different work top configuration or setup. The various controls are described below following a work trail and not in order of position on menu.

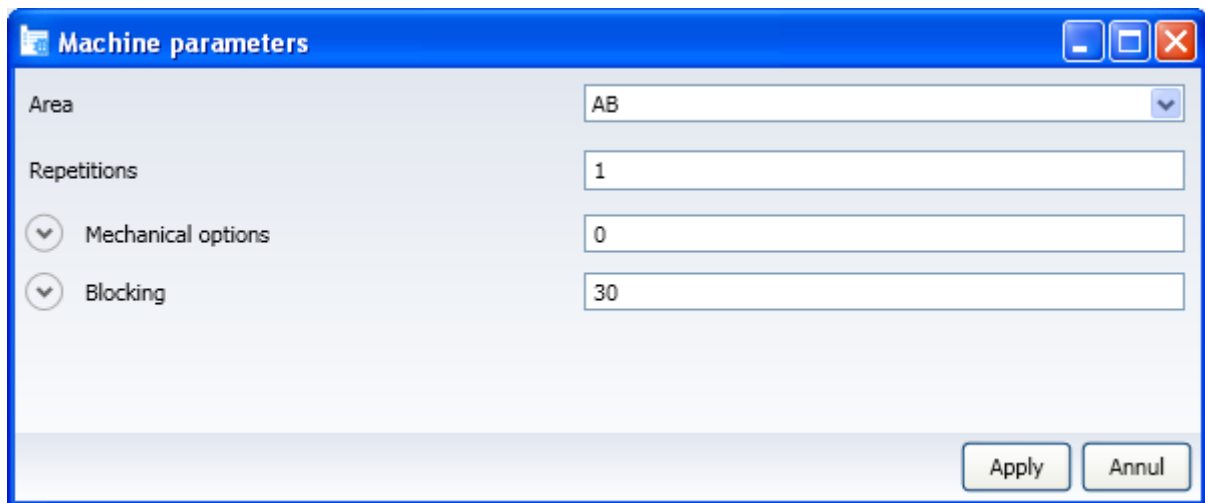
### 3.10.1 Choice of work area, references and devices

The machine has a work top divided by areas having various references and different blocking devices.

They are selected through "Machine parameters" control in the "Setup" menu.



The following menu appears after selection:

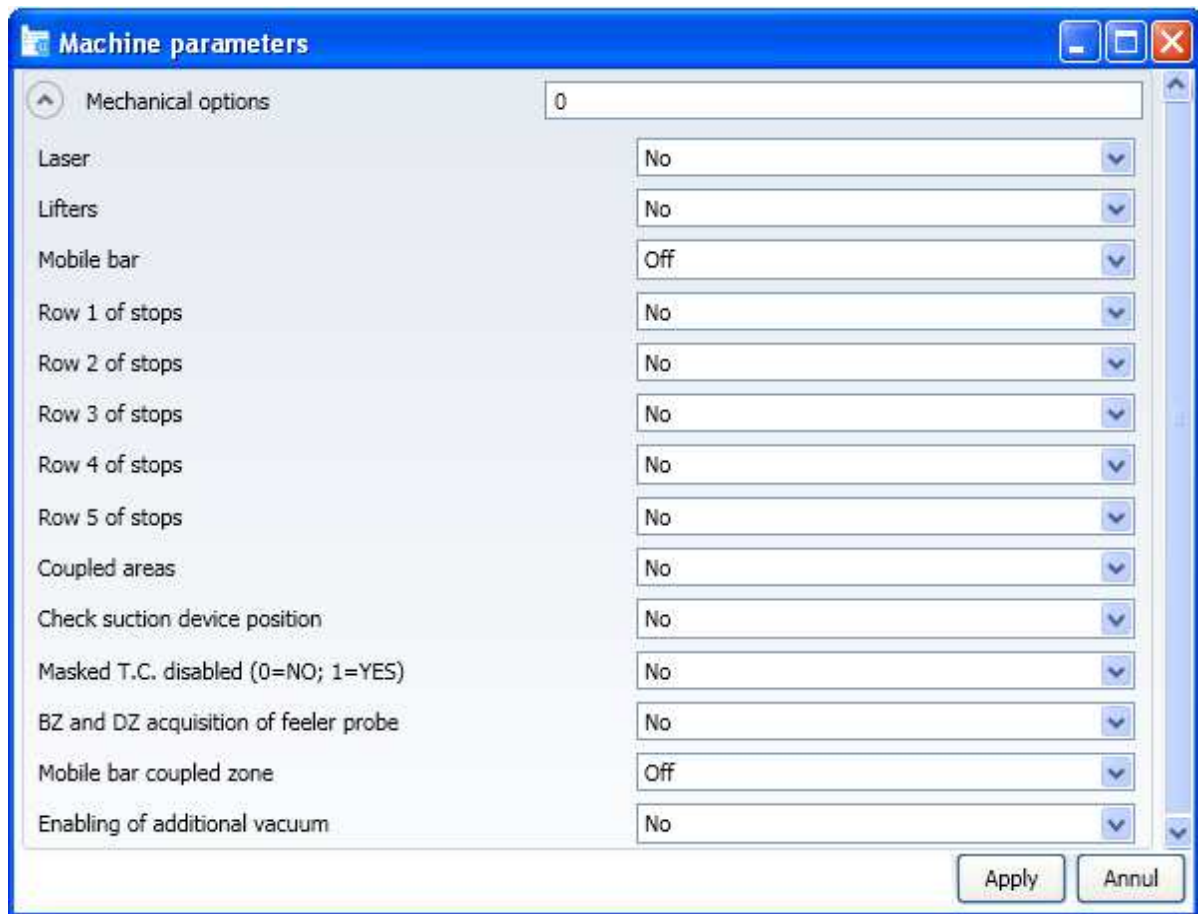
A screenshot of a 'Machine parameters' dialog box. The dialog has a blue title bar with the text 'Machine parameters' and standard window control buttons (minimize, maximize, close). The main area contains several input fields: 'Area' with a dropdown menu showing 'AB', 'Repetitions' with a text box containing '1', 'Mechanical options' with a dropdown menu showing '0', and 'Blocking' with a text box containing '30'. At the bottom right, there are two buttons: 'Apply' and 'Annul'.

#### 3.10.1.1 Work area selection

Use the right drop-down menu showing the available areas in a given machine configuration to select the work area.

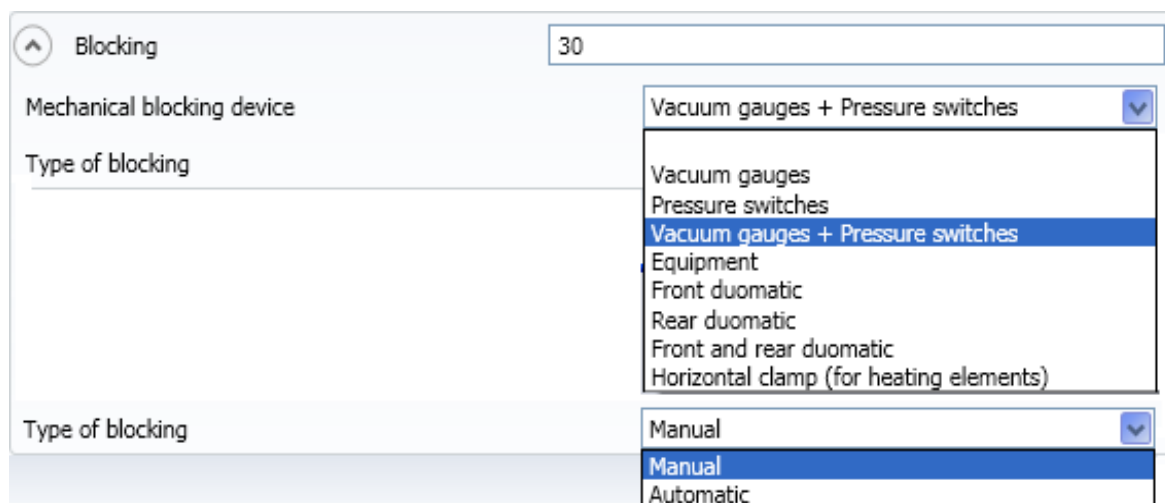
### 3.10.1.2 References Selection

Use the "Mechanical options" sub-menu to select the workpieces' references:



### 3.10.1.3 Type of blocking selection

Use the "Blocking" sub-menu to select the type of blocking:

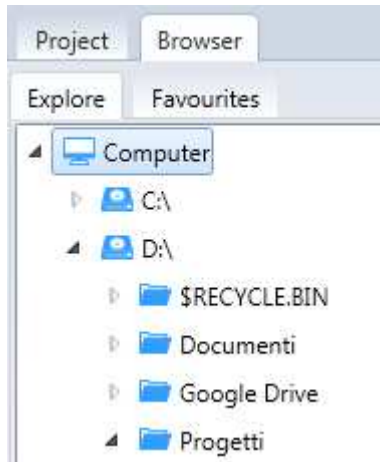


### 3.10.2 Adding pieces on the machine plane

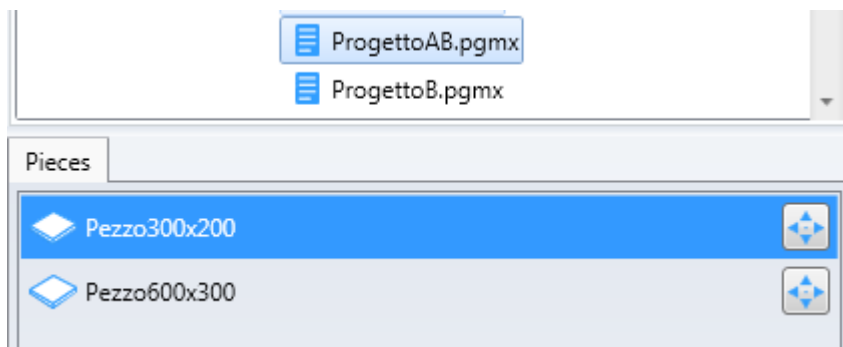
It is possible to add a piece on the machine plane by selecting it from an existing project. The starting project can also be empty.

The mode is as follows:

- Select a project using the "Browser" folder that allows you to search for it on your PC.



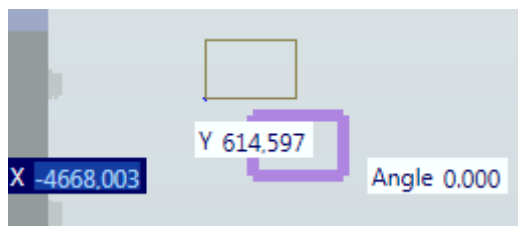
- Select one of the pieces in the project



- Set a distance to maintain with respect to existing pieces



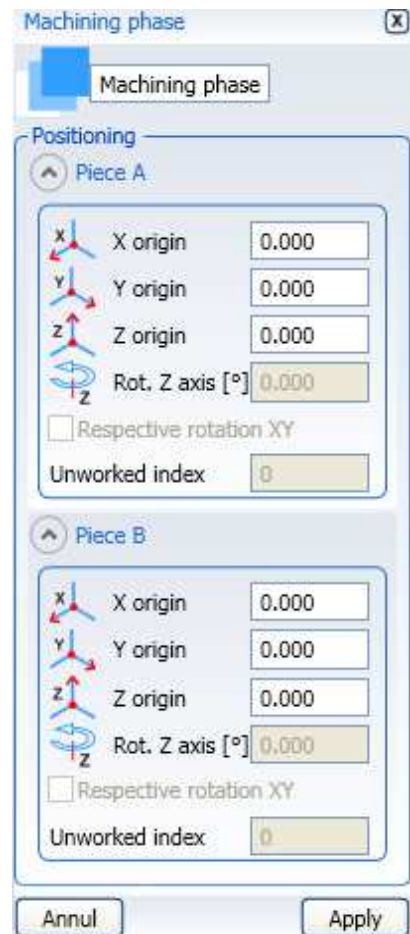
Add the new piece using the mouse



### 3.10.3 Workpieces positioning

Use the right-hand window to position the workpieces on the machine's work top:

The workpieces' position can be modified by changing the "source" values and clicking on "Apply".  
(Also see paragraphs 3.9.5)

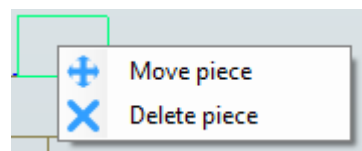


The pieces can also be positioned using the "Select piece" and "Move piece" commands



The "Select piece" command is necessary because the default of Arrange is to select suckers or clamps.

Once the piece has been selected, to move the piece, use the "Move piece" command available in the layout menu or in the context menu



In this case, the selected piece will move with the mouse respecting the set offset.

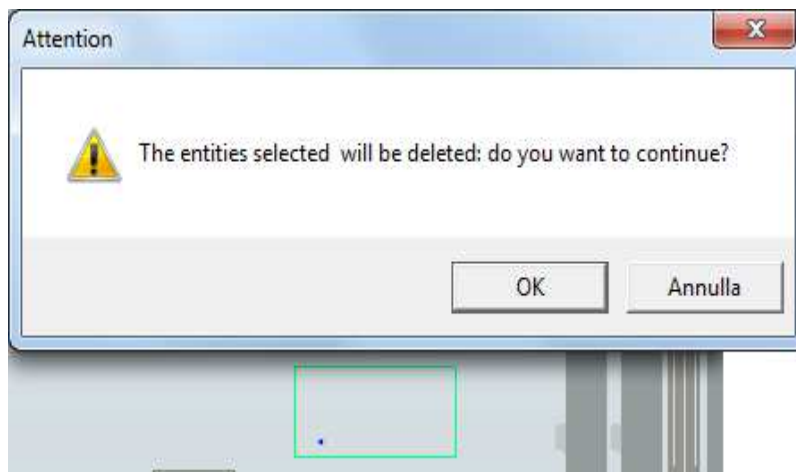
### 3.10.4 Elimination of pieces from the machine plane

The pieces can be removed using the "Select piece" and "Delete" commands

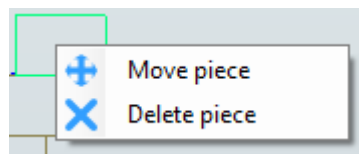


- Select the "Select piece" command
- Select the piece (the piece will assume a green outline)
- Select the "Delete" command

The confirmation message will appear before being deleted:



You can also select the piece (turning it green) click the right mouse button (the context menu appears) and select "Delete piece"



## 3.11 Definition of the blocking system

There are two types of blocking systems on the work centres:

1. Clamps blocking
2. Suction cups blocking

### 3.11.1 Workpiece blocking with clamps

Workpiece blocking using clamps normally happens to block window or door elements. Select the devices via "Pressure switches" or "Vacuum gauges + Pressure switches", as described in chapter 3.10.1.3.

Note: If working with machines not having motorised work top it is not necessary to position the clamps.

#### 3.11.1.1 Type of clamp

The types of clamps currently available are:

1. Clamps with round plate
2. Clamps with rectangular plate
3. Clamps with reference stops
4. Horizontal clamps

The first three types of clamp block the workpiece along axis Z.  
The horizontal clamps block the workpiece along axis Y.

A code is used to identify the various types of clamps.

The code consists of a maximum of 4 separate fields. The first two are the same for all types of clamps.

The code of the first two fields is:

Hzzz-U

Where

- zzz is the clamp work height (50/110)
- U is the type of use (M/A)(Manual/Automatic)

The remaining optional fields of the clamp code will be described in the following chapters.



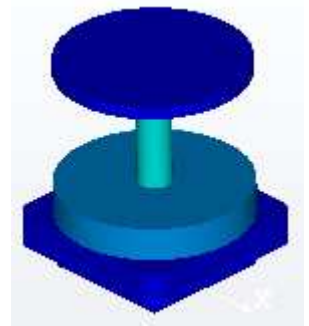
### 3.11.1.1.1 Clamps with round plate

For these clamps the third code field describes the plate diameter.  
The code for this type of clamp is:  
Hzzz-U-ddd

Where ddd can currently have a value of 145 or 185.

Examples:

H50-M-145            Clamp height 50mm manual with 145mm diameter  
H110-A-185         Clamp height 110mm automatic with 185mm diameter



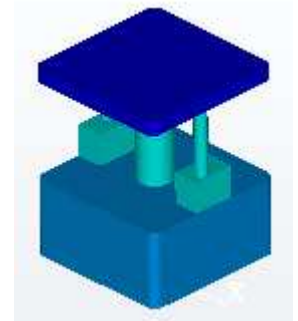
### 3.11.1.1.2 Clamps with rectangular plate

For these clamps the third code field describes the plate rectangular dimension.  
The code for this type of clamp is:  
Hzzz-U-bbbxyyy

Where bbb and yyy are the length and width of the clamp plate rectangle.  
Currently there is only 165X165

Examples:

H50-M-165x165     Clamp height 50mm manual with square plate 165x165mm  
H110-A-165x165    Clamp height 110mm automatic with square plate 165x165mm



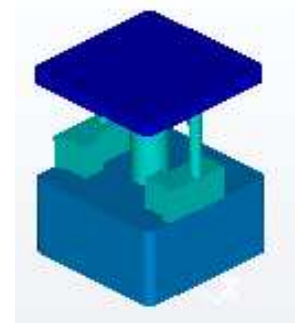
### 3.11.1.1.3 Clamps with reference stops

For these clamps the third code field describes the plate rectangular dimension and the fourth field the reference stop thickness.  
The code for this type of clamp is:  
Hzzz-U-bbbxyyy-Scc

Where bbb and yyy are the length and width of the clamp plate rectangle. Currently there is only 165x165.  
cc is the thickness of the reference stop on which the workpiece rests (values currently available: 10,20,25,30)

Examples:

H50-M-165x165-S10     Clamp height 50mm manual with square plate 165x165mm and 10mm stop.  
H110-A-165x165-S30    Clamp height 110mm automatic with square plate 165x165mm and 30mm stop.



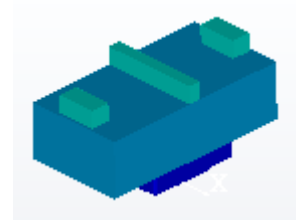
#### 3.11.1.1.4 Horizontal clamps

This type of clamp blocks the workpieces horizontally along axis Y and is only managed manually.

For these clamps the third code field describes the rectangular dimension of the whole clamp.

The code for this type of clamp is:

Hzzz-U-bbbxyyy



Where bbb and yyy are the length and width of the whole clamp. Currently there is only 150x294.

There is no parameter indicating whether it is an horizontal clamp.

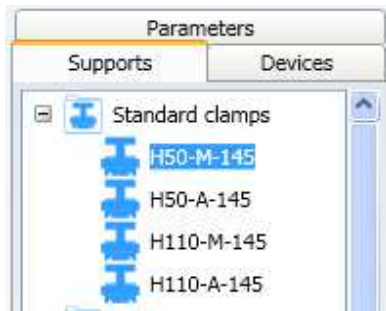
At the moment it can only be seen by the dimension of the rectangle 150x294.

Example:

H50-M-150x294    Clamp height 50mm manual with rectangular plate 150x294mm

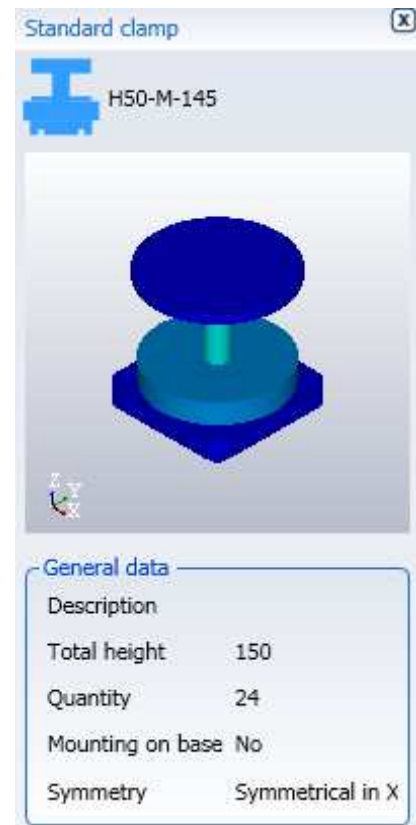
H110-M-150x294    Clamp height 110mm manual with rectangular plate 150x294mm

### 3.11.1.2 Type of clamp selection



Use the bottom-left list of available clamps to select the type of clamp:

The top-left window shows the clamps' preview:



### 3.11.1.3 Clamps positioning on the bars

Select the clamp to be used and drag it onto the manual bar or onto the motorised bars' plates.

Assemble the clamps on all plates in case of motorised bars. Do this using the context control and right-click the mouse on the clamp to be used.

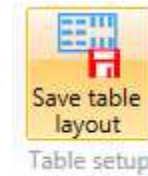


In certain cases there may be collisions during the assembly. This is because Maestro considers the total space of the machine elements (plate plus camp) in the collision control between mobile elements. To overcome this issue simply move the plates from each other before applying the clamp.

### 3.11.1.4 Saving plane layout

Once the plane has been set up with clamps or suction cups, it is possible to save the plane layout so that it can be used on other occasions.

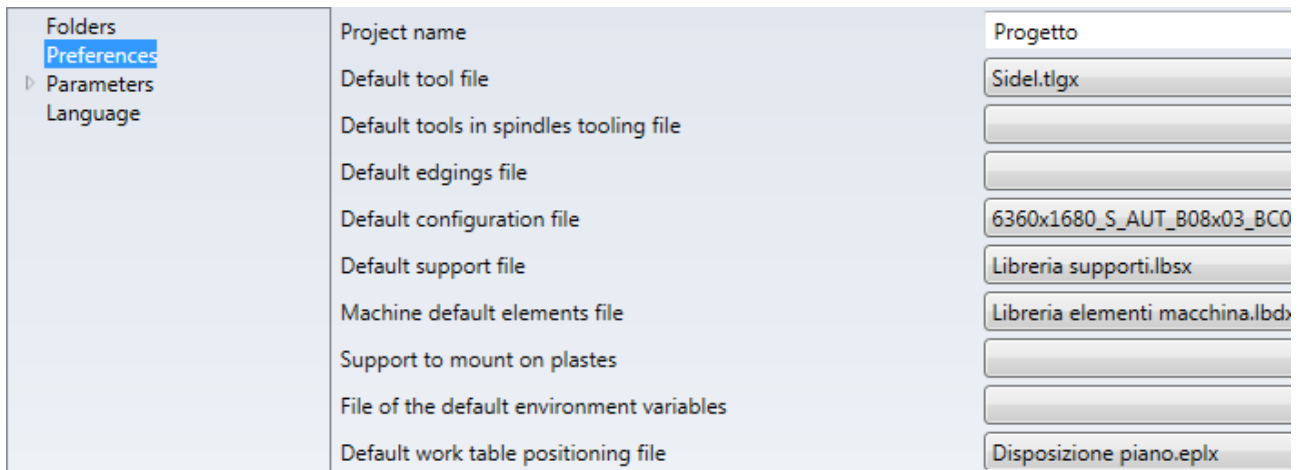
To save, the command "Save plane arrangement" command is used



A .eplx file will be created in its Eplx folder

### 3.11.1.5 Reusing the plane layout

To re-use the previously saved plane configuration, set the name to: Tools-Options-Preferences File of the default work plane positioning



After setting the field with the name of the eplx file, every time you enter the "Layout" session, the previously saved setup will appear.

### 3.11.1.6 Blocking states of clamps

Each clamp can have 3 blocking states:

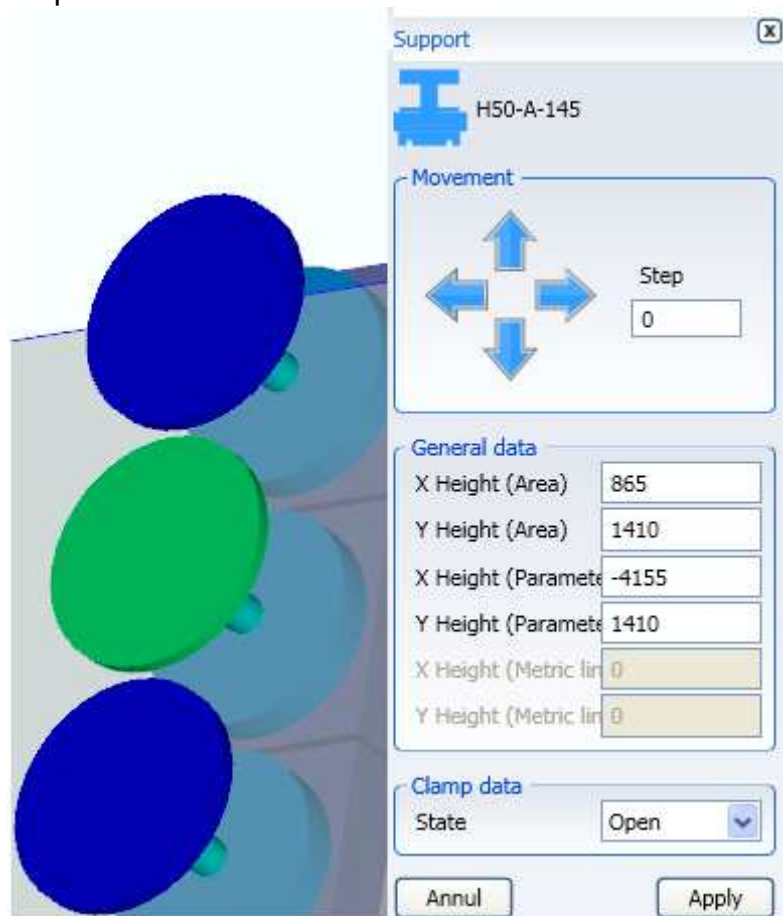
- 1) Open
- 2) Closed
- 3) Closed on the piece

The clamps state can change in every phase to manage the machining process.

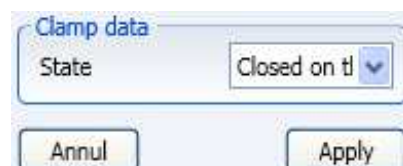
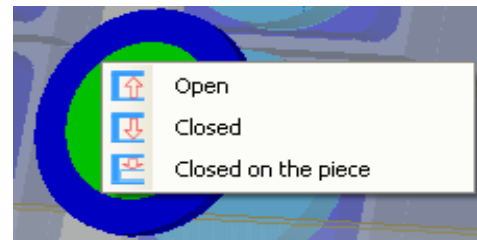
#### 3.11.1.6.1 Blocking state change

To change the blocking state of a clamp:

- select a clamp  
(it becomes light green)  
(a window appears on right)



- right-click the mouse (a context menu appears)
- select the wanted state entry by left-clicking the mouse
  
- or click on right arrow of the state
- select the wanted state entry by left-clicking the mouse
- click on "Apply"



### 3.11.1.6.2 Blocking state display

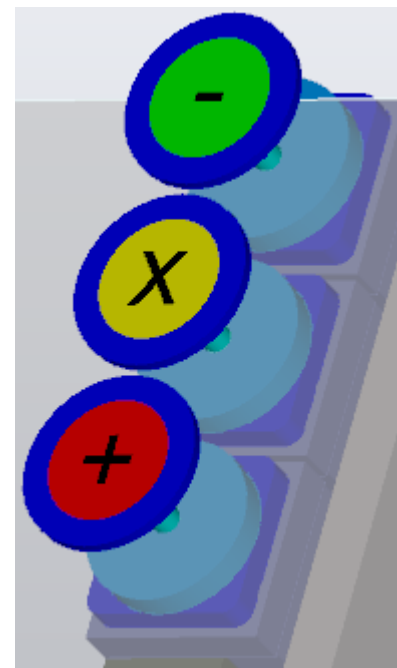
The blocking state is displayed through the "Clamp Data" – "State" menu, but this only shows the selected clamp's state.

Use the "Setup" menu's "Blocking states" control to see the overall clamps states.

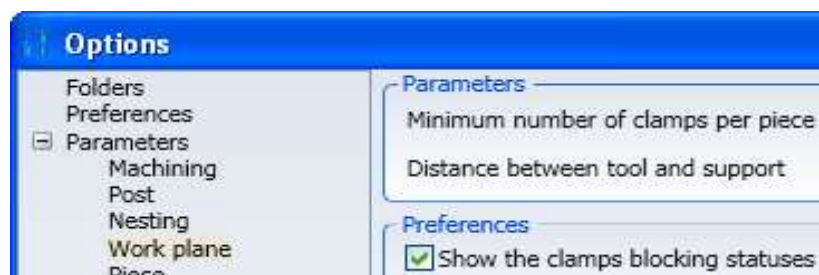


The clamps are displayed as follows:

- Open (green with symbol "-")
- Closed (yellow with symbol "X")
- Closed on workpiece (red with symbol "+")



Tick "Display the clamps' blocking states" in the "Options" menu's "Work top" "Parameters" to ensure the blocking state is always active when accessing "Setup".



### 3.11.1.7 Clamps positioning on workpiece

One or more workpieces having one or more work phases can be blocked on the machine's work top.

Every phase must contain information on position and states of clamps for machining workpieces with machines having motorised work top (not essential for machines with manual work top).

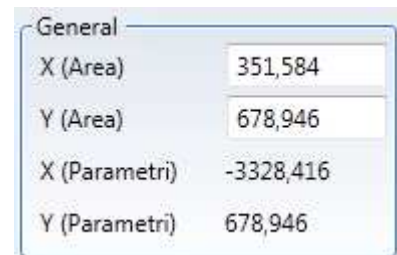
The clamps can be positioned:

- 1) Fully manually
- 2) Manually using the blocking profile
- 3) Automatically using the blocking profile

Once the clamp has been fitted, it can be moved in three ways:

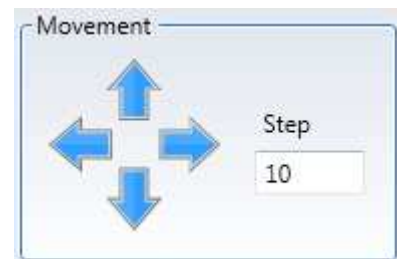
- Click on the clamp and drag it keeping the mouse button pressed;


- Enter the precise coordinates where the clamp must be positioned in the screen on the right displayed after clicking on the clamp (figure 8);



General	
X (Area)	351,584
Y (Area)	678,946
X (Parametri)	-3328,416
Y (Parametri)	678,946

- Enter the size of the movement to perform and clicking on the arrows on the right displayed after clicking on the clamp (figure 9).



Movement	
	Step 10

The clamp movement is restricted by an anti-collision control that does not allow movements if it detects collision between the clamp and other machine elements on the plane.

The following functions can be used by the operator during the positioning.



This ribbon button is used to block the movement of the bars to guarantee the positioning of the supports in Y without worrying about the movement in X.



This ribbon button is used to block the movement of the supports to guarantee the positioning of the supports in X without worrying about the movement in Y.

When a bar has a clamp closed on the workpiece, the movement of the bar is not allowed. Therefore the movement in X is inhibited when trying to move the bar and when moving a different support from the one blocked.

If there is only one clamp closed on the workpiece along the bar, its movement also moves the bar in X.

When a clamp is moved by dragging, its blocking state changes to closed. This function has been implemented to comply with standard clamps.

### 3.11.1.7.1 Manual clamps positioning

This positioning only has the screen viewing aid.

Position by dragging one clamp in the wanted position at a time.

Then change clamp state in "Closed on workpiece" (see paragraph 3.11.1.6)

Ensure the workpiece does not touch the clamps' rod.

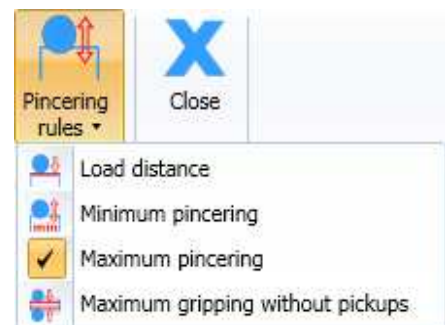
### 3.11.1.7.2 Manual clamps' positioning with blocking profile

The blocking profile is a polyline that can only currently be realised from "Script": writing an external file to be imported in Maestro.

As well as polyline geometry, the blocking profile also has a dimensional feature indicating minimum pincering.

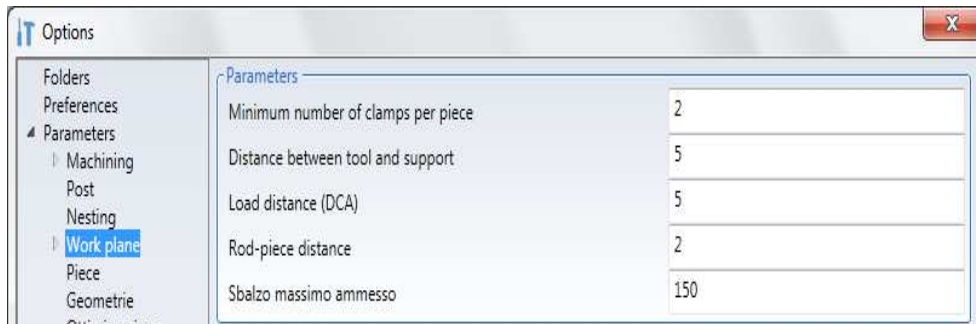
Different blockings can be made using the "Pincering rules" in the "Setup" menu, with a piece having a blocking profile.

- Load distance
- Minimum pincering
- Maximum pincering
- Maximum pincering without recoveries





- 1) The "Load distance" rule is normally used to position the clamps that must internally block a door to be profiled.  
The clamp's axis sticks to an axis that becomes blue when a clamp nears the blocking profile using the "Load distance" rule.  
This guarantees the clamp remains at a distance from the blocking profile.  
The clamp's state is automatically taken to "Open" when left on the blue axis.  
Use the "Tools" "Options" control to modify the parameter establishing the clamp's distance from the blocking profile (Load Distance (DCA)). The following screen appears:



The "Load distance (DCA)" parameter is found in the "Options" "Parameters" "Work top" folder.

- 2) The "Minimum pincering" rule is normally used to position the clamps that must block the workpiece, pincering minimum wood section.  
The clamp's axis sticks to an axis that becomes red when a clamp nears the blocking profile using the "Minimum pincering" rule.  
This guarantees the clamp pincers the blocking profile for minimum established height.  
The clamp's state is automatically taken to "Closed on workpiece" when left on the red axis.  
The minimum pincering dimension is written as blocking profile feature.
- 3) The "Maximum pincering" rule is normally used to position the clamps that must block the workpiece, pincering maximum wood section.  
The clamp's axis sticks to an axis that becomes green when a clamp nears the blocking profile using the "Minimum pincering" rule.  
This guarantees the clamp pincers the blocking profile for maximum established height.  
The clamp's state is automatically taken to "Closed on workpiece" when left on the green axis.  
The maximum pincering dimension depends on type of clamp and "Rod-workpiece distance" parameter found in the "Options" "Parameters" "Work top" folder.
- 4) The "Maximum pincering without recoveries" rule is normally used to position the clamps that must block the workpiece, pincering maximum wood section, guaranteeing exchange of clamps without recoveries to speed-up the cycle.  
This rule only remains active for horizontal, without arcs, blocking profiles.  
The clamp's axis sticks to an axis that becomes blue when a clamp nears the blocking profile using the "Maximum pincering without recoveries" rule.

This guarantees the clamp pincers the blocking profile for maximum established height.

The clamp's state is automatically taken to "Closed on workpiece" when left on the blue axis.

The pincering dimension depends on type of clamp, workpiece width, "Safety distance in Y" parameter found in the Bars' "Property" window configuration and on "Rod-workpiece distance" parameter found in "Options" "Parameters" "Work top" folder.

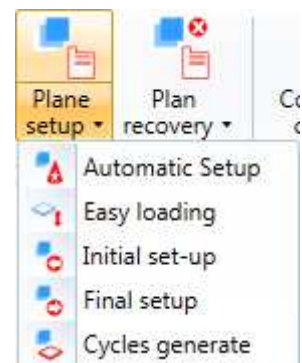
Notes:

- The "Load distance" rule deactivates the others and vice-versa
- The maximum pincering rules exclude one another
- "Minimum pincering" rule can be activated together with a maximum pincering rule. This enables clamp positioning in the area between the red and green axes or between the red and blue axes.

### 3.11.1.7.3 Automatic clamps' positioning with blocking profile

Use the "Automatic setup" control in the "Setup" menu's "Work top setup" sub-menu to automatically position the clamps on the blocking profile.

The clamps are automatically positioned on the blocking profiles of all workpieces in all phases without adding further information, after clicking on the function control.



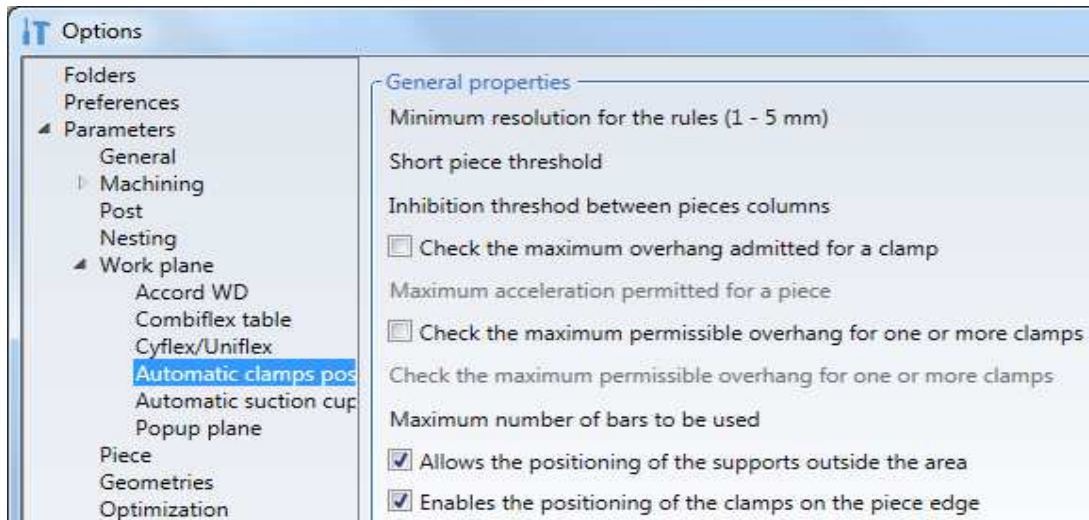
Note: This function saves time compared to manual positioning.

### 3.11.1.7.4 Positioning terminals outside the work area

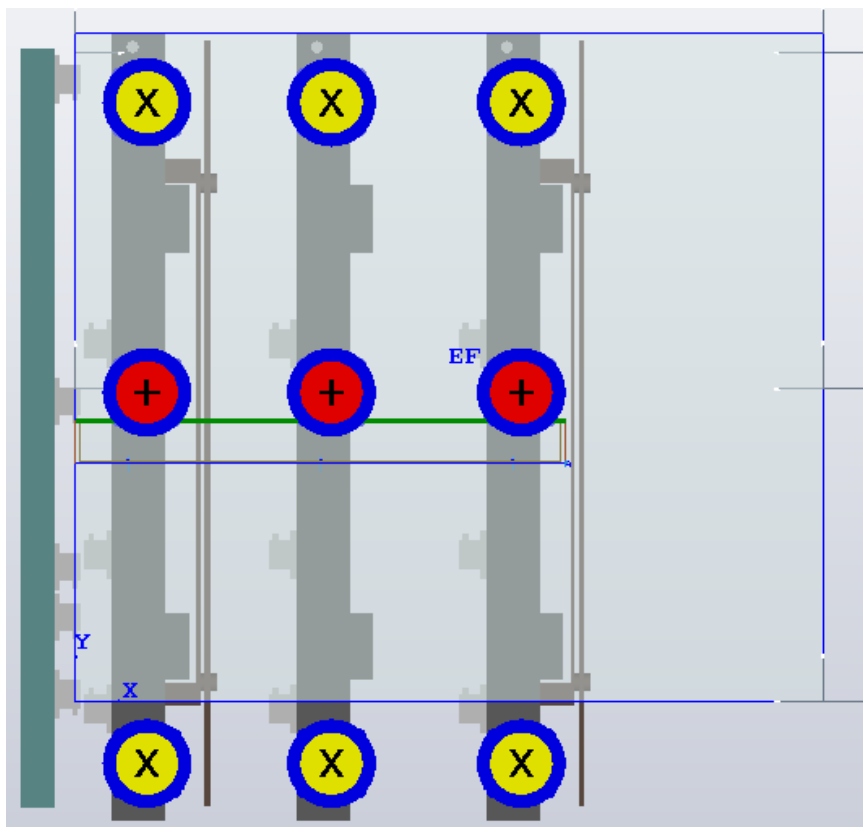
When you have a machine that has reference stops inside the work area, you may need to place the clamps outside the work area.

This feature is enabled by ticking the entry:  
Allows the positioning of the supports outside the area  
the item is located by selecting:

Tools-Options-Parameters-Work Plan-Automatic Positioning Terminals



Enabling functionality allows you to get this set up, allowing you to have more work space available:



### 3.11.1.7.5 Enables the positioning of the clamps on the piece edge

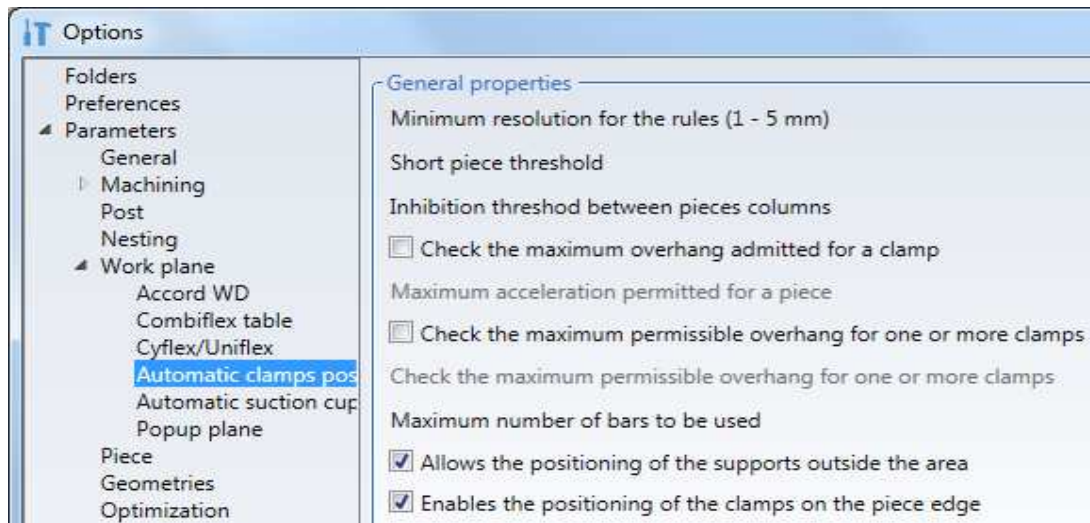
When working a particularly tedious material, it may damage the special high-pressure part that is created when the automatic positioner of the terminals places them on the edges of the workpiece.

the axes can be moved with the mobile control panel.

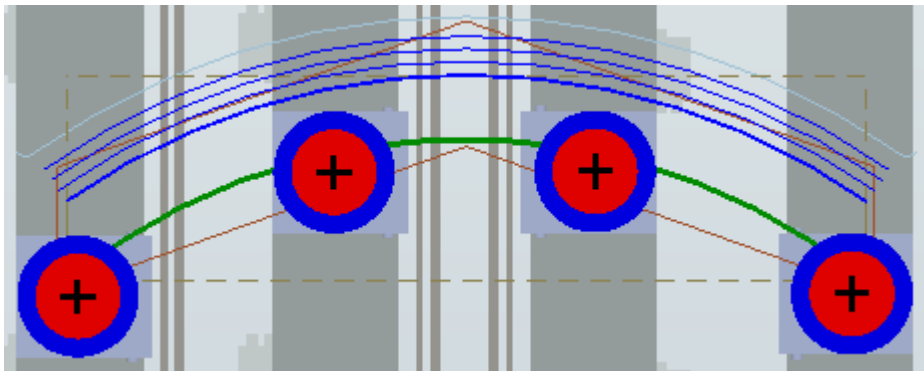
Enables the positioning of the clamps on the piece edge

the item is located by selecting:

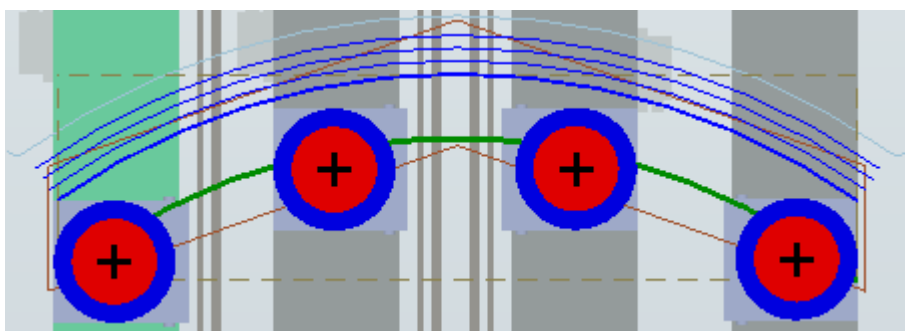
Tools-Options-Parameters-Work Plan-Automatic Positioning Terminals



Example that could crash the edges (With the flag selected)



Example that does not damage the edges (Without the flag selected)

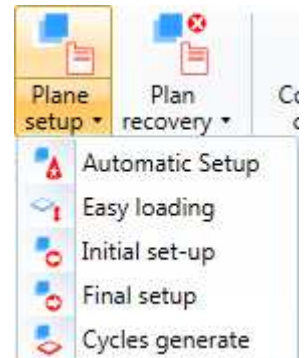


### 3.11.1.8 Initial setup creation

An "Initial phase" is required for loading the workpieces to complete the project.

Use the "Initial setup" control in the "Setup" menu's "Work top setup" sub-menu to do this.

A new "Initial Setup" phase, indicated in the first project phase with regard to clamps' positions (the clamps blocking the workpiece will have the "Open" state), is created after having selected the control.

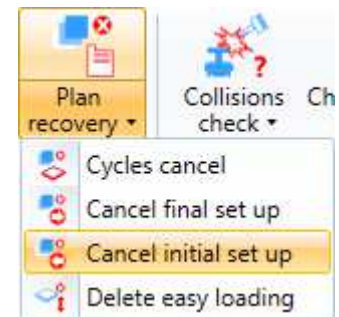


The initial setup phase must be manually completed if preparing initial setup for a closed structure workpiece, requiring internal blocking (door profiling).

Use the "Load distance" rule to position the clamps inside the door for loading. (see chapter 3.11.1.7.2 point 1)

Note: the "Initial Setup" phase can be eliminated like any other phase.

To do this, you can also use the "Cancel initial set up" command in the "Plan recovery" menu



### 3.11.1.9 Final setup creation

Unlike initial setup, this function is compulsory. It is used if clamps must be moved to remove the workpiece (internally blocked door) at program end.

Use the "Final setup" control in the "Setup" menu's "Work top setup" sub-menu to create a final setup phase.

A new "Final Setup" phase is created, identical to "Initial Setup", after having selected the control.

The function will not be successful if an initial phase is not already present.

Note: the "Final Setup" phase can be eliminated like any other phase.

To do this, you can also use the "Cancel final set up" command in the "Plan recovery" menu



### 3.11.1.10 Cycles generation for clamps exchange

Establish the clamps' sequence of movements to change from one project phase to another, once the clamps have been positioned in the various blocking phases.

This is done automatically using the "Cycles generate" control in the "Setup" menu's "Work top setup" sub-menu.

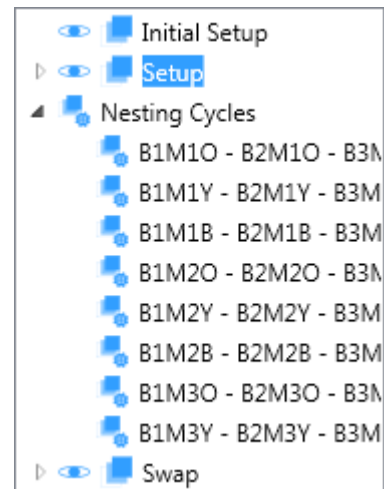
A cycles structure, a set of phases to manage the clamps' exchange, is created after having selected the control.

The following is a structure example:

The cycles names are composed of initials identifying the clamps, followed by a letter expressing a change.

Examples:

- B1M1O → (O) Opening of clamp 1 (M1) of bar 1(B1)
- B1M1Y → (Y) Long Y movement of clamp 1 (M1) of bar 1(B1)
- B1M1B → (B) Blocking on workpiece of clamp 1 (M1) of bar 1(B1)



### 3.11.1.11 Cycles elimination for clamps exchange

The already created cycles must be eliminated if modifying one or more positions of the clamps in any phase.

Use the "Cycles cancel" control in the "Setup" menu's "Plan recovery" sub-menu to eliminate all cycles.

This happens automatically if re-using the "Cycles generate" control.



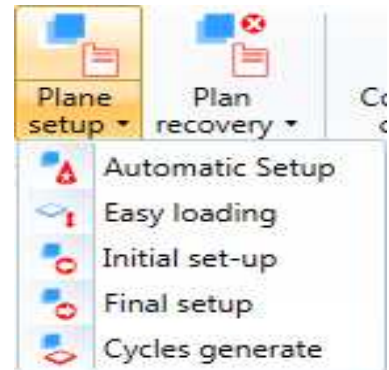
### **3.11.2 Workpiece blocking with suction devices**

### 3.11.3 Easier Load

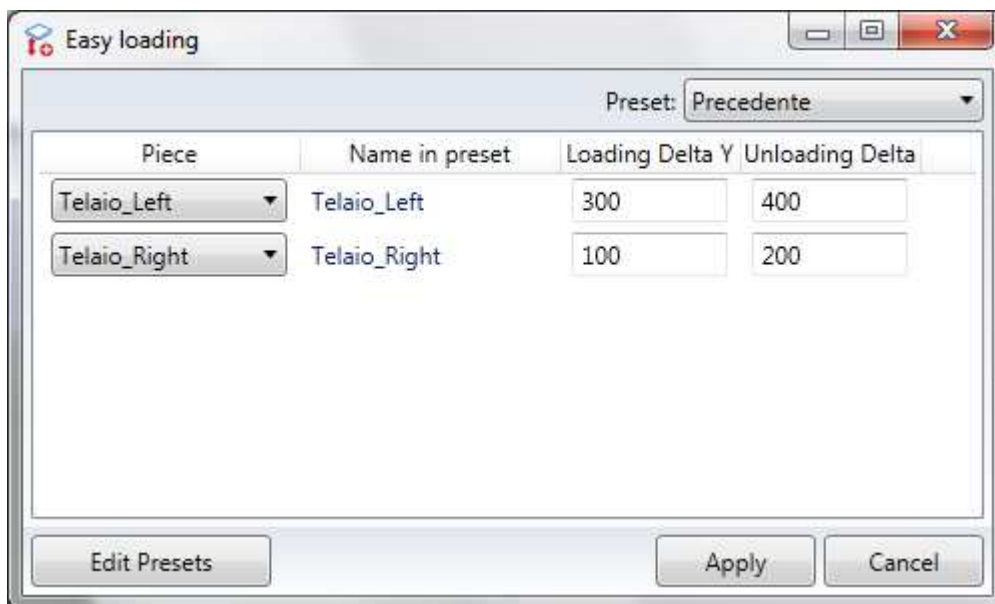
The "Easier Load" function can be used to machine workpieces in a different position to that loaded.

#### 3.11.3.1 Easier Load Insertion

Use the "Easier Load" control in the "Setup" menu's "Work top setup" sub-menu to make loading easier.



The following screen appears after the control is selected:



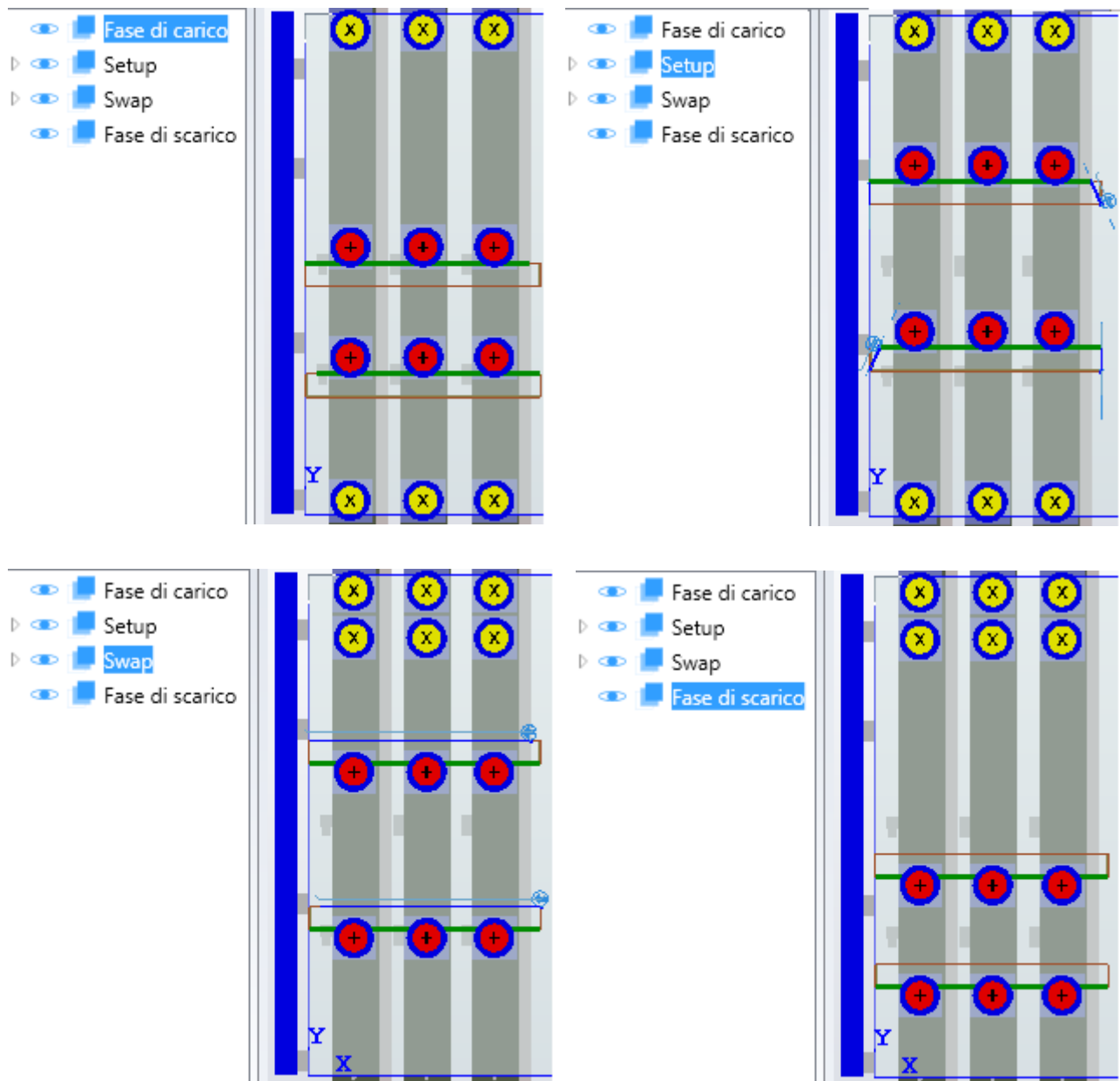
The names of workpieces in the project appear on left. Specify the "Load Y Delta" and the "Unload Y Delta" for each workpiece on right. The "Load Y Delta" indicates the distance the workpiece must be placed at compared to loading initial position. The "Unload Y Delta" indicates the distance the workpiece must be re-positioned at compared to positioned defined with initial "Load Y Delta".

Two new phases are created by clicking on "Apply": "Load phase" and "Unload phase".



- "Load phase" is identical to old "Setup" phase (without machining).
- The new "Setup" phase is like the old "Setup" phase plus "Load Y Delta".
- The new "Swap" phase is like the old "Swap" phase plus "Load Y Delta".
- The "Unload phase" is identical to the new "Swap" phase less "Unload Y Delta" (without machining).

Example:



### 3.11.3.1.1 Preset of easier load

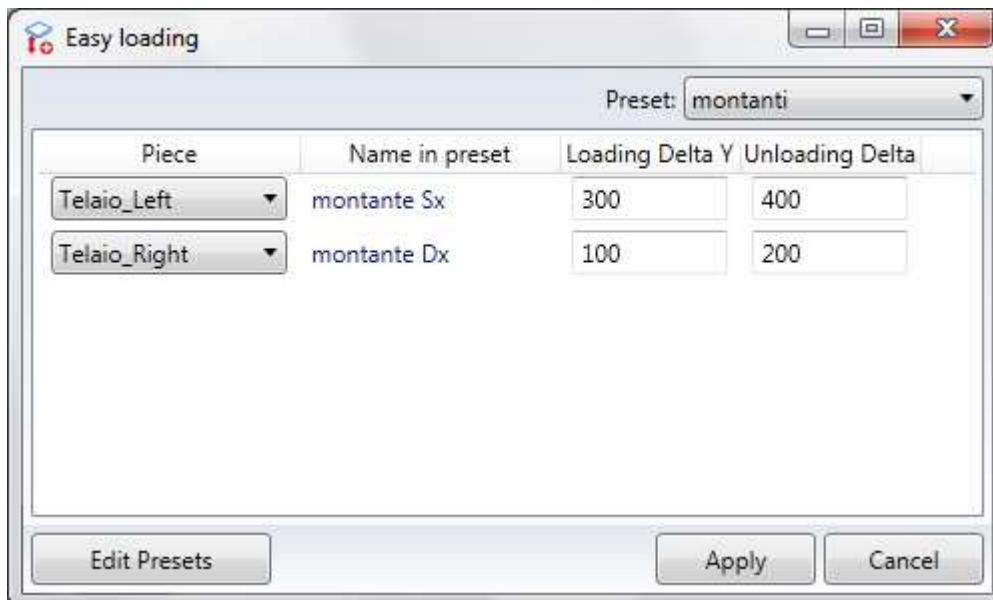
It may be necessary to use different easier load heights based on type of workpieces to be machined.

A range of "Presets" can be created to memorise the load and unload heights of the various workpieces to be machined.

### 3.11.3.1.2 Preset of easier load creation/change

Use the "Modify Presets" control in the "Easier load" screen to create or modify a new "Easier load preset".

The following screen appears after command is given:



The following screen appears by clicking on "New":



For entering a new preset name.

"Preset" field shows existing presets: for selecting the name of a preset to be modified.

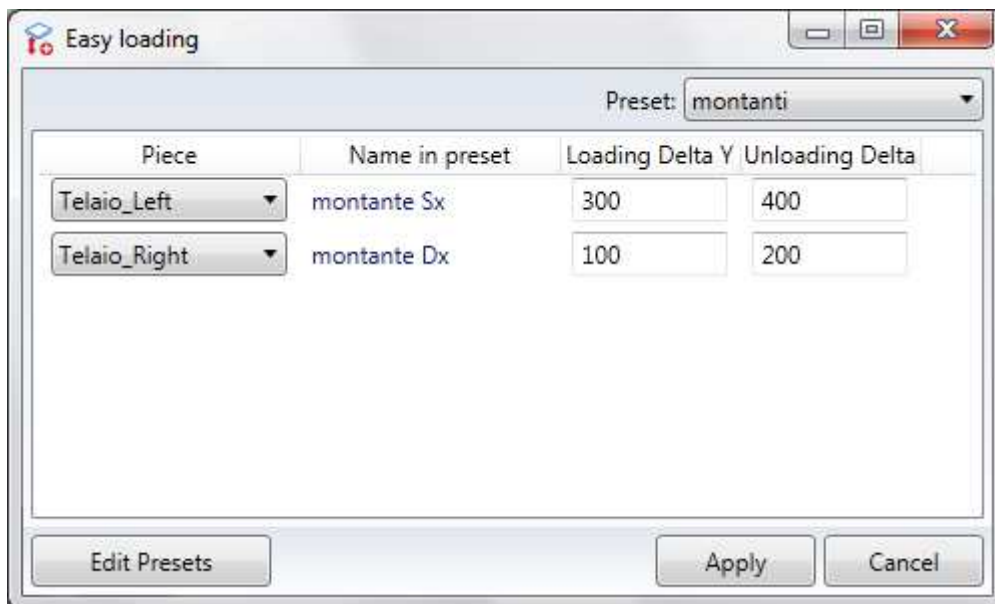
All screen fields can be modified with a visible preset.  
Click on "Add" to also insert a new workpiece.

Click on "Save" to memorise the made modifications.

### 3.11.3.1.3 Preset of easier load use

Select preset name from "Preset" box to associate those memorised to current project.  
Note: Visible presets are only those having the same number of workpieces of those of the current project.

The "Preset name" field is filled with names of workpieces described in preset and the "Y Delta" fields with relative values memorised in preset, once this is associated.



The workpieces' names are automatically associated if those memorised in preset and those in the current project coincide. If not, use the arrows right of the workpiece names to correctly associate them.

With "Apply", the "Easier load" function continues creating new phases as previously described.

### 3.11.3.2 Easier load elimination

Restore original phases if wanting to perform easier load again at different heights to those previously set. Eliminating the two "Load phase" and "Unload phase" added phases would be insufficient.

This is done using the "Delete easier loading" function in the "Setup" menu's "Plan recovery" sub-menu.

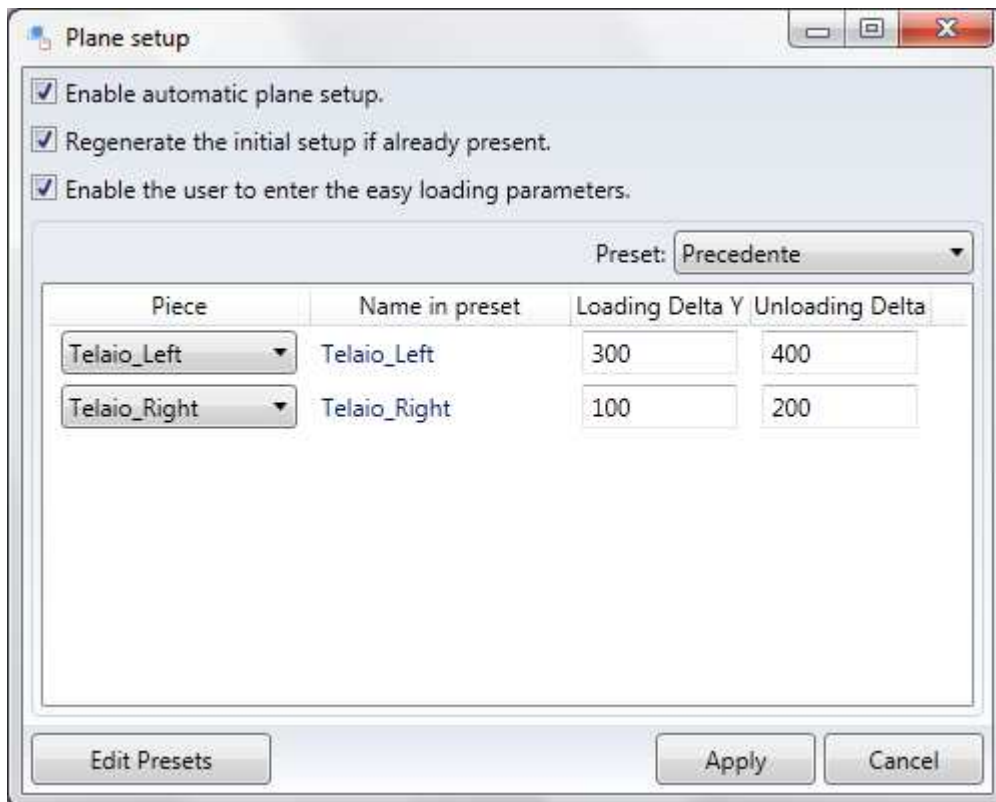


### 3.11.4 Work top automatic setup

This chapter describes many phases that can be automatically performed in sequence, without operator interaction.

This is done by directly clicking on the "Setup" menu's "Work top setup" control. Note the "Work top setup" icon is the same used as sub-menu for the previously seen individual functions.

The following screen appears after "Work top setup" command is given:



The following functions are performed in sequence once all is ticked and "Apply" is clicked:

- Automatic setup (see chapter 3.11.1.7.3)
- Easier load (see chapter 3.11.3.1)
- Initial setup (see chapter 3.11.1.8)
- Final setup (see chapter 3.11.1.9)
- Cycles generate (see chapter 3.11.1.10)

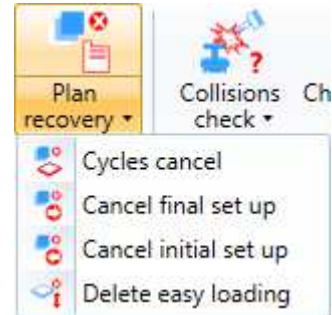
The project is enriched by the creation of all individual functions' phases, except "Final Setup" phase created only if required (for doors blocking).

### 3.11.5 Restoring the plan

This function is used when you want to eliminate all those phases or cycles that were automatically created with the "Plan set-up" function.

By clicking on the "Plan recovery" command in the "Setup" menu, all the operations visible in the drop-down menu will be performed automatically:

- Delete cycles
- Cancel final set up
- Cancel initial set up
- Clear facilitated load



The order of the cancellation operations is the exact reverse of the order of creation.

## 3.12 Definition of workpieces machined contemporaneously

In order to machine various workpieces define the following:

1. Number of workpieces to machine contemporaneously on the table
2. Whether to minimize the tool changes in the program

### 3.12.1 Number of workpieces machined contemporaneously

The number of workpieces to machine contemporaneously on the table depends on the dimension of the workpieces and the dimension of the machine table.

To enter various workpieces on the machine table you must create a project with various workpieces and display the position in machine with the "Arrange" function.

### 3.13 Project optimization

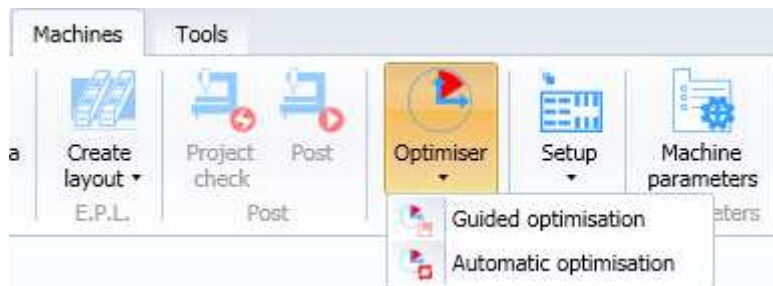
This option allows you to minimise the number of tool changes and the paths related to the transfer from one working process to another one.

Starting from a saved project, a new project will be created, with a working processes structure according to the modes indicated.

The project name will be "NameProjectOriginalOpt.pgm".

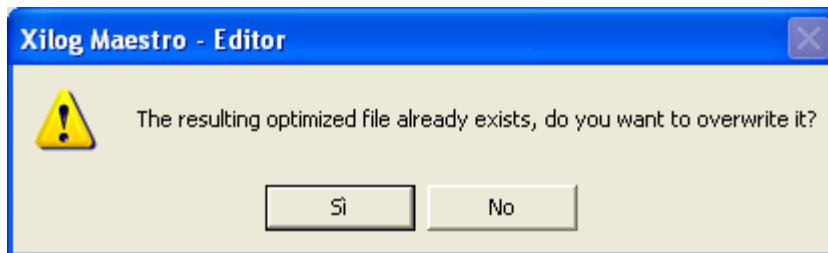
After the process, the source project will be closed and the optimised project will be opened.

In order to optimise a project, use commands "Guided optimization" or "Automatic Optimization" in the menu "Machines" "Optimizator".



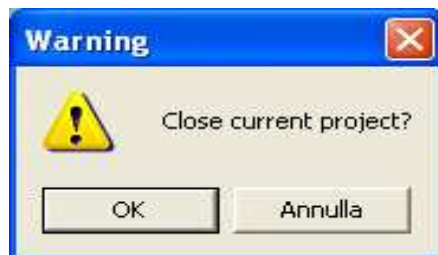
After applying the optimization modes described in following paragraphs, the file "NameProjectOriginalOpt.pgm". will be created.

If the optimised file was already been created, following message would appear:



By choosing "No", the optimization will end without saving any optimised file.

If you choose "Yes", following message will appear:



By choosing "Cancel", the optimization process will end.

By choosing "OK", the source project will be closed and the optimized project will open.

### 3.13.1 Standard projects optimisation

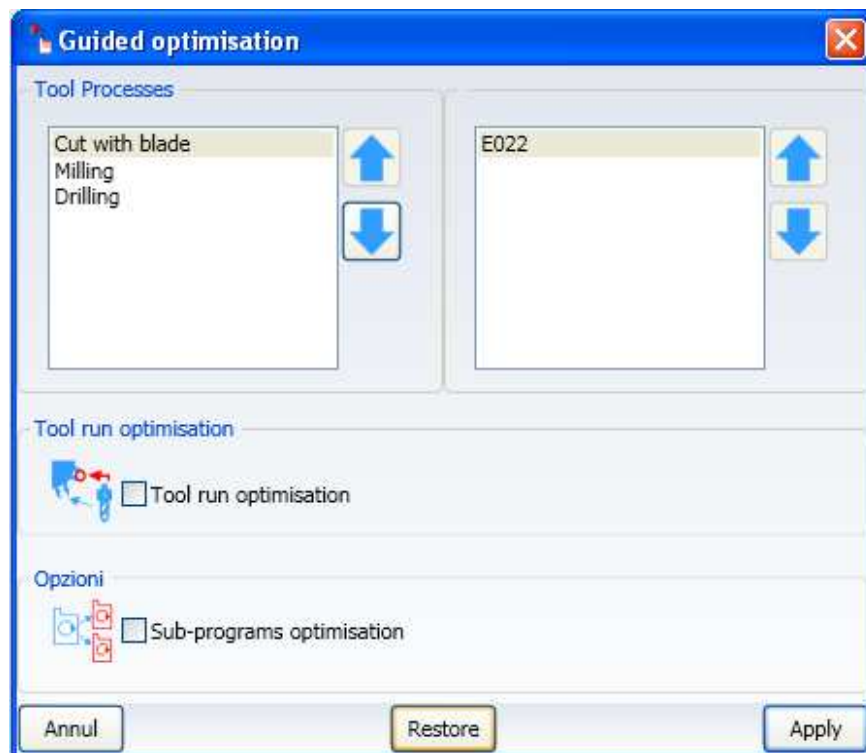
The mode for project optimisation can be:

- Guided
- Automatic

#### 3.13.1.1 Guided Optimization

This mode allows you to restructure the working process ordering in every step before optimizing the project.

By using "guided Optimization", following menu will open:



You can decide the optimization on one step or all steps by using the area "Working steps".

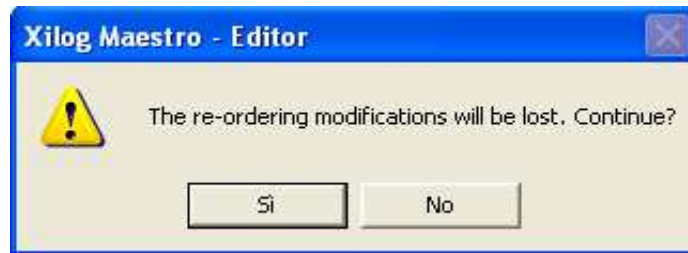
This workpiece of the menu will appear only if working processes on almost two steps exist.

In the area "Working processes", all working processes in the step or in the selected steps will be displayed. You can change the working process ordering with the arrows.

In the area "Tools", all tools in the selected type of working process will be selected. It is possible if you change the tool ordering with the arrows.



If you select the check "Sub-programs optimization", all working processes in the sub-programs will be drawn, following the same structure of the previous project. In order to avoid losing the ordering operations, following message will appear:

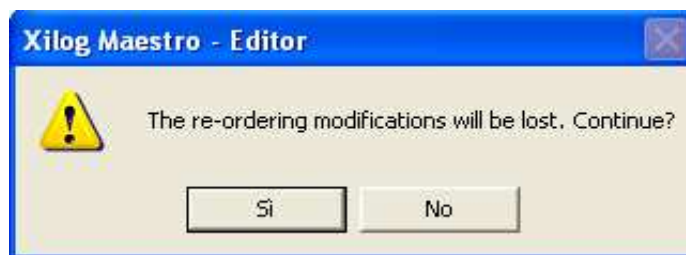


If you select the check "Optimization tool path", apart from tool change minimization, also the whole tool path is minimised among the different working processes.

After the selection, you can:

- Click on the command "Cancel" to avoid the optimization
- click on the command "Restore"
- Click on the command "Apply"

The command "restore" could restore the working tree as it was in the source project, so following message would appear:



With the command "Apply", the optimization for the tool change would be carried out without keeping the technological sequence on all working processes in the selected step.

### 3.13.1.2 Automatic optimization

By using the command "Automatic Optimization", following menu is open:



If you do not insert any check, the optimization for tool change is carried out on all project working processes without changing the technological sequence.

Tools change and overall tool path among the various machining are minimised if "Optimisation enable" is checked.

If you select the check "Optimization tool path", apart from tool change minimization, also the whole tool path is minimised among the different working processes.

If you select "sub-program optimization", before starting the process, all working processes in the sub-programs will be drawn. The working tree of the source project will be modified.

After the selection, you can:

- Click on the command "Cancel" to avoid the optimization
- Click on the command "Apply"

### 3.13.2 Nesting projects optimisation

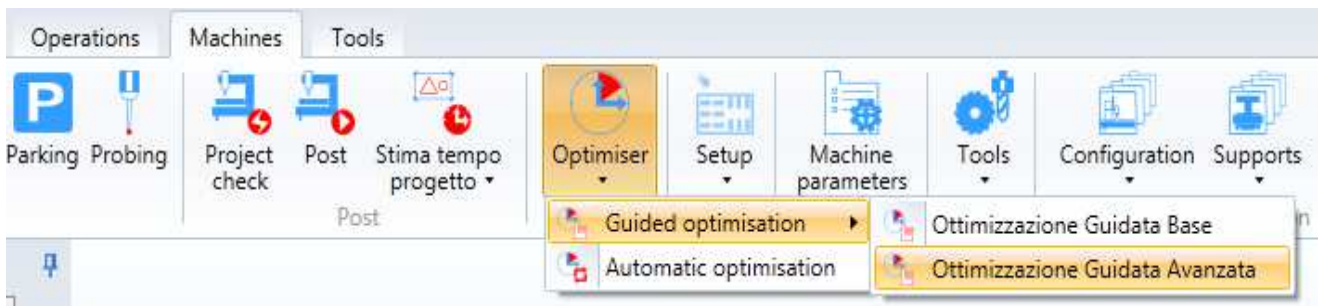
The mode for Nesting project optimisation can be:

- Guided
- Automatic

#### 3.13.2.1 Nesting Guided Optimisation

Guided optimisation is divided in another two modes:

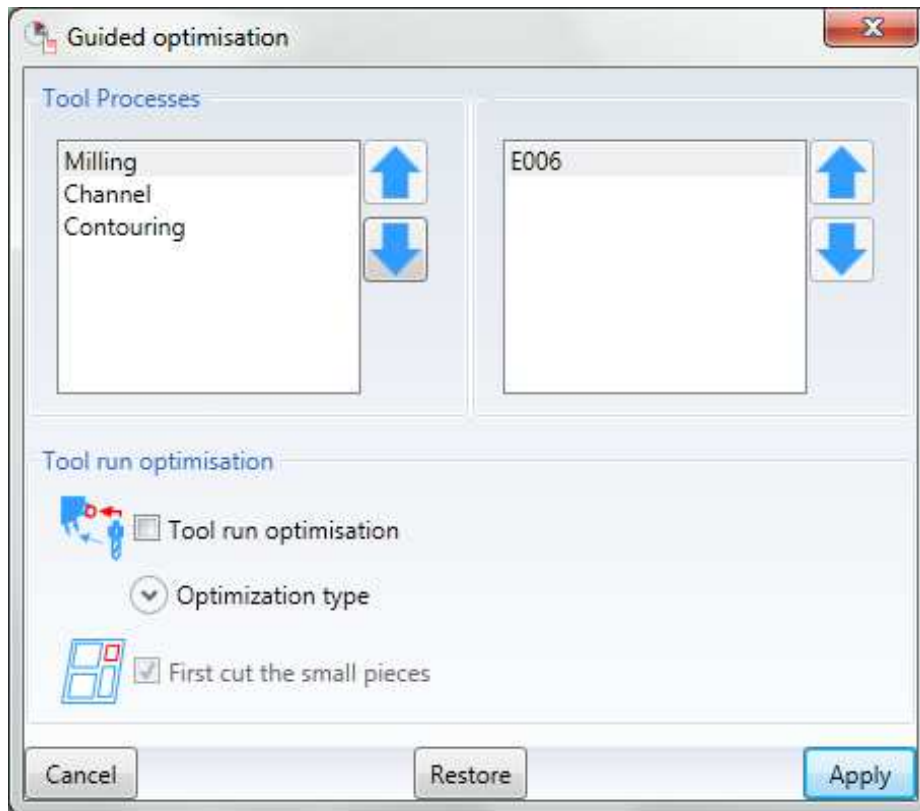
- Basic Guided Optimisation
- Advanced Guided Optimisation



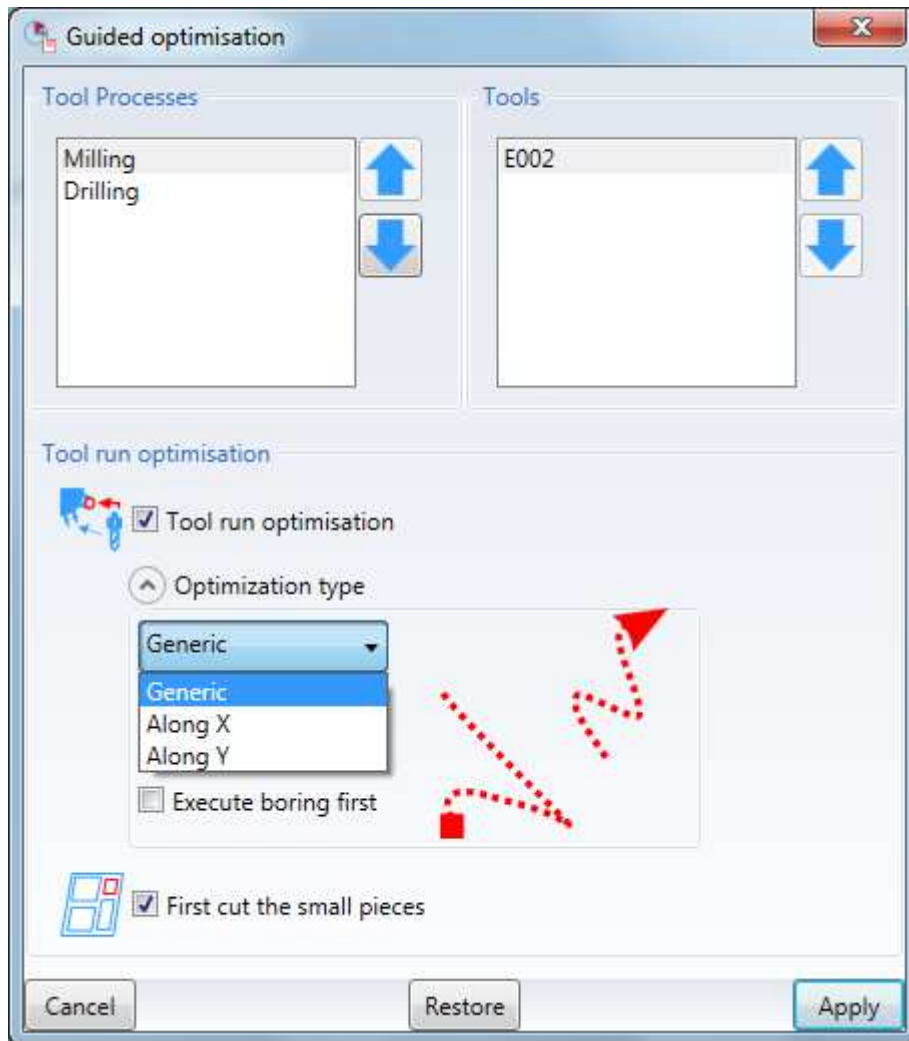
### 3.13.2.1.1 Basic Guided Optimisation

Before optimising the project, this allows manually re-organising the machining order within each phase.

The following menu opens using the "Basic Guided Optimisation" control:



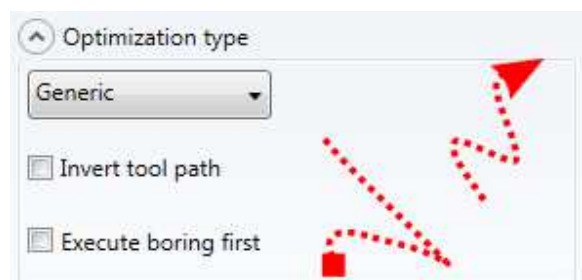
Basic optimisation is by machining sequence and a tool order can be indicated for every machining (the order can be manually modified for both *Machining* and *Tools* menus using the "UP" and "DOWN").



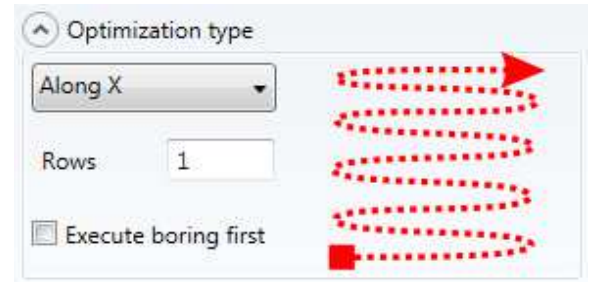
Enable *tool path optimisation* to:

- access the *Optimisation type* drop-down menu for specifying the tool path to be optimised according to *Generic* mode, *X Long* (optimises path according to movement along axis X), *Y Long* (optimises path according to movement along axis Y).
- Enable/disable the "Execute boring first" option.
- Enable/disable the *First cut the small pieces* option

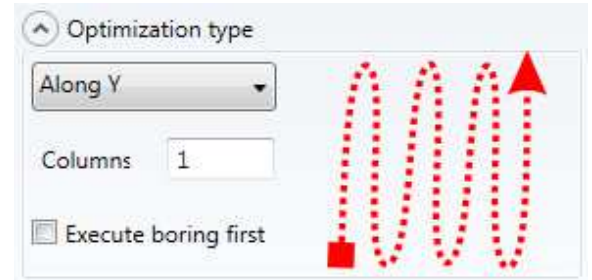
The generic optimisation allows the inversion of the machining tool path, by selecting the relative parameter.



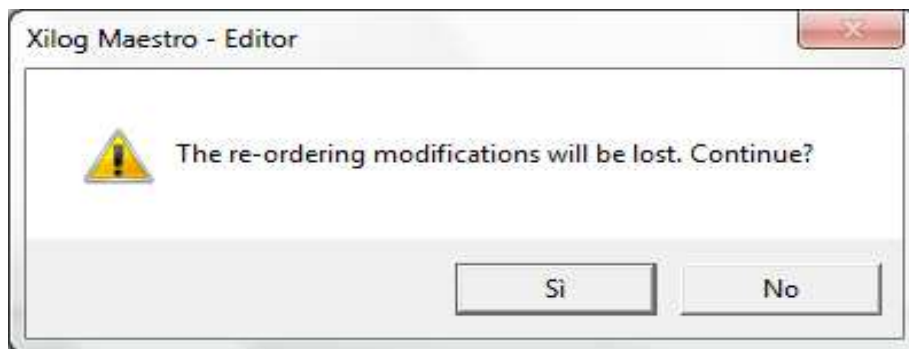
The optimisation along X allows the sheet to be virtually cut in horizontal strips that will have a Y dimension equal to the dimension of the sheet divided by the number of "Lines" set. The final result will be that the machinings after the optimisation will follow the progress displayed.



The optimisation along Y allows the sheet to be virtually cut in vertical strips that will have a Y dimension equal to the dimension of the sheet divided by the number of "Columns" set. The final result will be that the machinings after the optimisation will follow the progress displayed.



The guided optimisation window is closed using the "cancel" control. Initial condition, highlighting the following message, is restored using the "Restore" control.

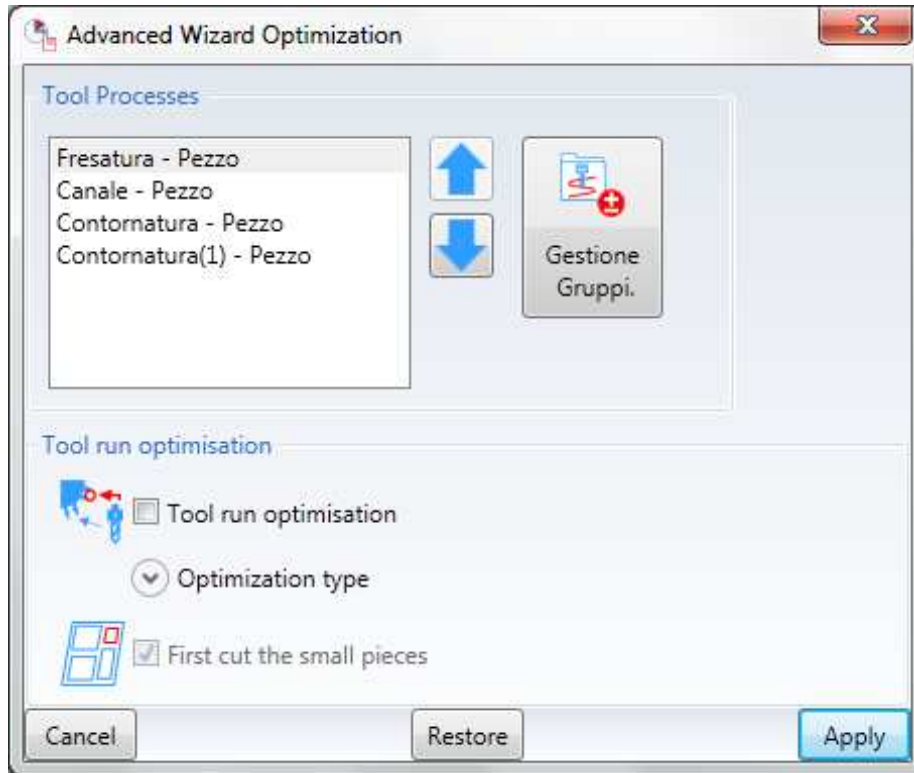


Optimisation for tool change is made with "Apply" control.

### 3.13.2.1.2 Advanced Guided Optimisation

Machining order is manually re-organised, creating groups containing different machining, through this mode.

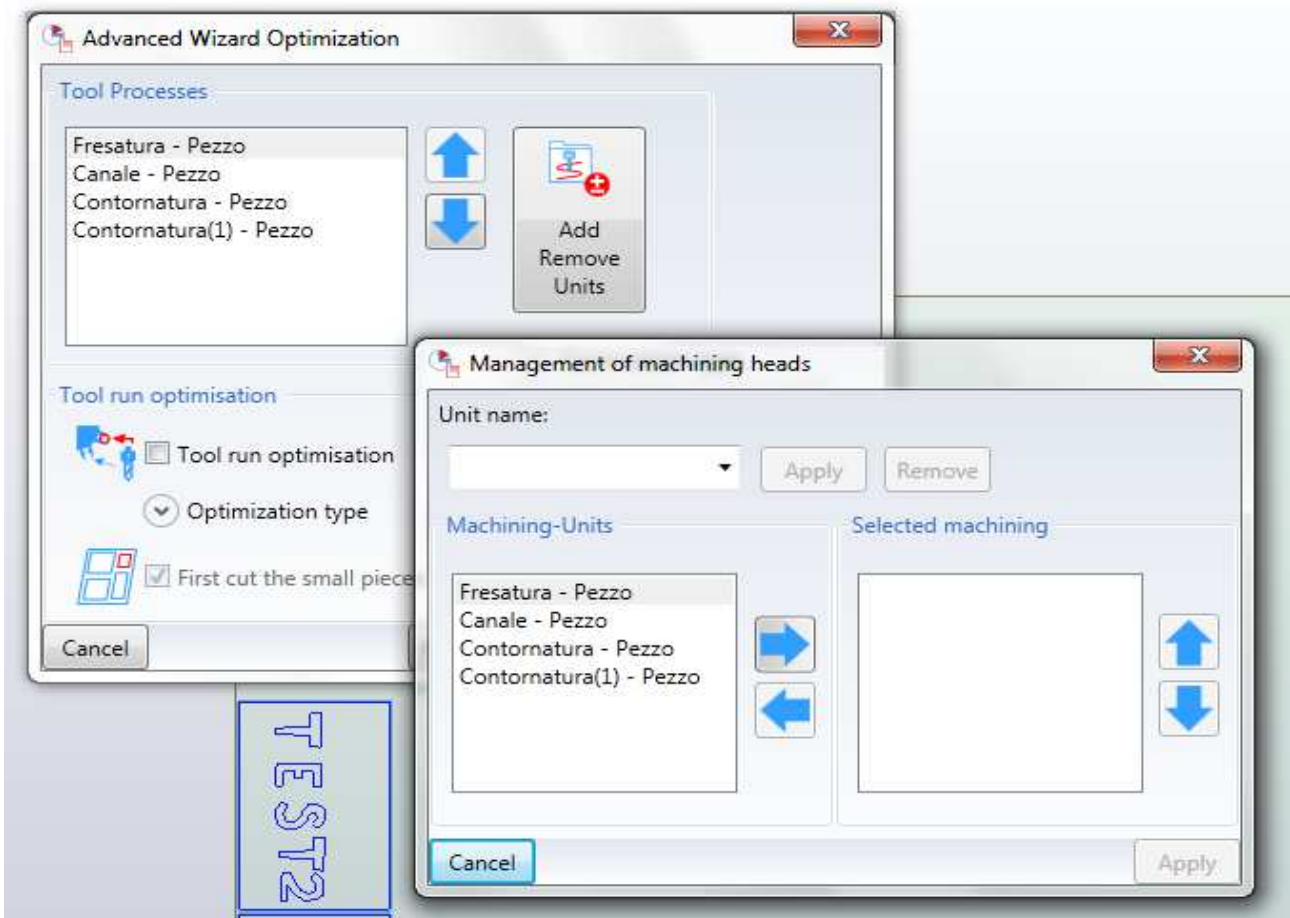
The following menu opens by selecting this optimisation mode:



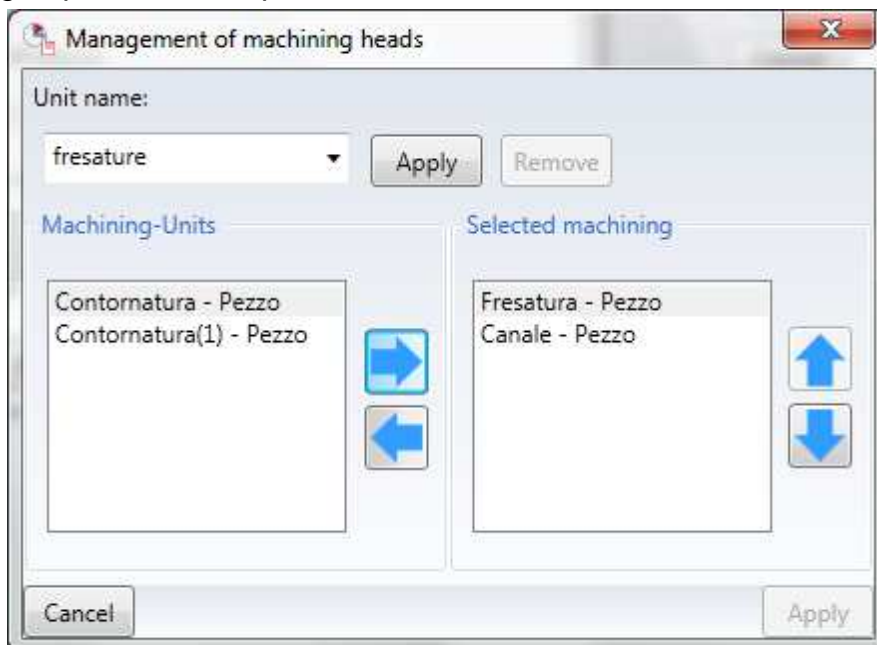
The machining section changes compared to basic optimisation: a button is present for groups' creation/removal.

A group is a set of machining (list on left) associated for managing its order priority compared to overall machining.

The following menu opens by selecting the Add/Remove Groups button:



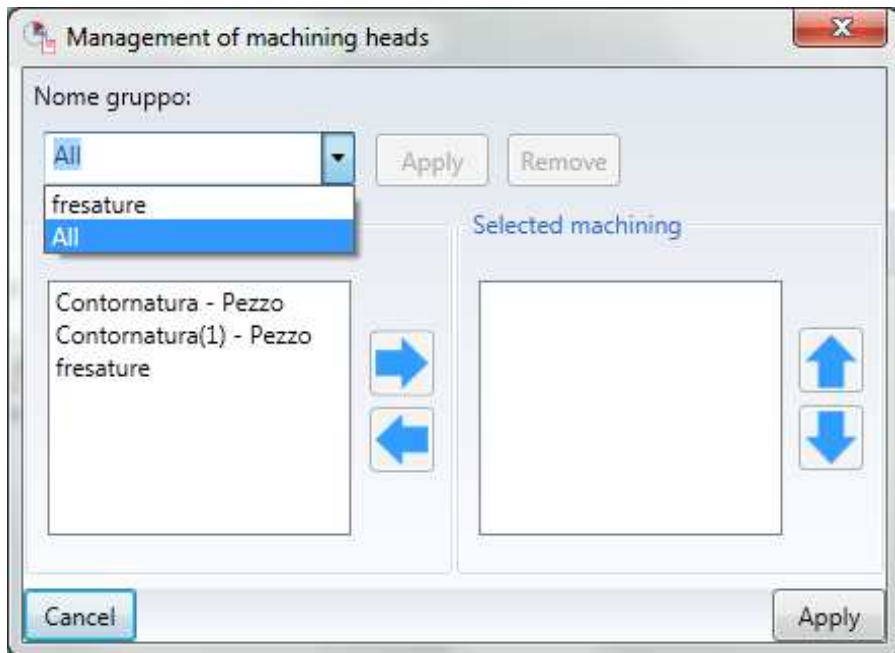
A group can be created containing machining in the list on right, using the *arrow towards right* button from the machining list on left. Enter the group's name in top-left box.



This group is added to the list (scrolled via the drop-down menu) using the Apply button.



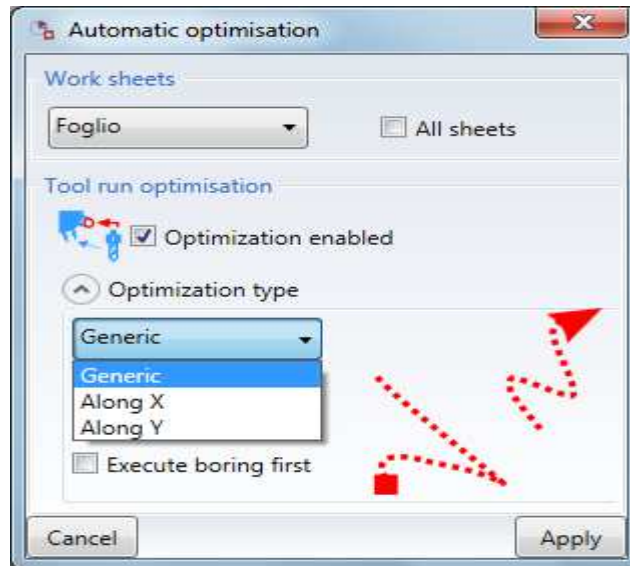
Advanced optimisation of all groups, following priority given during forming of the individual groups, is made by selecting "All".



All machining in *Milling* (with order established in the same group), then all *incisions* and finally the *Cuts* are performed in this case (above figure).

### 3.13.2.2 Nesting Automatic Optimisation

The following menu opens using the "Automatic Optimisation" control:



Tools change optimisation is performed on all project machining without altering the technological sequence if no check is inserted.

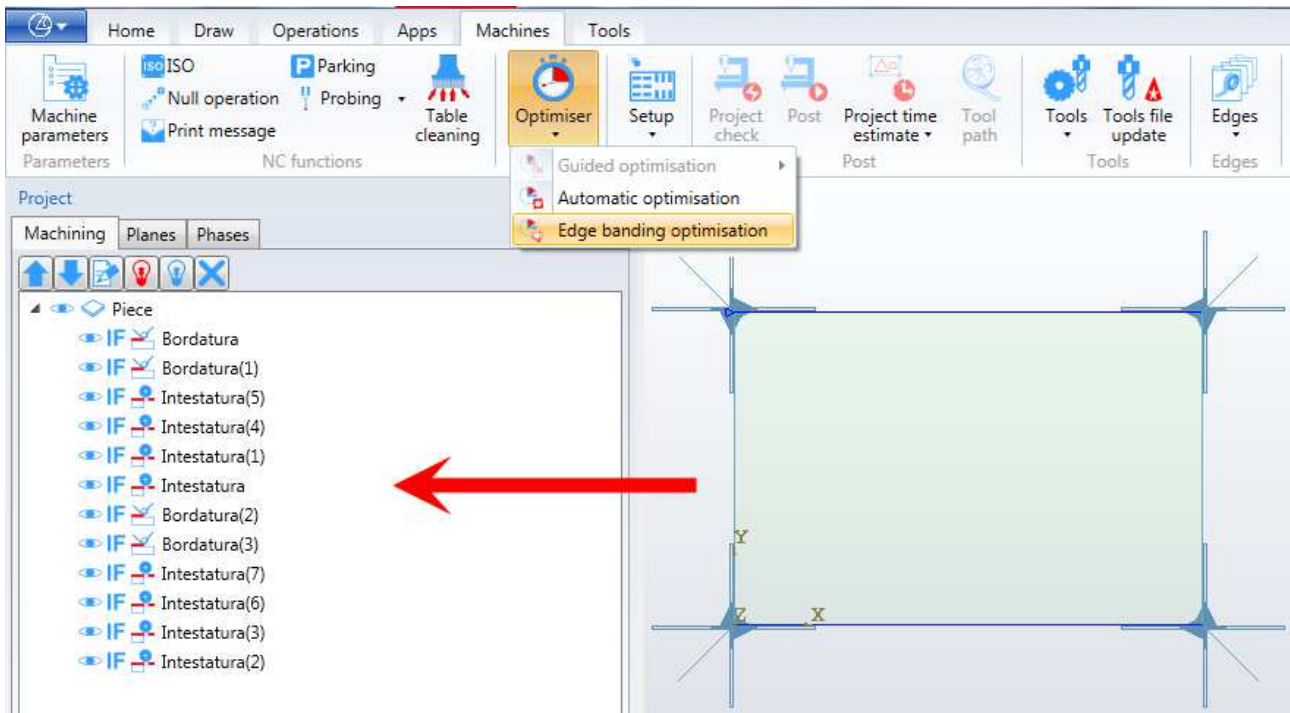
Tools change and overall tool path among the various machining are minimised if "Optimisation enable" is checked.

For a detailed description of the type of optimisation process see 3.13.2.1.1

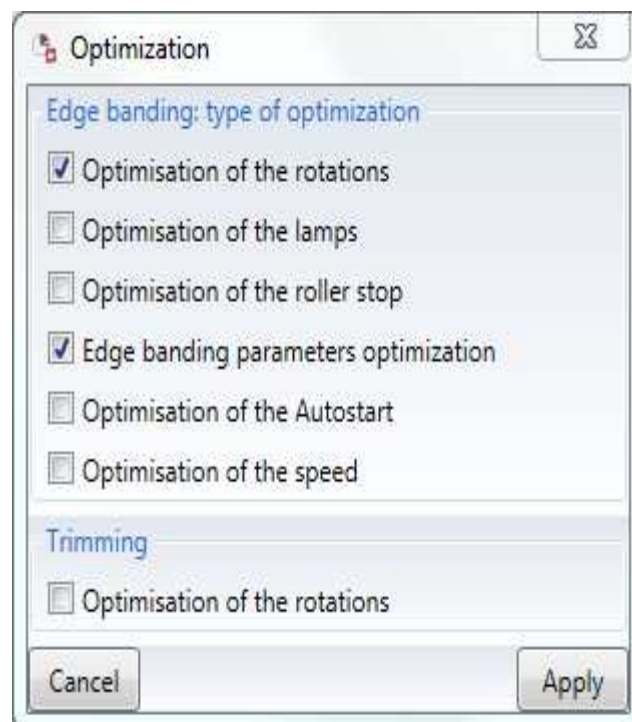
After the wanted selection is made:

- click on the "Cancel" control to miss optimisation
- click on the "Apply" control to proceed with optimisation.

### 3.13.2.3 Edge banding optimization

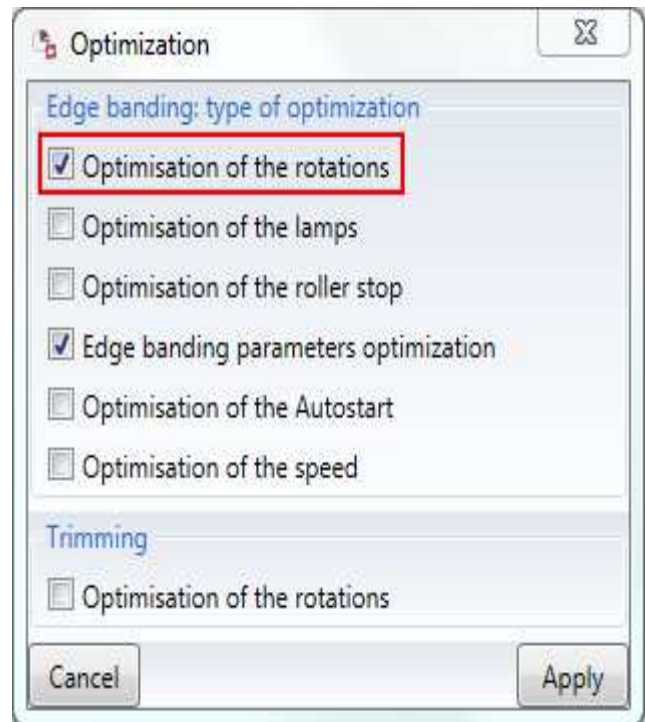


Once all the edge banding machinings have been created, they can be optimized by accessing machines, optimizer and select Edge Banding Optimizer. At this point a table will be displayed to select the various operations to optimize.



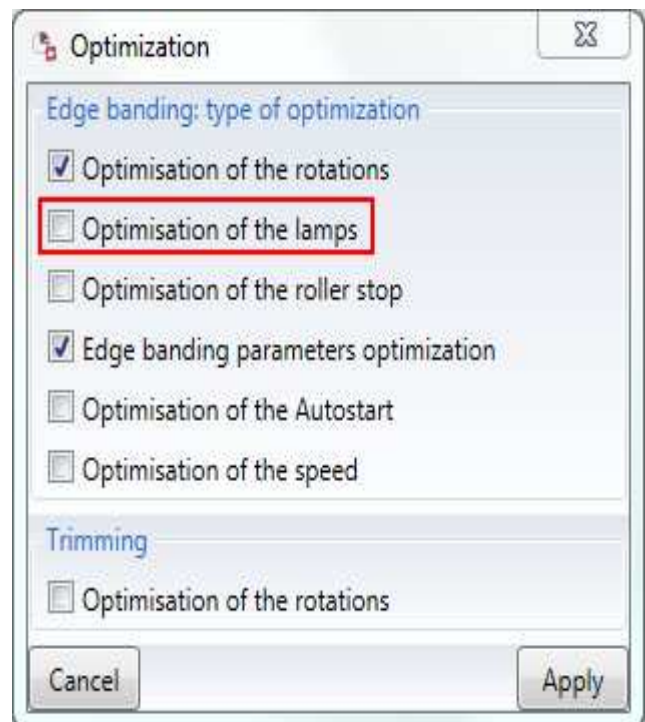
### 3.13.2.3.1 Rotations optimization

Select this optimization option to optimize the rotations of the edge banding head for the collisions check and to optimize the gluing between edge and panel.



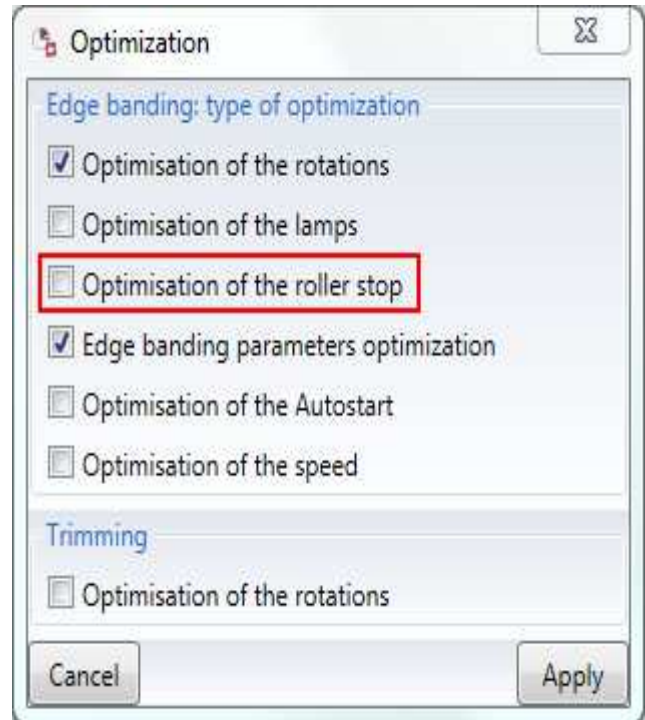
### 3.13.2.3.2 Lamps optimization

Select this optimization option to optimize the switch on of the lamps used to heat the edge, especially for geometries that include arcs.



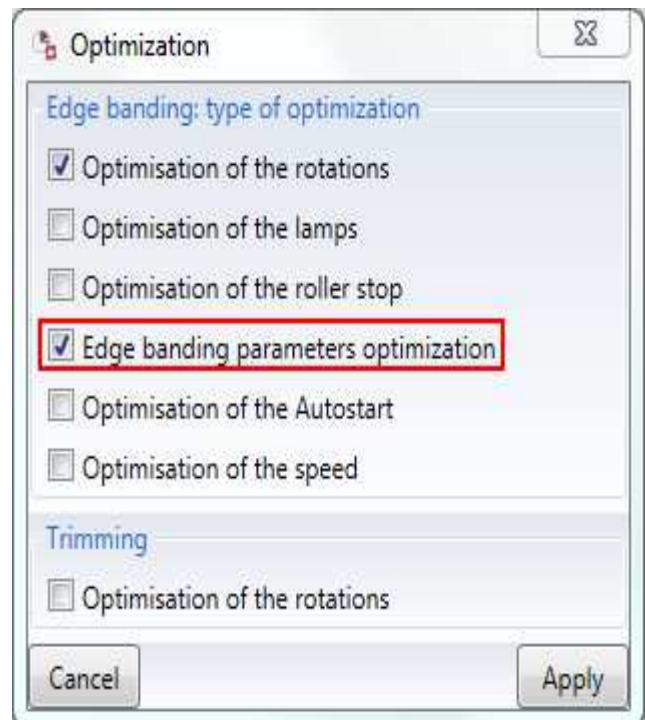
### 3.13.2.3.3 Stop roller optimization

Select this optimization option to optimize the roller stop of the edge banding head, especially for geometries that include arcs.



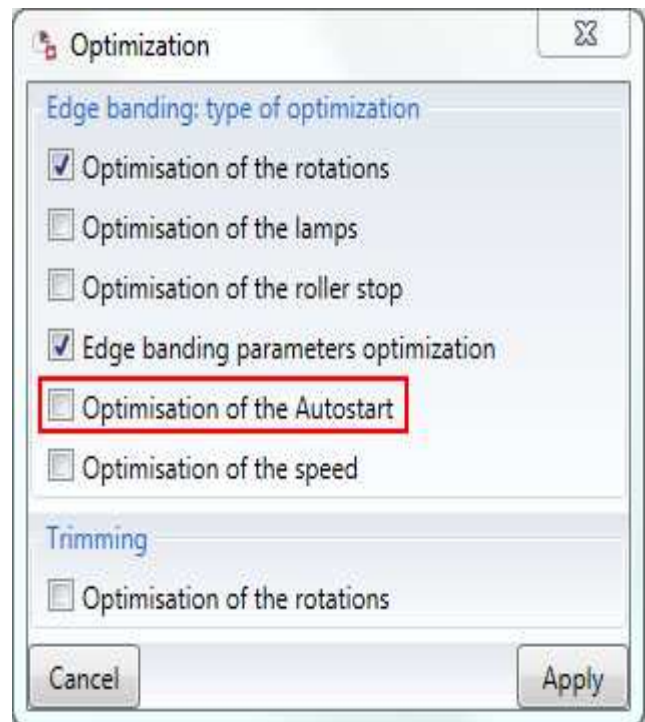
### 3.13.2.3.4 Edge banding parameters optimization

Select this optimization option to optimize the edge dispensing parameters and the infeed and outfeed wait times by using the technology data of the edge used for the edge banding.



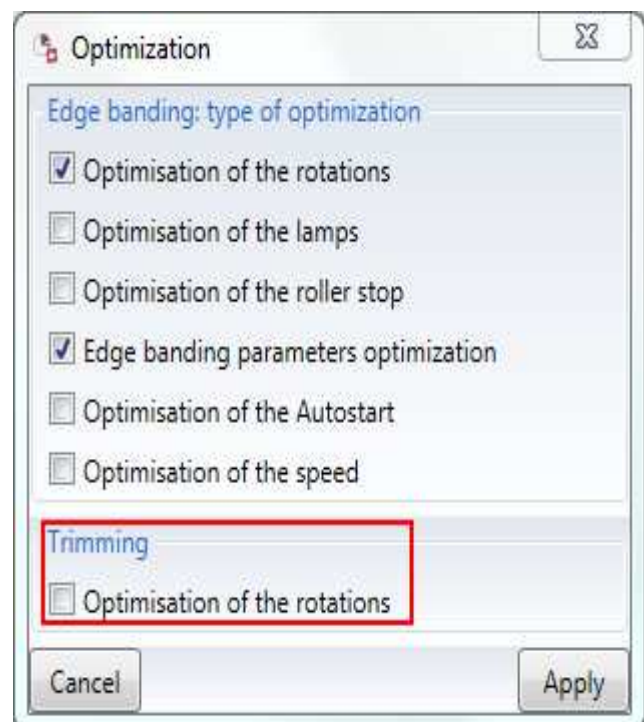
### 3.13.2.3.5 Autostart optimization

Select this optimization option to disable Autostart machine instruction for the first edge banding and enable it for the subsequent edge banding operations.



### 3.13.2.3.6 Trimmer rotations optimization

Select this optimization option to optimize the trimmer rotations to avoid collisions with the edge, especially when the trimmer changes direction.



### **N.B.:**

**The edge banding optimization will cancel all the attributes entered previously during the creation phase, replacing them with the optimized attributes.**

## 3.14 Automatic parts arrangement and optimization

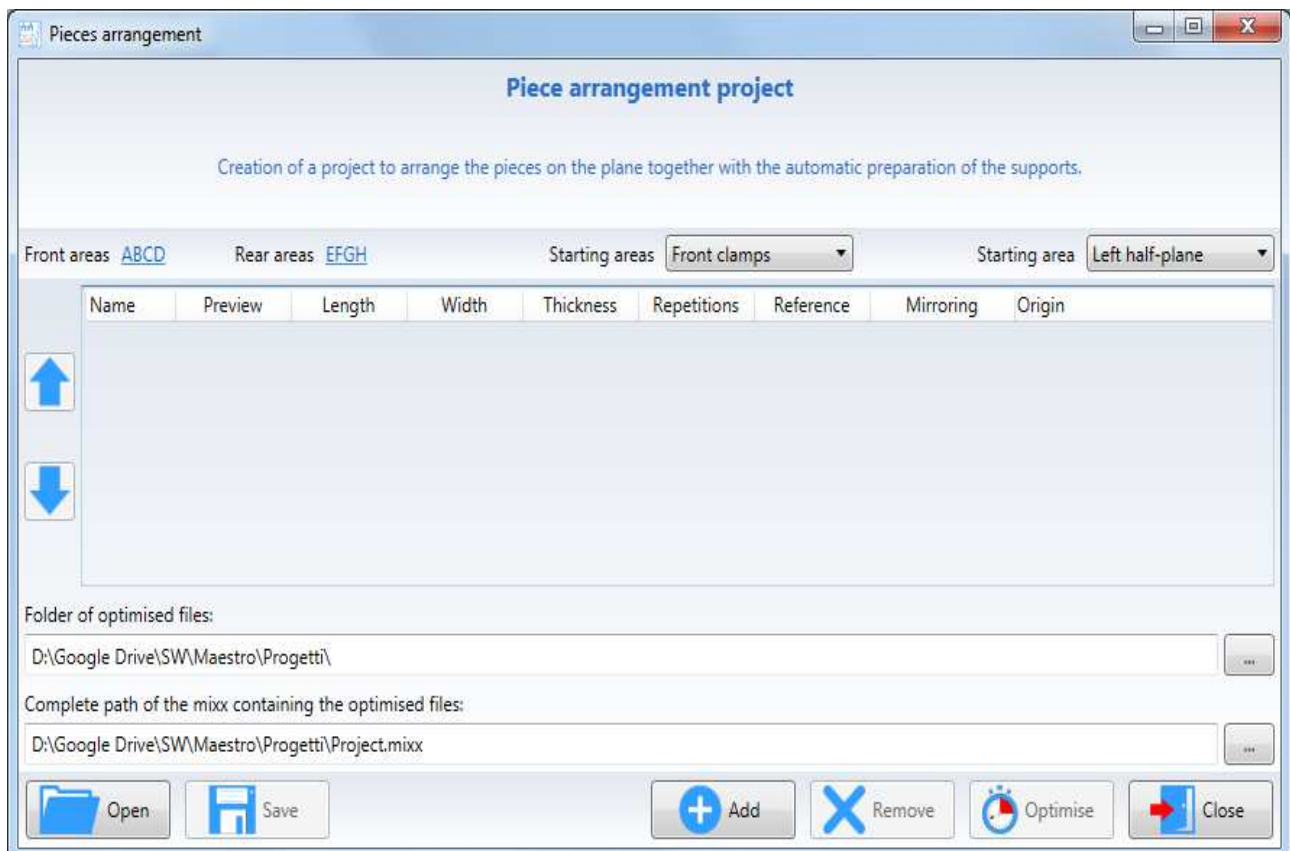
This function enables users to automatically place several pieces on the plane and to optimize the blocking and processing sequences

The premise is that a .eplx file has been previously saved defining how the plane is set up (see chapter 3.11.1.5)

To activate the function, use the command "Project-Pieces layout"



The following screen will appear:



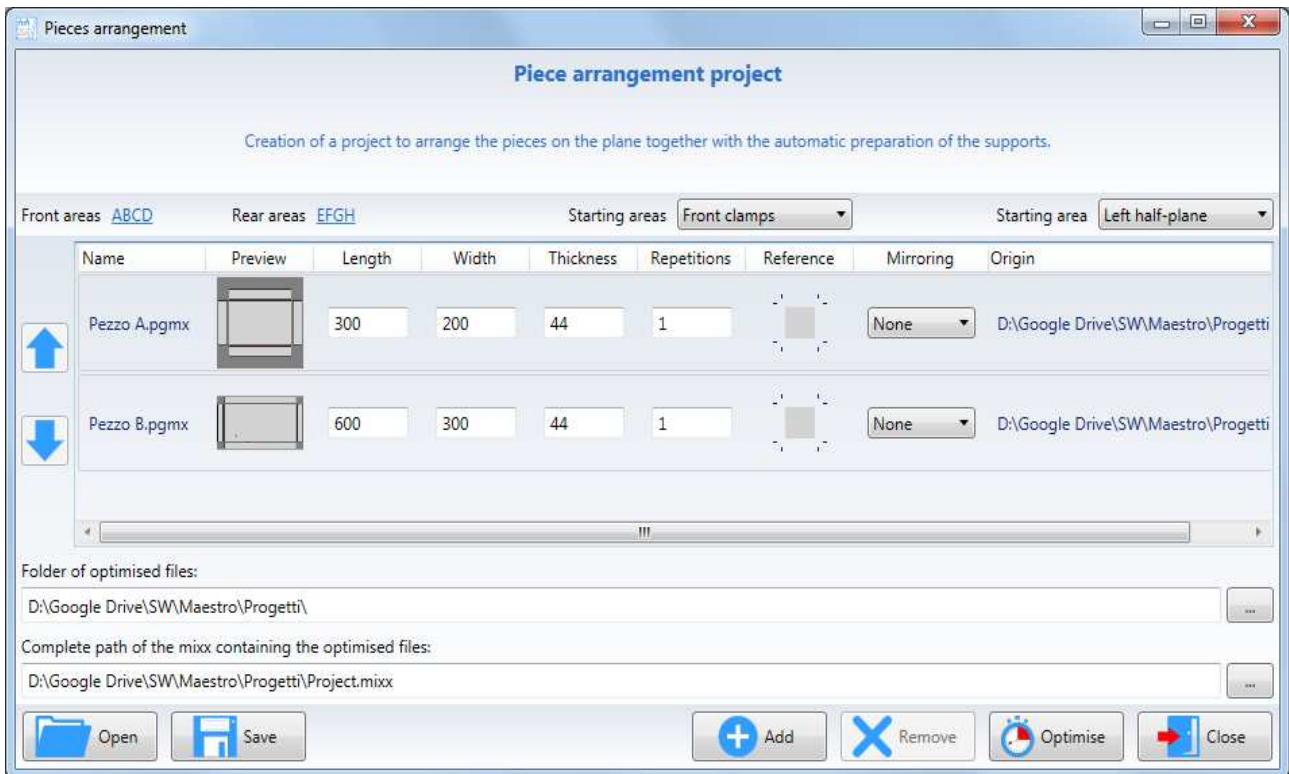
### 3.14.1 Add Project

With this command, you add a project to the list of projects to be optimized. The project must have a single piece inside.

The "Add" command is used






A list of pieces to be placed on the plane will appear:



### 3.14.2 Pgmx editor

By selecting a pgmx from the list, it will become blue so you can:

- Remove the pgmx with the key 
  - Move it to the top of the list with the key 
  - Move it down in the list with the key 
  - Change the quantity in the "Repetitions" box
  - Select one or two adjacent sides in the "Reference" field of the piece which must obligatorily be into abutment.
- The optimizer to decrease the generated pgmx could also rotate some pieces by 180 °.
- Select any mirroring in the "Mirroring" field



### 3.14.3 Saving and opening .wpx files

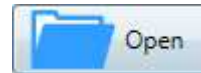
Files with the .wpx extension contain a list of projects to be optimized and blocked on the machine floor.

The wpx file is saved using the command



The save will create a file with the .wpx extension in the selected folder

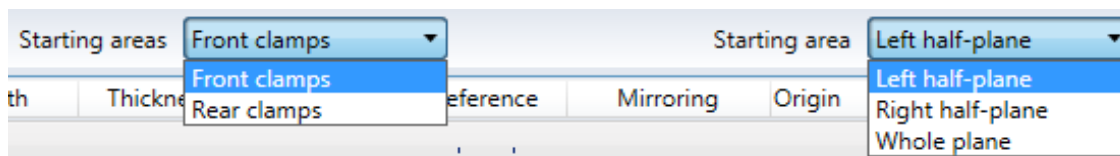
To open an existing .wpx file, use the command



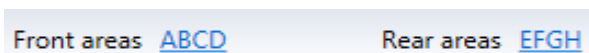
### 3.14.4 Optimisation criteria

The optimisation criteria determine from which part of the plane the pieces will start to be positioned.

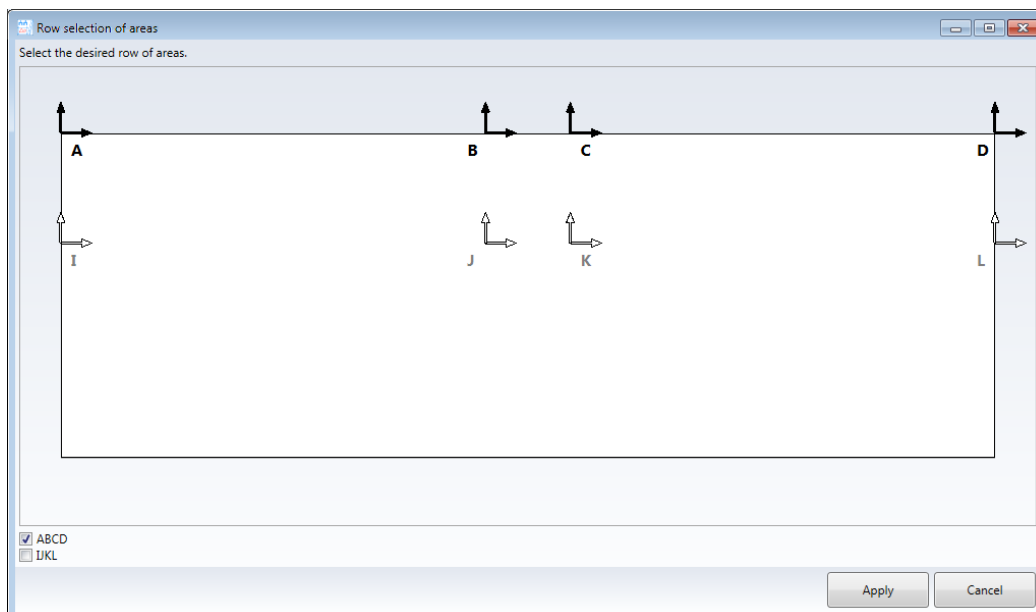
To this end, the "Departure areas" and "Starting area" drop-down menus are used.



The areas where you can position pieces are those read from the configuration:



The front areas are those near the operator, while those to the rear are those away from the operator. By clicking on ABCD, the following screen appears:



Where you can select or deselect the desired row of areas

The selection of the "Left half-plane" or "Right half-plane" will make the wizard choose the "pendulum" mode to calculate the area of the optimised programs. Selecting the "Whole Plane" will generate all programs in the AD area.

### 3.14.5 Optimisation

Piece layout optimisation is done with the button



The processing result will consist of:

- A .mixx file containing the list of generated pgmx
- many pgmx files containing one or more pieces

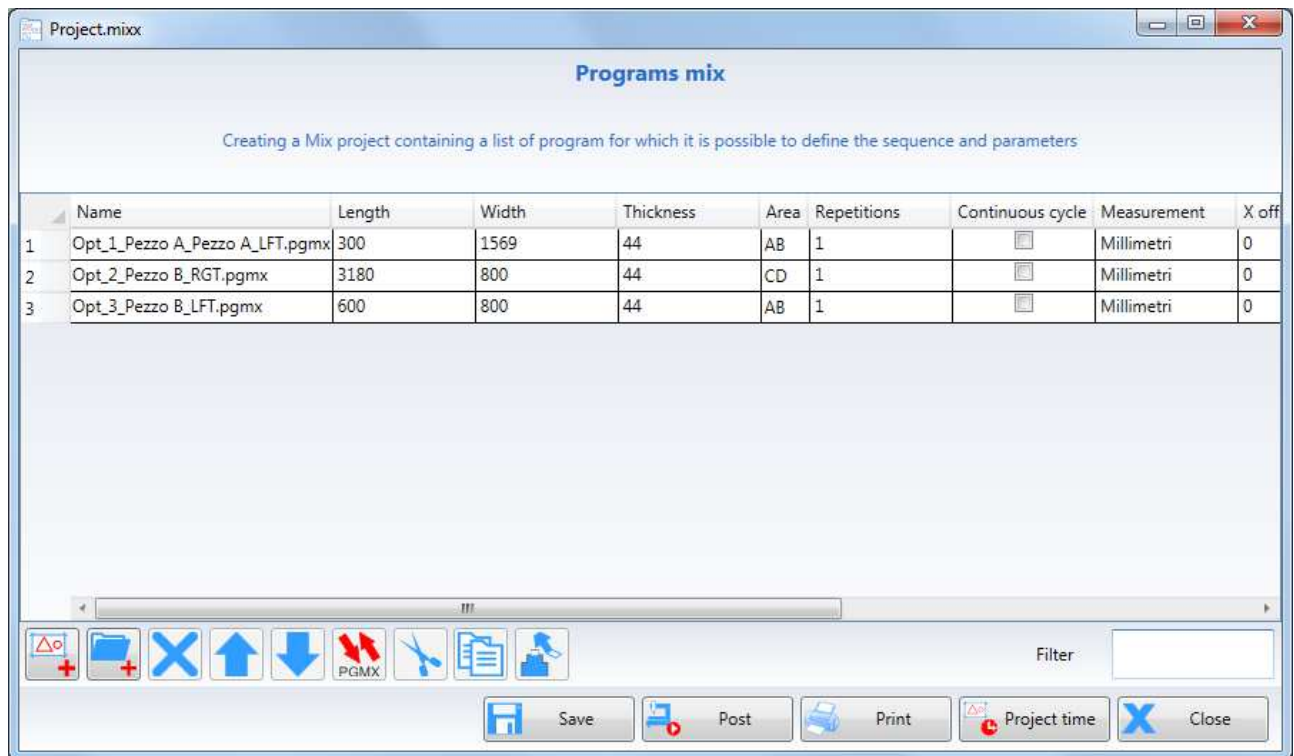
#### 3.14.5.1 Generated .mixx file

The generated .mixx file can be positioned and renamed using the field:

Complete path of the mixx containing the optimised files:

D:\Google Drive\SW\Maestro\Progetti\Project.mixx

The .mixx file will contain a list of all the .pgmx files generated during processing



#### 3.14.5.2 Generated .pgmx files

The generated .pgmx file can be placed in the desired folder using the field:

Folder of optimised files:

D:\Google Drive\SW\Maestro\Progetti\

Each generated .pgmx file will respect the set criteria.

## 3.15 Mathematical functions

Maestro features the following mathematical functions:

Abs(value)	(Absolute value)
Acos(value)	(Arcsine)
Asin(value)	(Arcsine)
Atan(value)	(Arctangent)
Tan(value)	(Tangent in degrees)
Tanh(value)	(Hyperbolic tangent in degrees)
Cos(value)	(Cosine in degrees)
Cosh(value)	(Hyperbolic cosine in degrees)
Sin(value)	(Sine in degrees)
Sinh(value)	(Hyperbolic sine in degrees)
Exp(value)	(Exponential)
Ceiling(value)	(Round up)
RU(value)	(Round up)
Floor(value)	(Round down)
RD(value)	(Round down)
Log(value)	(Natural logarithm)
Log(value, newBase)	(Basic logarithm newBase)
Max(val1; val2)	(Maximum)
Min(val1; val2)	(Minimum)
Pow(x; y)	(Raise x to the y power)
Round(value)	(Rounding off)
Sqrt(value)	(Square root)
-	(Subtraction 600-400 = 200)
+	(Addition 600+400 == 1000)
/	(Division 400/200 = 200)
%	(Remaining division (417% 100 = 17))
*	(Multiplication 400*2 = 800)
()	(Round brackets)
>	(Greater dx1>200)
<	(Lower dx1<200)
>=	(Greater equal dx1>=200)
<=	(Lower equal dx1<= 200)
<>	(Different 200 != 300)
=	(Equal 200 == 200)
^	(Power 2^2=2*2= 4, 2^3=2*2*2=8)
(Value1)AND(Value2)	(Operator AND (dx1>600)AND(dy1>400))
(Value1)OR(Value2)	(Operator OR (dx1>600)OR(dy1>400))
(Value1)XOR(Value2)	(OR exclusive (dx1>600)XOR(dy1>400))
NOT(Value)	(Negation NOT(dx1>1600))
IF(Condition;Value1; Value2)	(Condition IF (Condition == true) (Value1) else (Value2))

The functions listed can be used in the parameter or variable fields of the context menus.

## 4 Project Management for Cyflex

Cyflex machines have a different table compared to classic routers provided with a multifunctional table or a bars one; this is why the tools disposition on the machine needs to be treated separately.

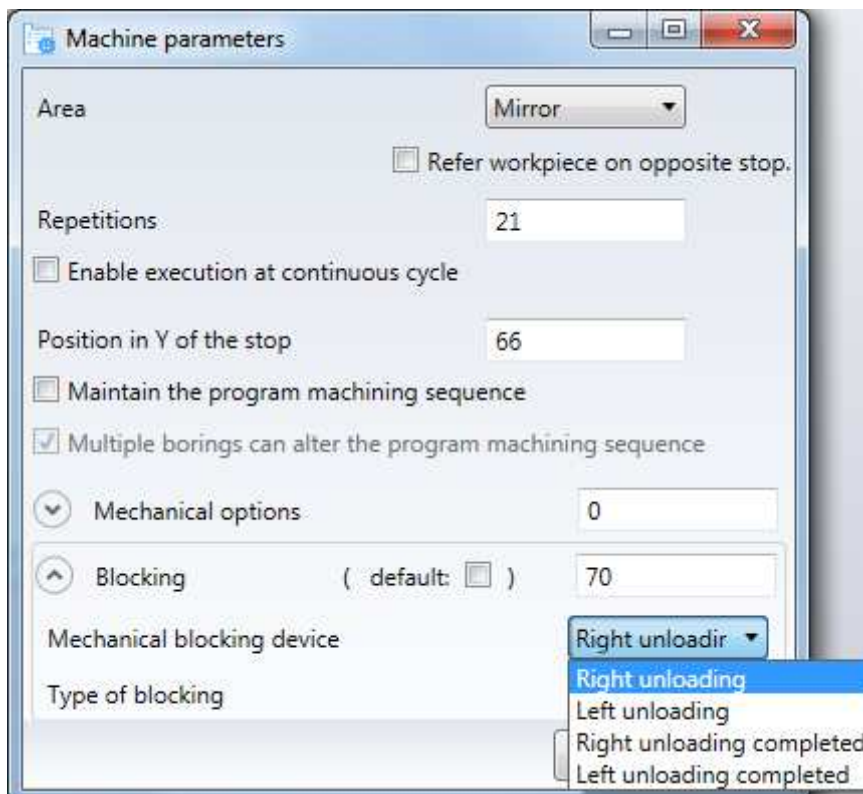
### 4.1 Loading panels on Cyflex.

The panels to be machined in Cyflex must be loaded manually, positioning them on the left side of the machine and moving them against the reference stop on the right.

### 4.2 Unloading panels on Cyflex.

The panels are unloaded from the machine automatically, but the side and unloading mode can be selected.

The selection is made on the "Machine parameters" - "Hold-down device"



The options in the drop down menu are:

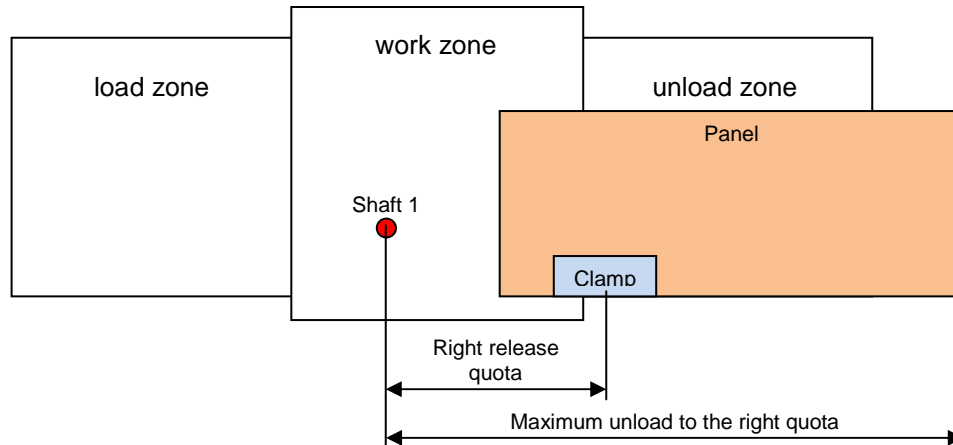
- Unload to the right
- Unload to the left
- Unload to the right complete
- Unload to the left complete

## 4.2.1 Unload to the right (V70)

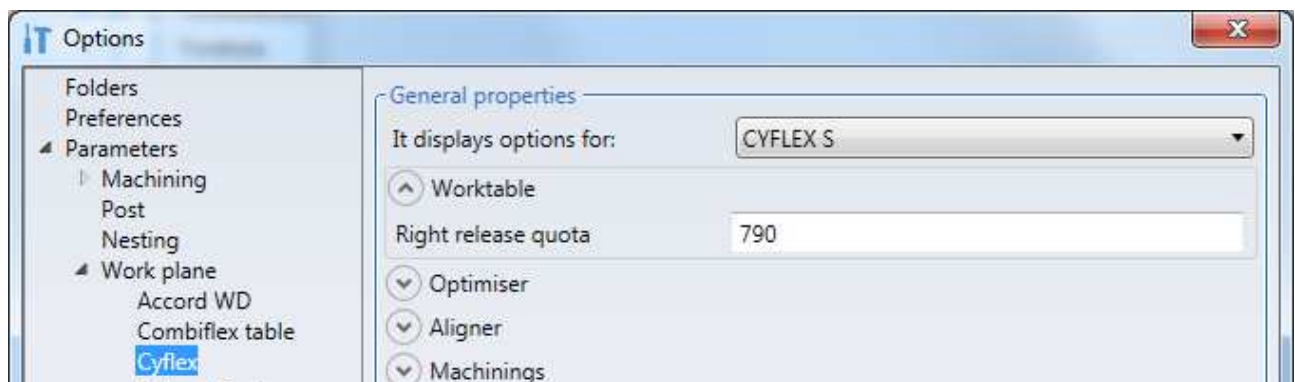
The unload to the right differs whether you are using a Cyflex S or a Cyflex HP.

The unload to the right for the Cyflex S moves the centre of the clamps to the "Right release quota", however the tail of the panel could remain the work zone when:

- the clamp reaches the positive limit switch
- the panel head reaches the "Maximum unload to the right quota"

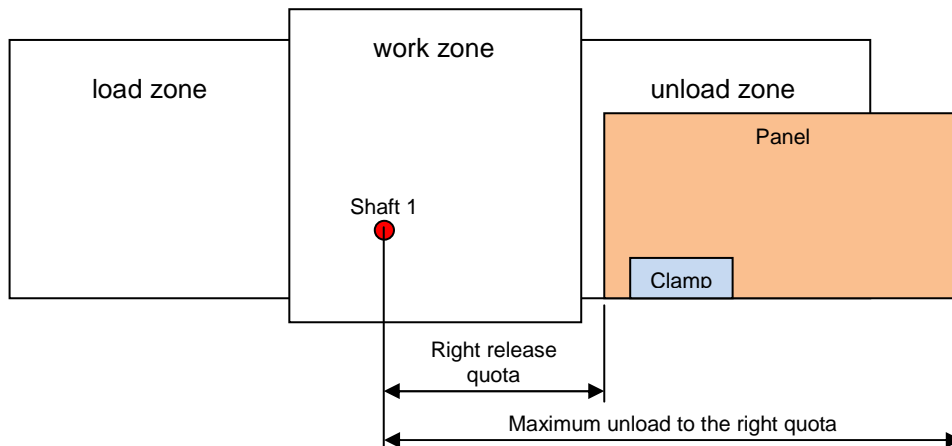


The "Right release quota" parameter for the Cyflex S can be changed in the session: "Tools-Options-Parameters-Work table-Cyflex -Work table" selecting "CYFLEX S"



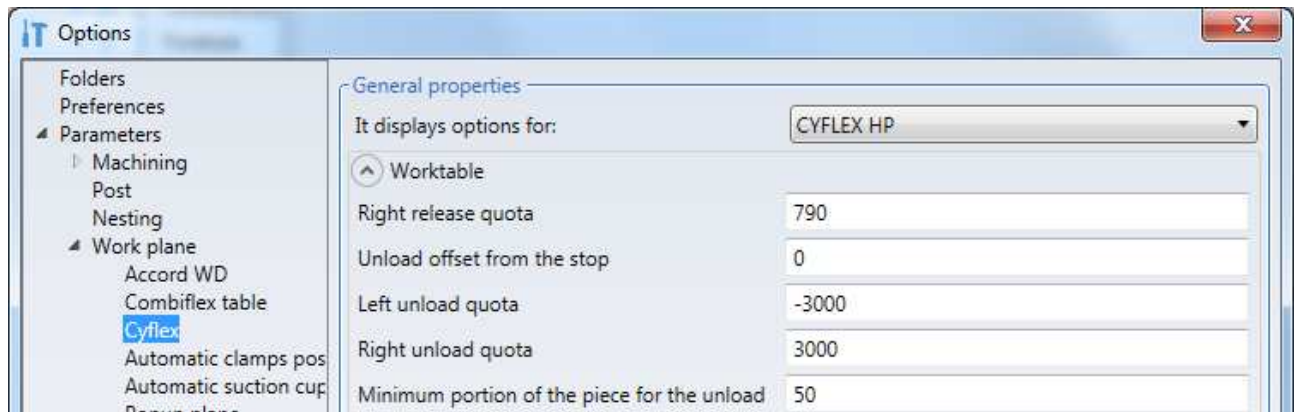
Whilst the "Maximum unload to the right quota" must be set in Xilog

The right discharge for the Cyflex HP is carried out by bringing both clamps to their right limit switch, opening the right clamp before moving. In this case, the "Right junction level" is not considered: if the clamps have enough running to bring the panel beyond that height, the movement is carried out without limitation. The only constraint is given by the parameter "Maximum discharge level on the right", which represents the limit beyond which the head of the panel cannot be carried.



The "Right release quota" and "Maximum unload to the right quota" parameters for the Cyflex HP are changed in the session:

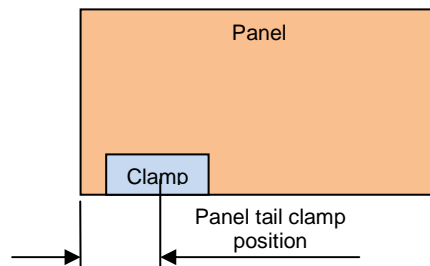
"Tools-Options-Parameters-Work table-Cyflex -Work table" selecting "CYFLEX HP"



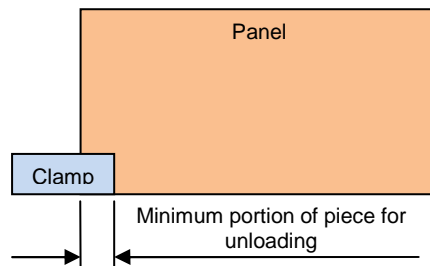
With this type of unloading, no ribbing is performed: the left clamp remains at the same coordinate used to perform the last machining, resulting in different unloading positions depending on the machining and optimization performed. To ensure that the panel is moved to a fixed coordinate, you must use "Full right unloading".

## 4.2.2 Unload to the right complete (V72)

For the Cyflex S, this unloading mode involves, like the V70, bringing the centre of the clamps to the "Right junction level", with the only difference being that before unloading the clamp will be repositioned to avoid the panel remaining in the machining area. In practice, you will bring the centre of the clamp to the "Panel tail clamp position" level (this level can only be modified by Xilog).



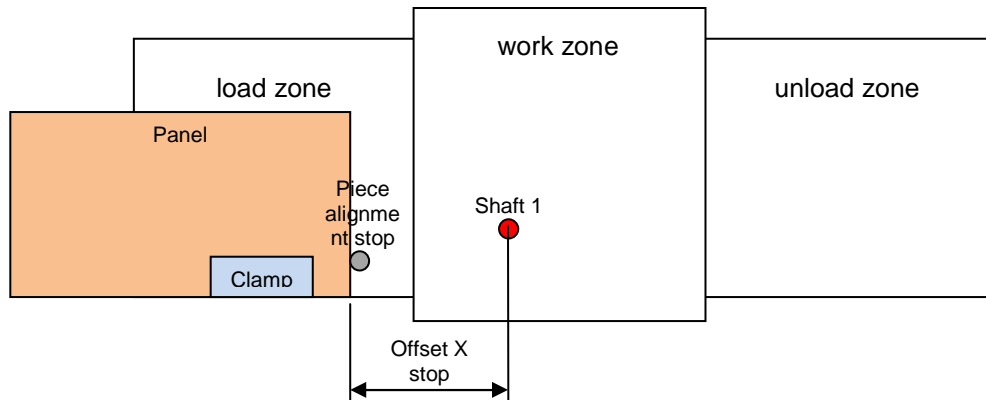
As for Cyflex HP, with this unloading mode, the tail of the workpiece is moved to the "Right junction level", taking into account also the "Minimum portion of the part to be unloaded". To ensure that the desired position can be reached, reinserts may be made before the panel is unloaded. As in the V70 mode, even in this case, do not take the head of the panel beyond the "Maximum right unloading level".



### 4.2.3 Unloading to the left (V71)

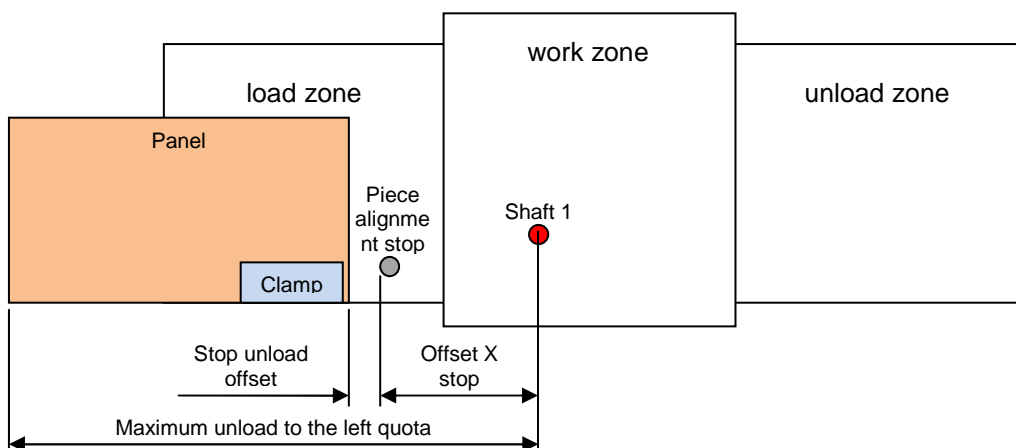
The unload to the left has a slightly different mode whether you are using a Cyflex S or a Cyflex HP.

The unload to the left for the Cyflex S is executed by trying to move the panel head to the "Offset X stop", however the panel head could remain in the work zone as no re-clamping is performed



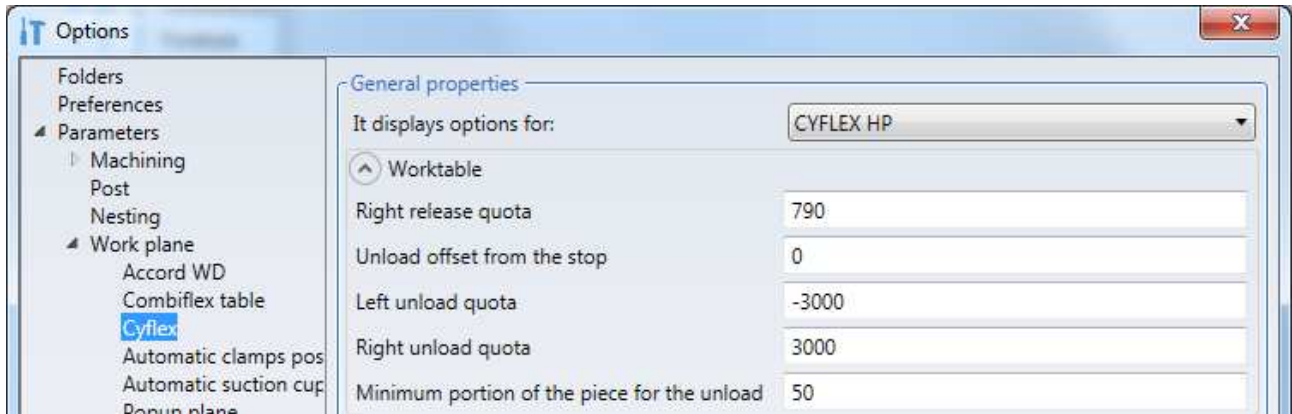
The "Offset X stop" parameter is a configuration parameter, changed in Xilog.

Left side unloading for the Cyflex HP is carried by bringing both clamps to their left hand limit switch, opening the left clamp before moving. In this case, the "Offset unloading stop" is not considered: if the clamps have enough running to bring the panel beyond that height, movement is carried out without limitation. The only constraint is given by the parameter "Maximum left unloading level", which represents the limit beyond which the head of the panel can not be carried.





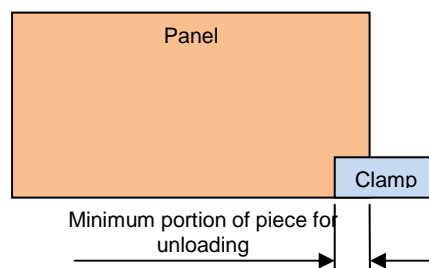
The "Stop unload offset" and "Maximum unload to the left quota" parameters for the Cyflex HP are changed in the session:  
 "Tools-Options-Parameters-Work table-Cyflex -Work table" selecting "CYFLEX HP"



With this type of unloading no ribbing is made: the right clamp remains at the same coordinate used to perform the last machining, resulting in different unloading positions depending on the machining and optimization performed. To ensure that the panel is moved to a fixed coordinate, you must use "Full left unloading".

#### 4.2.4 Unload to the left complete (V73)

This unloading mode is only available for Cyflex HP. With it the workpiece head is moved to match the "Stop unloading offset", also taking into account the "Minimum portion of the workpiece for unloading". To ensure that the desired position can be reached, insets may be made before the panel is unloaded. As in the V70 mode, even in this case, do not move the head of the panel beyond the "Maximum left unloading level".



## 4.3 Optimisation for Cyflex programs

The operator does not have to launch the optimisation, because when the pgmx is called from the machine panel, the optimisation process will be launched automatically.

Before the program is optimised proceed with the following:

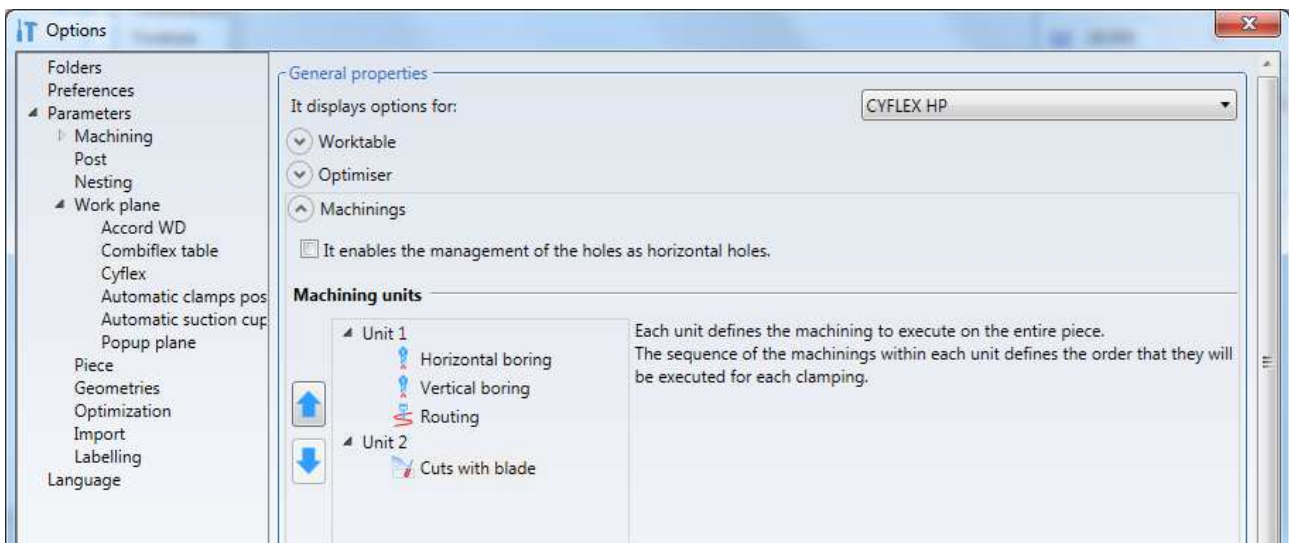
- Cluster the machining operations
- Set the clamps-machining operations distances

### 4.3.1 Machining clustering

The machining process on Cyflex machines requires one or more clampings to machine a piece completely.

For precision requirements it may be useful to cluster the same type of machining operations and clearly the machining times will increase in relation to the greater number of clampings.

The machining clustering is managed with the following dialogue window "Tools-Options-Parameters-Work table-Cyflex-Machining-Machining clusters"



The mode requires you to move up or down the four types of machining operations:

- Horizontal boring
- Vertical boring
- Routing
- Cuts with blade

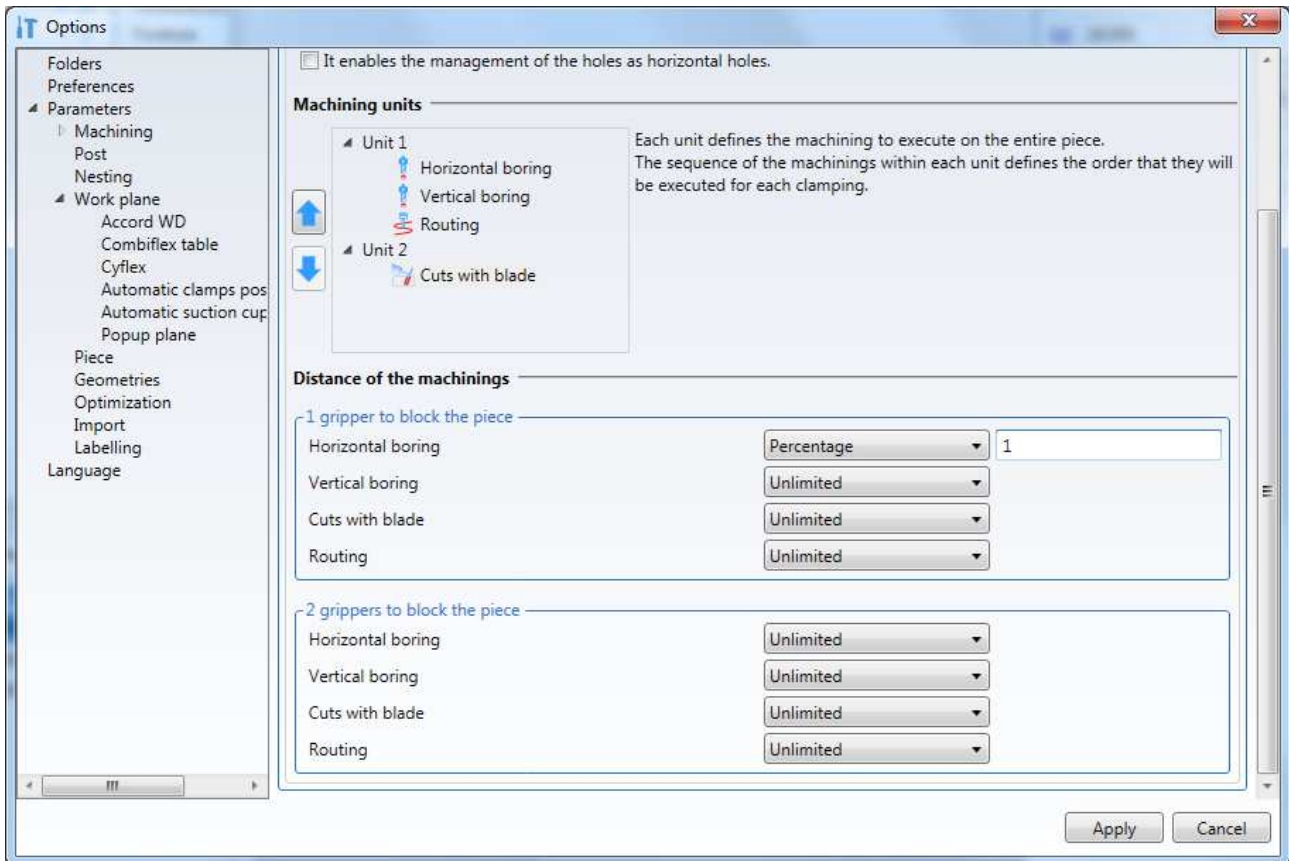
changing their order.

When a type of operation is the last in the cluster and you click on "Arrow down", it will be moved in the cluster below, that if it does not exist it would be created.

The maximum number of clusters that can be created are 4, but it is reduced to 3, when selecting "Enable the management of all borings as horizontal boring".

## 4.3.2 Clamps-machining distances setting

To set the distances between the clamps and machining use the session:  
"Tools-Options-Parameters-Work table-Cyflex-Machining-Distance from machinings"



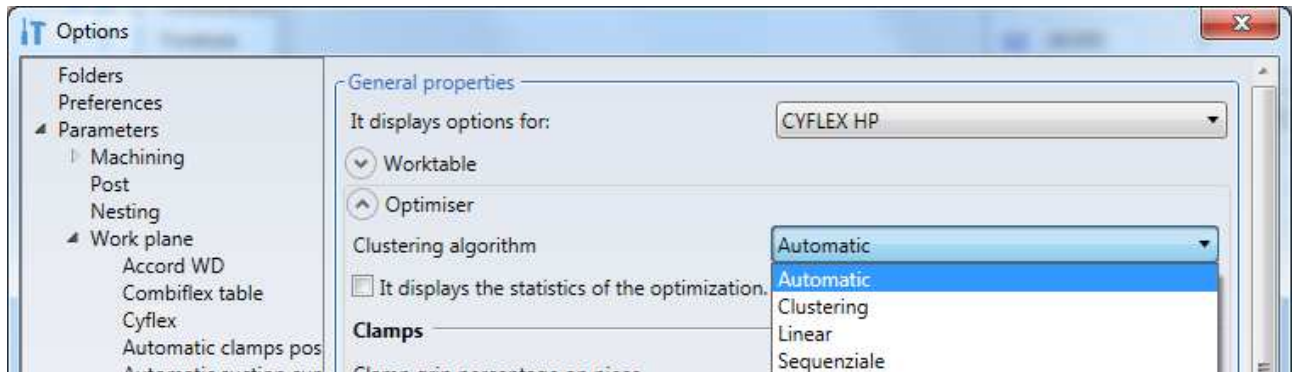
For each type of machining operation you can set a distance from the clamp:

- Unlimited
- Percentage
- Absolute

The setting can be made whether the piece is blocked with a one clamp or with two clamps (only for Cyflex HP).

### 4.3.3 Type of optimisation algorithm

The type of algorithm used to process the project can be selected in the menu:  
"Tools-Options-Parameters-Work table-Cyflex-Optimiser"



The following "Clustering algorithm" can be selected:

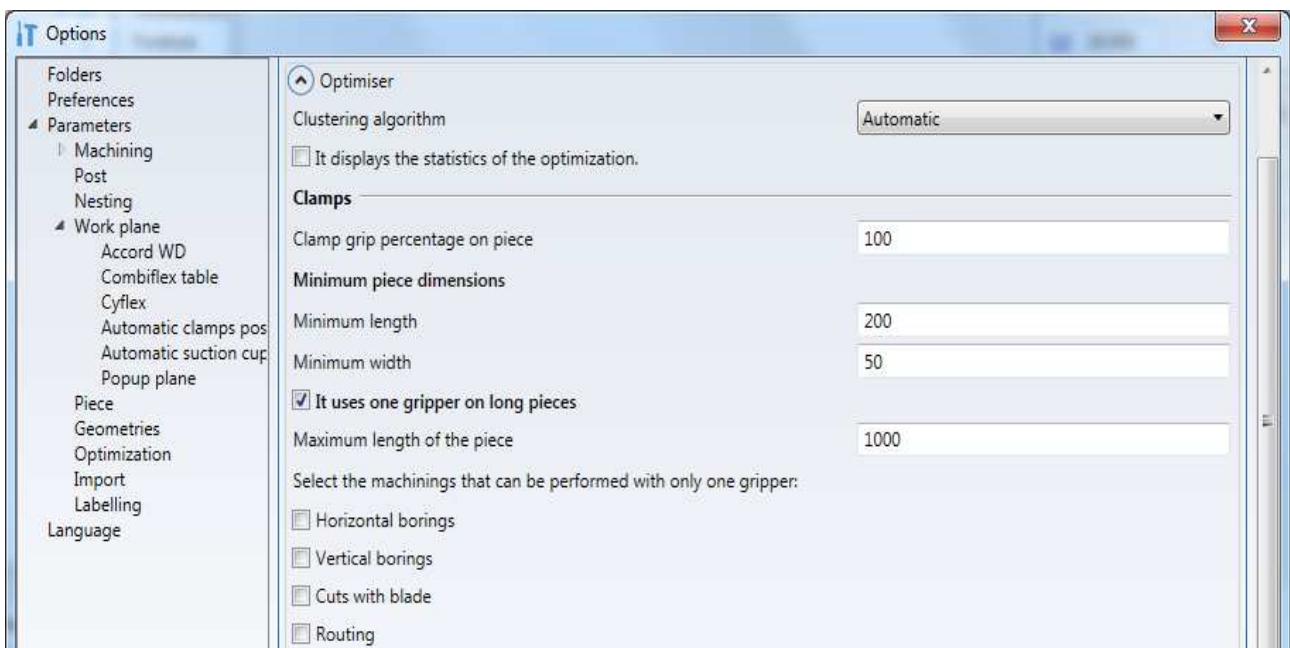
- Automatic
- Clustering
- Linear
- Sequential

To display a report of the phases generated select the "Display optimisation statistics" at the end of the optimisation.

### 4.3.4 Clamps options

With the session:

"Tools-Options-Parameters-Work table-Cyflex-Optimiser-Clamps"



to set the clamps intervention mode.

In the "Clamp grip percentage on piece" field enter a high value (max 100%) to block a piece very well or a lower value when the clamping is less significant.

The "Minimum piece dimensions" indicate the minimum piece that will be processed by the optimiser.

The Flag on "Use only one clamp on long pieces" is used only on Cyflex HP adding also the value in the "Maximum piece length" field.

The longer pieces will be blocked, if possible, with 2 clamps.

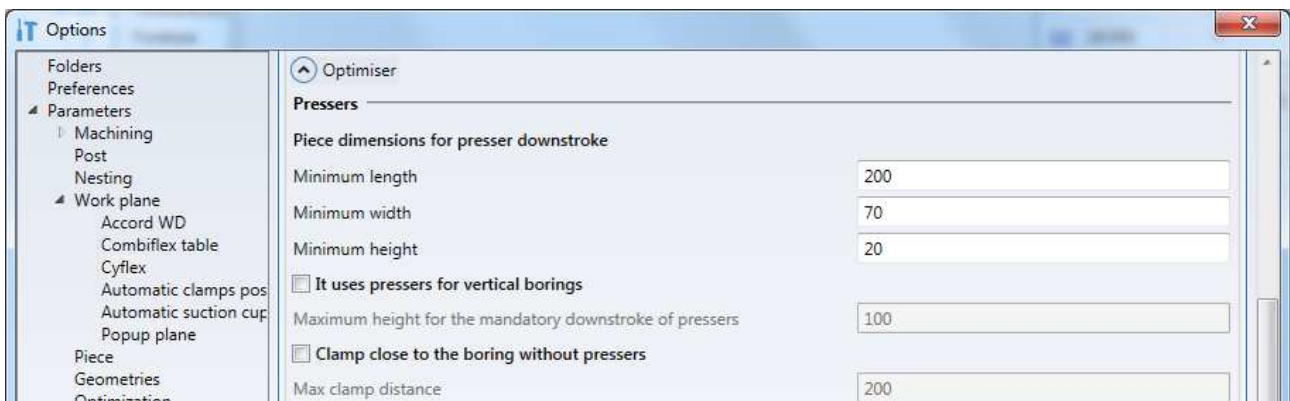
Selecting one or more types of machining operations means that during that type of machining only clamp will be used.

This is to have less interruptions during the machining.

### 4.3.5 Pressers options

With the session:

"Tools-Options-Parameters-Work table-Cyflex-Optimiser-Pressers"



to set the pressers intervention mode.

"Minimum length" and "Minimum with" are the minimum piece dimensions, below which the presser can move down also with one wheel.

"Minimum height" is the thickness below which the presser can move down in the lower part of the panel.

This is to prevent collisions between the presser and the panel support rollers.

Selecting "Use pressers for horizontal boring", with the value in the "Maximum height for pressers obligatory downstroke" field, forces the downstroke of the pressers for all horizontal boring operations that are at a lower Y quota than the one specified.

Selecting "Clamp close to boring without pressers", with the value in the "Maximum clamp distance" field, forces the presence of the clamp at a shorter or equal distance to the one set when the pressers cannot operate.

This mode is used to achieve a greater machining precision.

## 4.3.6 Probing

With the session:

"Tools-Options-Parameters-Work table-Cyflex-Optimiser-Probing"



to set the maximum difference for the panel to be optimised. Set the value in the "Probing margin" field.

## 4.3.7 Panel anti-fall

The anti-fall is made by avoiding opening the clamp or clamps that block the panel at the end of the discharge procedure. This behavior is conditioned by:

- using a Cyflex HP;
- to the left exhaust, complete and not.

The anti-fall feature can be enabled in the Maestro options or in the project's machine parameters.

The section on the Cyflex HP Options page has added the visible section in the following figure:

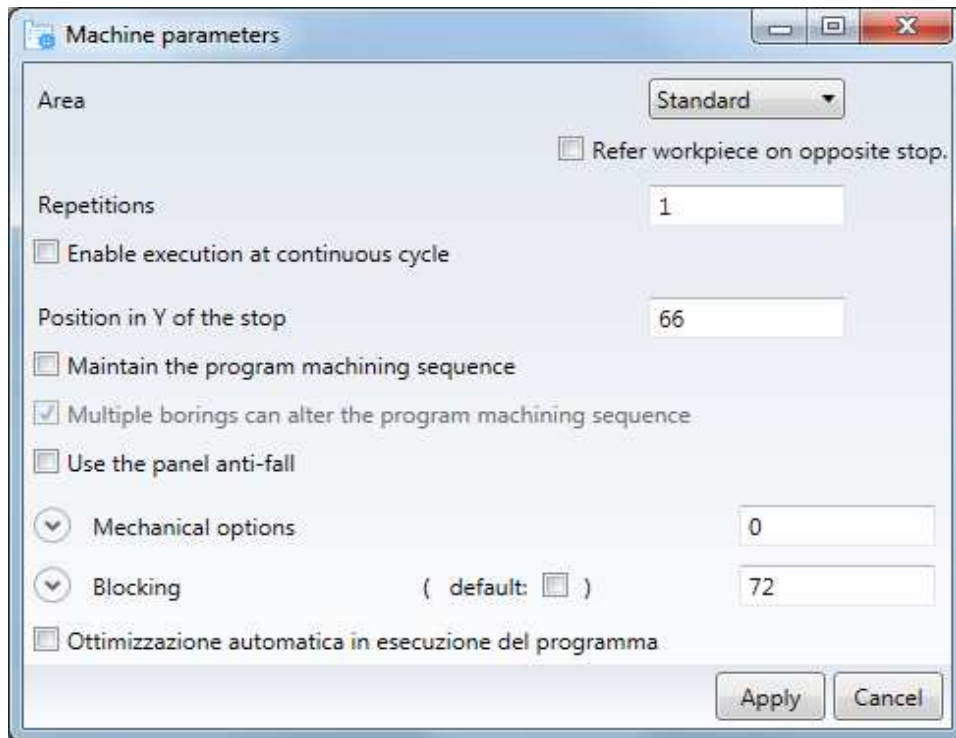


Through CheckBox, you can enable anti-fall functionality, which will become effective if at least two of the following conditions are met:

- Panel length < Minimum piece length value
- Panel width < Minimum piece width value
- Panel height > Maximum workpiece height value

Values set in Options will be active for all projects that will be optimized by Maestro.

When using a configuration for a Cyflex HP in machine parameters, the entry "Use panel anti-fall functionality" appears in the figure below:

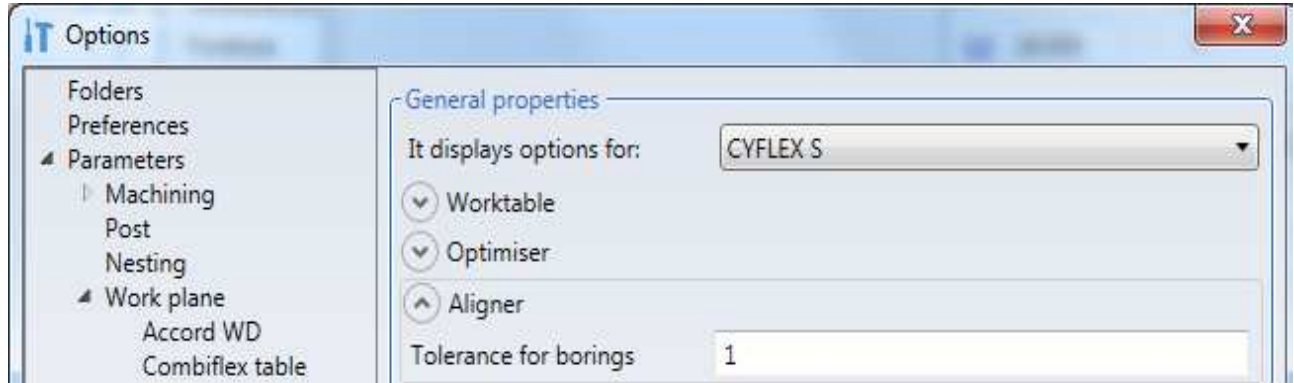


Through CheckBox, you can enable anti-fall functionality, which will only become effective for the current project.  
When activated by machine parameters, it has a higher priority and the anti-fall is applied regardless of the value in the Options.

## 4.3.8 Aligner

With the session:

"Tools-Options-Parameters-Work table-Cyflex-Optimiser-Aligner"



to set the aligner intervention mode. (Only for Cyflex S)

In the "Boring tolerance" field you can set the minimum material thickness, below which the aligner will not operate.

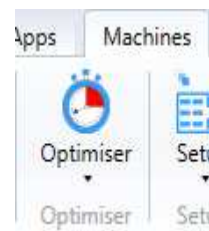
This parameter is used to prevent that a through machining operation in the high part of the panel can be damaged by the pressure of the aligner.

## 4.3.9 Processing with optimiser

This procedure provides the pgmx program with all the stages required by Cyflex to handle and work the piece.

Optimisation can be started through the "Optimisation" control from the "Machines" menu.

The result can be viewed in the "Arrange" section.





## 4.4 Layout for Cyflex programs

The "Layout" for Cyflex is used to see the piece in all the phases from loading to unloading.

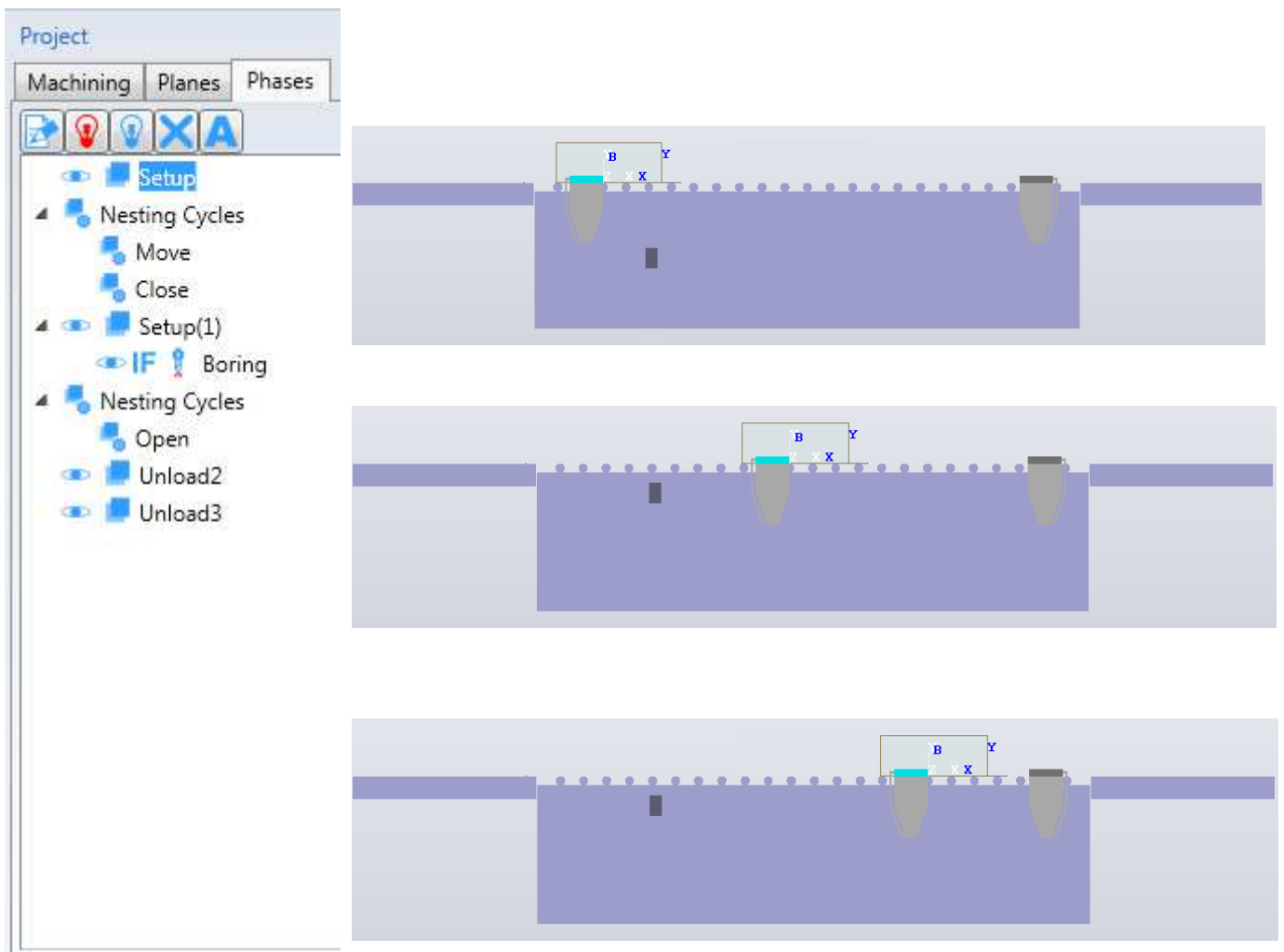
The procedure can be executed to check the work phases that have been produced.

To access the Layout environment use the "Layout" command in the "Machines" menu.



The "Layout" environment for a pgmx of a Cyflex machine appears as follows:

on the left there is the phases tree and in the centre of the screen there is the machine and the piece moved by the clamps:



Move through the phases tree to see the piece moved by the clamps in the machine.

# 5 Project Management for Uniflex

## 5.1 Create project

The Uniflex machine allows the machining of a single piece or two superimposed pieces with the same dimensions in X and Y. In the latter case, it is required to construct a project containing both pieces, suitably positioned on the plane.

To facilitate the implementation of the project, a tool was created using the "Uniflex" command in the "Home-Project" command:

Three different types of projects will be possible:

Auto Double

Double

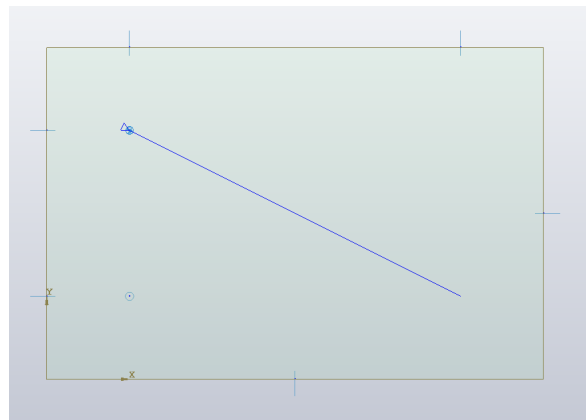
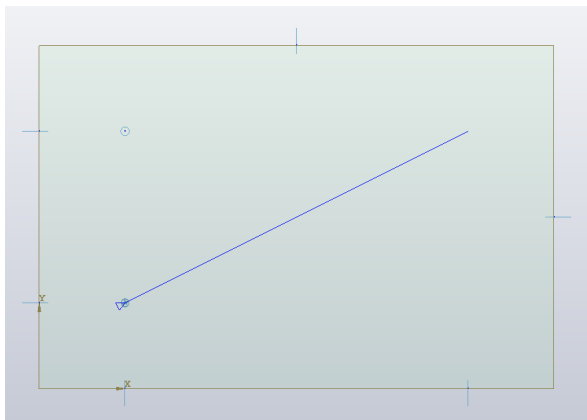
- Single



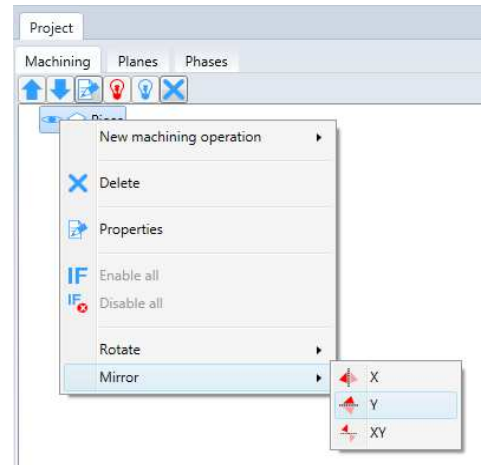
### 5.1.1 One piece mirror

In the paragraphs that will follow, when it comes to mirroring the workpiece, reference will be made to the combined operation of mirror machining and rollover of the workpiece. Both will be referenced to the Y coordinate: it is not permitted in the Uniflex to mirror or reverse over the X, due to a loss of precision, since the reference is on the side supporting the pliers.

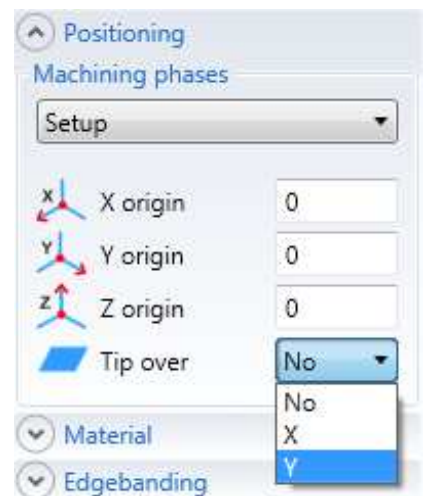
The mirror operation, as shown in the following pictures, works by overturning the workings leaving them on the same plane they belong to, except for those on the front and rear faces, which are exchanged:



This type of operation, if you want to do it manually, can be done through the "Specchia - Y" command of the workpiece context menu:



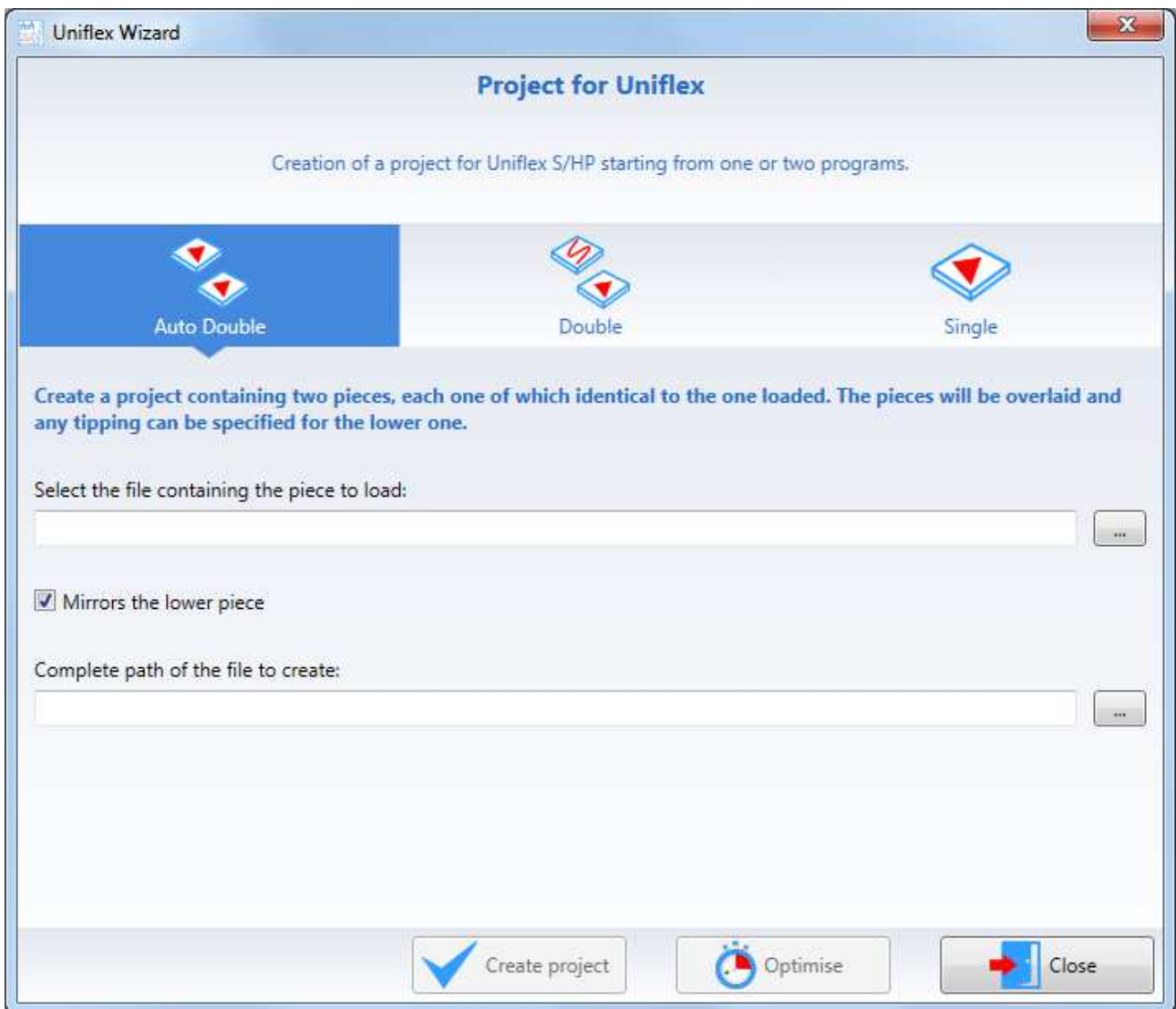
The tilting operation has no effect on the view of the workpiece in the Maestro CAD environment, but only in the Available. In practice, this is done by setting the roll over in the positioning properties of the workpiece.



By using these commands and appropriately setting the positions of the pieces in Z, you can manually construct, if you want to do so, the types of projects described below.

## 5.1.2 Creating the Double Dual Project

A Double Dual design contains two identical pieces, importing the same piece twice. The panel in the top position is oriented as by the original program, and the bottom panel can be mirrored. If it were, the machining of both pieces would be aligned, allowing them to be executed at the same time by the upper and lower heads.



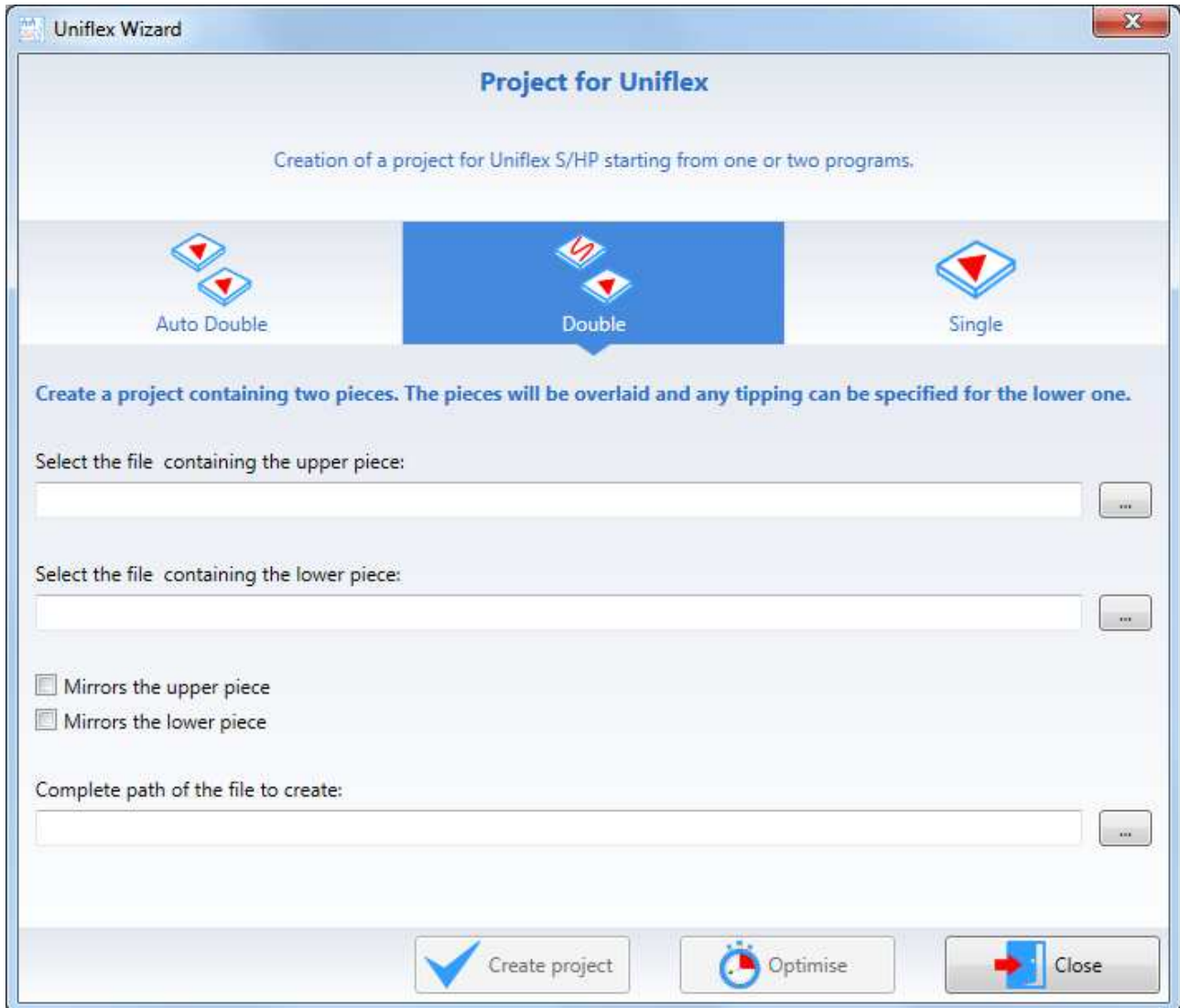
Entering the name of the project to be uploaded to its field will also automatically fill in the name of the file to be created.

Mirroring is only allowed for the lower part and is selected by default because it represents the most common operating mode.

Selecting "Create Project" will create the project with the pieces at the appropriate locations, while selecting "Optimize" will be produced directly the optimized file, with the work phases.

### 5.1.3 Creating the Double Dual Project

This type of project is useful for creating a program with two different and overlapping pieces.

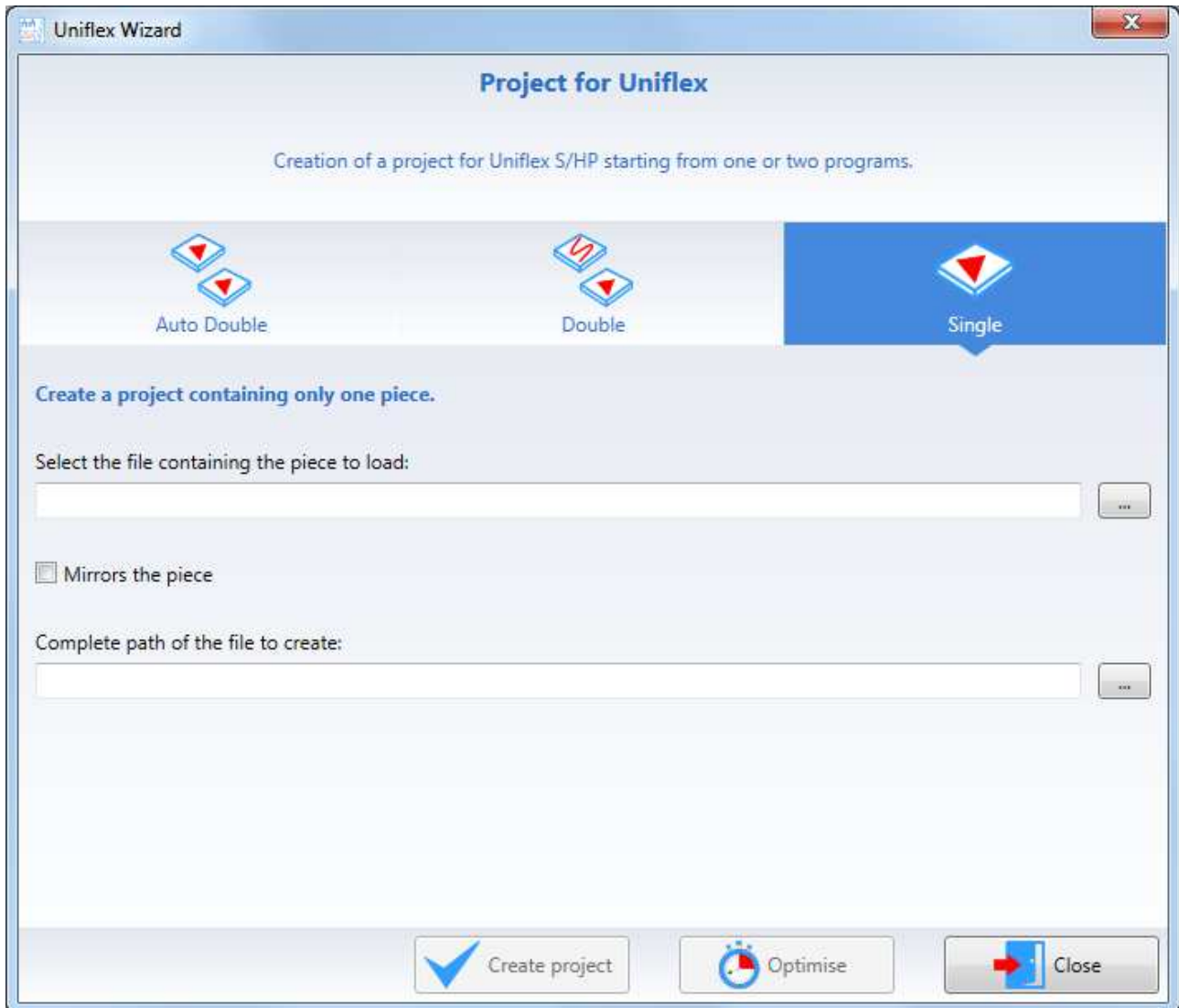


By entering the names of the projects to be uploaded in their respective fields, the name of the file to be created will also be automatically filled in.

The mirror can be applied to both pieces, to give the operator maximum freedom of use. Selecting "Create Project" will create the project with the pieces at the appropriate locations, while selecting "Optimize" will be produced directly the optimized file, with the work phases.

## 5.1.4 Creating the Single Project

The single project is the simplest one, that is, the one that contains the only piece to be optimized.



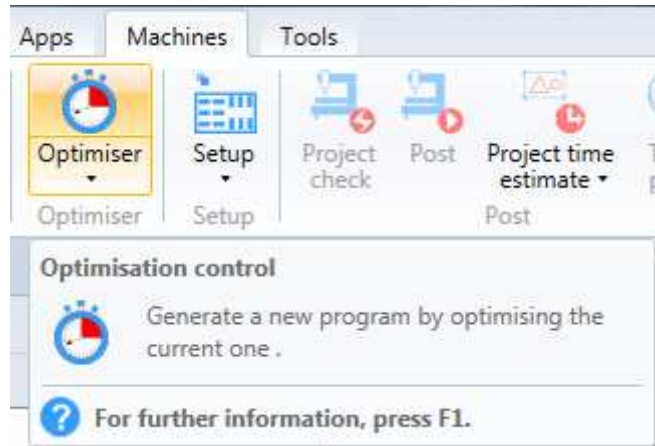
Entering the name of the project to be uploaded to its field will also automatically fill in the name of the file to be created.

It is possible to apply the mirror to the workpiece to be optimized.

Selecting "Create Project" will create the project with the piece in the right position, selecting "Optimize" will produce the optimized file directly with the work steps.

## 5.1.5 Optimization of the project by Maestro

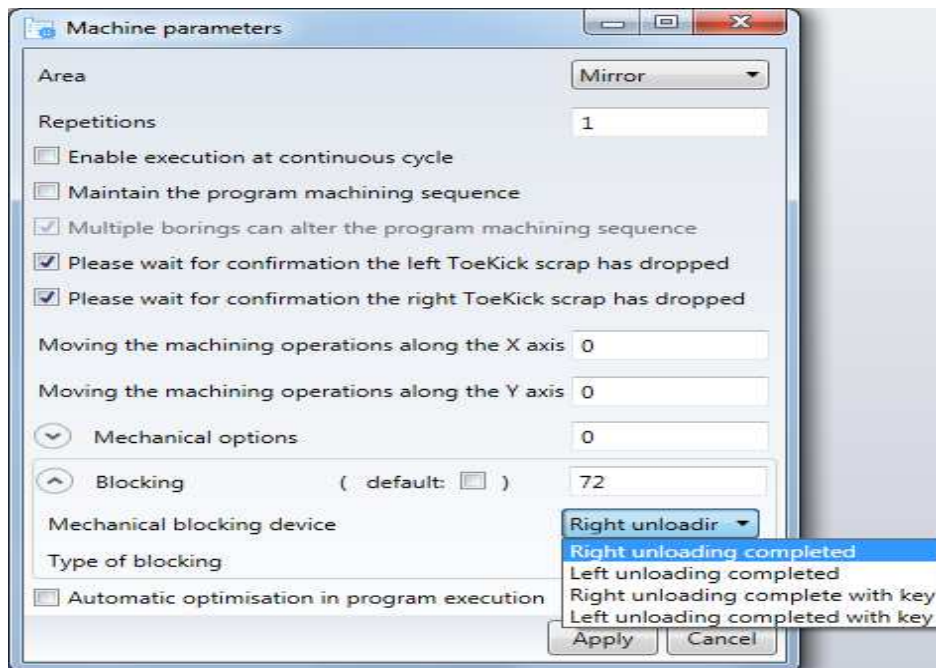
The optimization project can be created or opened directly into Master, without going through the tool just described. To optimize the current design, simply press the "Optimizer" button on the "Machines" tab of the ribbon.



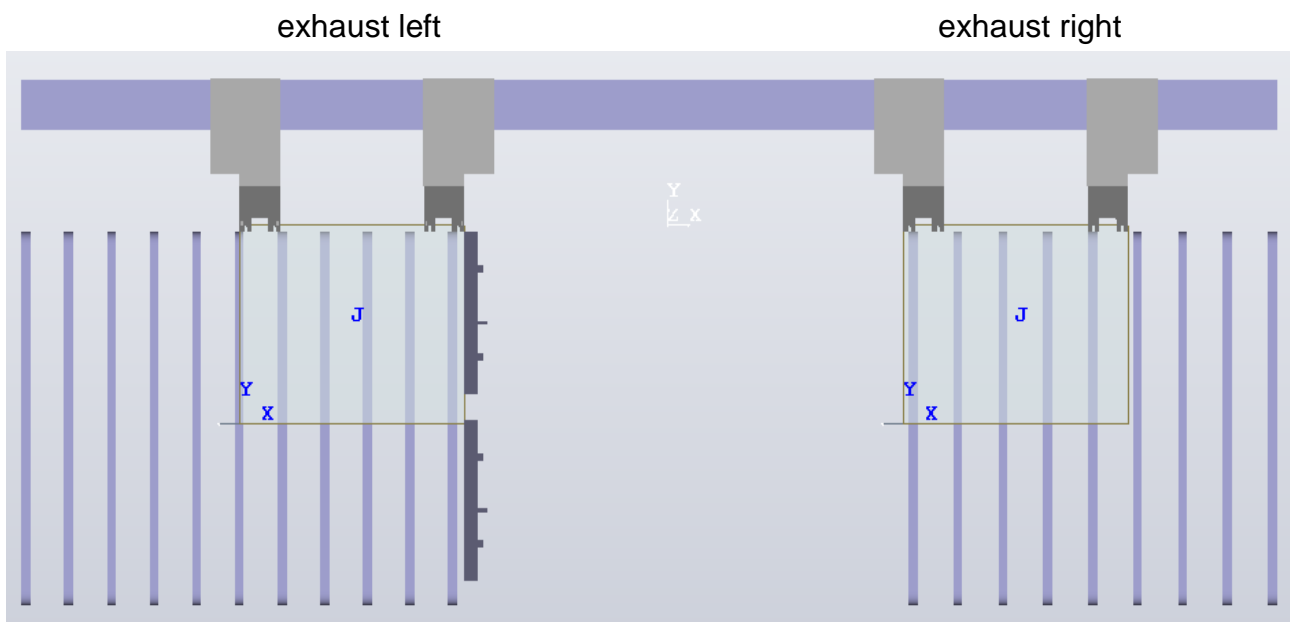
## 5.2 Types of drain

As the following image shows, there are 4 types of discharge:

1. exhaust to full right
2. exhaust to full left
3. discharge to the right with touch probe
4. full left exhaust with touch



The drain on the left carries the panel at the loading position, ie with the head over the bars. The drain on the right leads instead the tail of the panel at the junction to the right, as indicated in the Teacher options.



The "touch probe" modes enable the touch probe, which verifies the actual size of the panel.



## 5.3 ToeKick

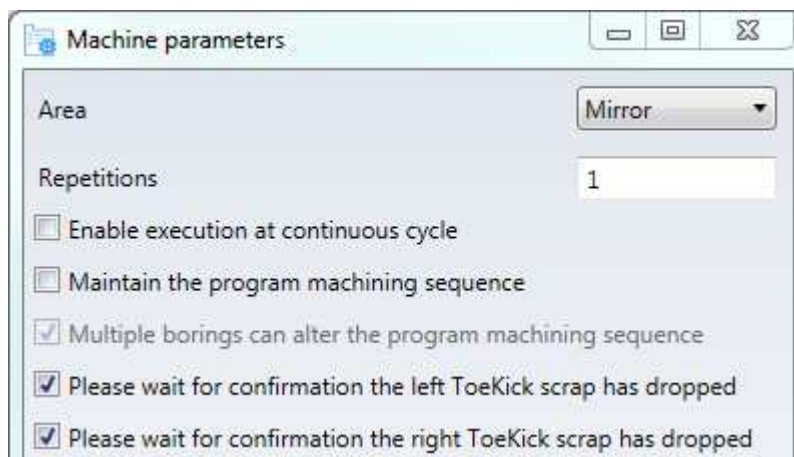
ToeKick machining can be built in two different ways.

When it does not produce shrinkage, ie when it is not passing or when it is made with jumper, it does not need special intervention: it is like any other milling work and can be done at the discretion of the optimizer at any point in the process.

When it generates fracture, as it can not fall inside the machine, damaging the same, the ToeKick is executed as the last work and the clamp is put from the optimizer to block both the piece and the sludge. In this case, it is absolutely necessary that the macro is constructed with machining in the tool center.

Obviously in this chapter you are referring to the ToeKick on the back of the piece, that is, those in contact with the pliers. Front ToeKicks can not generate friction, because the machine does not have the elements needed to drain residues outside the work plane.

The right ToeKick scrap is unloaded by bringing the workpiece to the position indicated in the Master options ("Right ToeKick Shredding Quota Discharge"), while the Left ToeKick residue is dropped into the unloading position of the workpiece. In both cases, you are expected to confirm that the sprayer has been dropped, so as not to damage the machine. These expectations can also be disabled by turning off the corresponding check in the machine parameter window:



## 5.4 Tool programming rules

To allow the correct behavior of the optimizer, you must follow some rules while building the tools.

### 5.4.1 Flat bit

If the diameter of the spindle is greater than the useful one, the diameter of the anti-collision must be equal to it and the other diameters must be the same as the one useful. The tool will be considered as a diameter equal to that useful for its entire length, then with a diameter equal to that of anti-collision.

Flat bit properties	
<b>General Data</b>	
Name	Flat bit
Description	
Code	
Quantity	1
Overall dimension (mm)	50
Tool total diameter (mm)	5
Tool anti-collision diameter (mm)	20
<b>Tool body data</b>	
Graver length (mm)	5
Diameter (mm)	5
Bit total length (mm)	50
Bit working length (mm)	40

In the case of a greater useful diameter than the spindle, all tool diameters must be equal to the tool diameter.

Flat bit properties	
<b>General Data</b>	
Name	Flat bit
Description	
Code	
Quantity	1
Overall dimension (mm)	50
Tool total diameter (mm)	35
Tool anti-collision diameter (mm)	35
<b>Tool body data</b>	
Graver length (mm)	5
Diameter (mm)	35
Bit total length (mm)	50
Bit working length (mm)	40

## 5.4.2 Launch tip

The same rules as the flat tip apply.

Conical bit properties	
<b>General Data</b>	
Name	Conical bit
Description	
Code	
Quantity	1
Overall dimension (mm)	50
Tool total diameter (mm)	5
Tool anti-collision diameter (mm)	20
<b>Tool body data</b>	
<input checked="" type="radio"/> Bit angle (degrees) <input type="radio"/> Cone Height (mm)	45
Diameter (mm)	5
Bit total length (mm)	50
Bit working length (mm)	40

## 5.4.3 Smooth flat tip

The useful diameter is that of the first part of the tip.

The maximum diameter of the counter must correspond to the total diameter of the tool.

The anti-collision diameter is equal to the spindle diameter and is used beyond the total length.

Countersink flat bit properties	
<b>General Data</b>	
Name	Countersink flat bit
Description	
Code	
Quantity	1
Overall dimension (mm)	50
Tool total diameter (mm)	15
Tool anti-collision diameter (mm)	20
<b>Tool body data</b>	
Graver length (mm)	5
Diameter (mm)	5
Bit total length (mm)	50
Bit working length (mm)	40
Countersink maximum diameter (mm)	15

If the counterfoil diameter is larger than that of the spindle, the anti-collision diameter must be equal to ace.

Countersink flat bit properties

**General Data**

Name	Countersink flat bit
Description	
Code	
Quantity	1
Overall dimension (mm)	50
Tool total diameter (mm)	30
Tool anti-collision diameter (mm)	30

**Tool body data**

Graver length (mm)	5
Diameter (mm)	5
Bit total length (mm)	50
Bit working length (mm)	40
Countersink maximum diameter (mm)	30

## 5.4.4 Countersunk spear tip

The same rules as the flat tip apply.

Countersink conical bit properties

**General Data**

Name	Countersink conical bit
Description	
Code	
Quantity	1
Overall dimension (mm)	50
Tool total diameter (mm)	15
Tool anti-collision diameter (mm)	20

**Tool body data**

Bit angle (degrees)   
  Cone Height (mm)

Bit angle (degrees) / Cone Height (mm)	45
Diameter (mm)	5
Bit total length (mm)	50
Bit working length (mm)	40
Countersink maximum diameter (mm)	15

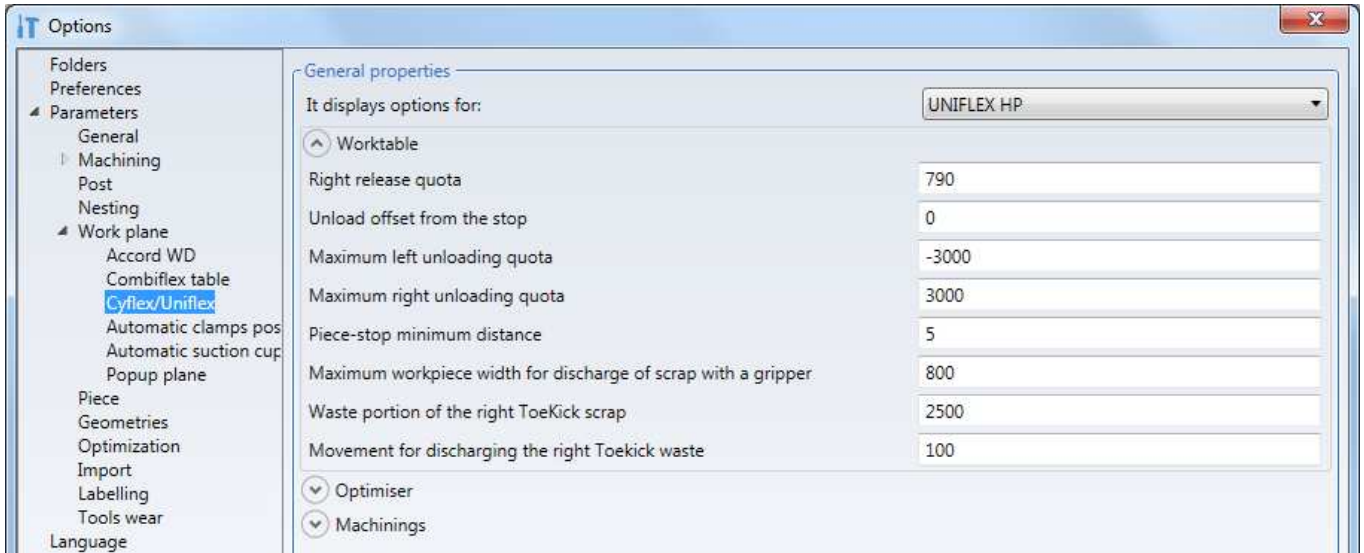
## 5.4.5 Blitz Tip

The same rules apply as for the countersunk tips, except that the "Blitz Maximum Countersink Diameter" is used instead of the "Maximum Countersink Diameter".

Property of Bliz flat drill	
<b>General Data</b>	
Name	Bliz flat drill
Description	
Code	
Quantity	1
Overall dimension (mm)	50
Tool total diameter (mm)	15
Tool anti-collision diameter (mm)	20
<b>Tool body data</b>	
Scoring length (mm)	0
Diameter (mm)	5
Bit total length (mm)	50
Bit working length (mm)	40
Bliz maximum diameter (mm)	15

## 5.5 Uniflex Options

This chapter contains information for the correct installation of machine parts that make up the Uniflex. For other options, please refer to the Cyflex section of the Maestro manual.



### Piece-stop minimum distance

The stops against which the right side of the workpiece rests during loading, lowered to bring the panel into the work area, can be raised while the programme is running, provided that the workpiece does not collide with them. This option is used to have a tolerance for comparing the size of the workpiece with that of the stops.

### Maximum workpiece width for discharge of scrap with a gripper

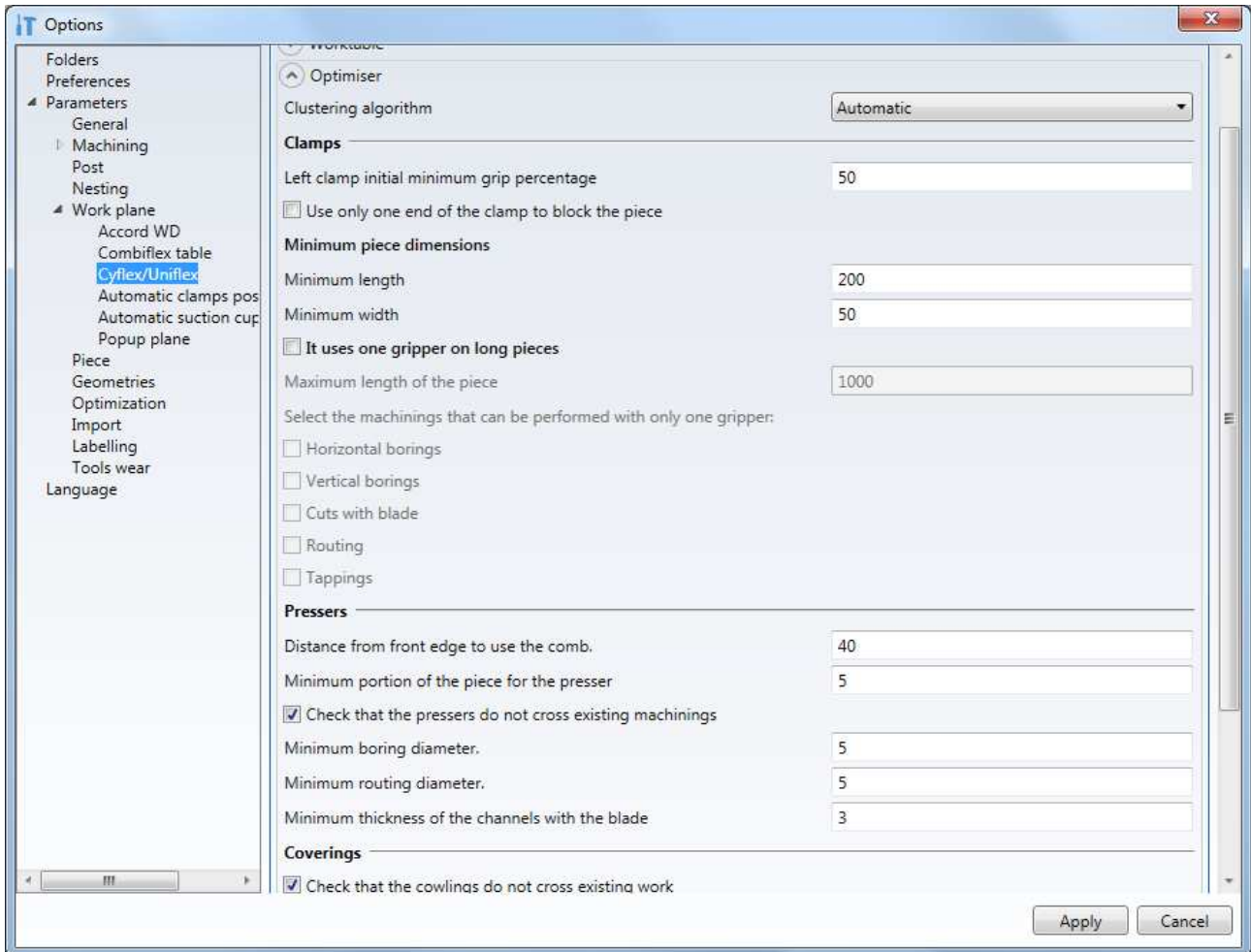
The unloading of the panel can be done by bringing the workpiece to the unloading position with both grippers, or with a single gripper, the one closest to the head, which pushes the workpiece out of the work area (this choice is made, when necessary, to avoid a re-punching). However, the latter technique cannot be used when the workpiece is wider than a certain threshold, otherwise it will rotate during unloading. Authorization is used to specify access rights.

### Waste portion of the right ToeKick scrap

It represents the X coordinate to which the workpiece head is carried to unload the residual ToeKick machining residue, when it is made without jumpers.

### Movement for discharging the right Toekick waste

It's the movement made to drop the residue of ToeKick's machining, when it is made without jumpers.



### **Left clamp initial minimum grip percentage**

Indicates the portion (percentage) of the length of the left clamp that should lock the workpiece during loading. This parameter must not be reduced beyond default, otherwise the clamp will collide with the workpiece alignment devices.

### **Use only one end of the clamp to block the piece**

Given the shaped shape of the clamp with two ends that can hold the workpiece, this option indicates whether only one of the two ends can be used for clamping. Obviously, if the workpiece is held by a single clamp, this option is automatically excluded, regardless of its value.

### **Distance from front edge to use the comb.**

Indicates the distance of spindle 1 from the front edge of the workpiece to force the use of the comb for clamping the workpiece during machining.

### **Minimum portion of the piece for the presser**

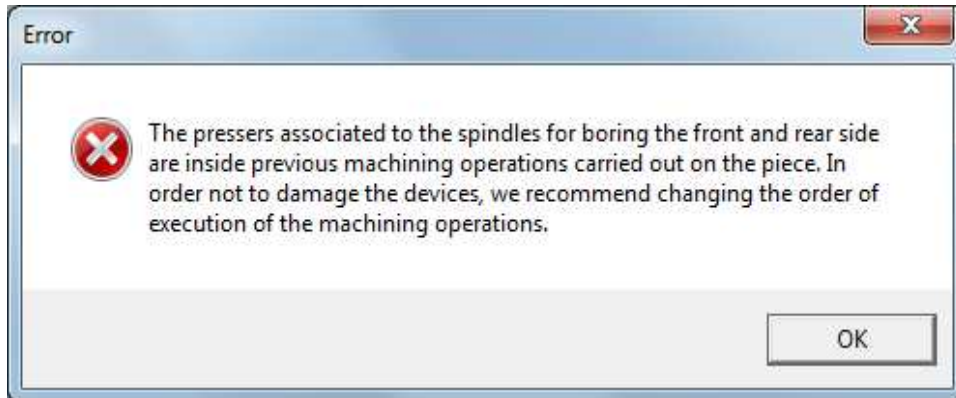
When the edge of one of the pressers is less than this threshold from the edge of the workpiece, the presser is not lowered. The expertise may be carried out in-house or externally.

### **Check that the pressers do not cross existing machinings**

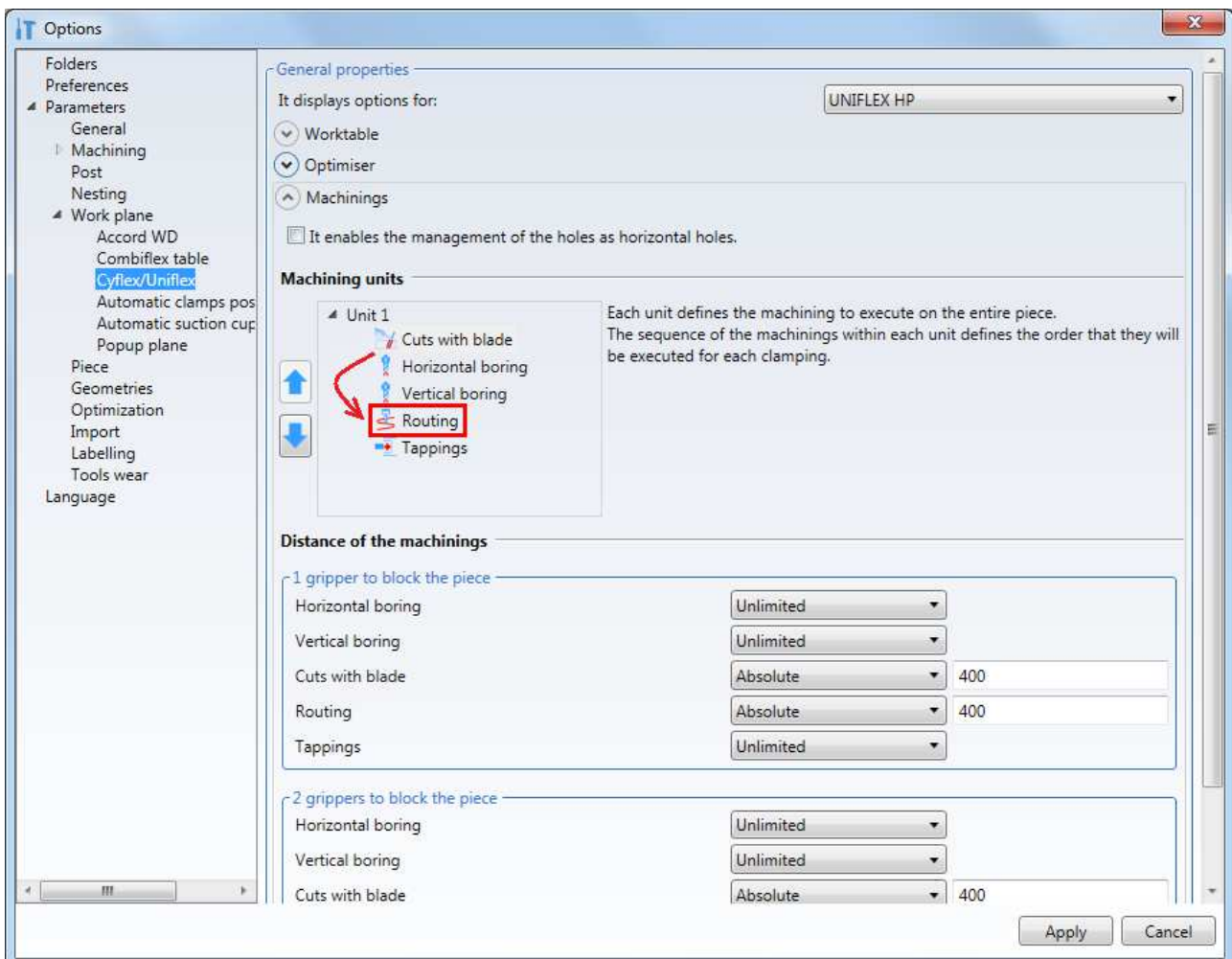
Check that the pressers associated with front and rear holes do not cross existing machinings and break.

This inspection shall only cover work carried out with tools of a diameter or thickness greater than those indicated.

When a presser is found to fall into a machining operation, the workpiece optimization is interrupted and the following error is displayed:



To solve the problem, as indicated by the same error message, it is sufficient to modify the order of execution of the machining operations, in order to perform the horizontal drilling before the incriminated machining operations (for example, in the following image, the milling damaged the presser). This order is changed in the Maestro options:





# 6 Project management for PowerFlex

This chapter describes Maestro's capabilities for optimizing part design for PowerFlex machines.

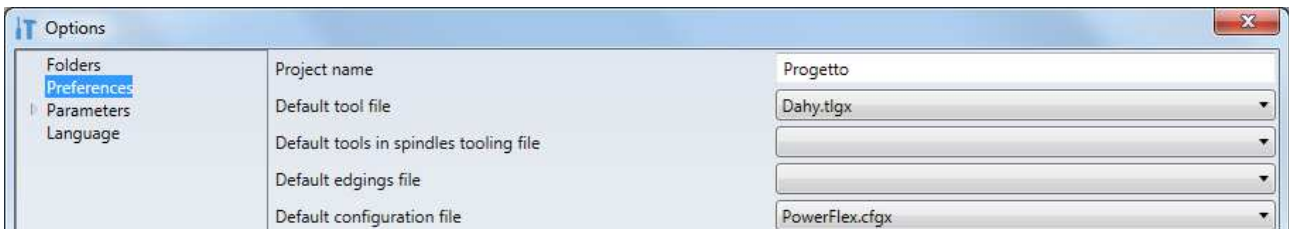
## 6.1 Master Settings

In order to successfully create and optimize a PowerFlex program, you must on your PC, you have installed the Albatros software environment and that Maestro is configured to work with PowerFlex machines.

In the Master Options in the Folders section, you must select the path where the Albatros installation resides.

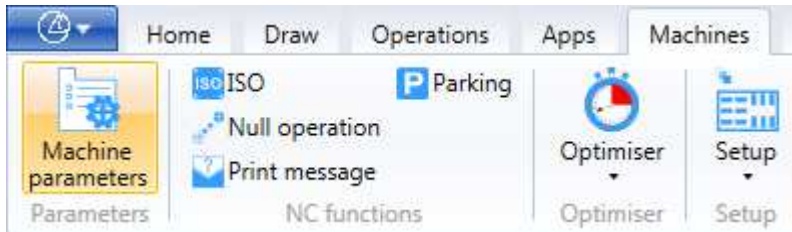


In the Master Options in the Preferences section, you must select your PowerFlex machine configuration (. cfgx file). This selection is essential for the correct operation of the machine program (interaction with PowerInterface).

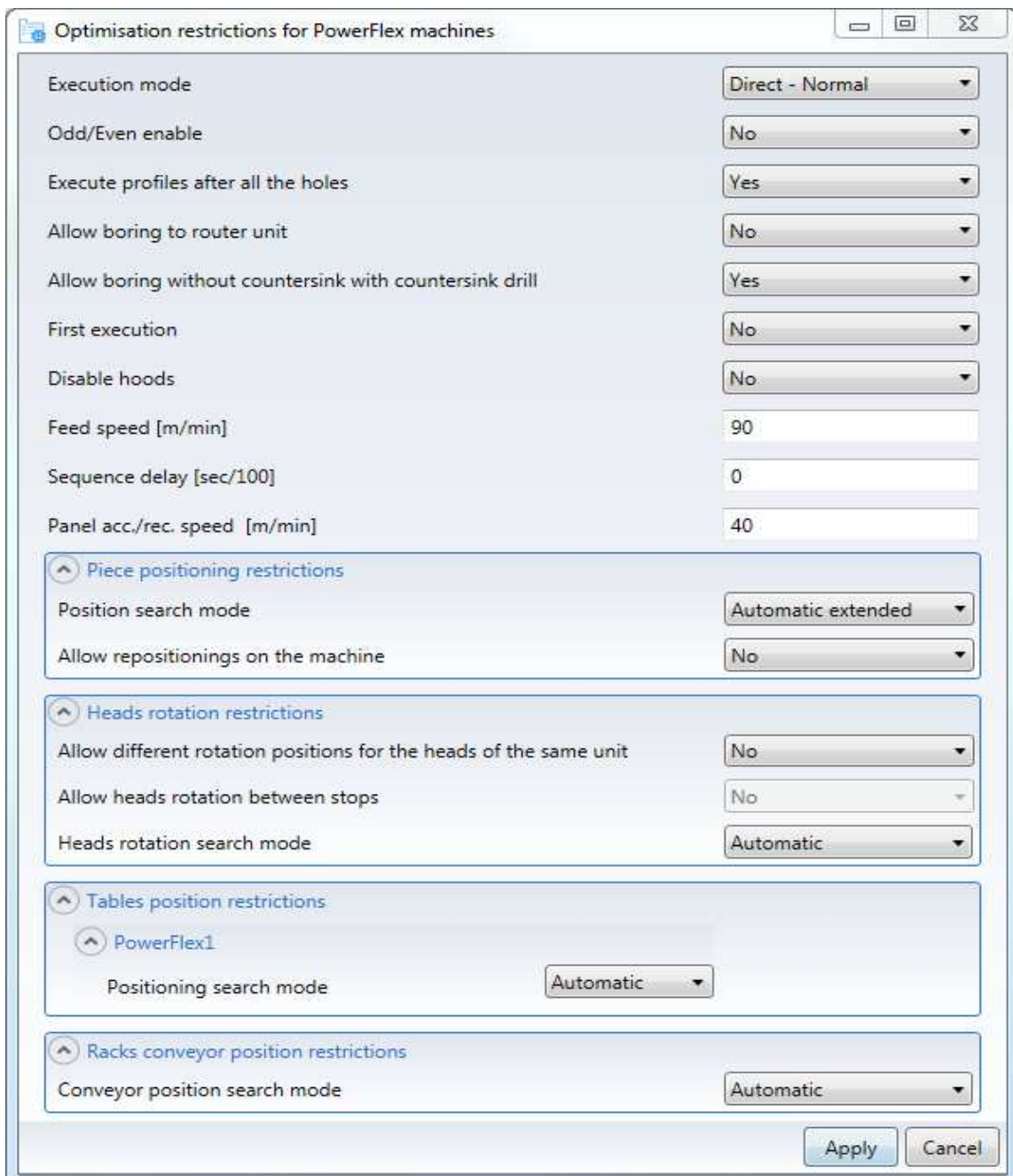


## 6.2 Optimization constraints

Before optimizing a program, it is necessary to set the program-specific optimization constraints. To access the commands enter the “Draw” area of the application menu and press “Surfaces”.



Having selected a PowerFlex machine configuration, the parameters shown to the user will be as follows:

A screenshot of the 'Optimisation restrictions for PowerFlex machines' dialog box. The dialog has a title bar with standard window controls. The main area contains several sections of settings, each with a blue expandable header. The settings are as follows:

- Execution mode:** Direct - Normal (dropdown)
- Odd/Even enable:** No (dropdown)
- Execute profiles after all the holes:** Yes (dropdown)
- Allow boring to router unit:** No (dropdown)
- Allow boring without countersink with countersink drill:** Yes (dropdown)
- First execution:** No (dropdown)
- Disable hoods:** No (dropdown)
- Feed speed [m/min]:** 90 (text input)
- Sequence delay [sec/100]:** 0 (text input)
- Panel acc./rec. speed [m/min]:** 40 (text input)
- Piece positioning restrictions:**
  - Position search mode:** Automatic extended (dropdown)
  - Allow repositionings on the machine:** No (dropdown)
- Heads rotation restrictions:**
  - Allow different rotation positions for the heads of the same unit:** No (dropdown)
  - Allow heads rotation between stops:** No (dropdown)
  - Heads rotation search mode:** Automatic (dropdown)
- Tables position restrictions:**
  - PowerFlex1:**
    - Positioning search mode:** Automatic (dropdown)
- Racks conveyor position restrictions:**
  - Conveyor position search mode:** Automatic (dropdown)

At the bottom right, there are 'Apply' and 'Cancel' buttons.

## 6.2.1 Generic Constraints

The generic constraints are described below.

### ***Execution mode***

Allows you to specify different optimization modes for the execution of the workpiece on the machine. You can choose from the following options:

- Direct - Normal: the project is optimized without undergoing mirroring transformations. The optimizer produces a single sequence of execution steps. All panels associated with the project will be processed in this sequence.
- Direct - MirrorX: the project is optimized after being mirrored in X. . The optimizer produces a single sequence of execution steps. All panels associated with the project will be processed in this sequence.
- Multiple - MirrorX: the project is optimized after being mirrored in X. The optimizer produces two sequences of execution steps, one for the executing the unmirrored project and one for the mirrored project. The panels associated with the project will be machined by alternating the use of the sequence obtained for the unmirrored project, then the one obtained for the mirrored project.
- Multiple – Normal:the project is optimised without mirroring transformation. The optimiser produces two sequences of execution steps, one for the outward panel and one for the return path panel. The final outward execution status will be the initial state of the return execution. The panels associated with the project will be machined by alternating the direct sequence, then the return sequence.
- Multiple - MirrorX: the project is optimized after being mirrored in X. The optimizer produces two sequences of execution steps, one for the execution of the outward and one for the execution of the panel in return path. The final outward execution status will be the initial state of the return execution. The panels associated with the project will be machined by alternating the direct sequence, then the return sequence.

For multiple execution modes, it is also possible to specify by flag whether the return path generated by the optimizer should be a simple backward execution of the outward path or may differ from it. In the first case, during production, it will be necessary to check the quality of only one panel, because in both paths the machining operations will be carried out by exactly the same tools. In the second case, it will be necessary to check the quality of two panels, because the optimizer could choose a return path that does not use the same tools, but which is more advantageous in terms of execution times.

### ***Odd/Even enable***

Significant functionality only in the case of a line composed of several machines. In the case of enabling, each machine in the line must be able to work independently the panel in its entirety and the optimizer must find a complete solution for each machine in the line. In the case of disabled functions, the optimizer can use all the machines in the line for the complete execution of a panel, dividing the machining operations between the machines.

### ***Execute profiles after all the holes***

Through this parameter it is possible to constrain the optimizer to perform all the machining operations that allow a profile (milled/lamated) following the execution of all drilling operations.

### ***Allow boring to router unit***

This parameter can be used to constrain the optimizer to use or not the milling unit to drill holes.

### ***Allow boring without countersink with countersink drill***

Through this parameter it is possible to constrain the optimizer to use or not the countersink drills to make holes that do not allow fading (using the tip part without flaring).

### ***Towing speed***

This parameter allows you to set the towing speed (m/min) to be used for the panel. The towing speed is not related to the line speed, but only affects the transport inside the machine.

### ***Sequence delay for panel clamping and unclamping***

Parameter that allows you to set the sequence delay (sec/100) to be used for panel locking and unlocking.

### ***Acc. /rip speed. pane***

Parameter that allows you to set the speed of approach and repositioning of the panel (m/min).

## **6.2.2 Piece positioning restrictions**

Through this section it is possible to condition the optimizer's choices for the search of the workpiece stop positions in the machine. The parameters of interest are described below.

### ***Position search mode***

Allows you to specify the search criteria for the stop positions of the panel in the machine, choosing between:

- Extended automatic search: the optimizer determines the panel stop positions independently, performing an accurate search for all possible stop positions.
- Limited automatic search: the optimizer determines panel stop positions independently, making a quick search for all possible stop positions in a limited search range.
- Manual search: the optimizer determines panel stop positions in an assisted manner, using the search range indicated by the user. For each machine, the user must specify the minimum and maximum stop position that the machine panel can take on. In case of coincident values, the optimizer will position the panel at the value indicated, allowing only one positioning of the panel in the machine.

### ***Allow repositionings on the machine***

This parameter can be used to constrain the optimizer to search for solutions that allow repositioning of the workpiece in the machine or not.

### 6.2.3 Heads rotation restrictions

Through this section it is possible to condition the optimization choices for the search of the rotation status of the heads present in the machine.

#### ***Allow different rotation positions for the heads of the same unit***

If the parameter is enabled, the optimizer can assign a different rotation status to the heads of the same group. For example, considering a group of two heads, in the case of parameter enabled, the optimizer can choose whether to rotate one of the two heads or both or none of them, while in the case of a disabled the optimizer can only choose to rotate both heads or not rotate them both.

#### ***Allow heads rotation between successive stops***

If the parameter is enabled, the optimizer can decide to change the rotation status of the heads between one machining step and the next, within the machining of the same panel. If not enabled, the optimizer assumes that the rotation states of the heads in the first step are such for the entire panel machining time.

#### ***Heads rotation search mode***

Allows you to specify the search criteria for head rotations, choosing between:

- Automatic search: The optimizer determines the rotation of the heads autonomously.
- Manual search: The optimizer uses user set head rotations. For each machine, the user must specify the rotation status assumed by the heads of each group.

### 6.2.4 Tables position restrictions

Through this section it is possible to condition the optimizer's choices for the search of the workpiece stop positions in the machine.

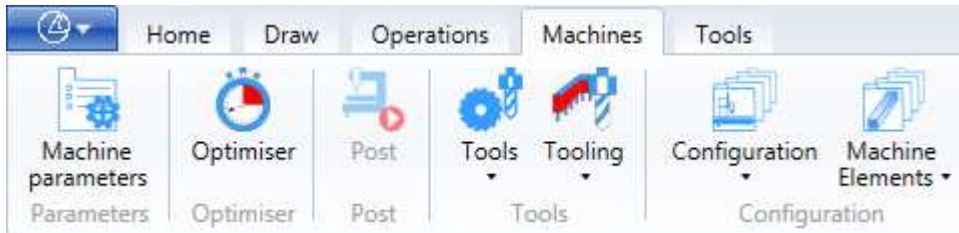
#### ***Positioning search mode***

For each machine, you can choose the positioning search mode by choosing between the following:

- Automatic search: The optimizer determines the positions of the planes autonomously.
- Manual search: The optimizer uses the positions of the planes set by the user. For each machine, the user must specify the minimum and maximum stop position that the machine panel can take on. However, the optimizer will check the feasibility of the workpiece and analyse possible collisions.

## 6.3 Optimizing execution

In order to optimize a PowerFlex program, a PowerFlex machine configuration must be activated in the Machines section of the interface, a valid tool database and tooling must be selected. Before performing optimization, it is good practice to set the specific program optimization constraints (terms described in the previous chapter). Optimizer execution is carried out by pressing the "Optimizer" button in the Machines tab.



## 6.4 Optimization results

Once the optimization has been completed, the user is informed of the result of the optimization operation.

In the case of a program that cannot be optimized, the user is informed of the error found.

In the case of an optimisable program, a short report is sent to the user with an indication of the number of positions in the machine needed to execute the panel, the number of steps (higher) and the time taken in optimization phase.

Subsequently, the user is led to choose whether to use the data found during the optimization process as new optimization constraints for the source program. This operation allows you to modify the source program in order to obtain a "pre-optimized" program whose optimization in the machine is faster.

Depending on the type of execution chosen in the optimization constraints, the successful optimization results in the generation of one or more optimized programs. For modes that allow only one execution sequence, only one optimized program is generated, while for modes that allow multiple execution sequences, e. g. multiple modes, an optimized program is generated for each obtained sequence.

It is important to note that optimized programs are not machine-executable programs. Their task is only to provide the user with a tool to evaluate the execution sequences obtained by the optimizer before running the source program on the machine.

## 6.5 PowerSet Tooling Optimizer

The PowerSet Tool Optimizer is a Master CAD/CAM tool which allows users to create, evaluate or integrate PowerFlex machine tooling.

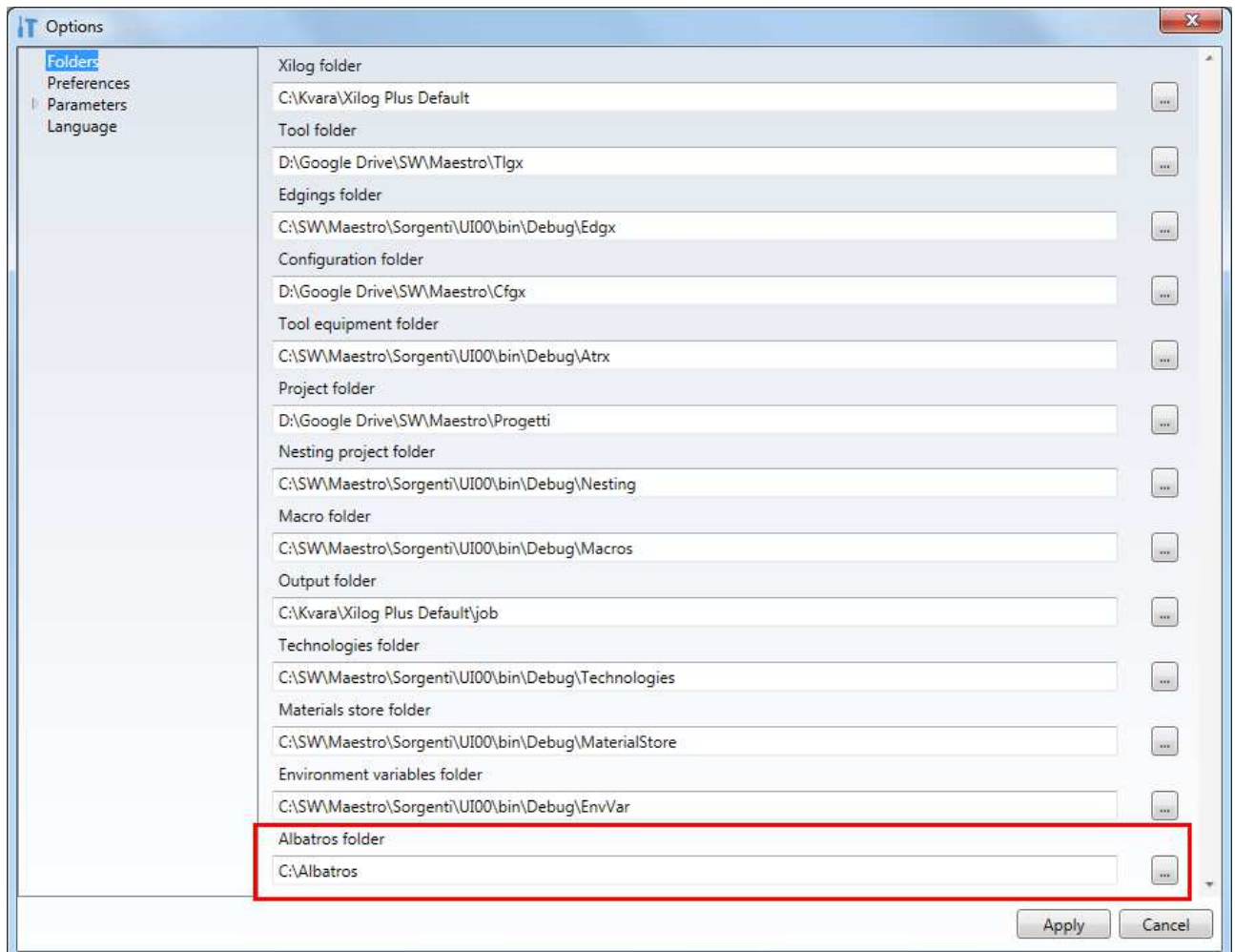
### 6.5.1 Operating requirements

To be able to use the instrument, you need to have:

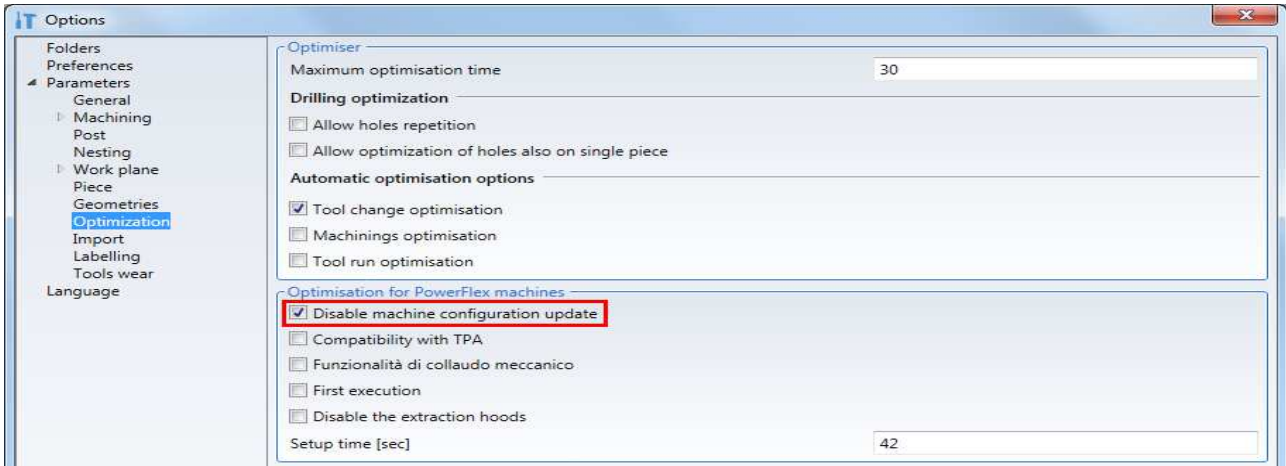
- the installation of the Albatros environment, with the machine/equipment parameters of interest;
- the configuration of machinery/equipment of interest (. cfgx file);
- tool database (. tlgx file);
- the mix of programs of interest (mixx file and related. pgmx files);
- a hardware key enabled to run the tooling optimizer;

In addition, if the integration or verification of a tooling is required, it is necessary to have the sample tooling (. atrx file).

To allow Maestro to interact properly with Albatros, ensure that the pathway where the Albatros installation resides is correct in the Maestro options.



As a rule, any changes in the Albatros parameter are saved at each optimization and the machine configuration files are updated accordingly. This ensures that optimization is always done with recent machine data. If you want to disable this sequence (recommended choice for debugging), disable the machine configuration update in Tools→Options→Parameters→Optimization.



## 6.5.2 Optimization Wizard

In order to display the PowerSet tool optimizer button, you must have the selection of a PowerFlex machine configuration enabled. To access the tool, click on the PowerSet button in the Tools tab.

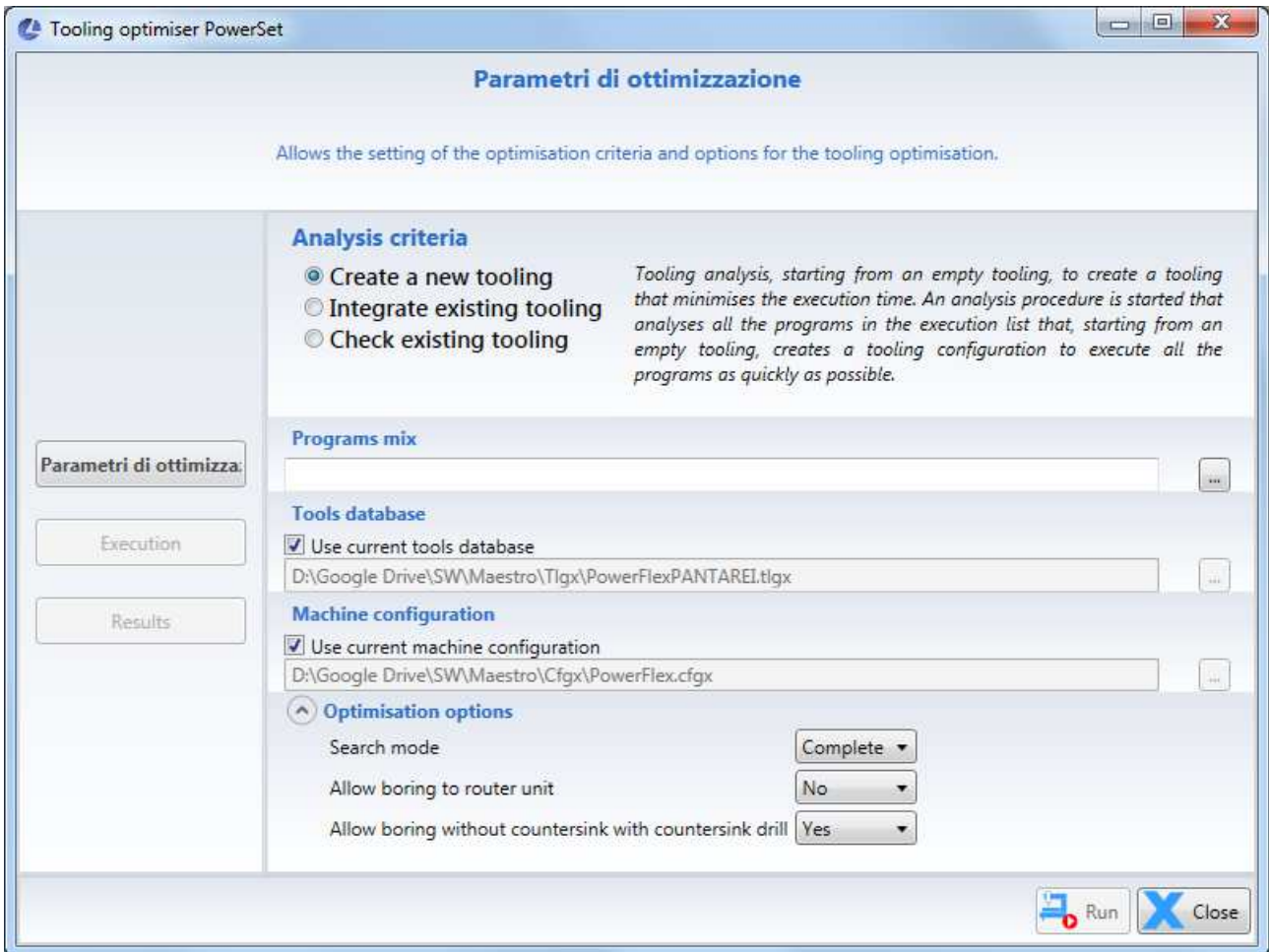


This opens a wizard consisting of three pages: parameters, execution, results. Each page is representative of a step in the optimizer execution flow and guides the user through the correct sequence of steps.



## 6.5.2.1 Page1: Parameters

On this page, the user must set the parameters of interest for the optimizer execution.



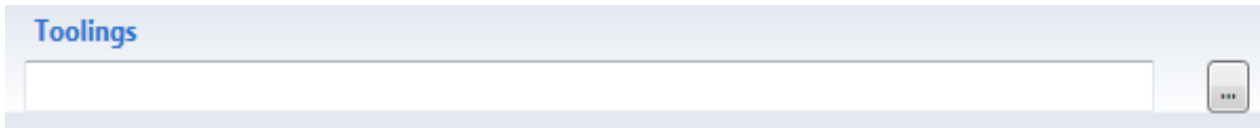
### Analysis criteria

PowerSet provides three different types of analysis:

- **Create new tooling:** An analysis procedure is started that analyses all the programs in the execution list which, starting from empty tooling, creates a tooling configuration to execute all the programs as quickly as possible.
- **Integrates existing tooling:** an analysis procedure of all the programs in the operating list which, starting from a loaded tooling, integrates the current composition, without replacing the drills already present, to minimise the programme execution time
- **Checking existing tooling:** an analysis procedure of all the programs in the operating list which, using the tooling loaded, produces a report with the execution times and the number of stops obtained for the programs. The loaded tooling is not changed.

## Toolings

This section appears only if one of the analysis criteria that requires starting equipment is selected. In this case, the user must choose the reference. atrx file for analysis.



## Programs mix

In this section, the user must choose the reference. mixx file for analysis. This file will contain the list of execution programs to be considered for optimization. Remember that for the correct functioning of the tooling optimizer, the file. mixx created as a folder import must be at the same level as the import folder. This is necessary to allow the optimizer to access the source programs with which it was created.

## Tools database

In this section, the user must choose the reference. mixx file for analysis. This file will contain the list of tools available for tooling.

## Machine configuration

In this section, the user must choose the reference. mixx file for analysis. This file will contain the representation of the machine/plant to be used for analysis.

## Optimisation options

This section describes the common tooling optimiser settings for the programs in the list. Research mode: the user can choose between a full run mode, and a fast one. The full mode provides accurate, but time-consuming analysis. Fast mode performs less accurate analysis, with shorter run times

Allows drilling in milling unit: you can specify whether or not to use the milling unit for drilling.

Allows users to make holes without cuts with flaring tool: you can use flaring tips to make holes without cuts, using the part of the tip before the countersink.

These options command any settings in the individual list programs. In addition, the tool optimizer takes into account individual programs in the list:

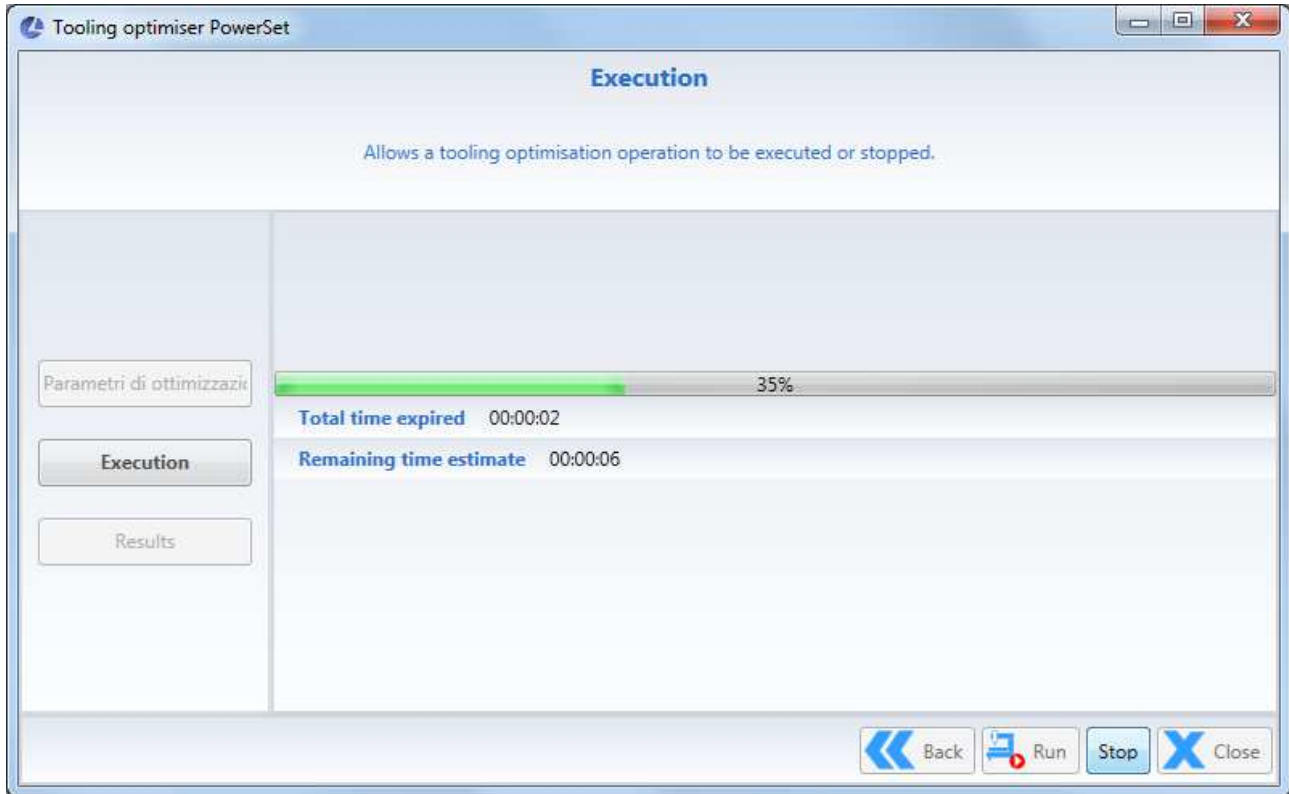
- always automatically search for panel positioning in the machine;
- panel re-positioning in the machine is always possible;
- always automatically search for the head rotation statuses;
- always automatically search for plane positions.

Constraints set on the individual program machining operations are respected by the tool optimizer.

Once the parameters have been compiled, the user can start the optimization run by pressing the Run button. The second page of the wizard is displayed as follows

### 6.5.2.2 Page2: Execution

On this page, the user can view the progress status of the execution through a progress bar with completion percentage. The user is also provided with the elapsed execution time and an estimate of the remaining run time.

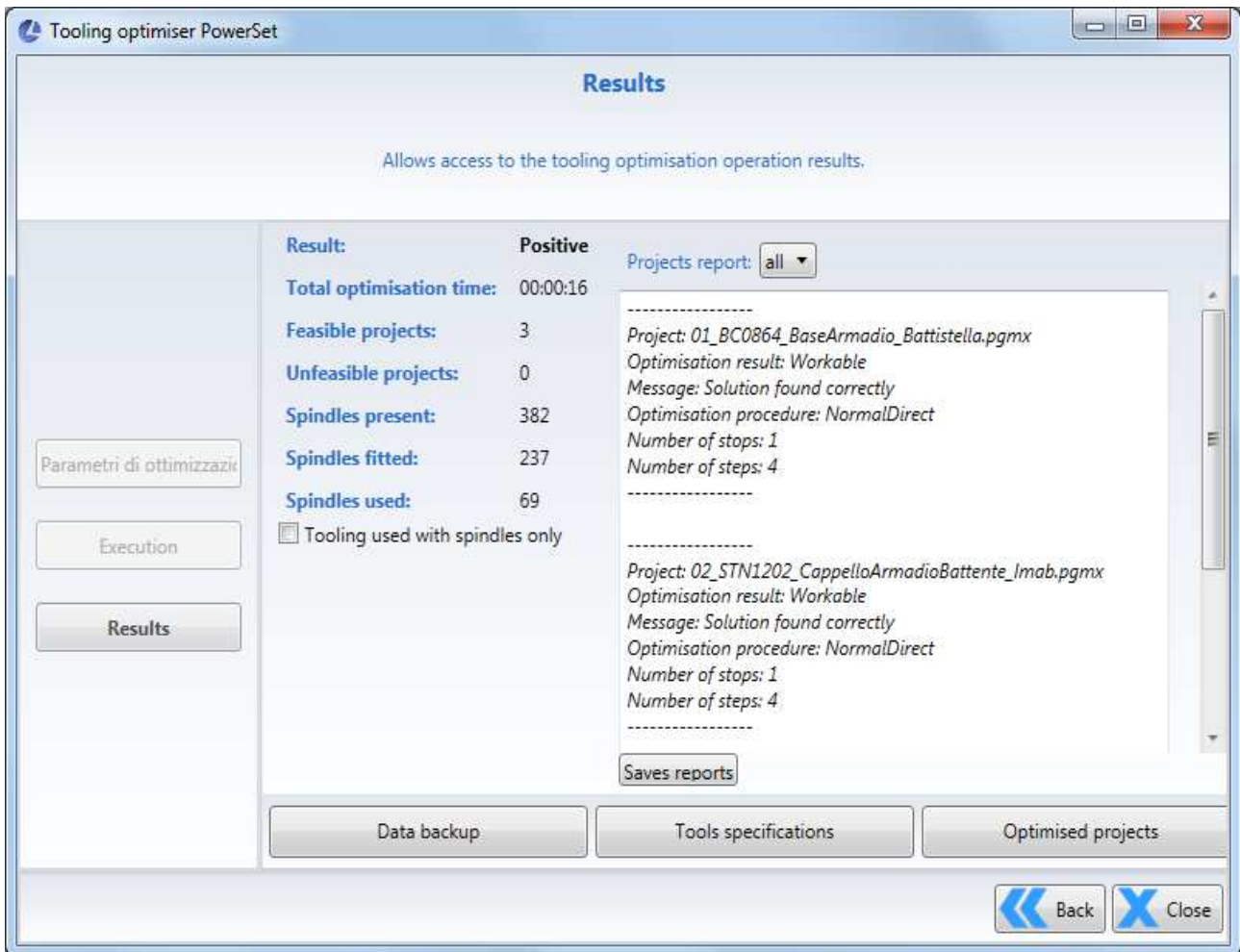


Optimization can be stopped by pressing the Stop button. This action stops the current operation with loss of processed data.

If the user does not interrupt the execution, the results page is loaded when the analysis is complete.

### 6.5.2.3 Page3: Results

This page displays the configuration result of the tool optimizer.



The central part of the page displays the general information about the analysis outcome, including:

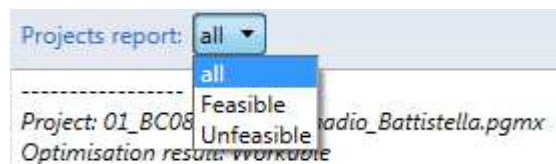
- outcome of execution (positive/negative)
- time required to perform optimization;
- Number of mix projects which are feasible;
- number of mix projects that are not feasible;
- total number of spindles in configuration;
- total number of spindles equipped;
- total number of spindles used for programme machining;
- individual project implementation reports.

If the number of spindles used is less than that of the equipped spindles, a display will give the user the possibility of choosing whether to consider the full setup or setup without tools unused in the mix.

Spindles fitted:	237
Spindles used:	69
<input checked="" type="checkbox"/> Tooling used with spindles only	

The choice of a setup with only the spindles used can be useful to identify any spindles with tools that are not useful for the mix execution. These spindles, if considered unfit, can be equipped with new tools in future optimizations of integration of the existing tooling.

If the result outcome highlighted the presence of unfeasible projects, it is possible to verify the reason for their unfeasibility in the project report. Using the drop-down menu, the user can choose to display the report of all projects, feasible projects or only those unfeasible ones.



You can save the report by copying the text content in the displayed pane.

In the lower part of the page, there are three buttons which allow you to respectively:

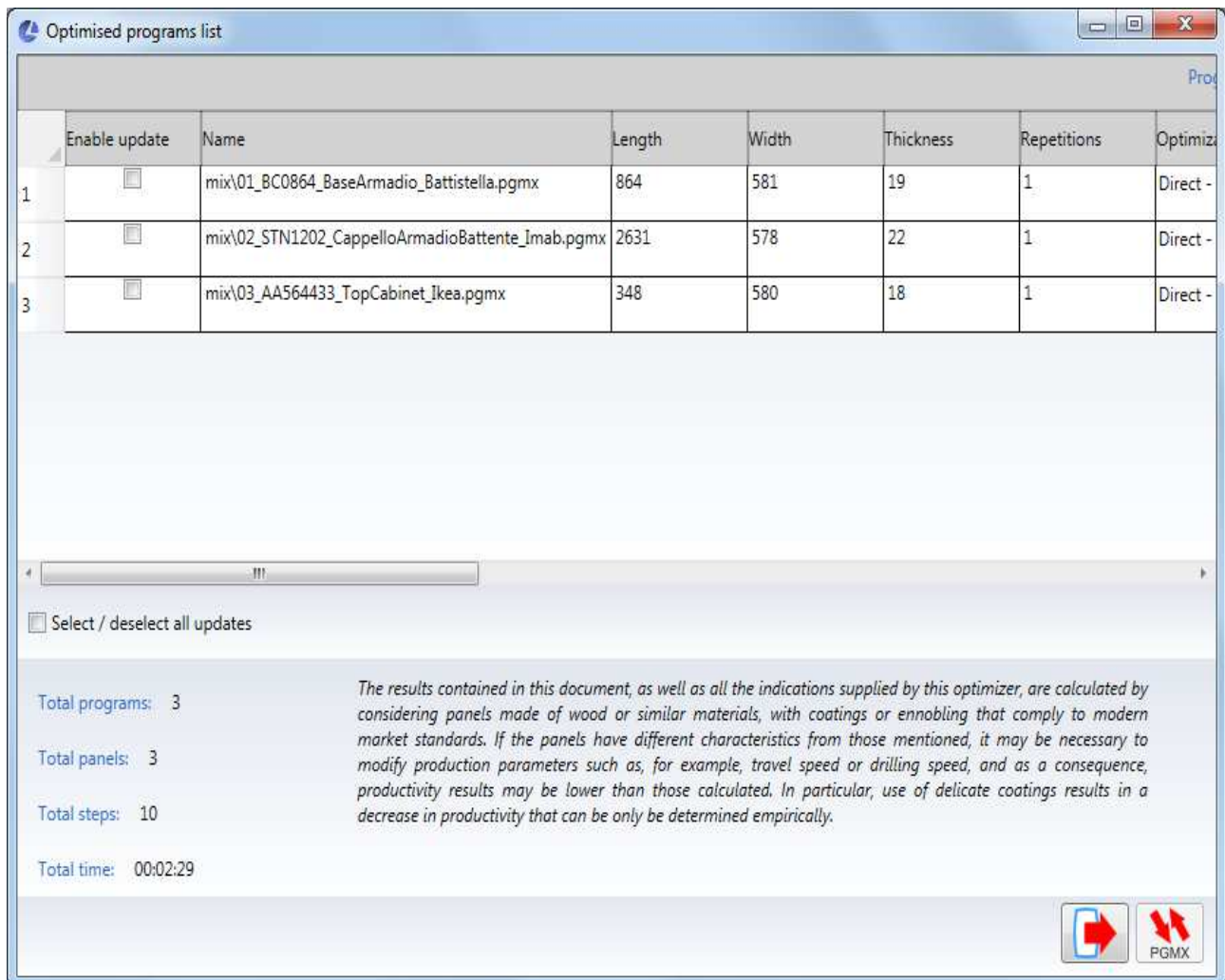
- Access the tool list;
- Access data from optimized projects;
- save the tooling.

## Tools specifications

By pressing this button, you can print the list of tools in the tool kit, including information on quantity, general tool characteristics and position on head (spindle number). For more details on the tooling specifications, please refer to the description of the tool printing functionality in the ToolEquipmentManager.

## Optimised projects

By pressing this button, it is possible to access the cyclical optimization data of the mix programs, obtained by using the optimised tooling.





The screenshot shows a window titled "Optimised programs list" with a table containing three rows of data. Below the table, there is a summary section with statistics and a disclaimer.

	Enable update	Name	Length	Width	Thickness	Repetitions	Optimiz
1	<input type="checkbox"/>	mix\01_BC0864_BaseArmadio_Battistella.pgm	864	581	19	1	Direct -
2	<input type="checkbox"/>	mix\02_STN1202_CappelloArmadioBattente_Imab.pgm	2631	578	22	1	Direct -
3	<input type="checkbox"/>	mix\03_AA564433_TopCabinet_Ikea.pgm	348	580	18	1	Direct -

Select / deselect all updates

Total programs: 3  
Total panels: 3  
Total steps: 10  
Total time: 00:02:29

*The results contained in this document, as well as all the indications supplied by this optimizer, are calculated by considering panels made of wood or similar materials, with coatings or ennobling that comply to modern market standards. If the panels have different characteristics from those mentioned, it may be necessary to modify production parameters such as, for example, travel speed or drilling speed, and as a consequence, productivity results may be lower than those calculated. In particular, use of delicate coatings results in a decrease in productivity that can be only be determined empirically.*

A table is presented at the top of the window, in which each line contains the optimization data of a mix program. For each program, the general program information and the optimization data obtained from the tooling are presented.

Below is a list of the main information provided for each item in the table:

- programme name;
- workpiece size;
- number of workpiece repetitions (quantity);
- optimization mode for execution on machine;
- Number of steps per quantity 1;
- execution time for quantity 1;
- productivity (pieces/minute);
- setup time considered (for change of workpiece);
- lot execution time (full quantity);
- panel positioning in the machine;
- Number of steps in the machine for quantity 1;
- run time in the machine for quantity 1.

This table can be saved in the electronic excel format, by pressing the export button.



If the user needs to transfer the cyclical information obtained to a source program, he can enable the program update by checking the "Enable update" box for the line of interest. If instead you want to select/deselect the update of all programs, you can use the cumulative box at the end of the table. The information transfer operation is activated by pressing the PGMX update button.



This operation has the effect of inserting optimization constraints in the source project, such as panel stop positions and the rotation status of the heads. The resulting program can thus be optimized in the machine with shorter processing times, as thought pre-optimised.

### **Save tooling**

By pressing this button, you can save the tooling generated by the optimizer.

## 7 Nesting Project Management

“Nesting” is a technique which allows the best arrangement of workpieces on a panel to be found, so as to minimize waste when the workpieces are produced.

A “Nesting Project” is a project including the definition of one or more workpieces to be produced, definition of the rough panel with all of the workpieces positioned, and the information describing methods for positioning the workpieces on the panel.

Hereinafter the “rough panel” is referred to simply as the “Sheet”.

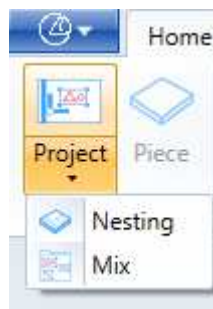
The “Nesting Project” file has a (.nstx) extension and is a compressed file containing:

pezzo1.pgm	Workpiece 1 design
pezzo2.pgm	Workpiece 2 design
pezzon.pgm	Workpiece n design
foglio1.pgm	Nesting Arrangement on sheet 1
foglio2.pgm	Nesting Arrangement on sheet 2
foglion.pgm	Nesting Arrangement on sheet n
Progetto Nesting.xml	information for positioning workpieces on panel

A Nesting Project can be:

- Created
- Opened
- Closed
- Saved

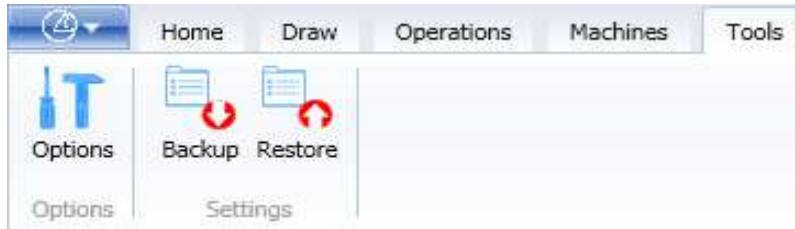
Using these icons present in the menu area:





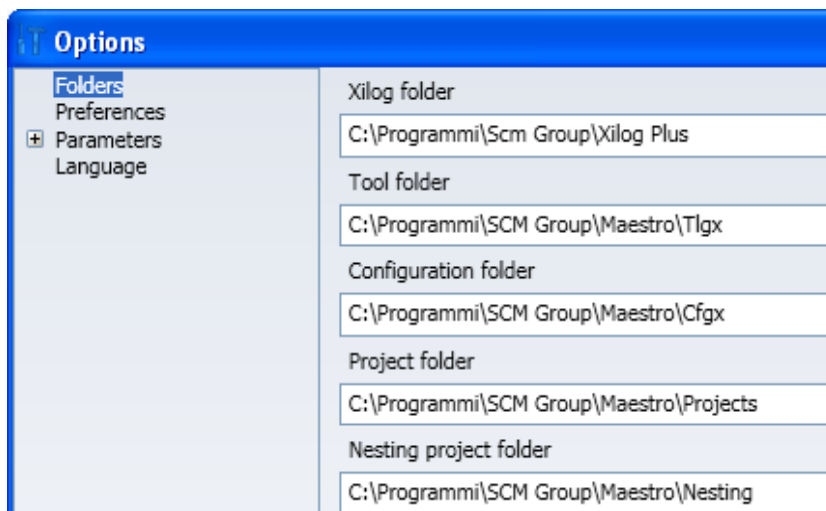
## 7.1 Defaults for “Nesting Projects”

It is possible to change the Defaults relating to Nesting Projects by clicking on the “Options” icon in the “Tools” tab.



### 7.1.1 Default Folder for “Nesting Projects”

To change the name of the folder in which the Nesting Projects are saved: Click on the “Options” icon and the following window opens:



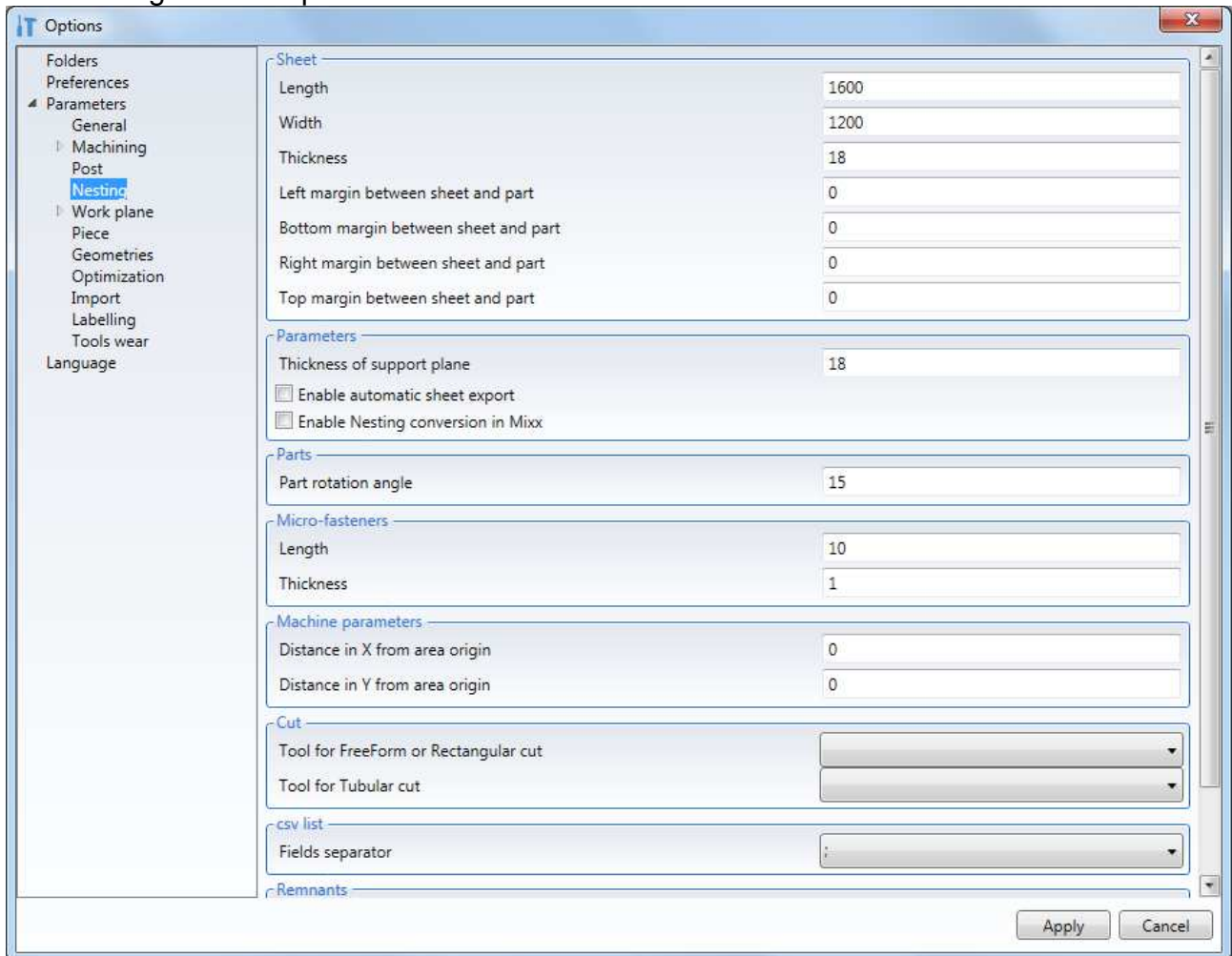
In which you can change the “Nesting Projects Folder” field.

## 7.1.2 Default Nesting Parameters

To change the value of the Nesting Parameters:

- Click on the “Options” icon and after the “Options” window opens
- Click on “+” to the left of “Parameters”
- Click on the “Nesting” item

The following window opens:



Where you can set:

Sheet:

- the size of the sheet that is normally used
- Margin dimensions of the sheet to be unused

Parameters

- the thickness of the martyr panel
- Choosing to export a pgmx for each nesting scheme
- Choosing to convert the nesting project to a .mixx file

Parts

- the angle of rotation of the parts to fit them into the Free Form nesting

### Microjoints

- Length and Thickness of the bridges to use to keep the ties together

### Machine parameters

- Work area where to place the sheet
- Distance to X and Y of the sheet from the origin of the workspace

### Cutting

- The tool to use by default for FreeForm or Rectangle cutting
- The tool to be used by default for tubular cutting

### Distinta csv

- The separator character to be used between the various fields of the csv file

### Remainders

- The number of decimals to be written in the code assigned to the rest of the reusable sheet

Once you have set the values you can:

- store them by clicking the "Apply" button
- cancel the changes by clicking the "Cancel" button

## 7.2 Creating a new Nesting Project

To create a new Nesting project, a tool Database must have been activated (see sec. 10.8), then click on the “Nesting” icon.



If there is a project already open, the “Close Project” function is automatically activated.

Select the “Nesting” command and you will be asked for information grouped into three parts of a dialogue box, in this sequence:

- 1) Parameters
- 2) Sheets
- 3) Parts Parts (not available for Manual Nesting)
- 4) Cutting
- 5) Parts (not available for Manual Nesting)
- 6) Remainders and scrap (not available for Tubular Nesting)
- 7) Labelling (not available for Tubular Nesting)
- 8) Loading unloading and cleaning (not available for Tubular Nesting)

The following buttons will be available on the bottom right of each session:



They can either be enabled or disabled.

“Back”, goes back to the previous dialogue box

“Next”, goes to the next dialogue box

“Finish” starts processing the Nesting

“Close” cancels the Nesting operation

## 7.2.1 Nesting Parameters

Nesting parameters are collected in the following dialogue box:

The screenshot shows the 'Parameters' dialog box of the Nesting Wizard. The window title is 'Nesting Wizard' and the main title is 'Parameters'. Below the title, it says 'Initialisation of parameters to be used for Nesting algorithm input'. On the left side, there is a vertical list of buttons: 'Parameters' (highlighted), 'Sheets', 'Parts', 'Cut', 'Optimisation', 'Remnants and scrap', 'Labelling', and 'Loading unloading and cleaning'. The main area is divided into two sections: 'Type of Nesting' and 'Parameters'. Under 'Type of Nesting', there are four radio buttons: 'Rectangular', 'Free Form' (selected), 'Tubular', and 'Manual'. Under 'Parameters', there is a 'Sequencing of parts' dropdown menu set to 'Area'. Below that are two input fields: 'Minimum distance between two nesting parts' with the value '0' and 'Spoilboard thickness' with the value '18'. There are also two checkboxes: 'Enable cutting outside sheet boundary' (unchecked) and 'Enables the possibility of nesting the parts in the holes of other parts' (unchecked). At the bottom right, there are four buttons: 'Back', 'Next', 'End', and 'Close'.

### 7.2.1.1 Type of Nesting

There are two possible types of Nesting:

- 1) Rectangular
- 2) Free Form
- 3) Tubular
- 4) Manual

Select one or the other by clicking in the related radio button.

### **7.2.1.1.1 Rectangular Nesting**

Rectangular Nesting allows only rectangular workpieces, which may even be rotated anti-clockwise through 90°, to be positioned on the sheet.

This type of nesting has a relatively low processing time, since it has a lot fewer degrees of freedom than Free Form Nesting.

### **7.2.1.1.2 Free Form Nesting**

Free Form Nesting allows workpieces having any shape and arranged at any angles to be positioned on the sheet.

This type of nesting has longer processing times than Rectangular Nesting because it involves many more degrees of freedom.

### **7.2.1.1.3 Tubular Nesting**

The Tubular Nesting is used to machine pieces from an unfinished bar. In this case the pieces must have the same section as the bar.

### **7.2.1.1.4 Manual Nesting**

This mode does not allow for a nesting project with automatic pieces positioning and relative optimisation.

At the end of the Wizard phase there won't be a complete nesting project, but the user will have to manually position the parts on the sheets and then optimise the program. (see 7.2.9 Manual Nesting completion)

### **7.2.1.2 Part sequencing**

Part sequencing allows definition of the sequence in which the workpieces must be arranged on the sheet.

Possible alternatives:

- 1) By Area
- 2) By Perimeter

which can be selected from the drop-down menu.

#### **7.2.1.2.1 Insertion in sequence by Area**

In this mode, the arrangement of the parts on the sheet is done starting with the workpiece which has the largest area and ending with the workpiece which has the smallest area.

#### **7.2.1.2.2 Insertion in sequence by Perimeter**

In this mode, the arrangement of the parts on the sheet is done starting with the workpiece which has the longest perimeter and ending with the workpiece which has the shortest perimeter.

### **7.2.1.3 Minimum distance between two nested parts**

This value represents the distance remaining between a workpiece and the tool machining the adjacent workpiece.

### **7.2.1.4 Spoil board thickness**

For setting the size of the panel placed beneath the rough sheet to be machined.

The default value of the spoil board thickness can be changed by setting the "Spoilboard thickness" in the session; Tools-Options-Parameters-Nesting-Parameters

### **7.2.1.5 Enable cutting outside the sheet**

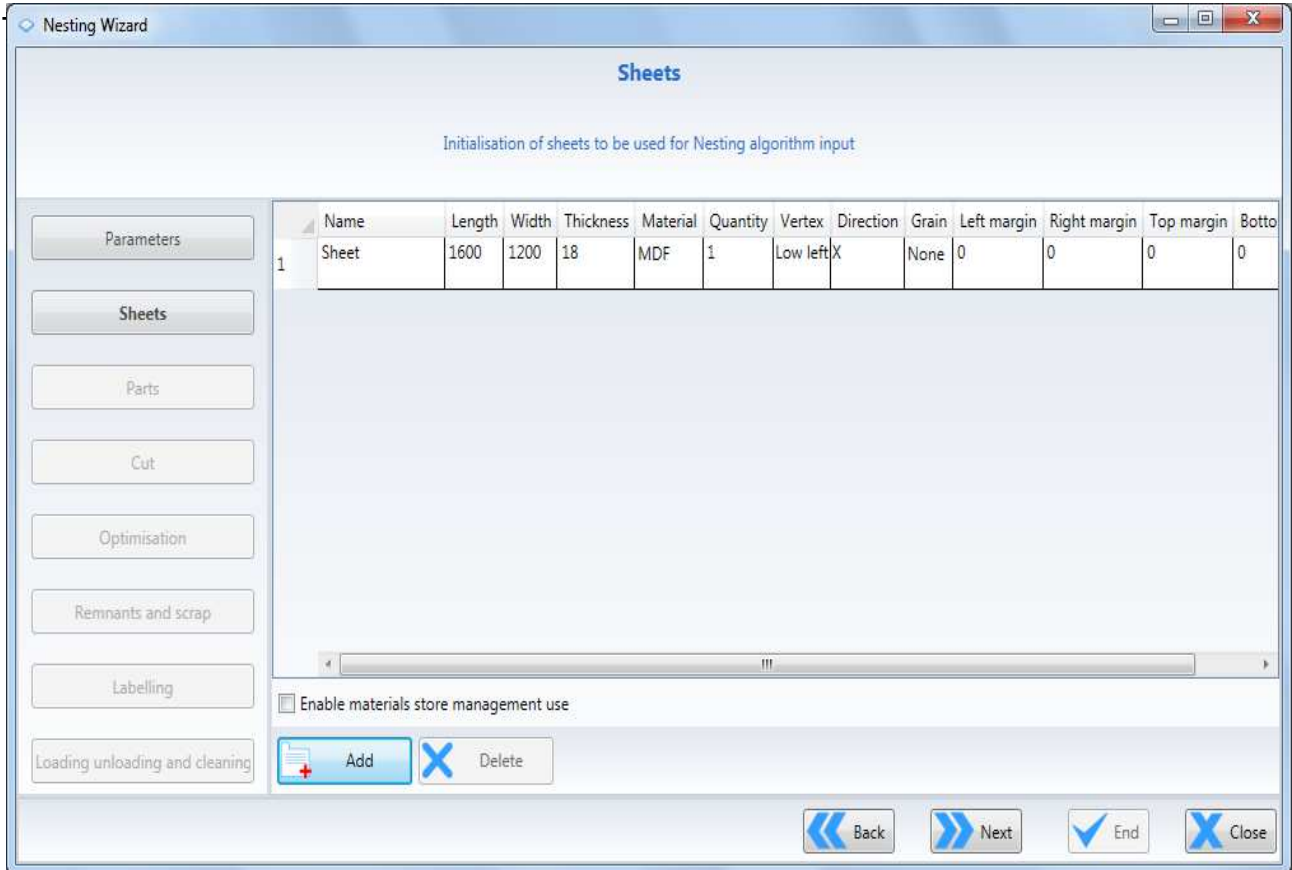
Tick this box and separation of the workpieces at the border of the sheet can be carried out even outside of the sheet.

### **7.2.1.6 Enables the possibility of nesting the parts in the holes of other parts**

Selecting this option allows a piece to be obtained from the scrap contained inside another larger part.

## 7.2.2 Nesting Sheets

In this session you can set the dimensions of the rough sheets to be used for Nesting machining.



The sheet displayed, when creating a new nesting project, will have the previously defined default dimensions. (see 8.1.1)

Use the “Back” command to return to the “Parameters” dialogue box

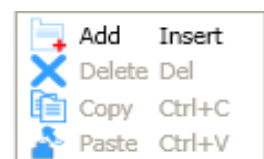
Use the “Forward” command to go to the “Parts” dialogue box

Use the “Close” command to cancel the Nesting operation.

In this context you can:

- 1) Add a sheet to the list
- 2) Delete a sheet from the list
- 3) Copy sheet data
- 4) Paste sheet data
- 5) Edit sheet data
- 6) Enable the use of materials magazine management

Right click with the mouse in the Sheets area and the following context menu appears:





### 7.2.2.1 Adding a sheet to the list

To add another sheet to the list:

- Left click with the mouse on the "Insert" item in the context menu.
- Or click on "Add" in the dialogue box

Or select a cell or the line of a sheet and press the "Ins" key

A new sheet will appear with the previously defined default dimensions. (see 8.1.1)

#### 7.2.2.1.1 Meanings of sheet data

Each sheet contains the following data:

- Name → Sheet name
- Length → Sheet length
- Width → Sheet width
- Thickness → Sheet thickness
- Material → Sheet material
- Quantity → Quantity of sheets available
- Vertex → Starting position for positioning workpieces  
(selectable from menu)
  - Top left
  - Top right
  - Bottom left
  - Bottom right
- Direction → Direction in which the workpieces will be positioned  
(selectable from menu)
  - X
  - Y
- Grain → Direction of the sheet grain  
(selectable from menu)
  - X
  - Y
  - None
- Left margin → Size of left margin in which workpieces cannot be positioned
- Right margin → Size of right margin in which pieces cannot be positioned
- Top margin → Size of top margin in which workpieces cannot be positioned
- Bottom margin → Size of bottom margin in which workpieces cannot be positioned

N.B.: The Vertex and Direction parameters are not available for the rectangular nesting.

### **7.2.2.2 Deleting a sheet from the list**

To delete a sheet from the list:

- Right click on the number to the left of the sheet you want to delete.
- Left click on the “Del” item in the context menu.
- Or click on "Delete" in the dialogue box
- Or press the “Del” key

### **7.2.2.3 Copying sheet data**

To copy the data from a sheet:

- Highlight the cells of the sheet to be copied
- Left click on the “Copy Ctrl+C” item
- Or simultaneously press the “Ctrl” and “C” keys

The selected data will be placed in a temporary memory so that it can be pasted.

### **7.2.2.4 Pasting sheet data**

To paste the data previously copied from one sheet to another sheet:

- Right click on the number to the left of the sheet you want to paste the data to.
- Left click on the “Paste Ctrl+V” item in the context menu.
- Or simultaneously press the “Ctrl” and “V” keys

### **7.2.2.5 Editing sheet data**

To edit the sheet data:

- Double click with the left mouse button on the data item to be edited
- Enter the new data item
- Press “Enter” or left click on another data item

Note: Some data items have preset values. In this case, the new value must not be typed in, but selected from a drop-down menu which can be displayed by clicking on the arrow to the right of the value or on the value itself.

### **7.2.2.6 Enable the use of materials magazine management**

By selecting this parameter the current sheets will be replaced with the ones previously defined in the materials magazine (see 8.9)

## 7.2.3 Nesting parts

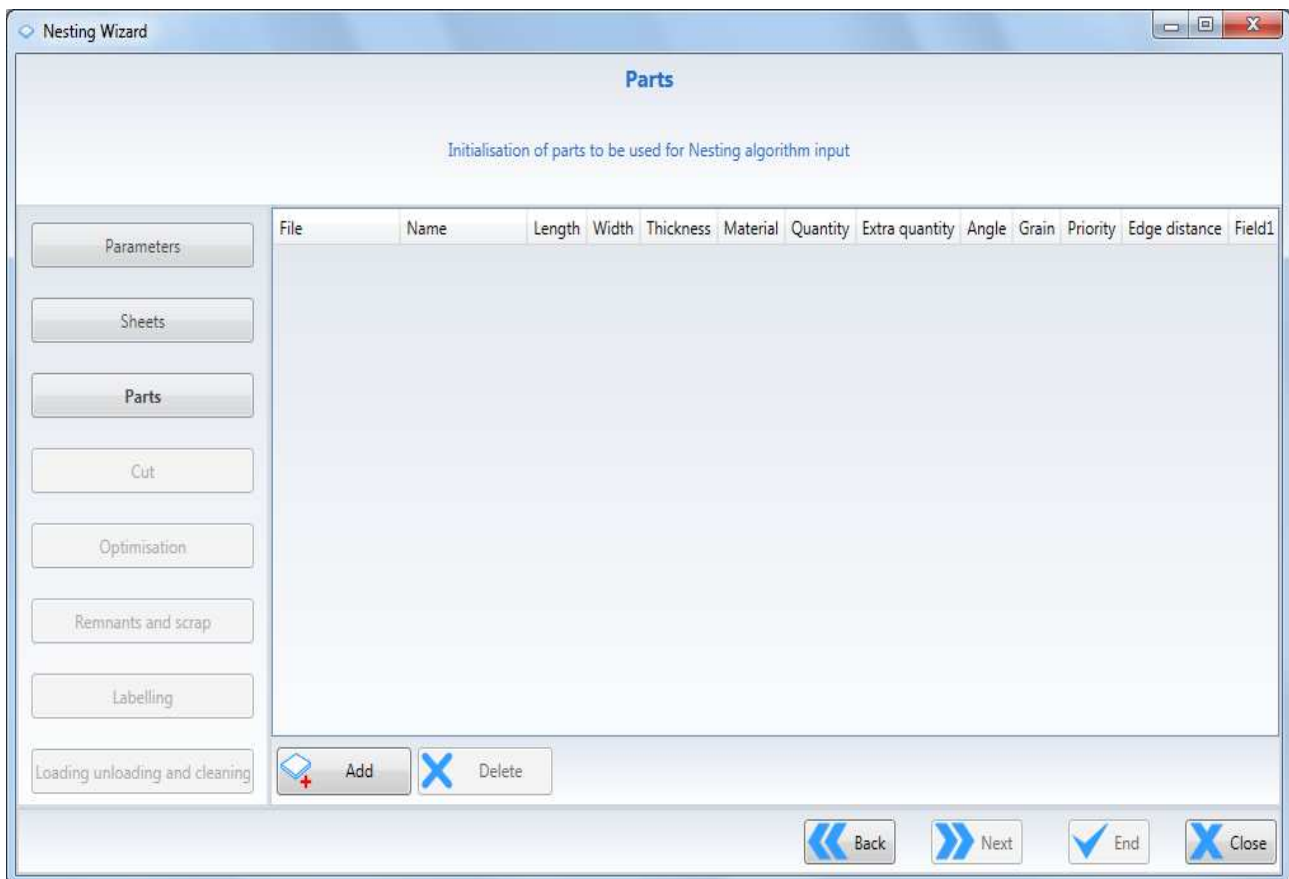
This session is for setting the list of workpieces to be machined.  
This session is not available for the Manual Nesting.

Parts to be entered in a Nesting project must be .pgmx type programs that must contain just one workpiece and one phase.

Non-rectangular shaped parts for nesting (Free form) should be created as "Extrusion" pieces and not as "Rectangular", otherwise the whole parallelepiped that encloses the non-rectangular piece would be considered as piece.

In this way the Nesting algorithm can slot the parts more efficiently.

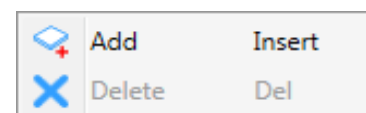
The following dialogue box appears:



In this context you can:

- 1) Add a workpiece to the list
- 2) Delete a workpiece from the list
- 3) Copy workpiece data
- 4) Paste workpiece data
- 5) Edit workpiece data

Right click with the mouse in the Parts area and the following context menu appears:



### 7.2.3.1 Enter workpiece in the list

To add a workpieces to the list:

- Left click on the "Insert" item in the context menu.
- Or click on "Add" in the dialogue box
- Or select a cell or the line of a workpiece and press the "Ins" key

The files that can be added must have the .pgmx extension.

The workpiece is added at the end of the list.

#### 7.2.3.1.1 Meanings of workpiece data

Each workpiece contains the following data:

- File → Workpiece file name
- Name → Workpiece name
- Length → Workpiece length
- Width → Workpiece width
- Thickness → Workpiece thickness
- Material → Workpiece material
- Quantity → Quantity of workpieces to be machined
- Extra quantity → Quantity of workpieces to be machined if space remains on the sheet after machining the required quantity of workpieces
- Angle → The angular increase used for tests when positioning the workpiece during Nesting processing
- Rotation → Workpiece 90° rotation (tick to select)
- Grain → Direction of workpiece grain (selectable from menu)
  - o X
  - o Y
  - o None
- Priority → Priority for insertion of workpiece on sheet (the workpiece with the highest value is inserted first)
- Distance from edge → Distance from edges in workpiece positioning
- Field1 - Fieldxx → Parameters available for the labels composition.

Notes:

- The parameters Angle, Priority and Distance from edge are only available in Free Form Nesting and Tubular Nesting
- The Rotation parameter is only available in Rectangular Nesting.
- The number of fields displayed us set in the labelling options. (see **Errore. L'origine riferimento non è stata trovata.**)

### 7.2.3.1.2 Inserting pieces from .CSV files

To insert multiple pieces at once, you can use a list of pgmx stored in a .Csv file

The list must consist of a line for each piece containing the following data:

Piece Code; (Ex: ARM21FSX;)

Type Piece; Not used, it can be "Blank" (Ex: F1

Description; Not used, it can be "Blank" (Ex: Left side;)

Amount; (Ex: 1;)

Extra Quantity; (Ex: 0;)

Length; (Ex: 1818;)

Width; (Ex: 478;)

Thickness; (Ex: 18;)

Material; (Ex: WALNUT;)

Grain; (0 = none 1 = long X 2 = Long Y) (Ex: 1;)

Pgmx file name, (Ex: Fianco\_sinistro.pgmx;)

Field1; Optional labeling fields (Ex: UserField1;)

Field fields are those set in the labeling options (see chapter 9.8.4.2)

Example :

ARM21FSX;F1;Fianco\_sinistro;1;0;1818;478;18;NOCE;1;Fianco\_sinistro.pgmx

ARM21FDX;F2;Fianco\_destro;1;0;1818;478;18;NOCE;1;Fianco\_destro.pgmx

ARM21CP1;T;Coperchio;1;0;864;479;18;NOCE;1;Coperchio.pgmx

ARM21BS;B;Fondo;1;0;864;479;18;NOCE;1;Fondo.pgmx

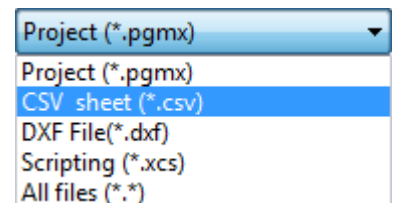
ARM21RPA;V;Divisorio\_orizzontale;1;0;864;467;18;VANIGLIA;1;Divisorio\_orizzontale.pgmx

ARM21SCH1;S;Schiena;1;0;1722;882;3;NAT;1;Schiena.pgmx

ARM21AZ2;Z2;Zoccolo\_frontale;1;0;864;80;18;NOCE;1;Zoccolo\_frontale.pgmx

ARM21AZ4;Z4;Zoccolo\_posteriore;1;0;864;80;18;NOCE;1;Zoccolo\_posteriore.pgmx

After selecting "Add" you select the filter "Bill CSV (\* .csv), then select the list containing the parts to be inserted.



The result that is obtained is the following:

	File	Name	Length	Width	Thickness	Material	Quantity	Extra quantity	Angle	Grain
1	Fianco_sinistro.pgmx	Pezzo	1818	478	18	NOCE	1	0	15	X
2	Fianco_destro.pgmx	Pezzo	1818	478	18	NOCE	1	0	15	X
3	Coperchio.pgmx	Pezzo	864	479	18	NOCE	1	0	15	X
4	Fondo.pgmx	Pezzo	864	479	18	NOCE	1	0	15	X
5	Divisorio_orizzontale.pgmx	Pezzo	864	467	18	VANIGLIA	1	0	15	X
6	Schiena.pgmx	Pezzo	1722	882	3	NAT	1	0	15	X
7	Zoccolo_frontale.pgmx	Pezzo	864	80	18	NOCE	1	0	15	X
8	Zoccolo_posteriore.pgmx	Pezzo	864	80	18	NOCE	1	0	15	X

### **7.2.3.2 Deleting a workpiece from the list**

To delete a workpiece from the list:

- Right click on the number to the left of the workpiece to be deleted.
- Left click on the “Del” item in the context menu.
- Or click on "Delete" in the dialogue box
- Or press the “Del” key.

### **7.2.3.3 Copying workpiece data**

To copy workpiece data:

- Highlight the cells of the workpiece to be copied
- Or simultaneously press the “Ctrl” and “C” keys

The selected data will be placed in a temporary memory so that it can be pasted.

### **7.2.3.4 Pasting workpiece data**

To paste the data previously copied from one workpiece to another:

- Right click on the number to the left of the workpiece you want to paste the data to.
- Left click on the “Paste Ctrl+V” item in the context menu.
- Or simultaneously press the “Ctrl” and “V” keys

Note: The data will not be pasted: File and Name

### **7.2.3.5 Editing workpiece data**

To edit workpiece data:

- Double click with the left mouse button on the data item to be edited
- Enter the new data item
- Press “Enter” or left click on another data item

Note :

- Note: Some data items have preset values. In this case, the new value must not be typed in, but selected from a drop-down menu which can be displayed by clicking on the arrow to the right of the value or on the value itself.
- The File and Name data cannot be changed.

## 7.2.4 Nesting Cut

In this session the user defines the way in which the pieces will be separated. The separation modes are:

- 1) Manual Cut
- 2) Automatic Cut

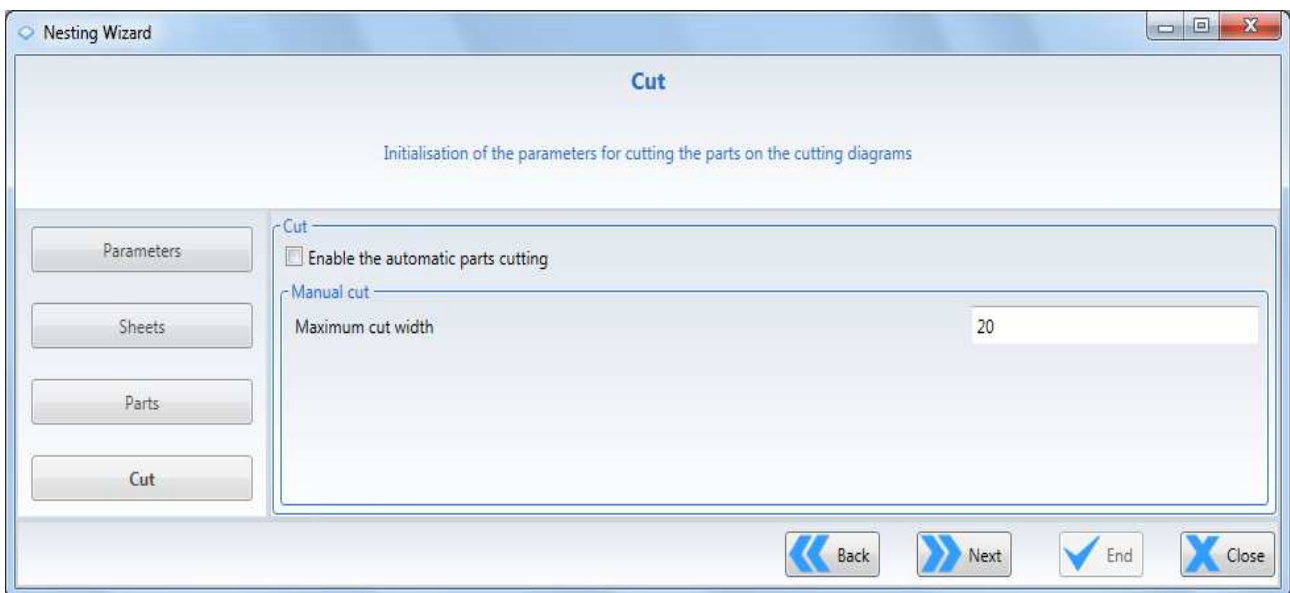
### 7.2.4.1 Manual Cut

This mode is used when the pieces in the nesting already have external trimming machining operations that effectively separate the pieces from each other.

It can also be used when the user does not want to do the separation during the nesting.

This session is accessed when the following parameter is not selected: "Enable the automatic cutting of the parts"

The following dialogue box appears:



The only thing that must be specified is the "Maximum cutting width"

During the nesting calculation the pieces will be placed at the specified distance.

## 7.2.4.2 Automatic Cut

This mode is used when the pieces in the nesting do not have external trimming machining operations that can separate the pieces from each other.

This session is accessed when the following parameter is selected:  
"Enable the automatic cutting of the parts"

The following dialogue box appears:

The screenshot shows the 'Cut' dialog box in the Nesting Wizard. The window title is 'Nesting Wizard' and the main title is 'Cut'. Below the title is the subtitle 'Initialisation of the parameters for cutting the parts on the cutting diagrams'. On the left side, there is a vertical list of buttons: Parameters, Sheets, Parts, Cut (highlighted), Optimisation, Remnants and scrap, Labelling, and Loading unloading and cle. The main area is divided into several sections: 'Cut' with a checked checkbox 'Enable the automatic parts cutting'; 'Tool' with a dropdown menu set to 'E003(NESTING-E003)'; 'Cutting technology' with fields for 'Cut speed', 'Correction of the cutting depth' (set to 0), 'Enable Cad compensation' (unchecked), 'Automatic start point' (set to None), 'Cutting direction' (set to Geometry), and 'Automatic cut of the parts holes' (unchecked); 'Incision' with 'Enable incision' (unchecked); 'Lead In' with 'Enable' (unchecked); 'Lead Out' with 'Enable' (unchecked); and 'Machine functions' with 'Mechanical presser' and 'Tool blower' (both unchecked). At the bottom, there are four buttons: 'Back', 'Next', 'End', and 'Close'.

Here you can:

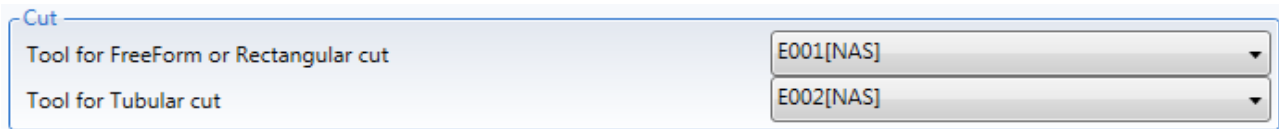
- 1) Tool
- 2) Cutting technology
- 3) Scoring
- 4) Approach
- 5) Retraction
- 6) Machine functions



### 7.2.4.2.1 Tool for Nesting cut

A tool can be selected from the current tooling to be used to separate the Nesting pieces.

The default tools can also be set by filling in the relative fields in the visible cutting session in "Tools-Options-Parameters-Nesting":



### 7.2.4.2.2 Cutting technology

In this session the following can be set:

- Cutting speed

- Cutting depth correction

the value is the distance between the tool and the lower face of the piece.

The following can be entered:

- a positive value (to drop with the tool below the lower face of the piece)

- zero (to drop with the tool level with the lower face)

- a negative value (to remain with the tool higher than the lower face of the piece, without detaching the piece from any scrap)

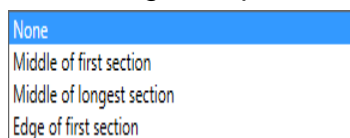
In the first two cases the piece is separated, whilst in the third case, the piece remains joined.

- Enable CAD compensation

Select this parameter for the CAD correction

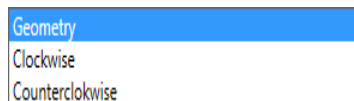
- Automatic start point

For each piece in the nesting, the machining start point can be forced in the following points:



- Cutting direction

For each piece in the nesting, the cutting direction can be forced in the following points:



Selecting geometry the cutting direction will be the one in the piece geometry

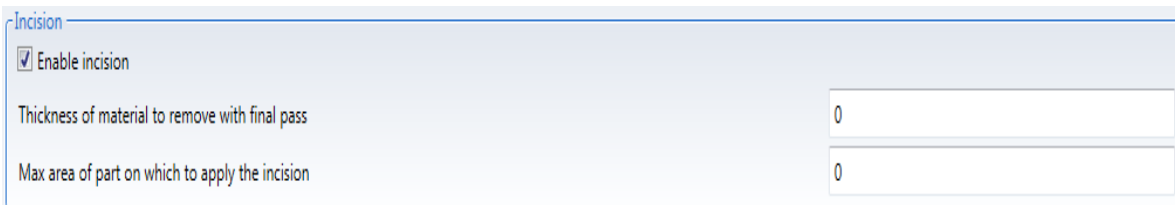
- Automatic cut of the holes of the parts

Selecting this parameter the holes of the parts that inside do not receive any machining will also be machined

### 7.2.4.2.3 Scoring

This function is used to machine the piece without detaching it from the sheet, leaving a minimum thickness of material that will be removed with the final pass. The aim is to reduce the cutting stress in the final pass so that when the piece is separated from the sheet it does not move

The following parameters are used for this:



Incision

Enable incision

Thickness of material to remove with final pass: 0

Max area of part on which to apply the incision: 0

The first sets the thickness of the material that keeps the piece joined to the sheet. The second links the scoring to the size of the pieces, as the movement problem is found only with small parts.

### 7.2.4.2.4 Approach

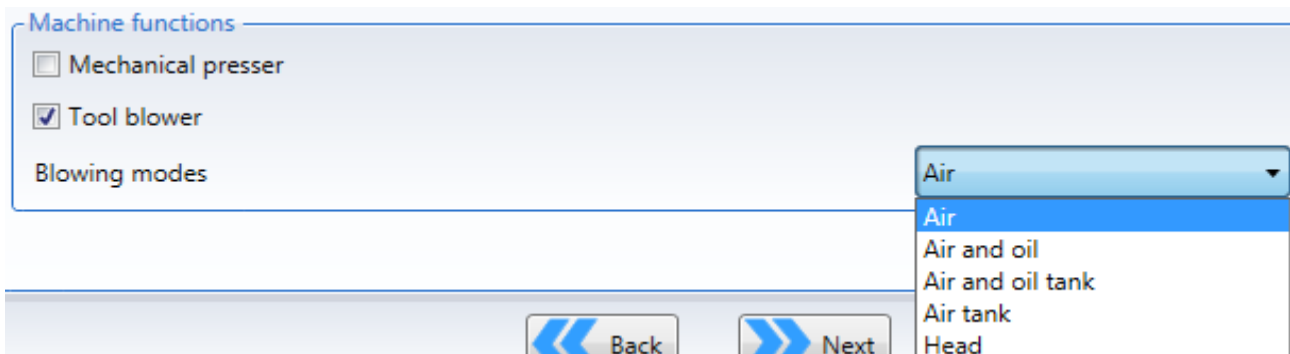
Managing the initial cutting section (see 3.5.14 Approach/Retraction (shared by the machining operations))

### 7.2.4.2.5 Retraction

Managing the final cutting section (see 3.5.14 Approach/Retraction (shared by the machining operations))

### 7.2.4.2.6 Machine functions

The following machine functions can be set:



Machine functions

Mechanical presser

Tool blower

Blowing modes: Air

- Air
- Air and oil
- Air and oil tank
- Air tank
- Head

Back Next

## 7.2.5 Nesting optimisation

This session can be used to set the same parameters that are set for the guided or automatic optimisation (see **Errore. L'origine riferimento non è stata trovata.**)  
The information is written in this sheet so that the nesting document is also optimised.

The screenshot shows the 'Nesting Wizard' window with the 'Optimisation' step selected. The window title is 'Nesting Wizard' and the main title is 'Optimisation'. Below the title, it says 'Initialisation of the parameters for the cutting diagrams machining optimisation'. On the left side, there is a vertical list of buttons: 'Parameters', 'Sheets', 'Parts', 'Cut', 'Optimisation' (which is highlighted), 'Remnants and scrap', 'Labelling', and 'Loading unloading and'. The main area contains the following settings:

- Enable machining optimisation
- Enable cut of small parts before the others
- Enable optimisation of the boring operations before all the other machining operations
- Type of optimisation: Along X (dropdown menu)
- Rows: 1 (text input field)

At the bottom of the window, there are four buttons: 'Back' (with a left arrow), 'Next' (with a right arrow), 'End' (with a checkmark), and 'Close' (with an X).

## 7.2.6 Nesting remainders and scrap

This session is used to manage the remainders and scrap machining. The remainder, unlike the scrap, is a portion of sheet that could be returned to the materials magazine (see 8)

The screenshot shows the 'Remnants and scrap' dialog box within the 'Nesting Wizard' application. The dialog is titled 'Remnants and scrap' and contains the following sections and controls:

- Remnants management:**
  - Enable remnants recovery
  - Tool:** A dropdown menu set to 'E001(NAS-E001)'.
  - All information:** A button with a help icon.
  - Min dimension of sheet to consider as remnant:** A text input field containing '0'.
  - Type of remnant:** A dropdown menu with 'Rectangular' selected. A tooltip is visible showing options: 'Rectangular', 'Single cut', and 'Rectangular'.
- Scrap management:**
  - Enable scrap machining
  - Enable scrap hogging
  - Tool:** A dropdown menu set to 'E001(NAS-E001)'.
  - All information:** A button with a help icon.
  - Min area for the scrap to be cut and not hogged:** A text input field containing '0'.
  - Overlap percentage between hogging passes:** A text input field containing '5'.
  - Residual material in relation to profile:** A text input field containing '0'.
  - Enable cut of scrap that cannot be hogged

At the bottom of the dialog, there are four buttons: 'Back', 'Next', 'End', and 'Close'.

The tool used to machine the remainders can be selected. The default tool is the one set in the cutting session displayed in "Tools-Options-Parameters-Nesting".

The pieces with both dimensions larger than the one set in "Minimum sheet dimension to consider as remainder" will be considered "Remainders".

The type of remainder can be "Rectangular" or "Single cut", so that a portion of sheet will be separated from the pieces machined with an open routing.

The scrap can be either be shred or cut by setting the parameters in the "Scrap management" table.

The tool used to machine the scrap can be selected.

The minimum area above which the scrap will be cut and not shred can be set. For the shredding the overlap of the passes can be set.

The "Remaining material in relation to the profile" parameter is used to complete the shredding of the scrap that would leave small parts at the edges. Setting a value equal to "-tool radius" the scrap is trimmed with the tool centre so that the scrap is completely shred.

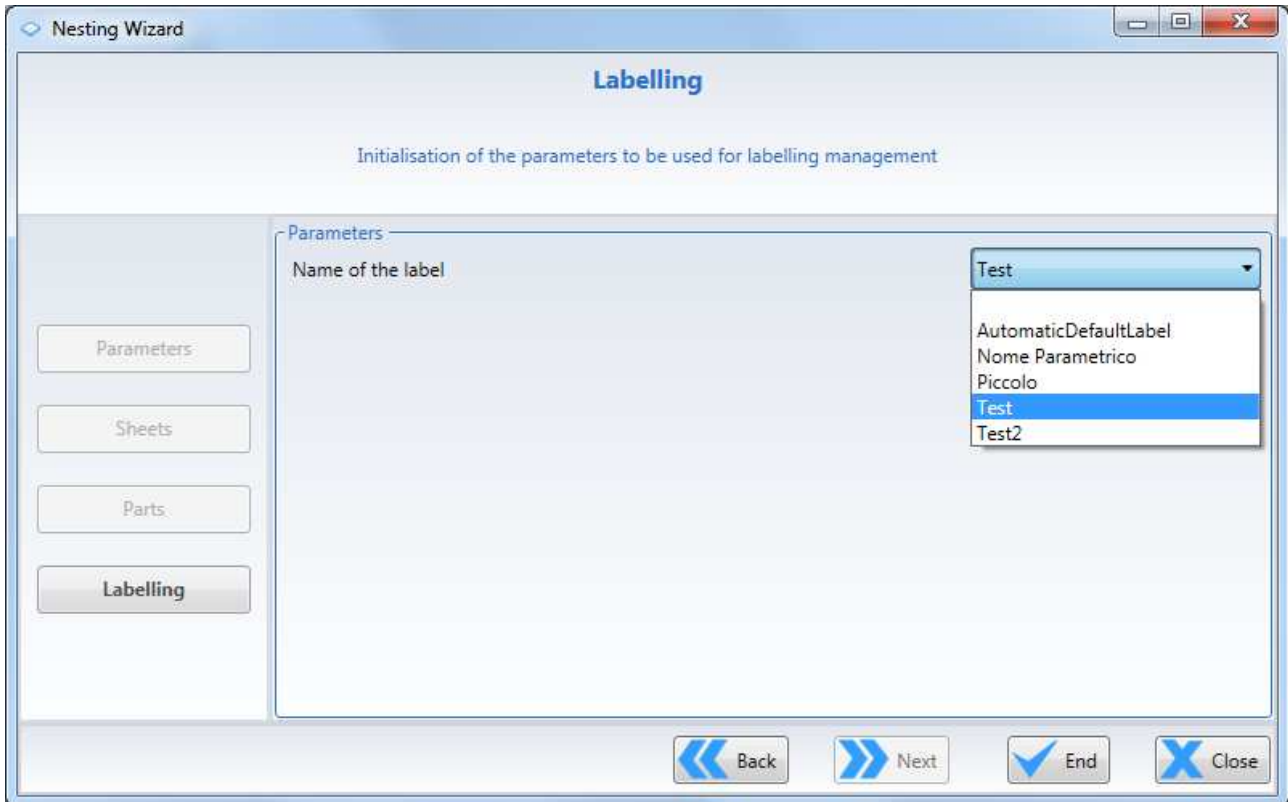
Selecting "Enable the scrap pre-cut" ensures that the scrap being cut does not, falling, ruin the finished pieces.

Therefore the pre-cut detaches the scrap or the remainder at a safety distance from the finished pieces so that the pieces are not damaged. It can be set in the "Pre-cut distance from the scrap profile" parameter.

## 7.2.7 Nesting labelling

This session defines the type of label to use for all the parts in the sheet.

The following dialogue window will be displayed:



Use "Back" to go back to the "Parts" dialogue window

Use "Finish" to start the processing (active if at least one piece has been added)

Use "Close" to cancel the nesting operation.

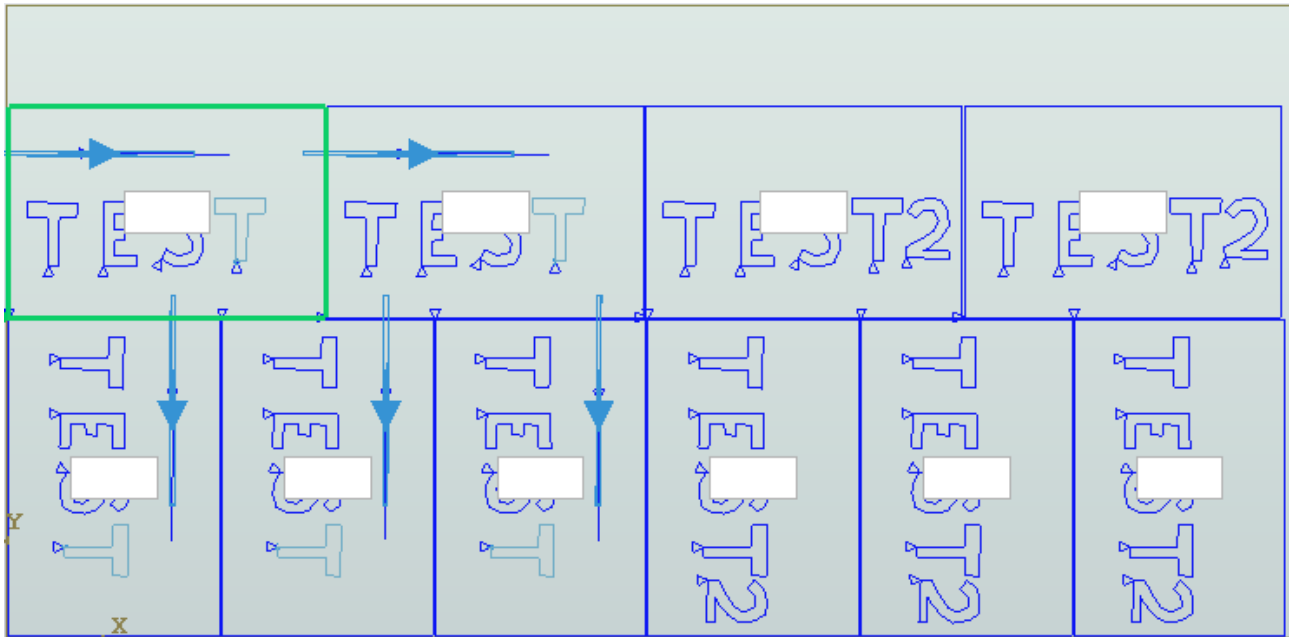
In this context you can select the label model that will be applied to all the pieces in the sheet.

The name of the label that appears by default is the one specified in the labelling options.

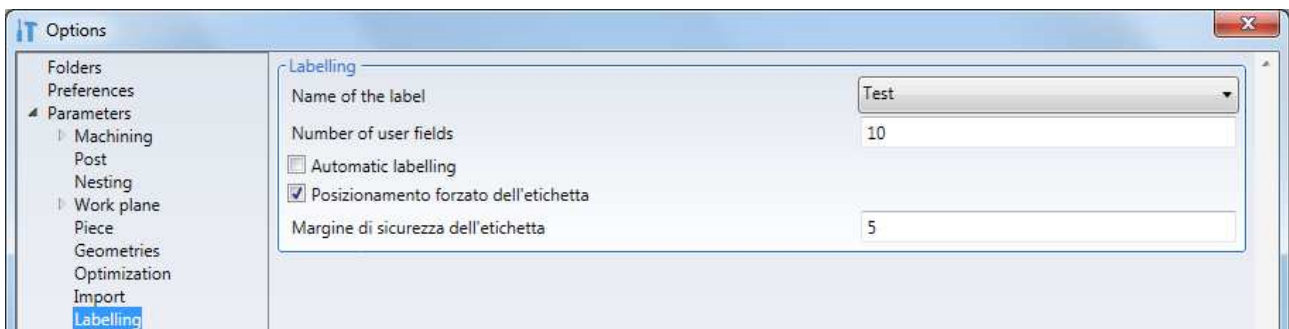
Note:

The label model is the same for each piece, but if the parametric label type has been selected, the data of each label can be different for each piece.

Click on "Finish" to see the layout of the pieces on the sheet and of the labels positioned on each piece.



The labels positioning depends on the options that have been set:



With the "Forced label positioning" selected the label will always be positioned in the middle of the piece.

Deselecting the "Forced label positioning" will position the label inside the piece and out of the machining on the piece, respecting the "Label safety margin".

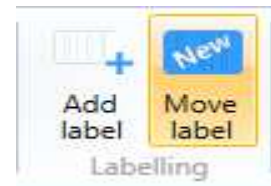
If there is no solution the label is not positioned.

If the result is not satisfactory you can:

- Move labels
- Delete labels
- Enter labels

### 7.2.7.1 Move labels

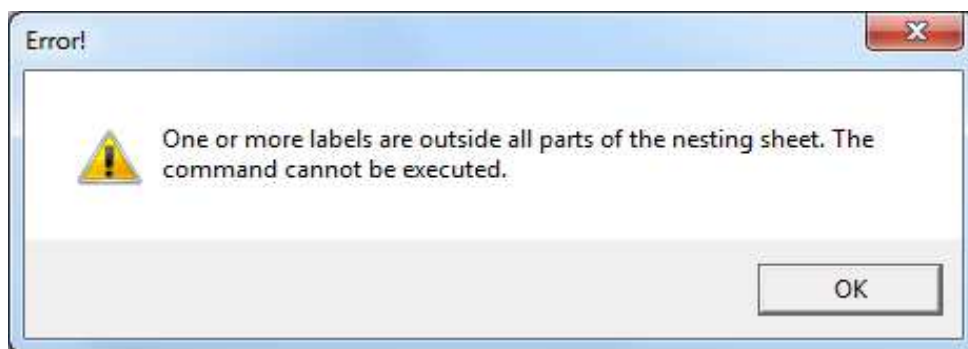
The labels can be moved with the "Move label" command in the "Drawing" menu.



Click on "Move label" to display the context menu that is used to specify the new position. A rectangular block appears with the dimension of the label being positioned.



If the label is positioned out of the piece the following message is displayed:



If the label is positioned on a different piece from the previous one the following message is displayed:



- Click on "Ok" to move the label
- Click on "Cancel" or on the red cross and the label will not be moved

To exit the command press "Esc".

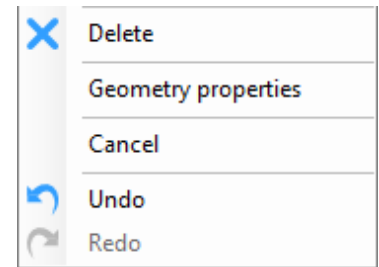


### 7.2.7.2 Delete labels

To delete a label use the context menu, similarly to any other element.

To activate the context menu, right click with the mouse on the label to be deleted and then click on "Delete" in the menu.

Or select the label to delete and press "Canc".

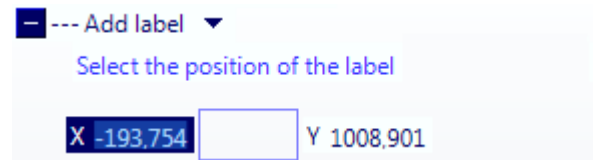


### 7.2.7.3 Enter labels

To enter a new label on the nesting pieces use the "Add label" command in the "Drawing" menu.



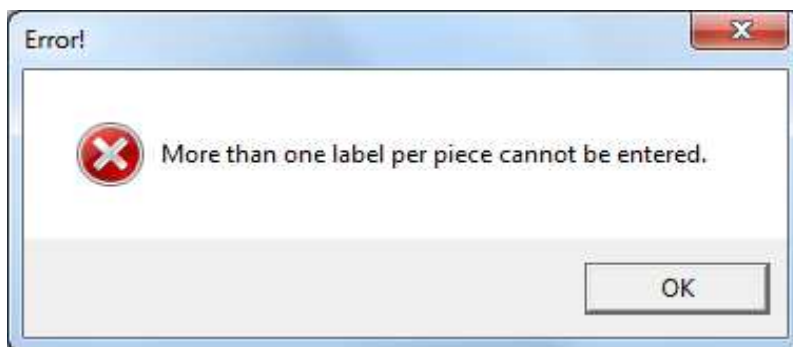
Click on "Add label" to display the context menu that asks you to select the label application point. A rectangular block appears with the dimension of the label being positioned.



If the label is positioned out of the piece the following message is displayed:



If the label is positioned on a piece that already has a label the following message will appear:



To exit the command press "Esc".

## 7.2.8 Loading, unloading and Nesting cleaning

This session defines the loading, unloading and cleaning modes of the machine table.

The following dialogue box appears:

The screenshot shows a software window titled "Nesting Wizard" with a sub-header "Loading unloading and cleaning". Below the sub-header is the text "Initialisation of the parameters for the sheets loading, unloading and cleaning management". On the left side, there is a vertical list of buttons: "Parameters", "Sheets", "Parts", "Cut", "Optimisation", "Remnants and scrap", "Labelling", and "Loading unloading and cleaning" (which is highlighted). The main area is divided into two sections: "Loading" and "Unloading and cleaning".

**Loading section:**

- Enable the sheet loading management
- Sheet loading method: Pickup on horizontal stop of elevator tabl
- Alignment method: Photocell
- Number of suction cups for alignment: Pickup with 2 suction cups and alignment
- Number of panels to load: 1
- Elevator table position offset: 0
- Unloader position offset: 0
- Enable piece presser

**Unloading and cleaning section:**

- Enable the operation of the sheet unloading and/or cleaning
- Piece pickup quota for unloading start: 0
- Type of table cleaning: None
- Cleaning start quota: Only piece area
- Unloader position offset: 0
- Enable the unloading only on the last sheet

At the bottom of the dialog, there are four buttons: "Back", "Next", "End", and "Close".

### 7.2.8.1 Load

The sheet can be loaded as follows:

Pickup on horizontal stop of elevator table and dragging with low suction cups  
Pickup with panel lifting and dragging with high suction cups  
Pickup with dragging and lifting

the sheet alignment method moves the sheet as far as a photocell or a mechanical stop.

The suction cups used for the pickup and alignment are managed by the parameter: number of suction cups for alignment and it has the following options:

- Pickup with 2 suction cups and alignment with 1 suction cup
- Pickup with all suction cups alignment with 2 suction cups

The following parameters can be set:

- Number of panels to be loaded.

N.B.: If the quantity is greater than 1 the automatic labelling is not available, and certain functions such as "Scoring" (Onion Skin) cannot be used because they are incompatible with the multiple loading.

- Elevator table position offset
- Unloader position offset

The piece presser can be enabled.

## 7.2.8.2 Unpacking and cleaning

The following parameters can be set:

- Piece pickup quota for unloading start

- Type of table cleaning with one of the following options

- None
- Semiautomatic with start wait
- Automatic at unloading end

- Cleaning start quota with one of the following options

- Only piece area
- All the table

- Unloader position offset

The unloading function can be executed only for the last nesting sheet.

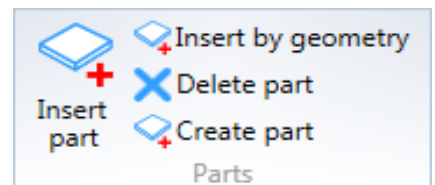
## 7.2.9 Manual Nesting completion

At the end of the Manual Nesting Wizard the project will have to be completed with: -  
Manual positioning of the parts on the sheet  
- Program optimisation

### 7.2.9.1 Manual positioning of the parts on the sheet

The parts on the sheet can be managed with the parts menu which allows the following operations

Create new part  
Insert new part  
Insert existing part  
Part rotation

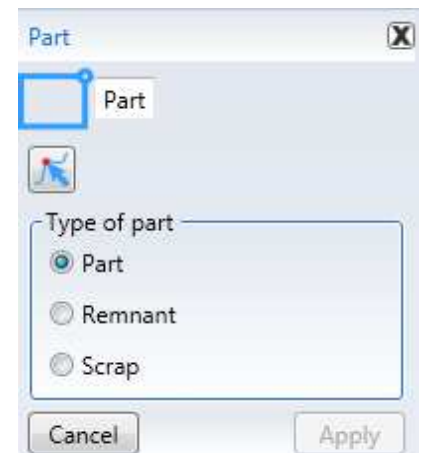


#### 7.2.9.1.1 Create new part

To create a new part use the "Create part" command  
This command assumes that there is a geometry on the sheet  
(imported or just created)

Select the geometry  
Decide the type of part to be created:  
Part  
Remainder  
Scrap

Click on Apply to create the desired part.

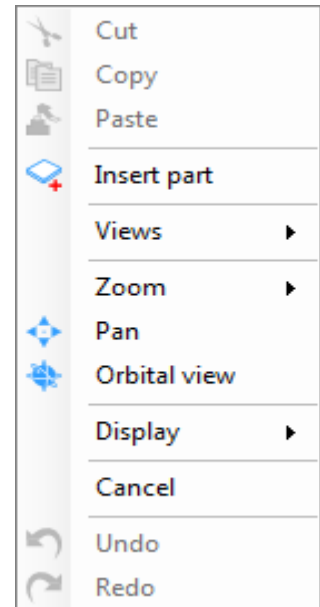


#### 7.2.9.1.2 Insert new part

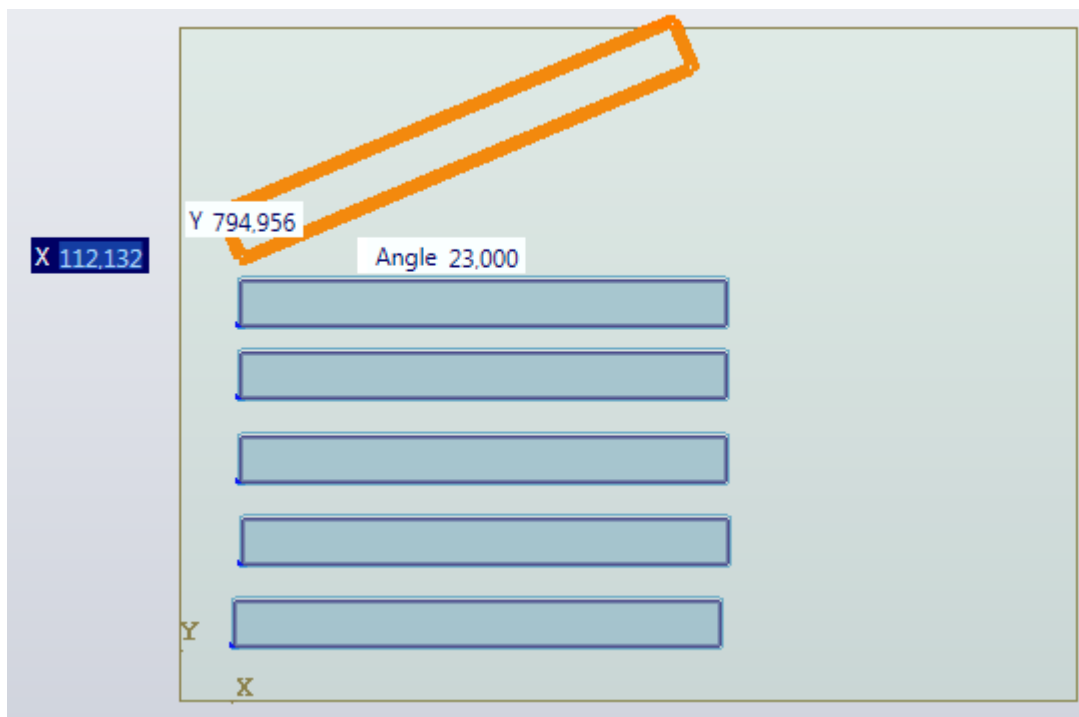
To enter a new part use the "Insert from geometry" command  
This command, unlike the "Create part" command, can be used to:  
- create more than part from a single geometry  
- position the parts in different positions  
- rotate the parts being inserted (see 7.2.9.1.4 Part rotation)

### 7.2.9.1.3 Insert existing part

To insert an existing part use the "Insert part" command that can also be found in the context menu that will appear by right clicking on the mouse on the sheet where the pieces are to be positioned.



Click on "Insert part" to display the part on the sheet, along with the fields of the X Y position and the orientation angle.



To move the part on the sheet you can:

- enter the values in the X Y fields
- move the mouse

To rotate the parts see the next section

The part insertion can be repeated: the same part and angle used previously will be maintained.

To interrupt the parts insertion press the "Esc" key.

#### 7.2.9.1.4 Part rotation

To rotate a part:

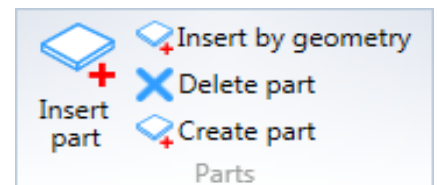
- enter a value in the angle field
- keep the Ctrl key pressed whilst rotating the mouse wheel  
In this case the rotation will be of 1°
- keep Shift+Ctrl pressed whilst rotating the mouse wheel  
In this case the rotation will be of 5°

When the parts comes out of the sheet or collides with another part during the rotation it will turn orange.

#### 7.2.9.1.5 Part deletion

To delete a part use the "Delete part" command

One or more parts can be selected for deletion.



## 7.3 Opening an existing Nesting Project

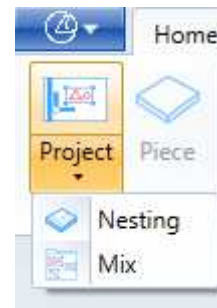
To open an existing Nesting Project, click on the “Open” icon.



A window appears in which you can select a file. Change the “File type” (whose default setting is: “Project (\*.pgmx)”) to Nesting (\*.nstx). Then select a .nstx file, followed by the “Open” key. Or double click on the .nstx file.

If there is a project already open, the “Close Project” function is automatically activated.

To change the data of a Nesting project with the Wizard use the “Create” – “Project” - “Nesting” command.



## 7.4 Closing a Nesting Project

Nesting project closing is the same as for a normal project (see sec. **Errore. L'origine riferimento non è stata trovata.** )

## 7.5 Saving a Nesting Project

Nesting project saving is the same as for a normal project (see sec. **Errore. L'origine riferimento non è stata trovata.** )

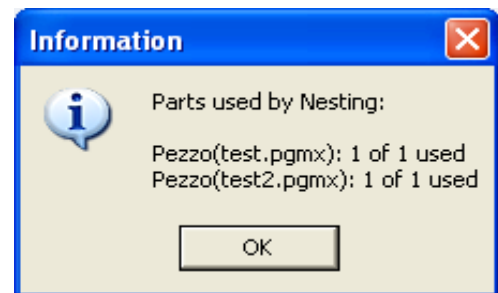
## 7.6 Nesting Processing

Nesting project processing takes place when you click on the “Finish” command in the “Parts” dialogue box (see sec. 7.2.3 ).

You access the “Parts” dialogue box both when a Nesting project has been created and when an existing Nesting project is opened, since all of the data entered at the creation stage was saved in the .nstx file.

After giving the “Finish” command, Nesting processing begins and it ends with a report message which summarizes the result:

Click on “OK” and the project graphics appear, showing all of the workpieces positioned on the sheet.

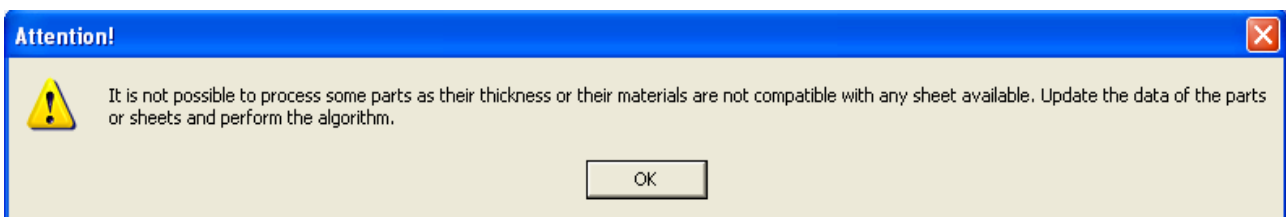


### 7.6.1 Conditions to be complied with in Nesting

If the data relating to:

- Thickness
- Material
- Grain

is not compatible between the workpieces and the sheets, Nesting processing is not successfully completed and the following message appears:





## 7.6.2 Recurring strategies in Nesting

### 7.6.2.1 Positioning small workpieces at the center

A recurring technique in Nesting is that of positioning small workpieces at the center of the panel to overcome hold-down problems.

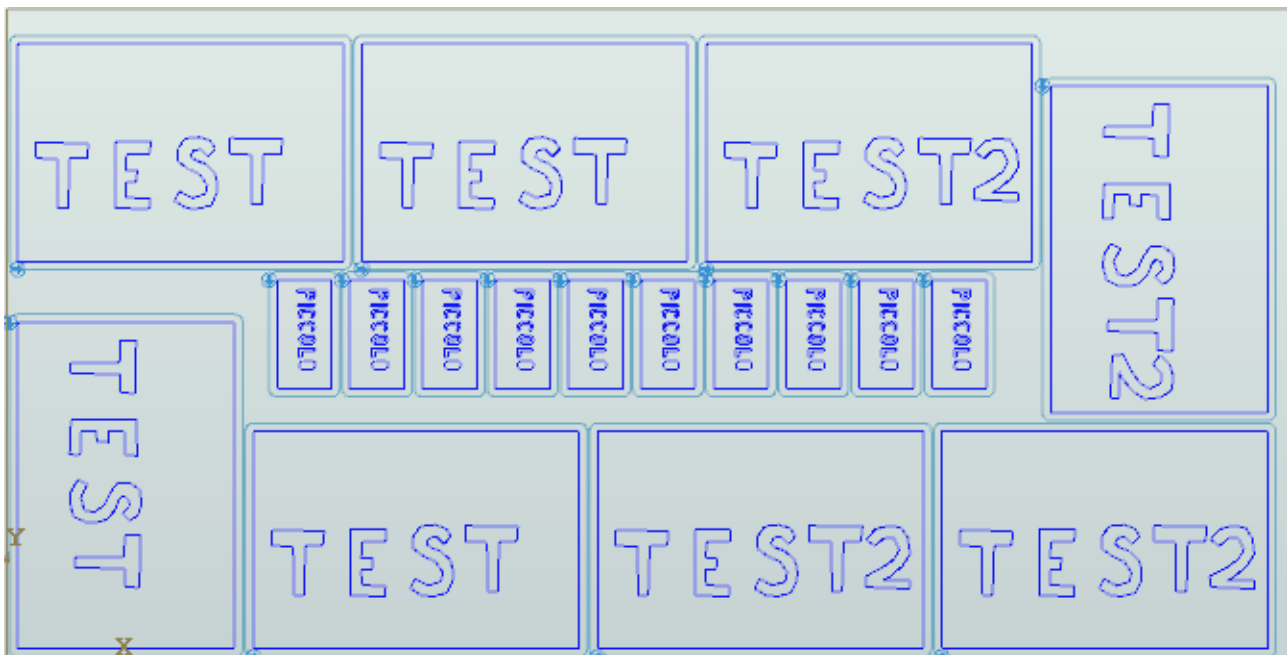
To do this:

- assign small workpieces a priority value higher than that of larger workpieces
- for the small workpieces set a "Distance from edge" value which places them at the center.

Example :

File	Name	Length	Width	Thickness	Material	Quantity	Extra quantity	Angle	Vein	Priority	Edge distance
1	test.pgm	300.000	200.000	18		1	0	15	None	0	0
2	test2.pgm	300.000	200.000	18		1	0	15	None	0	0
3	piccolo.pgm	100.000	50.000	18		1	0	15	None	0	0

The result obtained is as follows:

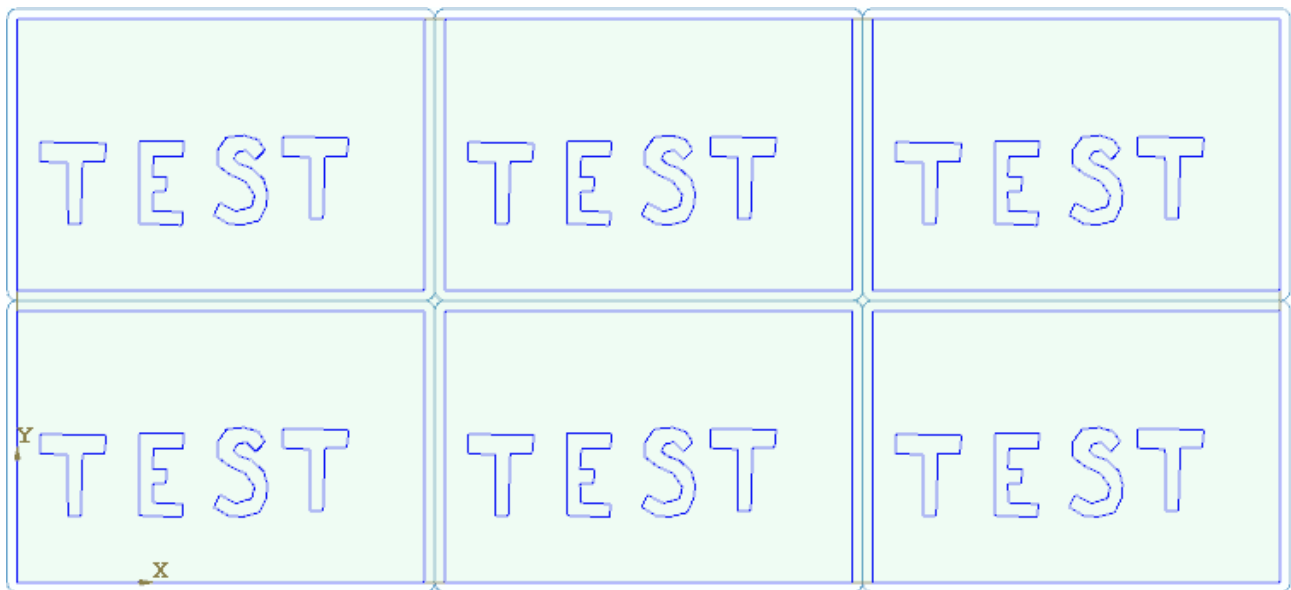


### 7.6.2.2 Contouring outside the sheet

To increase the number of workpieces which can be obtained from a sheet, you can use the technique of producing workpiece contours which are on the edges of the sheet outside of the sheet.

To do this, simply put a tick by the “Enable cutting outside the sheet” parameter (see sec. 7.2.1.5)

Example :



## 7.7 Optimizing Nesting machining

After Nesting processing and before the workpieces are machined on the machine, it is a good idea to optimize the Nesting project produced.

(see sec. **Errore. L'origine riferimento non è stata trovata.** Optimizing Projects)

## 7.8 Labelling pieces in machine

The pieces in the machine can be labelled in two ways, based on the labeller available.

### 7.8.1 Automatic labelling of pieces in machine

This type of labelling requires a labeller that can perform the following:

- Read the label data (what to print and where to position)
- Print labels
- Peel labels from support
- Position the label on the nesting panel before machining.

After the machining all the machined pieces separated by the sheet will have their own label attached.

To use this mode, the following parameters in the "Labelling options" must be set:



- Select the "Automatic labelling" parameter
- Set the "Label height and width"
- Select the labeller position:
  - Front (operator side labeller)
  - Rear (opposite operator side labeller)

### 7.8.2 Manual labelling of pieces in machine

This type of labelling requires a labeller that can perform the following:

- Read the label data (what to print)
- Print labels

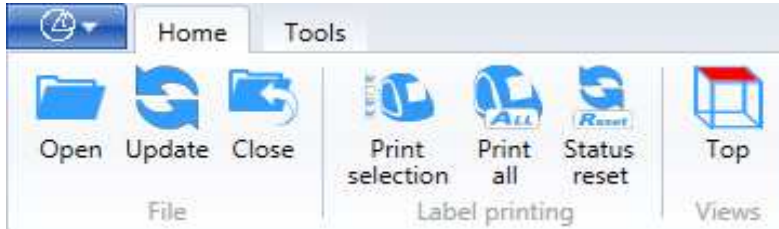
It also requires an operator to attach the labels on the various machined pieces.

This type of labelling is used if there is no automatic labeller, but it can also be used with the automatic labeller to attach damaged labels or labels not attached automatically.

### 7.8.2.1 Manual labelling of pieces with LabelPrinterManager

This application is used to print labels relative to one or all the pieces in the nesting sheet machined by the machine.

The interface is as follows:



It allows the operator to:

- Open a project
- Print one or all the labels in the open project

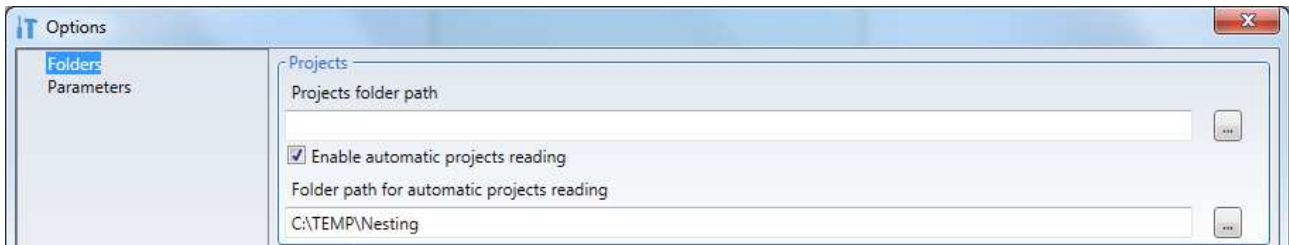
A project can be opened with the "Open" command and any project.

It can also be opened automatically with the "Refresh" command.

This mode lets you open in sequence all the projects executed from the PanelMac.

The programs executed are copied in an exchange folder with the prefix consisting of 5 digits to maintain the memory of the sequence of machined pieces.

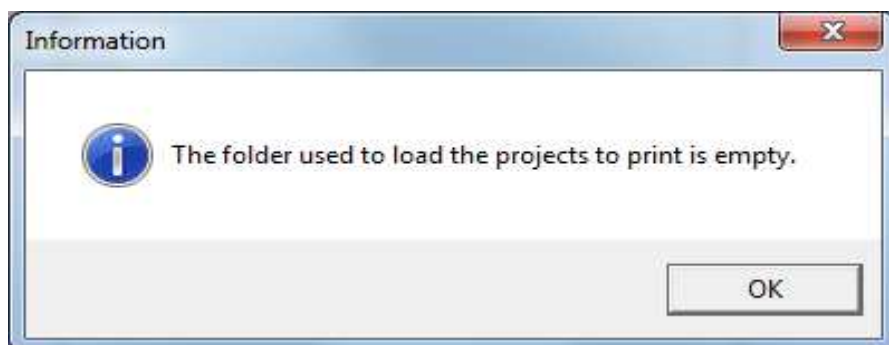
This exchange folder must be configured in the LabelPrinterManager options at "Path of folder for automatic projects reading".



Each time that "Refresh" is pressed:

- The next project in the exchange folder can be viewed
- One or more labels of the project open can be printed
- The project is deleted from the exchange folder after printing.

When there are no more projects in the exchange folder, the following message will be displayed:



One or more labels can be printed with the "Print selection" command.

This command is used to select one or more pieces and print the relative labels.



To print all the labels in the project use the "Print all" command.

This command does not require any selection.

After each label is printed the colour of the piece whose label has been printed changes.

The "Reset state" command is used to visually reset the state of the print, as if nothing had been printed.

### 7.8.3 Nesting .nstrptx report file

Each time a Nesting file is saved, in the Maestro\Report\Nesting folder, a name file is automatically saved equal to the name of the .nstx file and with extension .nstrptx.

The report file can be read with an external software to extract the information on sheets that have been nested and which and how many parts have been entered in the sheet. One of the various uses, for example, is to print labels to glue to each workpiece of the sheet.

The Nesting report file is a text file where each line contains the information of the part that is nested in the sheet.

Each line consists of elements defined by an identification key in square brackets "[]" performed by the symbol "=" as separator between key and value.

Each value ends with the separator ";".

Example:

```
[Key1]=Value1;[Key2]=Value2;[Key3]=Value3;[Key4]=Value4;
```

The following keys are present in a line:

[SheetName] : Name of the sheet  
[PartName] : Name of the part  
[PartFileName] : Name of the file containing the part  
[PartLength] : Length of the part  
[PartWidth] : Width of the part  
[PartDepth] : Depth of the part  
[PartQuantity] : Part quantity in the sheet  
[PartMaterial] : Material of the part

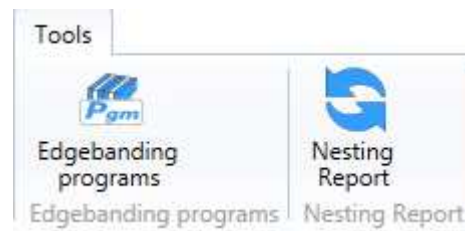
This is the .nstrptx report of the previous example :

```
[SheetName]=sheet;[PartName]= Pz500x200;[PartFileName]=drilled workpiece 500x200.pgm;  
[PartLength]=500;[PartWidth]=200;[PartDepth]=18;[PartQuantity]=3;[PartMaterial]=MDF;
```

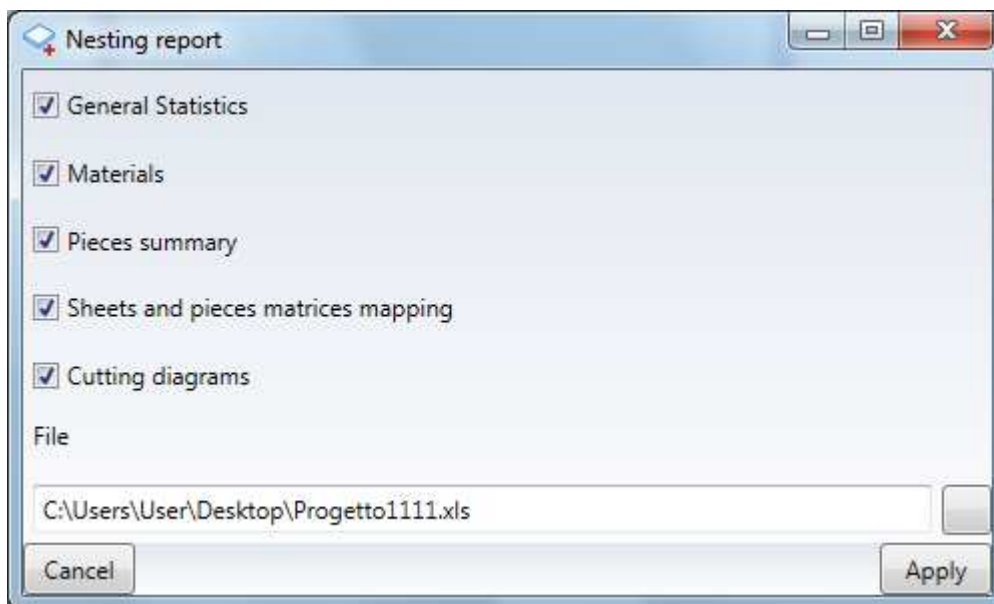
```
[SheetName]=sheet;[PartName]= Pz450x180;[PartFileName]=routed workpiece 450x180.pgm;  
[PartLength]=450;[PartWidth]=180;[PartDepth]=18;[PartQuantity]=3;[PartMaterial]=MDF;
```

## 7.9 Nesting report

Once the Nesting project processing has been completed the statistics can be viewed by pressing "Nesting Report" in the "Tools" menu.



Click on the command to display the following window where you can select the data that you want to examine.



For each topic a sheet will be created in an .xls file containing the relative data.

### 7.9.1 General statistics

The general statistics report contains the following values:

	A	B	C	D	E	F	G	H	I	J
1	<b>General Statistics</b>									
2	Type	Thickness	Time	% Parts	% Remainder	% Scrap	M <sup>2</sup>	M <sup>3</sup>	M <sup>2</sup> /h	M <sup>3</sup> /h
3		18	00:30:06	15,78	0	43,66	6,49	0,12	12,937	0,23
4	Summary		00:30:06	15,78	0	43,66	6,49	0,12	12,937	0,23

## 7.9.2 Materials

The materials report contains the following values:

	A	B	C	D	E	F	G	H	I	J
1	<b>Material</b>									
2	Type	Thickness	R	DY	Used	Quantity	Residual	M <sup>2</sup>	M <sup>3</sup>	
3		18	1600	1200	6	88	82	11,52	0,21	
4	<b>Summary</b>									
					6	88	82	11,52	0,21	

## 7.9.3 Pieces summary

The pieces summary report contains the following values:

	A	B	C	D	E	F	G	H	I
1	<b>Pieces summary</b>								
2	Shapes list							Total requested	Total obtained
3								200	200
4	Code	Description	R	DY	Thickness	Material	Quantity	Obtained	Grain
5	Pezzo	Cerchio.pgmx	203,27	203,27	18		200	200	

## 7.9.4 Sheets and pieces matrices mapping

The mapping report contains the following values:

	A	B	C	D	E	F	G	H	I	J
1	<b>Mapping</b>									
2	Type	Thickness		18						
3	<b>Statistical data</b>									
4	Number of patterns		Gross used		Net used		% Remainder		% Scrap	
5	2		11,52		6,49		0		43,66	
6										
7										
8										

## 7.9.5 Cutting diagrams

The cutting diagrams report contains a sheet for each sheet processed.

In this example 2 sheets have been used to execute the nesting of 200 circular pieces:

Sheet 1

	A	B	C	D	E	F	G	H							
1	<b>Cutting diagrams</b>														
2															
3	Pattern number	1 of 2													
4	Material code						5*1600*1200*18								
5	Description	Foglio(1)													
6	% Remainder	0 % Scrap		39,15											
7	Tot. area x 1	1,92		Tool area x 1	1,359										
8															
9															
10															
11															
12															
13															
14															
15															
16															
17															
18															
19															
20															
21															
22															
23															
24															
25															
26															
27															
28															
29															
30															
31	Code	Description	R	DY	Tot. Qty.	Inc.(%)	Done	To do							
32		Pezzo Cerchio.pgm		203,27	203,27	180	90	180 20							



Sheet 2

35	Pattern number	2 of 2						
36	Material code							1*1600*1200*18
37	Description	Foglio(1)						
38	% Remainder	0	% Scrap	66,2				
39	Tot. area x 1	1,92	Tool area x 1	0,755				
40								
41								
42								
43								
44								
45								
46								
47								
48								
49								
50								
51								
62								
63	Code	Description	R	DY	Tot. Qty.	Inc.(%)	Done	To do
64		Pezzo Cerchio.pgm	203,27	203,27	20	10	200	0

## 8 Materials magazine management

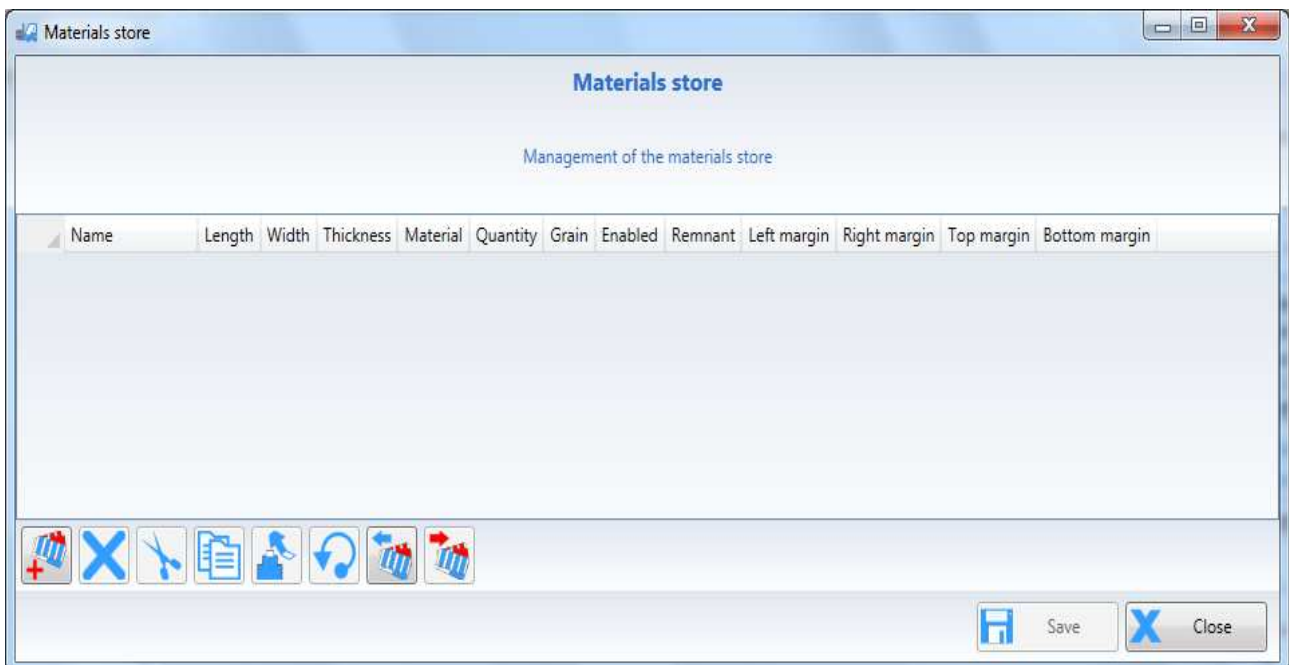
the materials magazine consists of a set of unfinished pieces used to produce the finished pieces.

The materials magazine is used mainly for nesting machining operations, but it may also be used in the future for other types of machining operations.

To access the "Materials magazine" application click on "Materials magazine" icon in the "Tools" menu.



The materials magazine interface looks like this:



The buttons on the bottom left can be used to:

- add new sheets
- delete existing sheets
- cut existing sheets
- copy existing sheets
- paste cut or copied sheets
- rotate the sheet
- import a materials magazine
- export a materials magazine

The materials magazine displayed when accessing this session is the last one that was saved.

For the meaning of the sheet data see **Errore. L'origine riferimento non è stata trovata.**

## 8.1 Add new sheets

To add new sheets use the "Add" button



Click on "Add" to add a new sheet in queue to the current materials magazine.

The name field will be "Sheet" for the first sheet entered "Sheet(1)" for the second sheet entered and so on.

The "Length, Width and Thickness" dimensions will be the default settings of the Nesting parameters.

The "Quantity" will be set as 1 and the sheet will have the "Enable" flag.

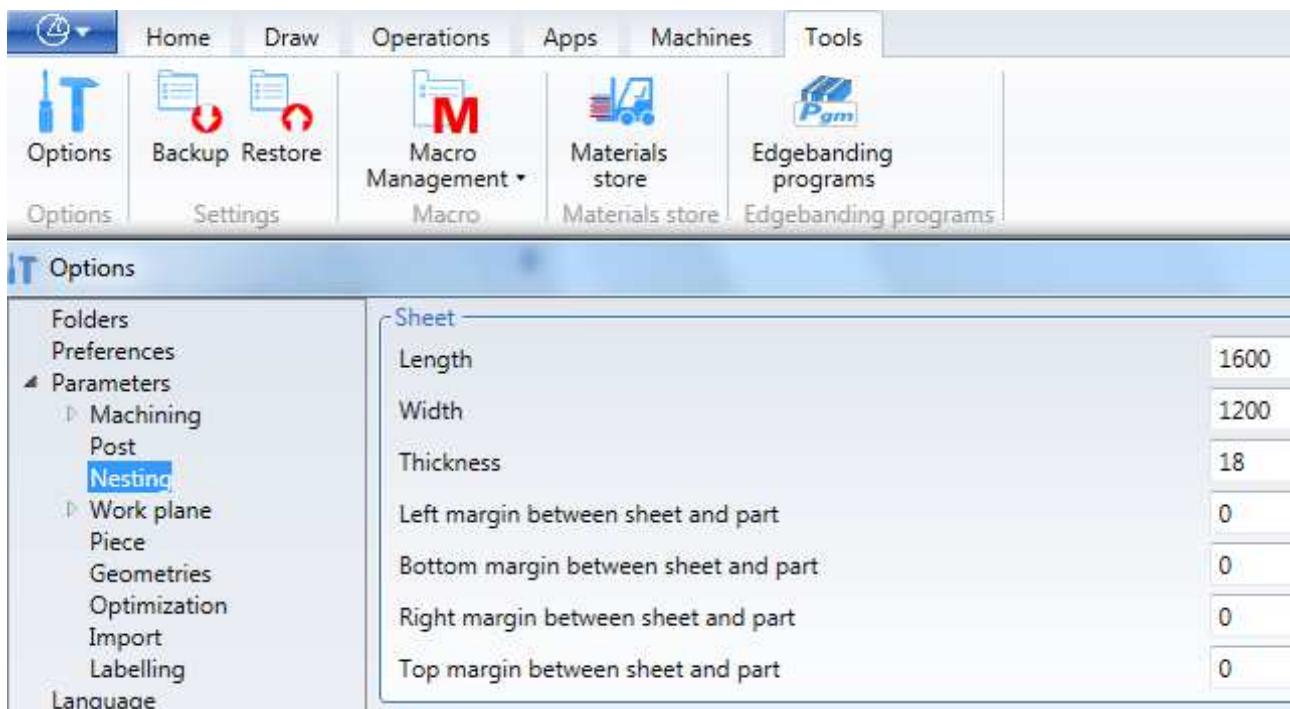
If the quantity is set as "-1" the sheet will be considered of unlimited quantity.



### 8.1.1 Default dimensions for Nesting sheets

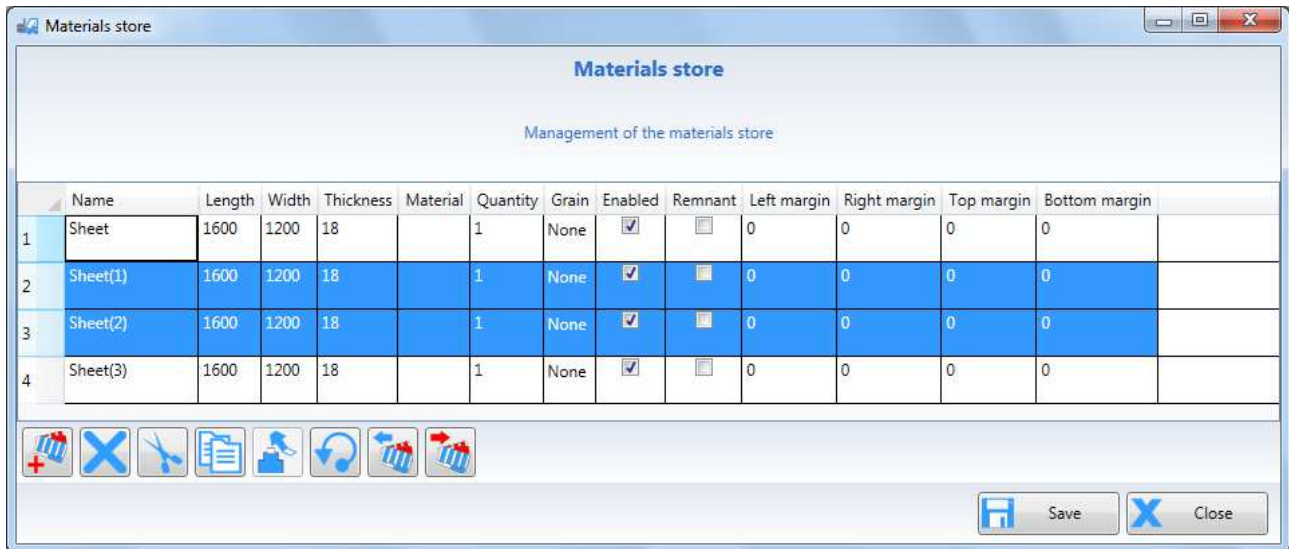
To set the default dimensions of a new Nesting sheet fill in the relative fields in the "Sheet" table displayed with:

"Tools-Options-Parameters-Nesting"



## 8.2 Delete existing sheets

To delete existing sheets, select one or more sheets (turning them blue) and then click on "Delete".



## 8.3 Cut existing sheets

To cut existing sheets, select one or more sheets (turning them blue) and then click on "Cut".



## 8.4 Copy existing sheets

To copy existing sheets, select one or more sheets (turning them blue) and then click on "Copy".



## 8.5 Paste cut or copied sheets

To paste previously cut or copied sheets, select the sheet below which they are to be pasted and then click on "Paste".



The screenshot shows a window titled 'Materials store' with the subtitle 'Management of the materials store'. It contains a table with the following data:

	Name	Length	Width	Thickness	Material	Quantity	Grain	Enabled	Remnant	Left margin	Right margin	Top margin	Bottom margin
1	Sheet	1600	1200	18		1	None	<input checked="" type="checkbox"/>	<input type="checkbox"/>	30	10	5	20
2	Sheet(3)	1600	1200	18		1	None	<input checked="" type="checkbox"/>	<input type="checkbox"/>	30	10	5	20
3	Sheet(1)	1600	1200	18		1	None	<input checked="" type="checkbox"/>	<input type="checkbox"/>	30	10	5	20
4	Sheet(2)	1600	1200	18		1	None	<input checked="" type="checkbox"/>	<input type="checkbox"/>	30	10	5	20

Below the table is a toolbar with icons for 'Paste', 'Cut', 'Copy', 'Paste', 'Rotate', 'Copy', and 'Paste'. At the bottom right, there are 'Save' and 'Close' buttons.

### 8.5.1 Copy-Paste Cells

To copy and paste cells between sheets:

- select one or more adjoining sheets from the same sheet
- press Ctrl-C (copy cells)
- select the equivalent cells in a different sheet
- press CTRL-V (paste cells)

## 8.6 Sheet rotation

Rotating the sheet switches the length value with the width value of the sheet.

To rotate existing sheets, select one or more sheets (turning them blue) and then click on "Rotate material".



The rotation not only switches the length with the width but also switches any grain from X to Y or from Y to X.

## 8.7 Import materials magazine

This operation is used to import a text file with the .csv extension containing the information on the sheets in the magazine.

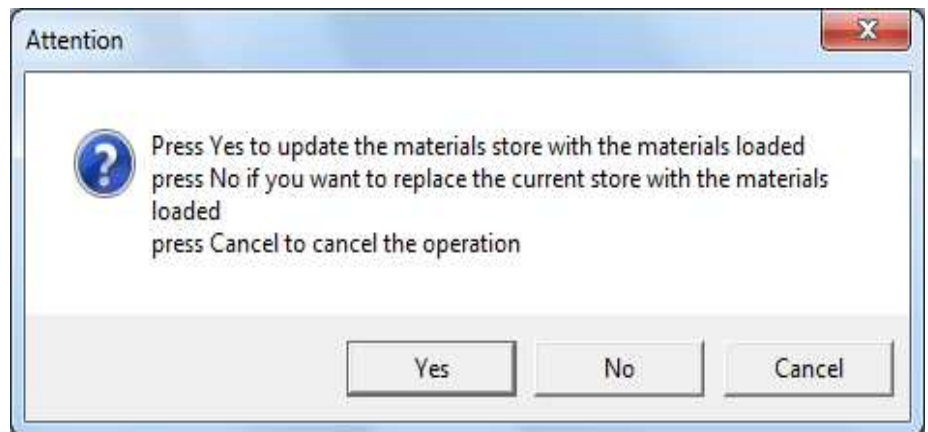
To import a materials magazine click on "Import materials magazine".  
Only a .csv file, that contains all the necessary information, must be selected.



Example of .csv file:

```
Name Piece;Length;Width;Thickness;Material;Quantity;Grain;Enable;
Remainder;LeftCollar;RightCollar;UpperCollar;LowerCollar;
Sheet;500;400;18;Mdf;5;None;1;0;10;20;30;40;
Sheet 1;500;500;18;Walnut;4;X;0;0;10;20;30;40;
Sheet 2;500;600;18;Mahogany;3;Y;1;1;10;20;30;40;
Sheet 3;3200;1600;18;Oak;3;None;1;1;10;20;30;40;
Sheet 4;3200;1600;18;Cherry;3;None;1;1;10;20;30;40;
Sheet 5;3200;1600;18;Ebony;3;None;1;1;10;20;30;40;
```

Once the file has been selected a message with the following options will appear:



This is the result of selecting "No" (the magazine has been replaced)

	Name	Length	Width	Thickness	Material	Quantity	Grain	Enabled	Remnant	Left margin	Right margin	Top margin	Bottom margin
1	Foglio	500	400	18	Mdf	5	None	<input checked="" type="checkbox"/>	<input type="checkbox"/>	10	20	30	40
2	Foglio 1	500	500	18	Noce	4	X	<input type="checkbox"/>	<input type="checkbox"/>	10	20	30	40
3	Foglio 2	500	600	18	Mogano	3	Y	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	10	20	30	40
4	Foglio 3	3200	1600	18	Quercia	3	None	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	10	20	30	40
5	Foglio 4	3200	1600	18	Ciliegio	3	None	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	10	20	30	40
6	Foglio 5	3200	1600	18	Ebano	3	None	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	10	20	30	40

Selecting "Yes" (the magazine is updated adding sheets to the existing ones).  
Sheets with the same name and dimensions will be added together.

## 8.8 Export materials magazine

This operation is used to export the current materials magazine in text files with the .csv extension formatted identically to the .csv file used as import.

To export the current materials magazine in .csv format, click on "Export materials magazine".



## 8.9 Save materials magazine

To save the materials magazine click on "Save".

The last magazine saved will be the one active when the "Materials magazine" session is opened once again.

The file containing the information of the materials magazine will be saved in the folder specified in the "Materials magazine folder" field in the session: Tools-Options-Folders.

The name of the file is always the same: "MaterialStore.mstx" and is written in a format that cannot be edited.

## 8.10 Unload materials

Once the nesting has been processed with the "Enable the use of materials magazine management" in the "Sheets" session (see 7.2.2.6) the materials used in the nesting machining can be unloaded.

To do this use the "unload materials" function in the "Tools-Options".



The sheets unloading will be confirmed with the following message:



The function will update the quantity of sheets used in the materials magazine.

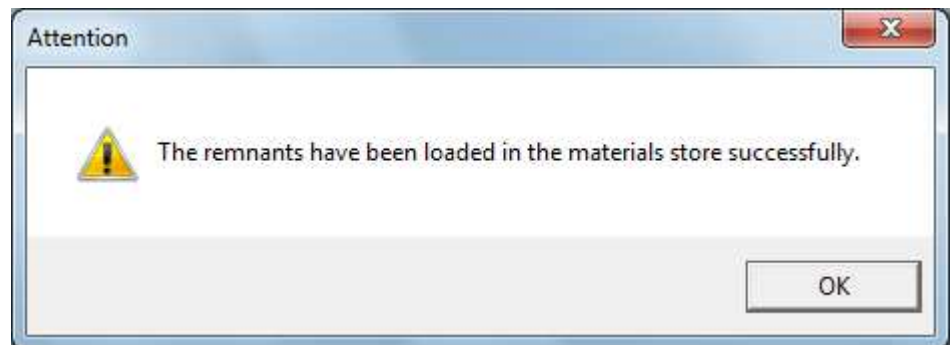
## 8.11 Load remainders

Once the nesting has been processed with the "Enable materials magazine management use" in the "Sheets" session (see 7.2.2.6) the remainders produced during the nesting machining can be loaded.

To do this use the "unload materials" function in the "Tools-Options".



The remainders loading will be confirmed with the following message:



The function will insert the remainders in the materials magazine.

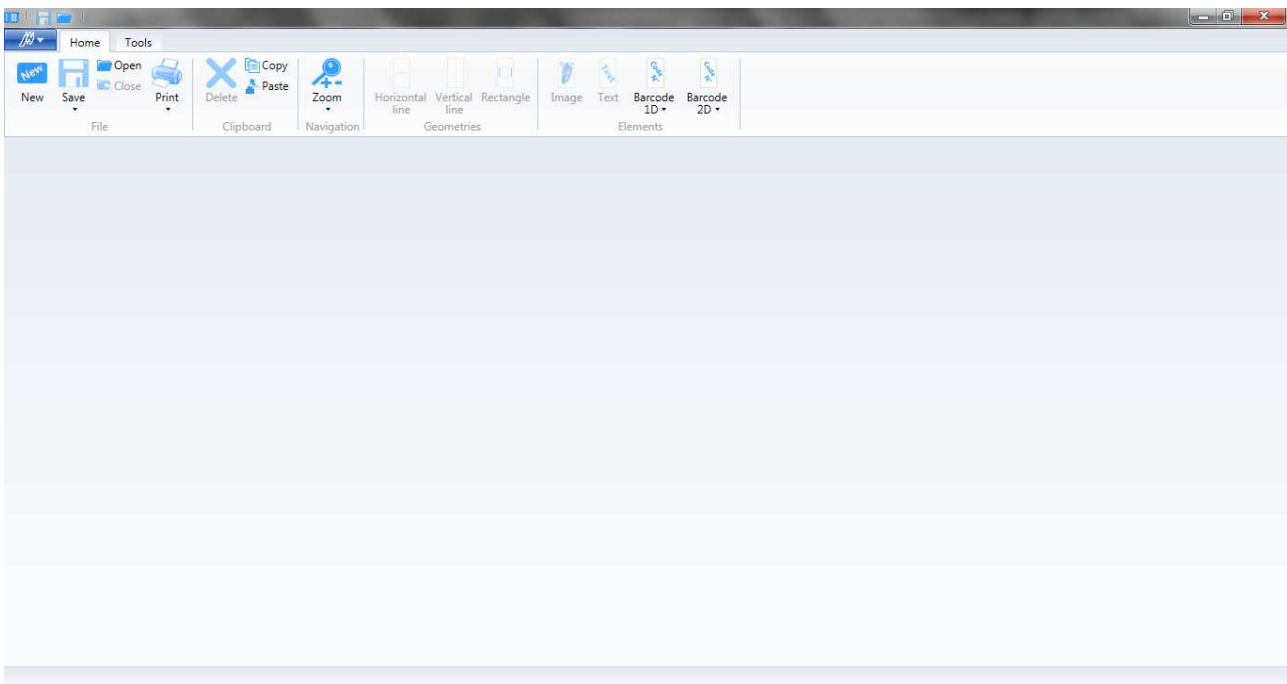


## 9 Labels database management with LabelEditor

The labels database contains all the information needed to define the labels used to label the pieces generated with the nesting.

LabelEditor can be launched from the installation directory, by clicking on “LabelEditor.exe”.

The LabelEditor interface looks as follows:



### 9.1 Create label

To create a new label use the “New” command in the “Home” menu → “File”.



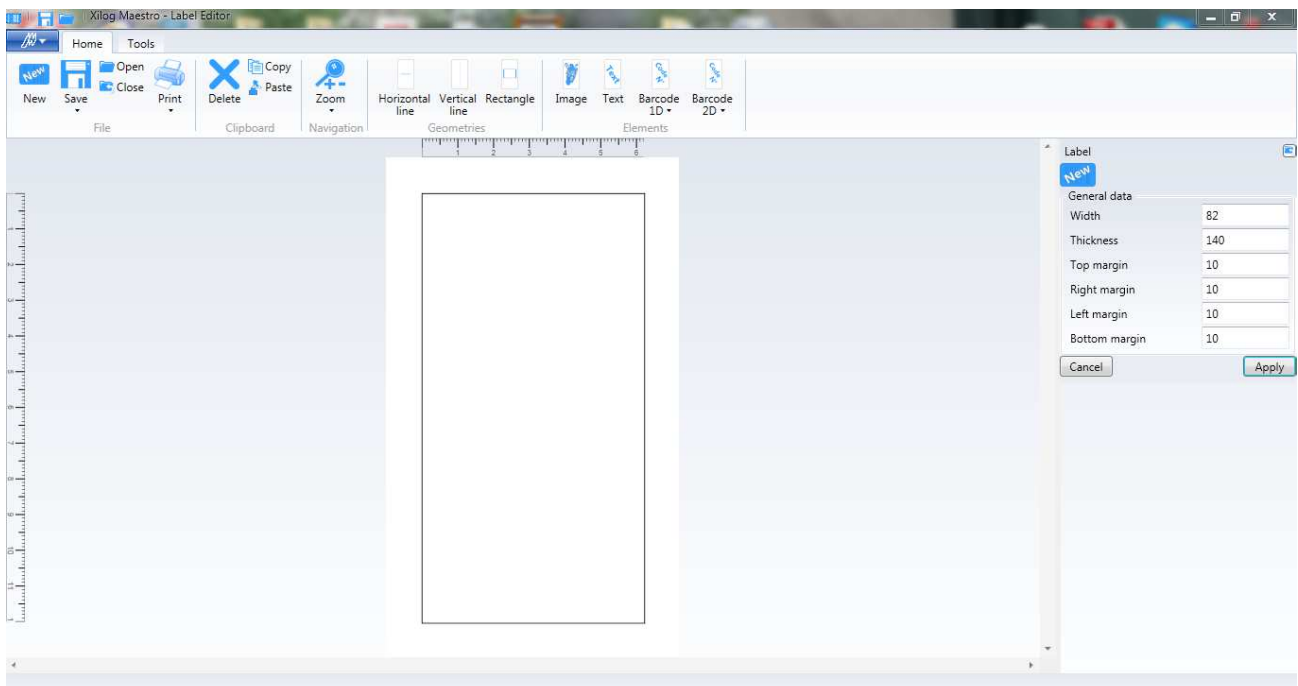
A list with the label attributes is displayed. The characteristic label data are:

- Width → width
- Height → height
- Top margin → upper edge
- Right margin → right edge
- Left margin → left edge
- Bottom margin → lower edge



The measurements are expressed in the unit of measurement set in the options.

To create the label press "Apply". To restore the initial values press "Cancel". Select a label to display its attributes. They can be changed and the changes rendered effective by pressing "Apply". Any graphic objects are maintained.



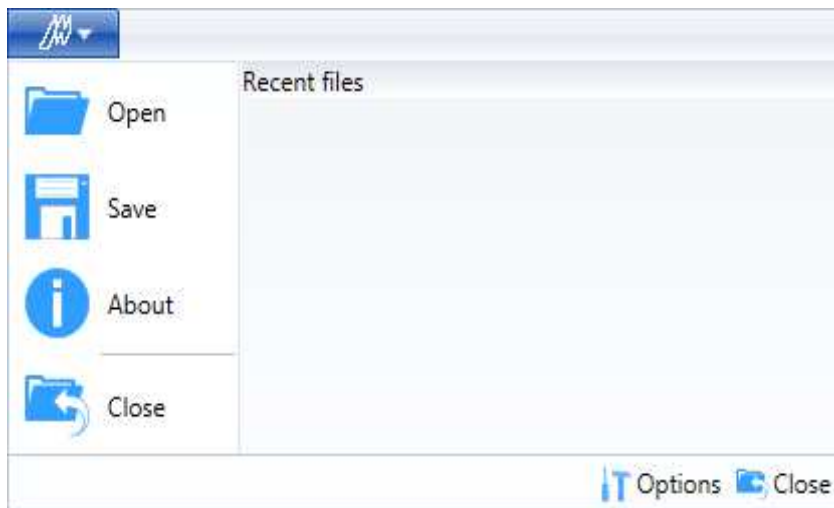
## 9.2 Open label

To open an existing label use the “Open” command in the “Home” menu → “File”.



The default path used to open the labels is defined in the “Options” → “File”.

A label can also be opened in the office button. The ones saved recently will appear in the list to be opened directly, without being searched.

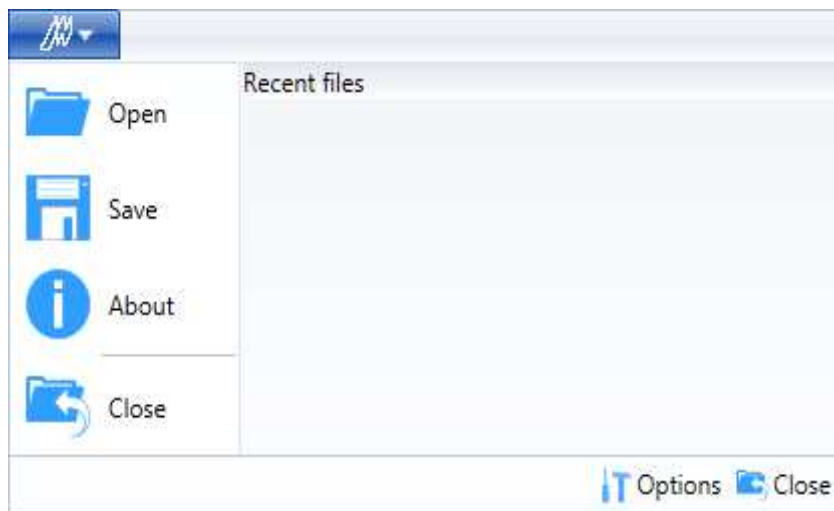


## 9.3 Close label

To close the current label use the “Close” command in the “Home” menu → “File”.



A label can also be closed from the office button.



## 9.4 Save label

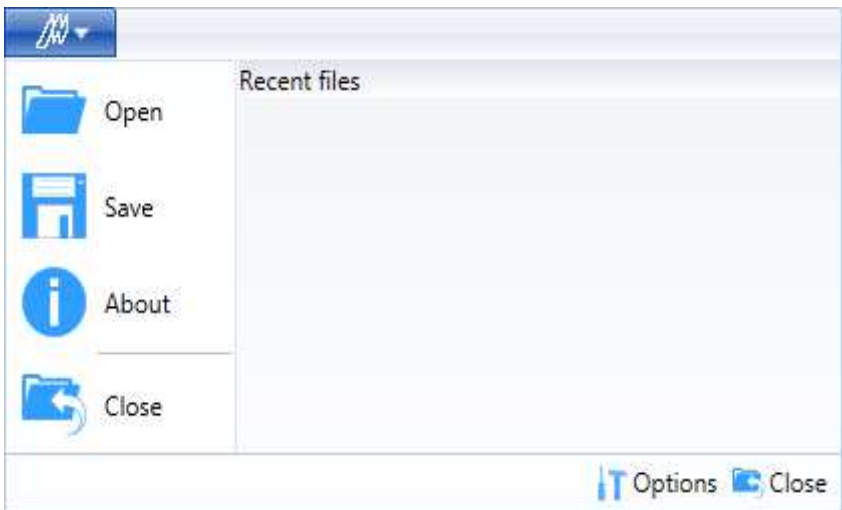
To save the current label use the “Save” command in the “Home” menu → “File”. If it is a label without a name yet, specify the label path and name.



To save the label with another name use the “Save with name” command in the “Home” menu → “File”.



A label can also be saved from the office button.



## 9.5 Print label

### 9.5.1 Print label on Zebra printer

To print the current label on the printer use the “Print zebra” command in the “Home” menu→“File”.



### 9.5.2 Print label on office printer

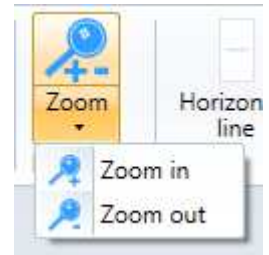
To print the current label on the printer based on the matrix diagram defined in the options, use the "Print office" command in the “Home” menu→“File”.



## 9.6 Tools

### 9.6.1 Zoom

Zooms in and out of the label display with the “Zoom” command in the “Home” menu → “Navigation”. It can also be executed with the mouse scroll when the mouse is positioned inside the label.



### 9.6.2 Pan

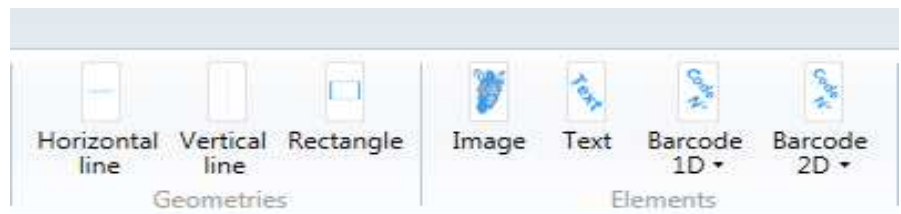
Once the label is selected, keep the left mouse key pressed to move the label with the mouse.

## 9.7 Drawing objects management

### 9.7.1 Add object

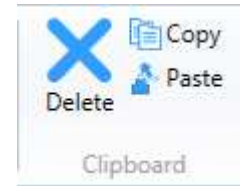
The following objects can be added to the label:

- Geometries
  - o Horizontal line
  - o Vertical line
  - o Rectangle
- Elements
  - o Image
  - o Text
  - o Barcode 1D
    - Code 39
    - Code 93
    - Code 128
  - o Barcode 2D
    - Aztec
    - Datamatrix
    - QR
    - PDF417



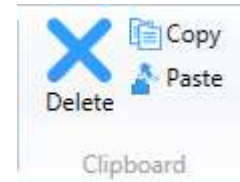
## 9.7.2 Delete object

Use the “Cancel” command in the “Home” menu → “Clipboard” to delete the element selected. The command is also available on the keyboard with the "Canc" key.



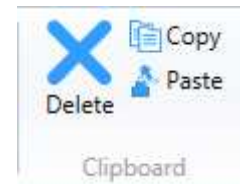
## 9.7.3 Copy object

Use the “Copy” command in the “Home” menu → “Clipboard” to copy the element selected. The command is also available on the keyboard with the “Ctrl”+“c” combination.



## 9.7.4 Paste object

Use the “paste” command in the “Home” menu → “Clipboard” to paste the previously copied element. The command is also available on the keyboard with the “Ctrl”+“v” combination.



## 9.8 Drawing objects

### 9.8.1 Horizontal line

Use the "Horizontal line" command in the "Home" menu → "Geometries" to create a horizontal line.



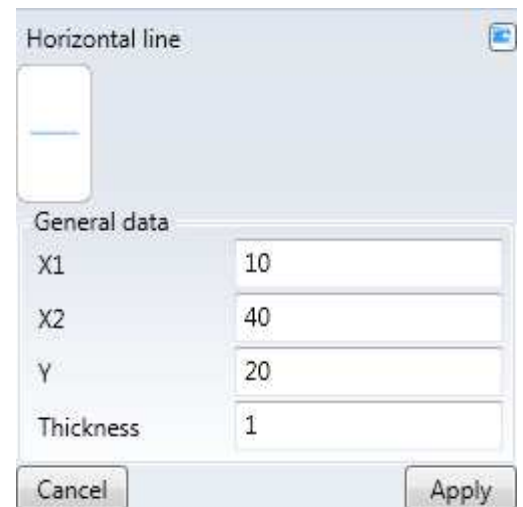
A list with the horizontal line attributes is displayed. The characteristic data of the horizontal line are:

- X1 → initial X coordinate
- X2 → final X coordinate
- Y → Y coordinate
- Thickness → line thickness

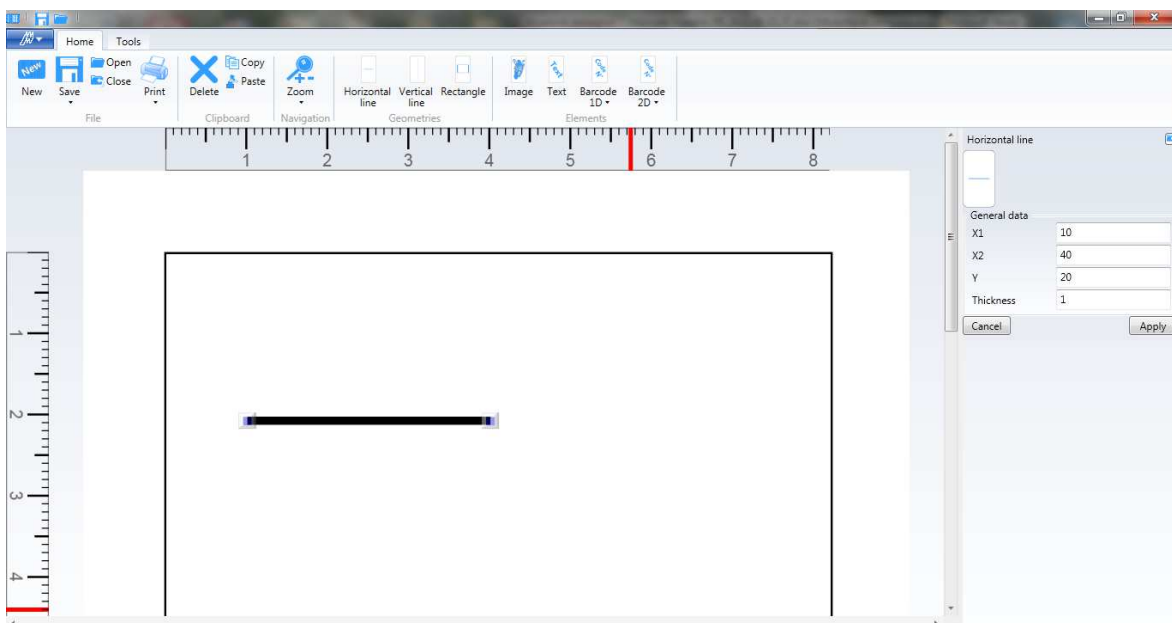
X1 must have a lower value than X2.

The thickness must have a higher value or the same value as the minimum value set in "Options" → "Parameters".

The measurements are expressed in the unit of measurement set in the options.



Press "Apply" to add the horizontal line to the label. To restore the initial values press "Cancel". Select the horizontal line graphic object in the label to display its list of attributes. They can be changed and the changes rendered effective by pressing "Apply". The element can be moved also by selecting it and dragging it with the mouse. The element can be resized also by selecting and dragging the end points with the mouse.





## 9.8.2 Vertical line

Use the "Vertical line" command in the "Home" menu → "Geometries" to create a vertical line.



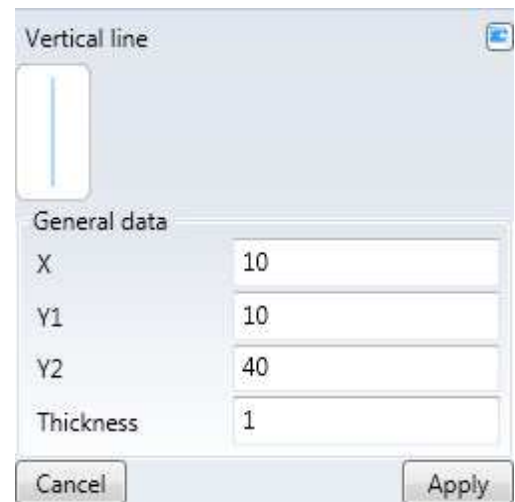
A list with the vertical line attributes is displayed. The characteristic data of the horizontal line are:

- X → X coordinate
- Y1 → initial Y coordinate
- Y2 → final Y coordinate
- Thickness → line thickness

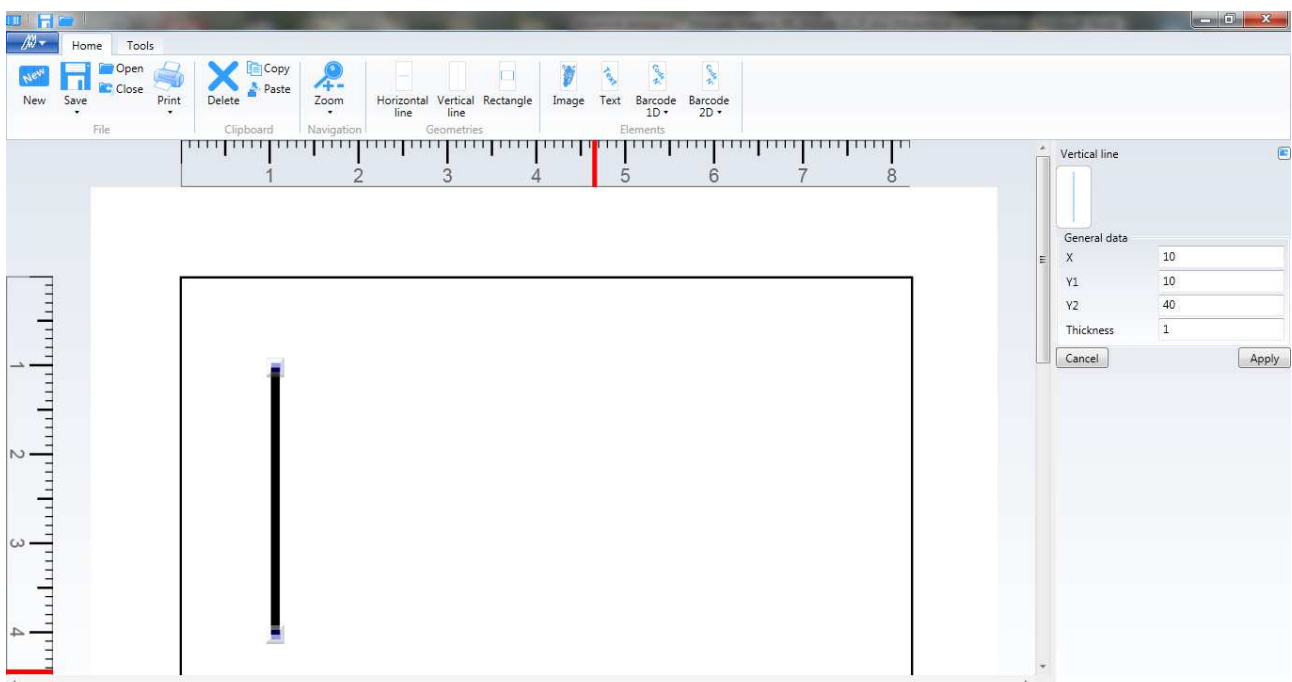
Y1 must have a lower value than Y2.

The thickness must have a higher value or the same value as the minimum value set in "Options" → "Parameters".

The measurements are expressed in the unit of measurement set in the options.

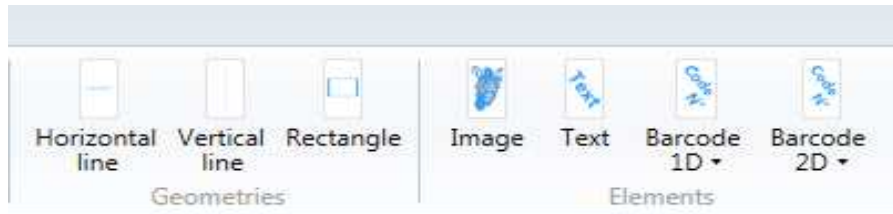


Press "Apply" to add the vertical line to the label. To restore the initial values press "Cancel". Select the vertical line graphic object in the label to display its list of attributes. They can be changed and the changes rendered effective by pressing "Apply". The element can be moved also by selecting it and dragging it with the mouse. The element can be resized also by selecting and dragging the end points with the mouse.



## 9.8.3 Rectangle

Use the "Rectangle" command in the "Home" menu → "Geometries" to create a rectangle.



A list with the rectangle attributes is displayed. The characteristic data of the rectangle are:

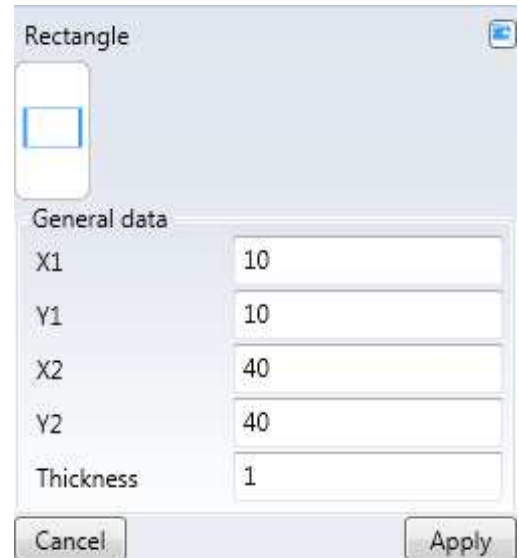
- X1 → initial X coordinate
- X2 → final X coordinate
- Y1 → initial Y coordinate
- Y2 → final Y coordinate
- Thickness → line thickness

X1 must have a lower value than X2.

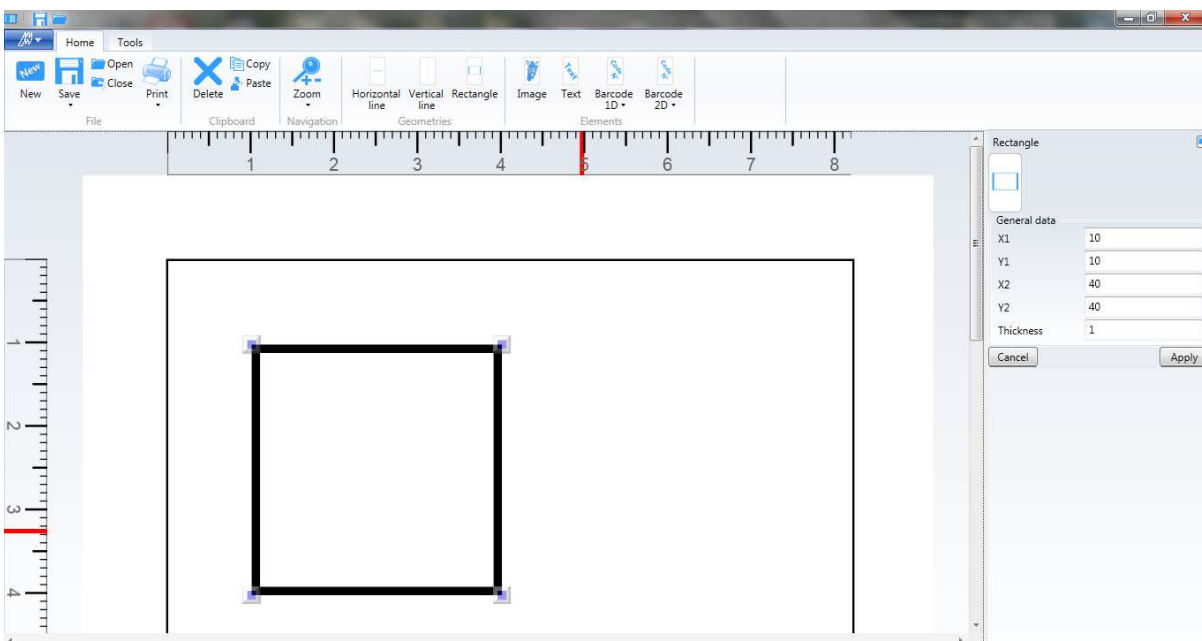
Y1 must have a lower value than Y2.

The thickness must have a higher value or the same value as the minimum value set in "Options" → "Parameters".

The measurements are expressed in the unit of measurement set in the options.

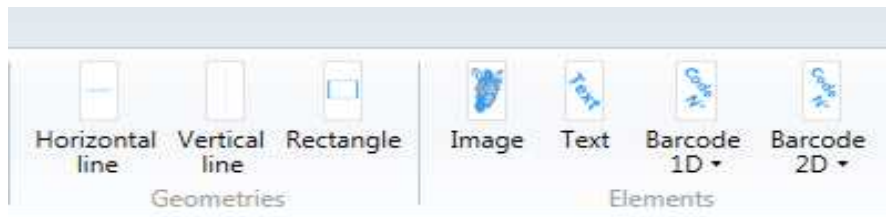


Press "Apply" to add the rectangle to the label. To restore the initial values press "Cancel". Select the rectangle graphic object in the label to display its list of attributes. They can be changed and the changes rendered effective by pressing "Apply". The element can be moved also by selecting it and dragging it with the mouse. The element can be resized also by selecting and dragging the end points with the mouse.



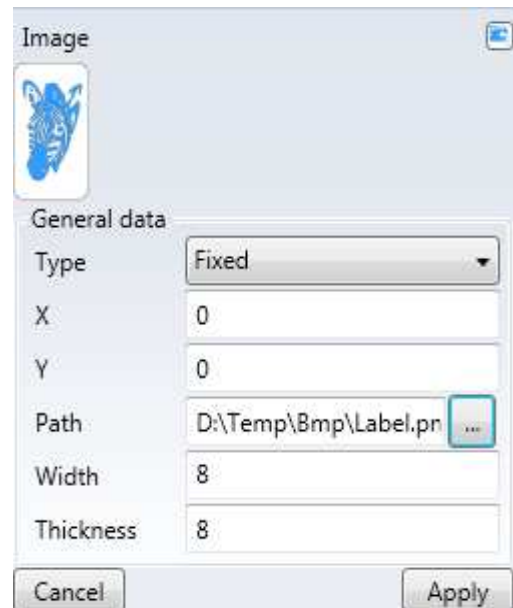
## 9.8.4 Image

Use the "Image" command in the "Home" menu → "Elements" to create an image.



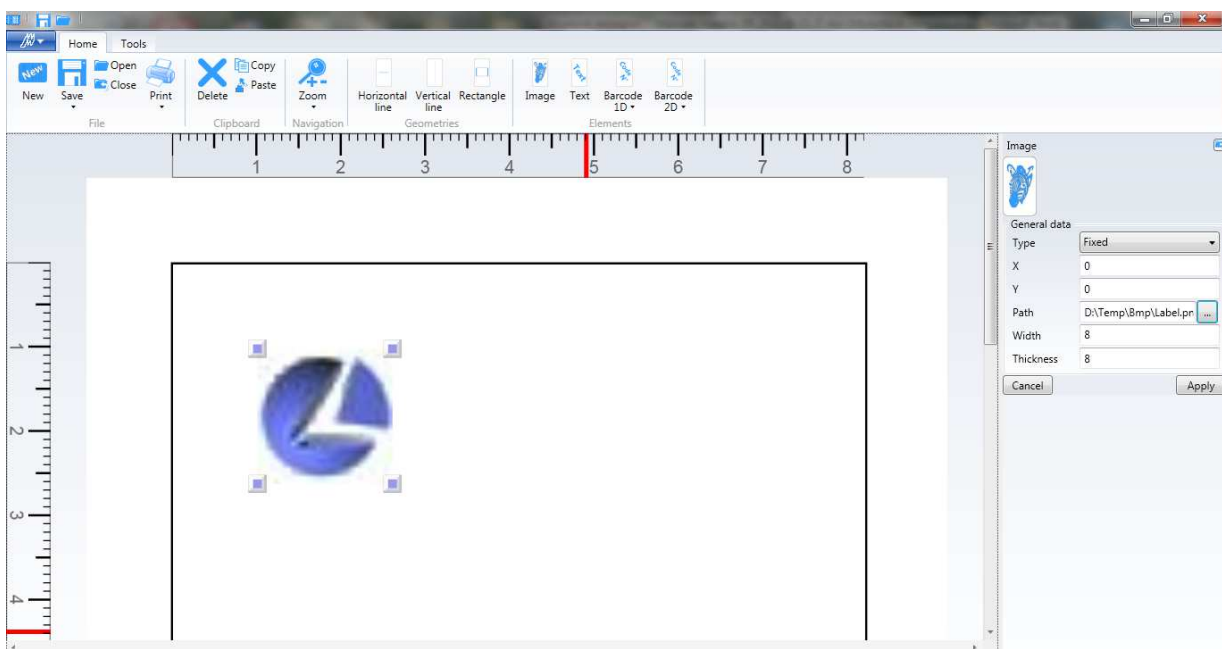
A list with the image attributes is displayed. The characteristic data of the image are:

- Type: fixed, parametric
- X → initial X coordinate
- Y → initial Y coordinate
- Path → image file name defined with the complete path
- Width → image width
- Height → image height



The measurements are expressed in the unit of measurement set in the options.

Press "Apply" to add the image to the label. To restore the initial values press "Cancel". Select the image graphic object in the label to display its list of attributes. They can be changed and the changes rendered effective by pressing "Apply". The element can be moved also by selecting it and dragging it with the mouse. The element can be resized also by selecting and dragging the end points with the mouse.



### 9.8.4.1 Fixed

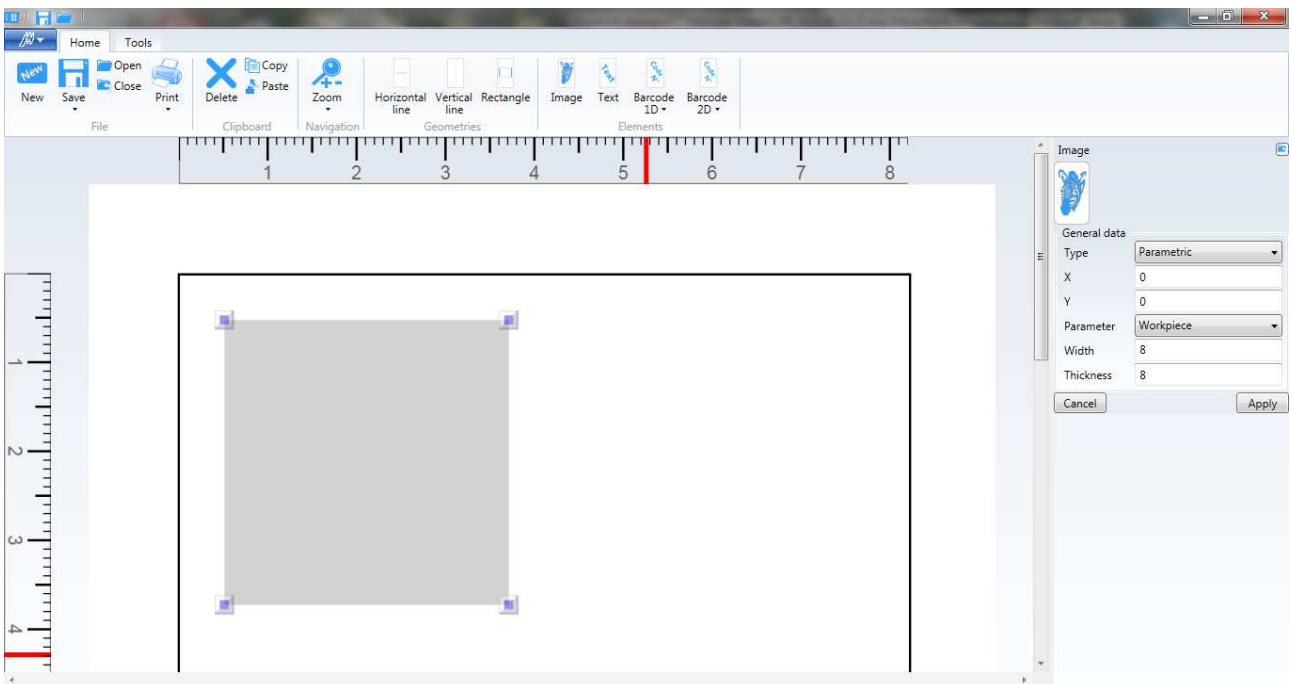
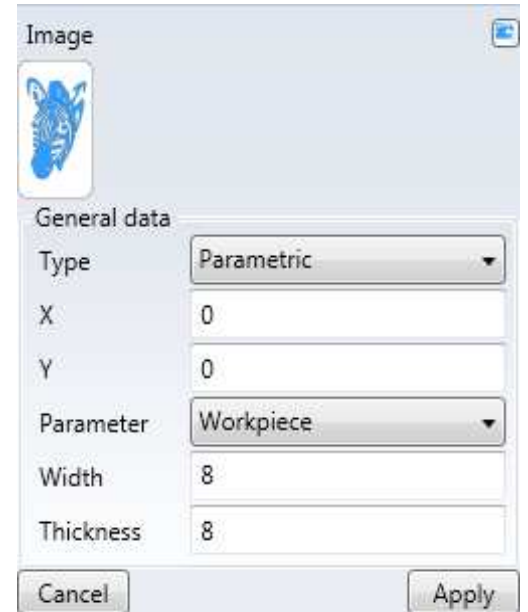
If the image is "Fixed" it means that the image is displayed in the label as it will be printed.

### 9.8.4.2 Parametric

In this case an additional "Parameter" attribute is displayed in the list of image attributes.

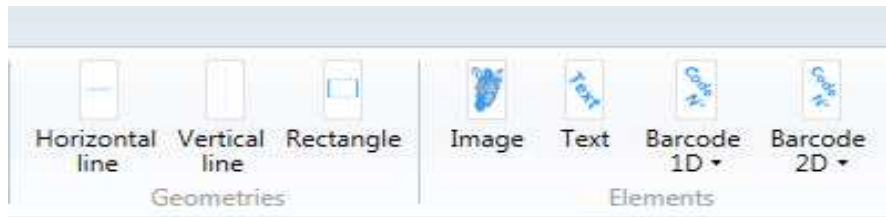
If the image is "Parametric" it means that an empty image is displayed in the label. The content of the image is filled during printing with the piece data from the nesting based on the type of parameter:

- Piece → the piece image is loaded
- FieldN (1 ≤ N ≤ 20) → parametric field, the image corresponding to the index N parameter value is loaded (file path)



## 9.8.5Text

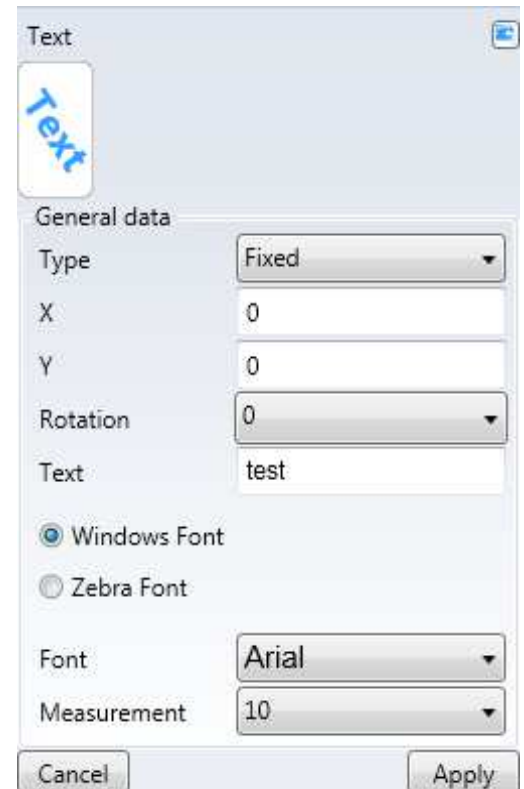
Use the "Text" command in the "Home" menu → "Elements" to create a text.



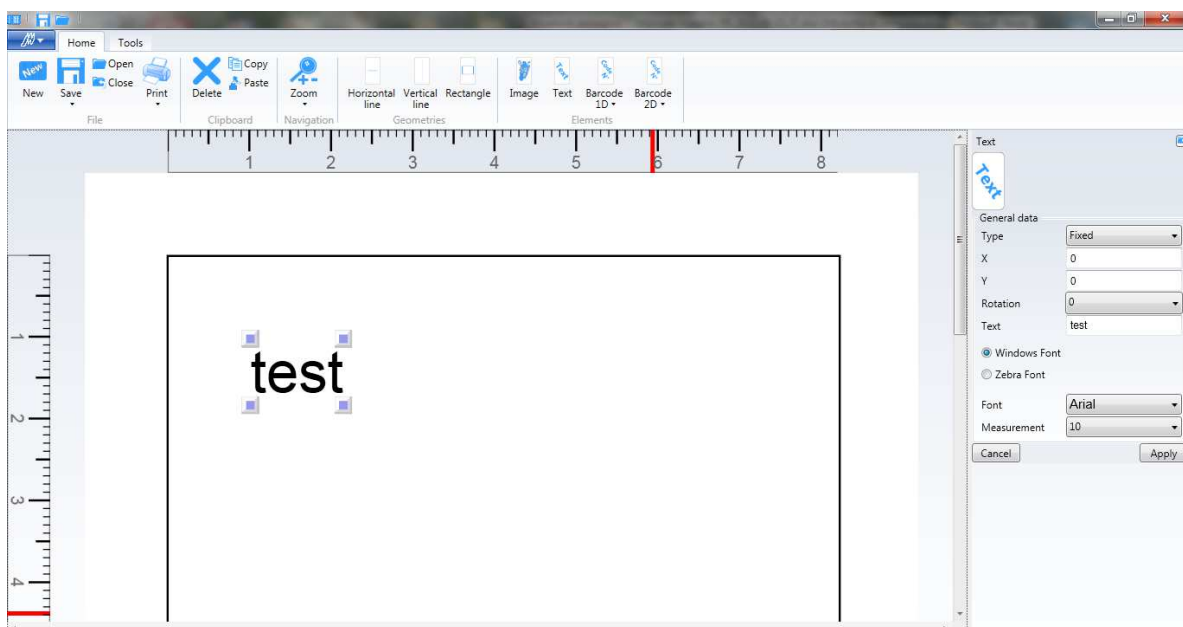
A list with the text attributes is displayed. The characteristic data of the text are:

- Type: fixed, parametric, date or time
- X → initial X coordinate
- Y → initial Y coordinate
- Rotation → text rotation angle
- Text → text
- Windows font → displays Windows font
- Zebra font → displays Zebra font
- Font → name of the font
- Measurement → font dimension (if Windows font it is expressed in point, if Zebra font it is a multiplicative factor of the base dimension of the font selected)

The other measurements are expressed in the unit of measurement defined in the options.



Press "Apply" to add the text to the label. To restore the initial values press "Cancel". Select the horizontal line graphic object in the label to display its list of attributes. They can be changed and the changes rendered effective by pressing "Apply". The element can be moved also by selecting it and dragging it with the mouse.



A Windows font or a Zebra font can be used for a text. The Zebra fonts are: ZebraA, ZebraB (only upper case characters), ZebraD, ZebraE, ZebraF, ZebraH (only upper case characters) and are fonts that reside in the printers. In the label the Zebra fonts are displayed with a grey background to distinguish them from the Windows font.



As they are not available as system fonts the zebra font text is displayed accurately in terms of space but not in style of font. This means that prints with zebra font texts on a zebra printer use texts with similar fonts and with the same space. Prints on an office printer, which has no zebra font, uses texts with the same font and with the same space. Unicode characters cannot be printed with the zebra font.

The Windows fonts are those contained in the "Windows\Fonts" system folder and, the ones that are used in the label texts, to be printed correctly, must have been previously downloaded in the printer memory with a Zebra utility. Once downloaded in the printer a file with the "FNT" extension is allocated to the downloaded font. In the printer, by default there is already the font Windows Swiss721 whose file is called "TT0003M\_.FNT". To use it in the "Windows\Fonts" folder there must be the font "Swiss721 BT" (file called "tt0003m\_0.ttf"). All this information must be set in the options of the zebra printer parameters. With the font Swiss721 unicode characters can be printed (e.g. texts in Greek, Russian, Turkish), to print Chinese; Japanese or Korean texts other fonts are required. They can be downloaded in "Windows\Fonts" and in the printer memory (Simplified Chinese - ANMDS.TTF, Traditional Chinese - ANMDT.TTF, Japanese - ANMDJ.TTF, Korean - ANMDK.TTF).

### 9.8.5.1 Fixed

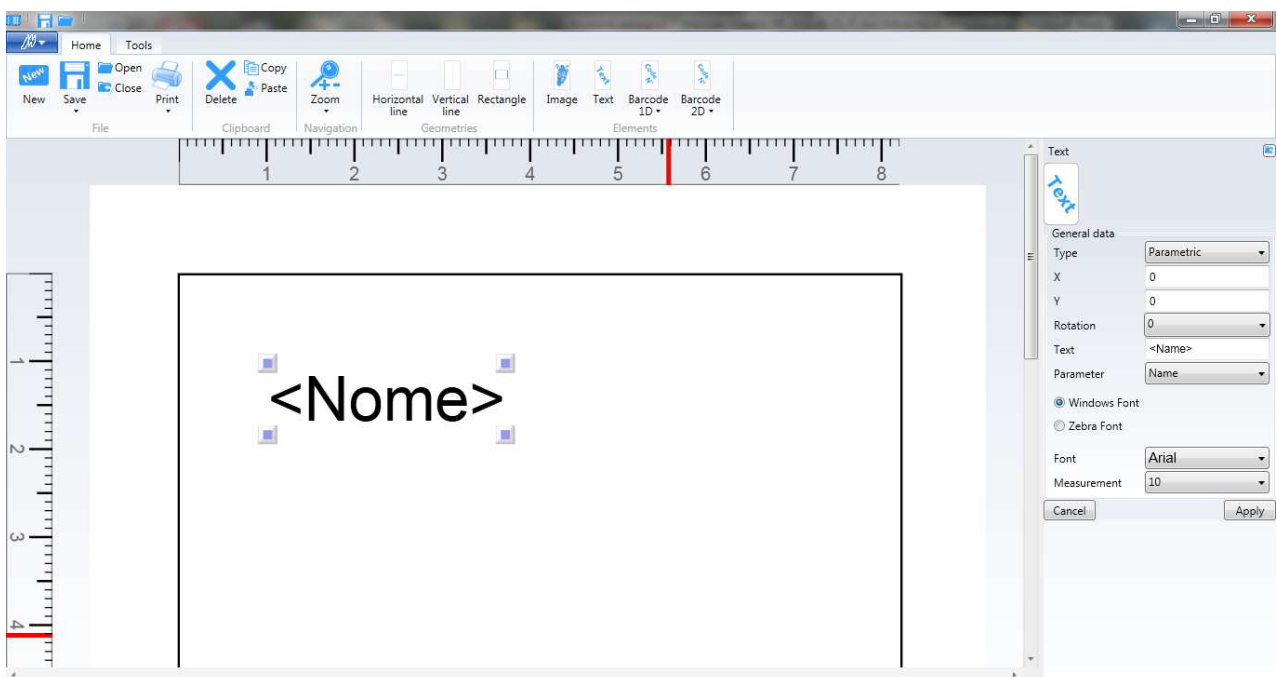
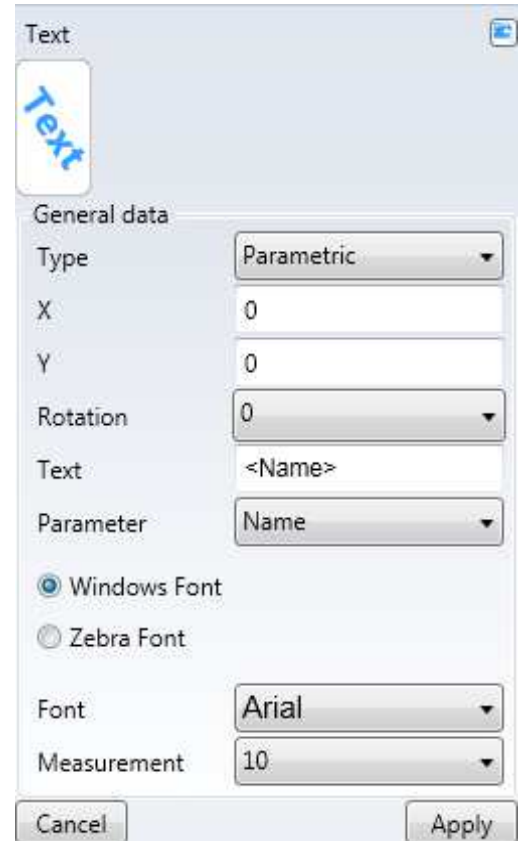
If the text is "Fixed" it means that the text is displayed in the label as it will be printed.

### 9.8.5.2 Parametric

In this case an additional "Parameter" attribute is displayed in the list of text attributes.

If the text is "Parametric" it means that a temporary text is displayed in the label. The content of the text is filled during printing with the piece data from the nesting based on the type of parameter:

- Name → the name is loaded
- Dimension X → the width is loaded
- Dimension Y → the height is loaded
- Dimension Z → the thickness if loaded
- Material → the material is loaded
- Program side 1 → the side 1 machining program code is loaded
- Program side 2 → the side 2 machining program code is loaded
- Program side 3 → the side 3 machining program code is loaded
- Program side 4 → the side 4 machining program code is loaded
- FieldN (1<=N<=20) → parametric field, the index N parameter value is loaded

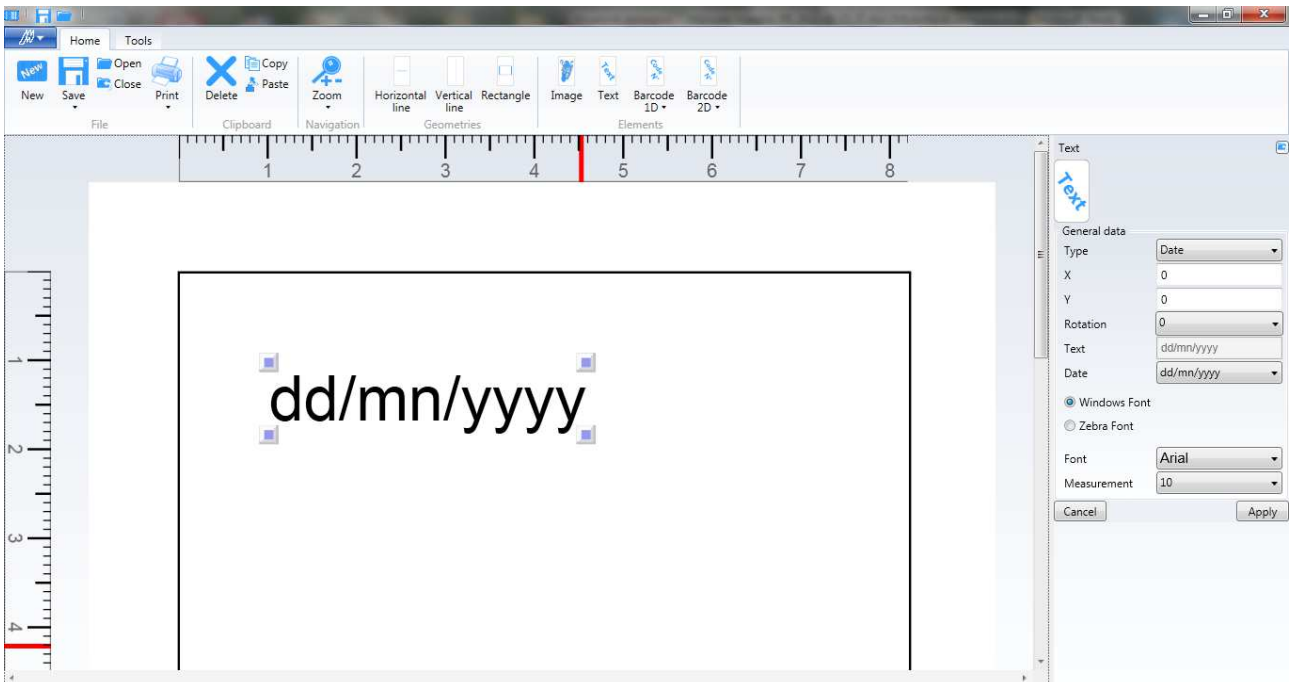
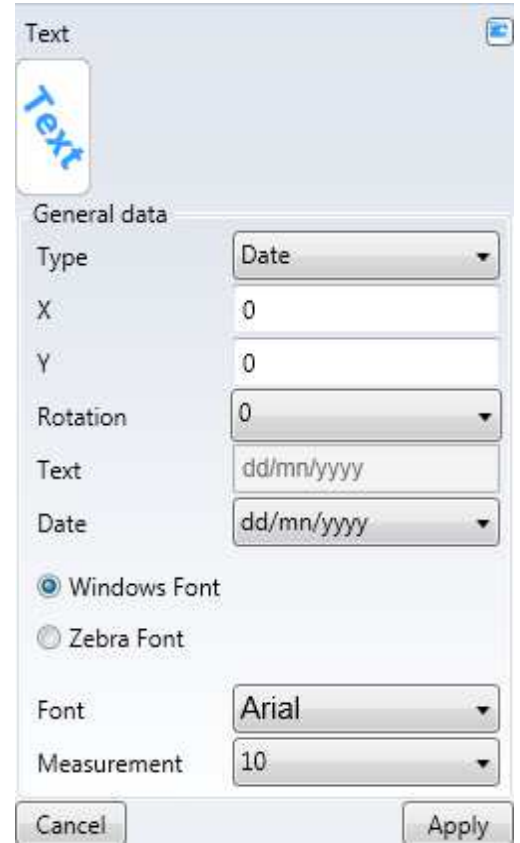


### 9.8.5.3 Date

In this case an additional "Date" attribute is displayed in the list of text attributes.

If the text is "Date" it means that a temporary text is displayed in the label. The content of the text is filled during printing with the current system date with the format selected:

- dd/mn/yyyy
- yyyy/mn/dd
- ddmnyyyy
- yyyymndd



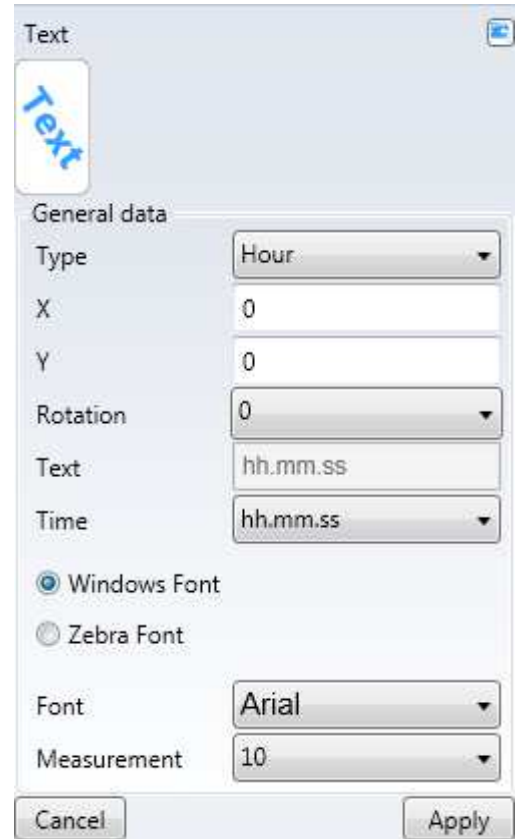


### 9.8.5.4 Time

In this case an additional "Time" attribute is displayed in the list of text attributes.

If the text is "Time" it means that a temporary text is displayed in the label. The content of the text is filled during printing with the current system time with the format selected:

- hh.mm.ss (24h)
- hh:mm:ss (24h)
- hh.mm.ss XX → XX assumes the "AM" or "PM" value (12h)
- hh:mm:ss XX → XX assumes the "AM" or "PM" value (12h)

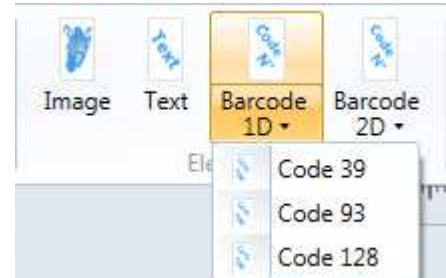


## 9.8.6 Barcode

### 9.8.6.1 Unidimensional

#### 9.8.6.1.1 Code 39

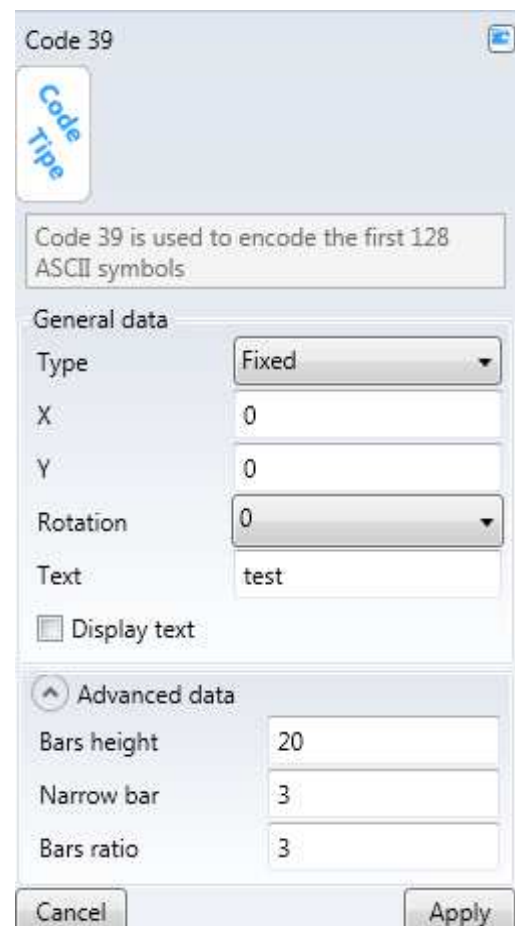
Use the “Code 39” command in the “Home” menu → “Elements” → “Barcode 1D” to create a code 39.



A list with the code 39 attributes is displayed. The characteristic data of the code 39 are:

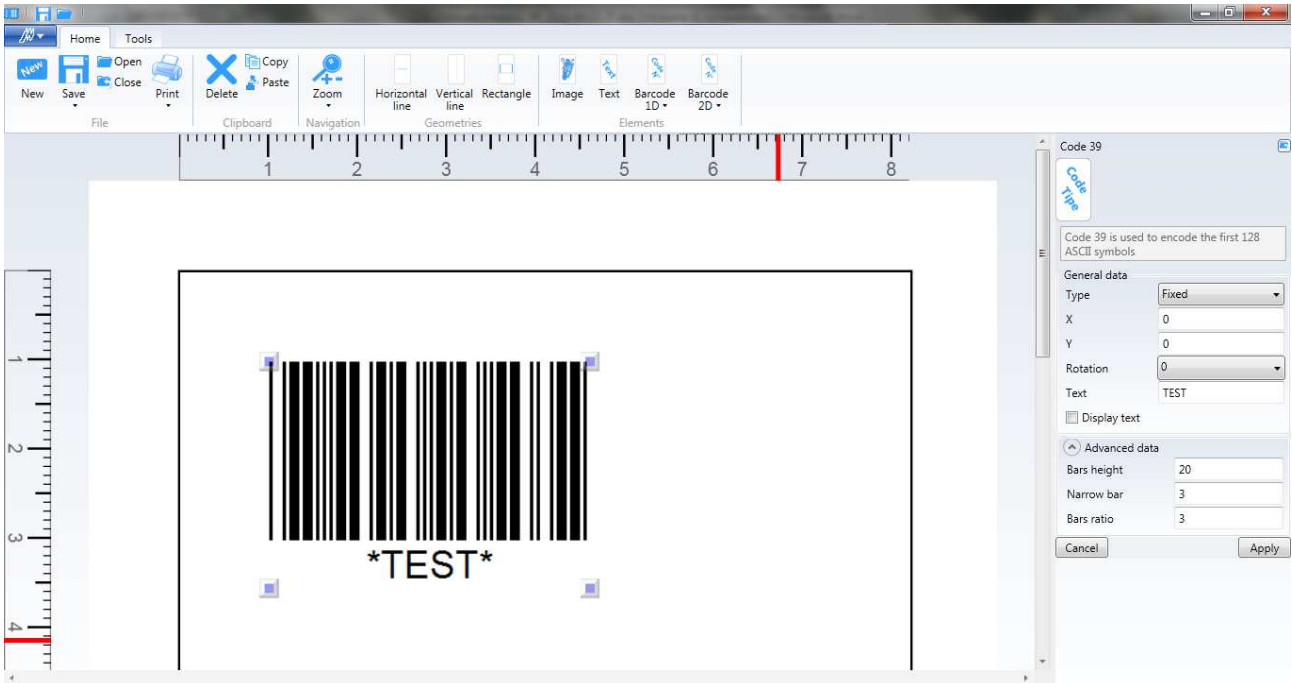
- Type: fixed, parametric, date or time (same as for the "Text" element)
- X → initial X coordinate
- Y → initial Y coordinate
- Rotation → code 39 rotation angle
- Text → text
- Display text → displays the text under the code
- Bars height → height of the bars
- Narrow bar → width of the bar expressed in points
- bars ratio → ratio between the bars

The other measurements are expressed in the unit of measurement defined in the options.



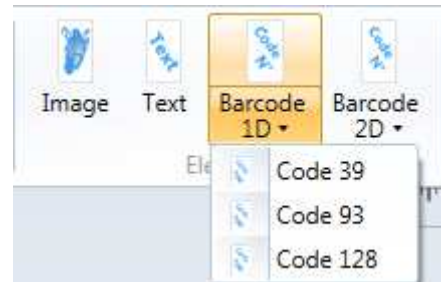
The code 39 is used to encode the first 128 symbols ASCII.

Press "Apply" to add the code 39 to the label. To restore the initial values press "Cancel". Select the code 39 graphic object in the label to display its list of attributes. They can be changed and the changes rendered effective by pressing "Apply". The element can be moved also by selecting it and dragging it with the mouse. The element can be resized also by selecting and dragging the end points with the mouse.



### 9.8.6.1.2 Code 93

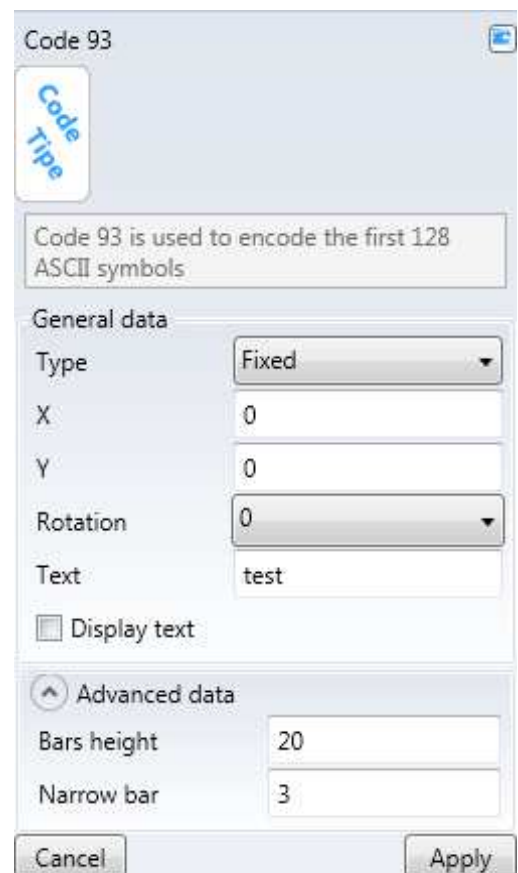
Use the “Code 93” command in the “Home” menu → “Elements” → “Barcode 1D” to create a code 93.



A list with the code 93 attributes is displayed. The characteristic data of the code 93 are:

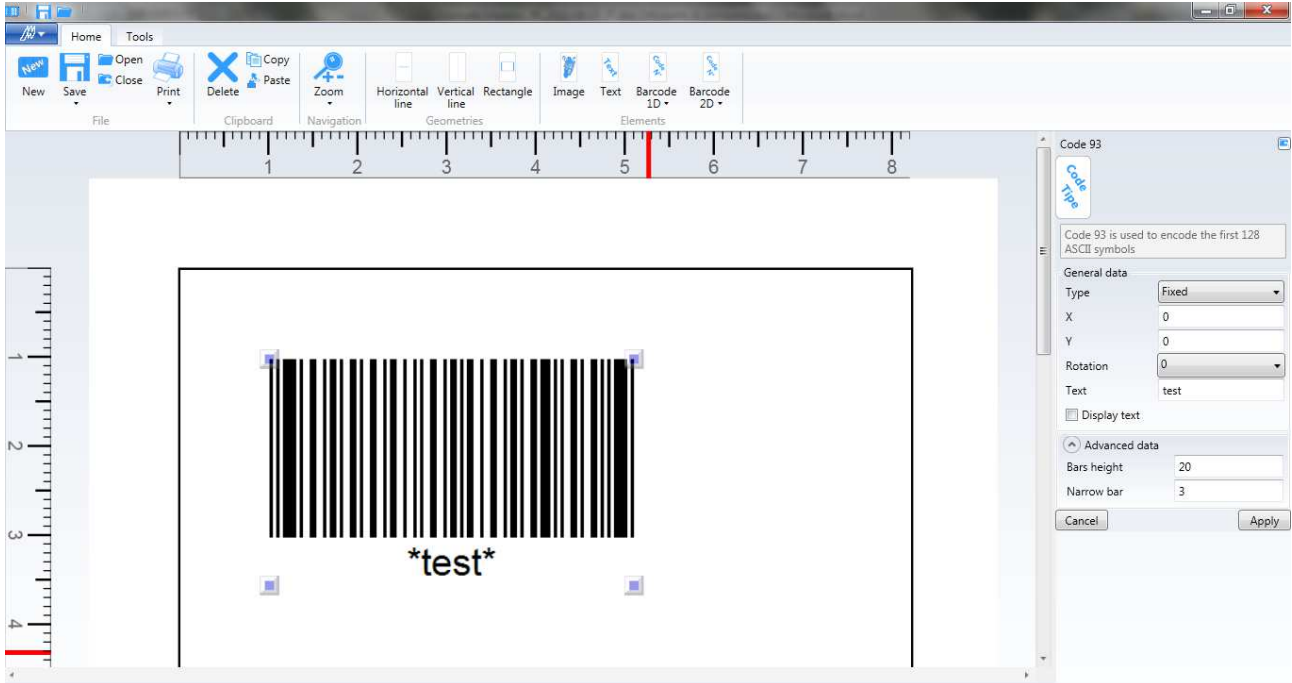
- Type: fixed, parametric, date or time (same as for the "Text" element)
- X → initial X coordinate
- Y → initial Y coordinate
- Rotation → code 93 rotation angle
- Text → text
- Display text → displays the text under the code
- Bars height → height of the bars
- Narrow bar → width of the bar expressed in points

The other measurements are expressed in the unit of measurement defined in the options.



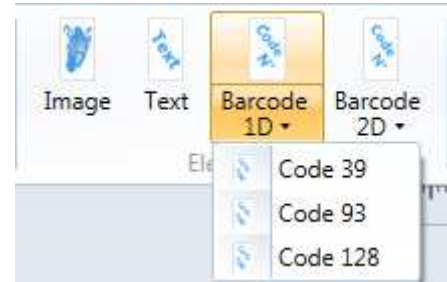
The code 93 is used to encode the first 128 symbols ASCII.

Press "Apply" to add the code 93 to the label. To restore the initial values press "Cancel". Select the code 93 graphic object in the label to display its list of attributes. They can be changed and the changes rendered effective by pressing "Apply". The element can be moved also by selecting it and dragging it with the mouse. The element can be resized also by selecting and dragging the end points with the mouse.



### 9.8.6.1.3 Code 128

Use the “Code 128” command in the “Home” menu → “Elements” → “Barcode 1D” to create a code 128.

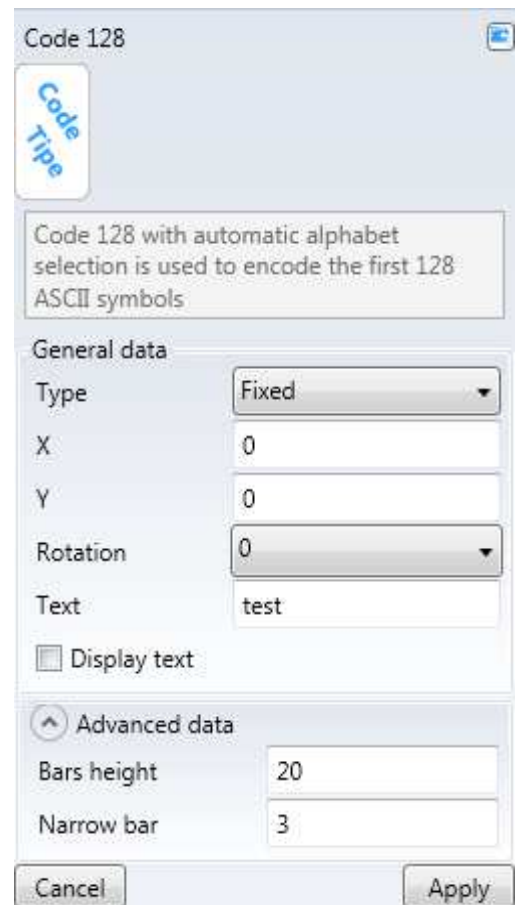


A list with the code 128 attributes is displayed. The characteristic data of the code 128 are:

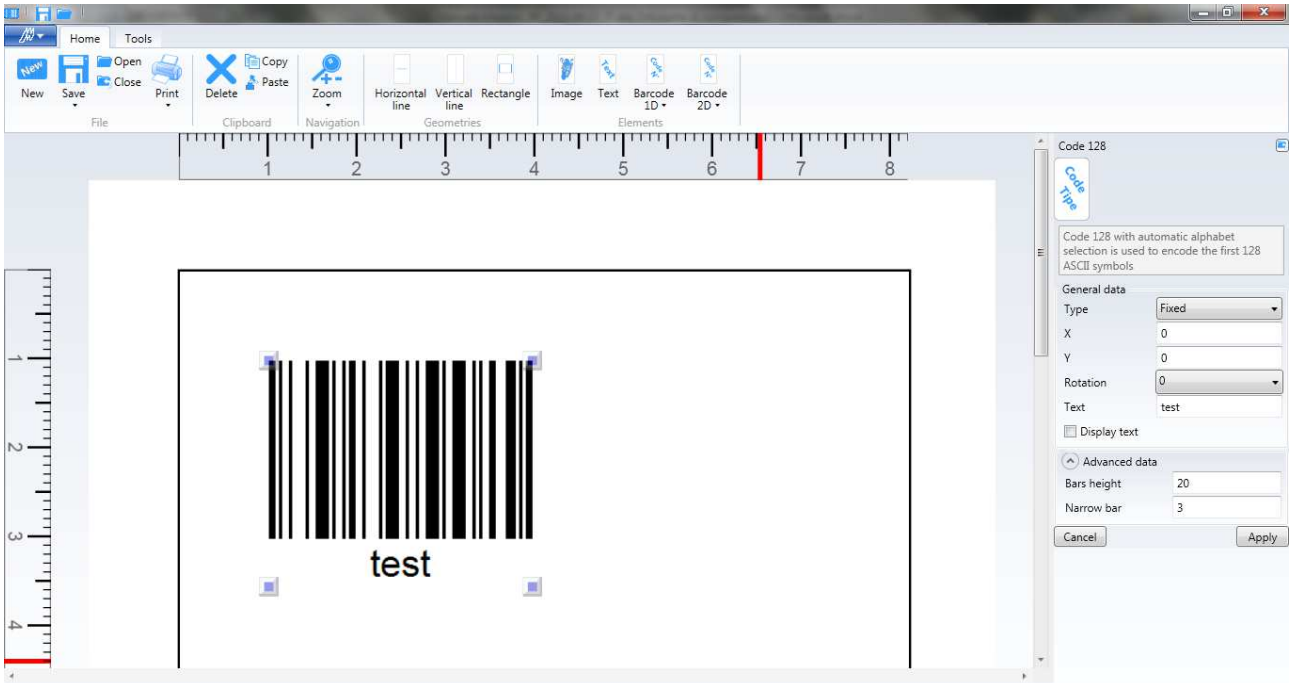
- Type: fixed, parametric, date or time (same as for the "Text" element)
- X → initial X coordinate
- Y → initial Y coordinate
- Rotation → code 128 rotation angle
- Text → text
- Display text → displays the text under the code
- Bars height → height of the bars
- Narrow bar → width of the bar expressed in points

The other measurements are expressed in the unit of measurement defined in the options.

The code 128 with automatic alphabet selection is used to encode the first 128 symbols ASCII.



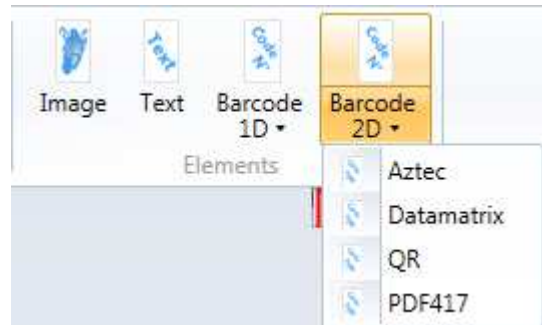
Press "Apply" to add the code 128 to the label. To restore the initial values press "Cancel". Select the code 128 graphic object in the label to display its list of attributes. They can be changed and the changes rendered effective by pressing "Apply". The element can be moved also by selecting it and dragging it with the mouse. The element can be resized also by selecting and dragging the end points with the mouse.



## 9.8.6.2 Two-dimensional

### 9.8.6.2.1 Aztec

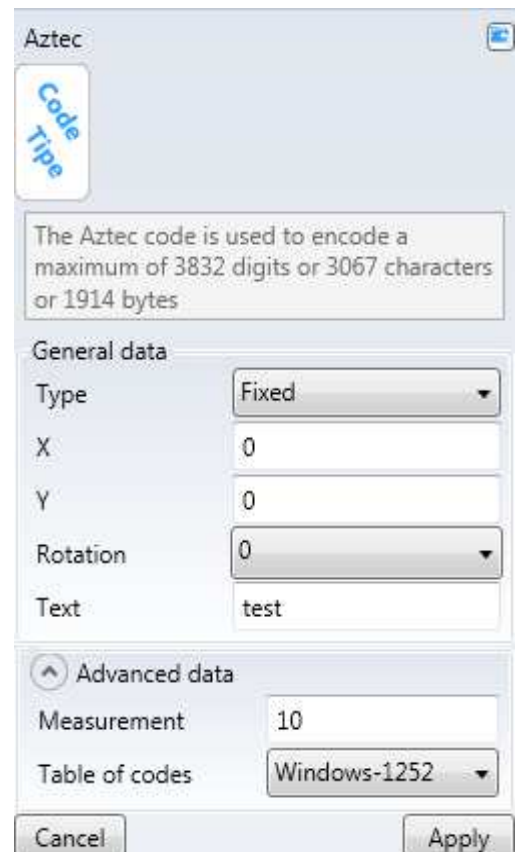
Use the “Aztec” command in the “Home” menu → “Elements” → “Barcode 2D” to create an Aztec code.



A list with the Aztec code attributes is displayed. The characteristic data of the Aztec code are:

- Type: fixed, parametric, date or time (same as for the "Text" element)
- X → initial X coordinate
- Y → initial Y coordinate
- Rotation → Aztec code rotation angle
- Text → text
- Measurement → adimensional code measurement (the value must be between 1 and 10)
- Codes table → page of the codes used to encode the text

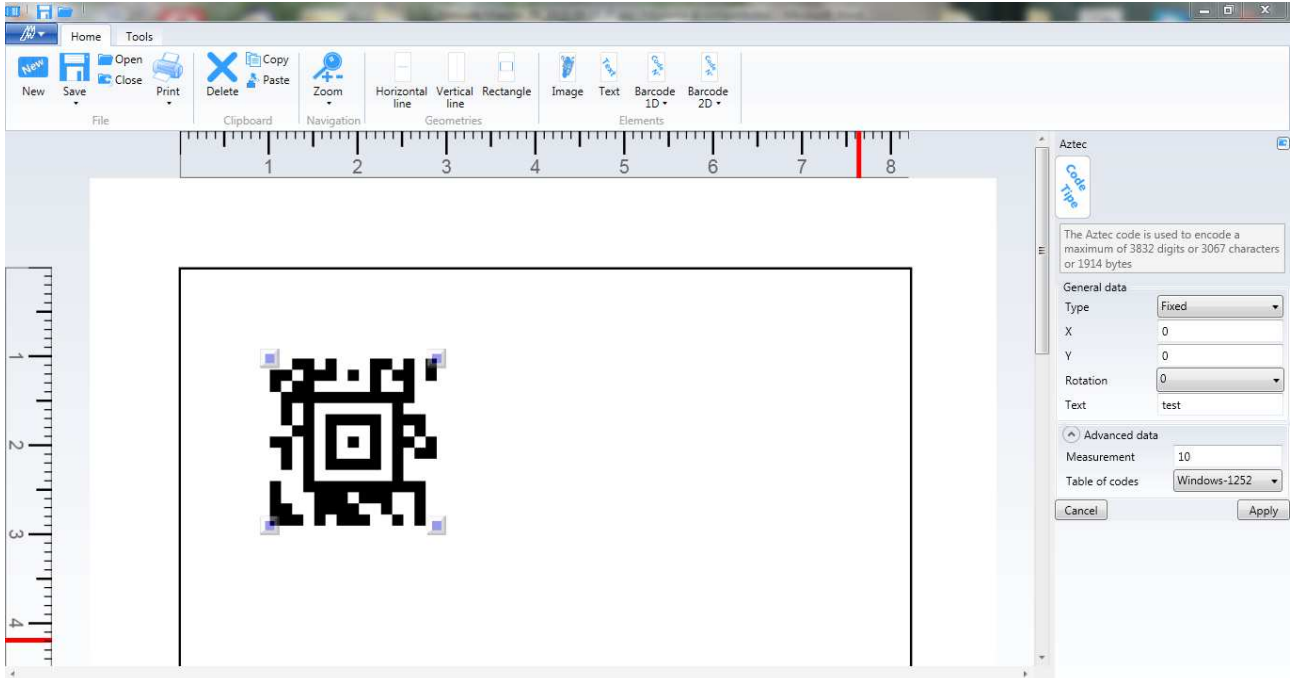
The other measurements are expressed in the unit of measurement defined in the options.



The Aztec code is used to encode a maximum of 3832 digits or 3067 characters or 1914 bytes.

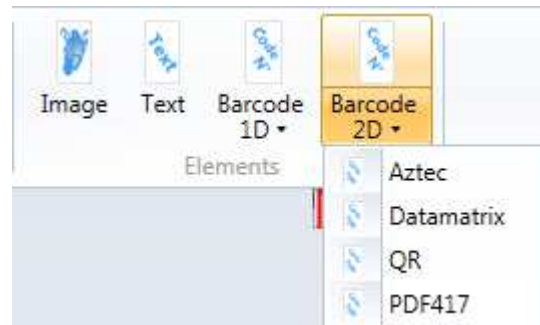


Press "Apply" to add the Aztec code to the label. To restore the initial values press "Cancel". Select the Aztec code graphic object in the label to display the list of attributes. They can be changed and the changes rendered effective by pressing "Apply". The element can be moved also by selecting it and dragging it with the mouse. The element can be resized also by selecting and dragging the end points with the mouse.



### 9.8.6.2.2 Datamatrix

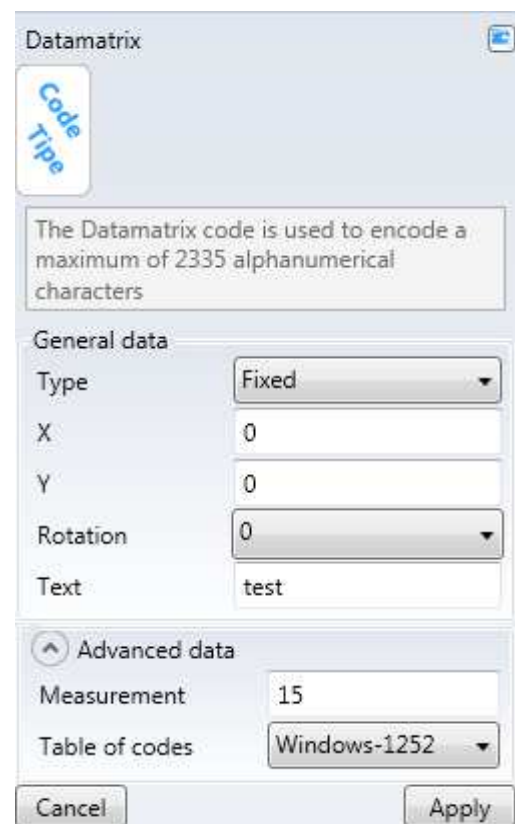
Use the “Datamatrix” command in the “Home” menu → “Elements” → “Barcode 1D” to create a Datamatrix code.



A list with the Datamatrix code attributes is displayed. The characteristic data of the Datamatrix code are:

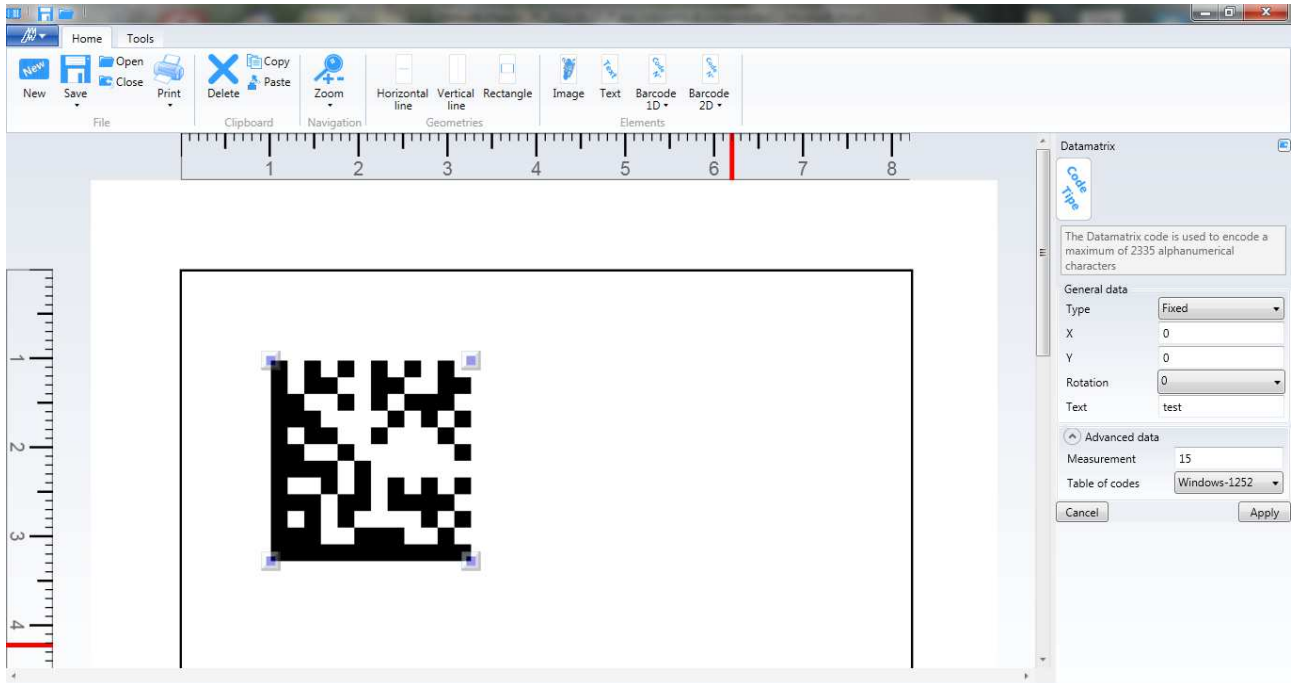
- Type: fixed, parametric, date or time (same as for the "Text" element)
- X → initial X coordinate
- Y → initial Y coordinate
- Rotation → Datamatrix code rotation angle
- Text → text
- Measurement → adimensional code measurement (the value must be between 1 and 50)
- Codes table → page of the codes used to encode the text

The other measurements are expressed in the unit of measurement defined in the options.



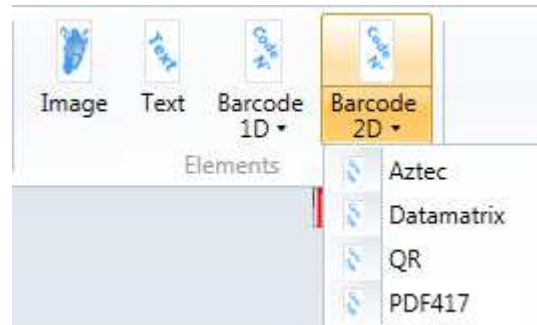
The Datamatrix code is used to encode a maximum of 2335 alphanumerical characters.

Press "Apply" to add the Datamatrix code to the label. To restore the initial values press "Cancel". Select the Datamatrix code graphic object in the label to display the list of attributes. They can be changed and the changes rendered effective by pressing "Apply". The element can be moved also by selecting it and dragging it with the mouse. The element can be resized also by selecting and dragging the end points with the mouse.



### 9.8.6.2.3 QR

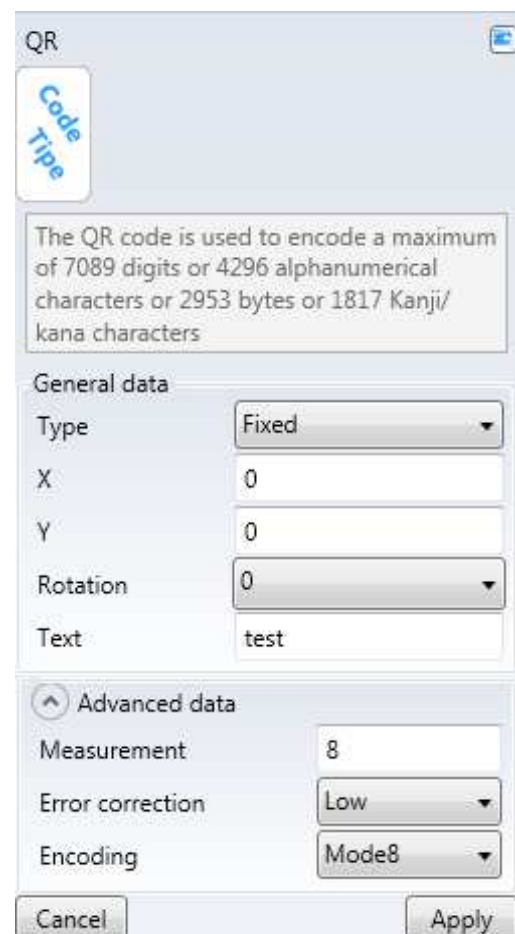
Use the “QR” command in the “Home” menu→ “Elements” → “Barcode 2D” to create a QR code.



A list with the QR code attributes is displayed. The characteristic data of the QR code are:

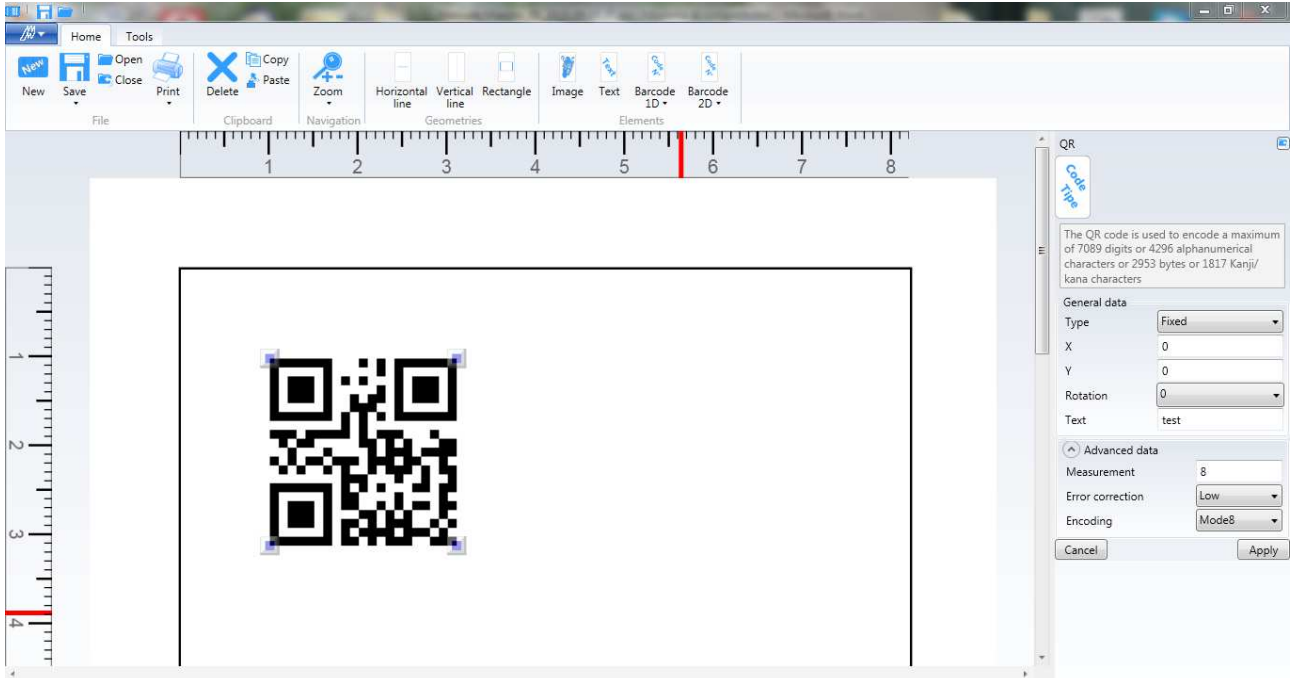
- Type: fixed, parametric, date or time (same as for the "Text" element)
- X → initial X coordinate
- Y → initial Y coordinate
- Rotation → Datamatrix code rotation angle
- Text → text
- Measurement → adimensional code measurement (the value must be between 1 and 100)
- Error correction → level of errors correction
- Encoding → type of encoding

The other measurements are expressed in the unit of measurement defined in the options.



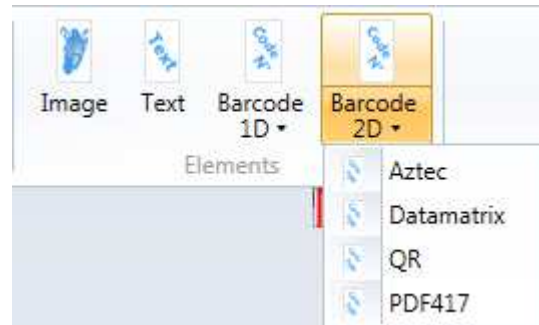
The QR code is used encode a maximum of 7089 digits or 4296 alphanumerical characters or 2953 bytes or 1817 Kanji/kana characters.

Press "Apply" to add the QR code to the label. To restore the initial values press "Cancel". Select the QR code graphic object in the label to display the list of attributes. They can be changed and the changes rendered effective by pressing "Apply". The element can be moved also by selecting it and dragging it with the mouse. The element can be resized also by selecting and dragging the end points with the mouse.



#### 9.8.6.2.4 PDF417

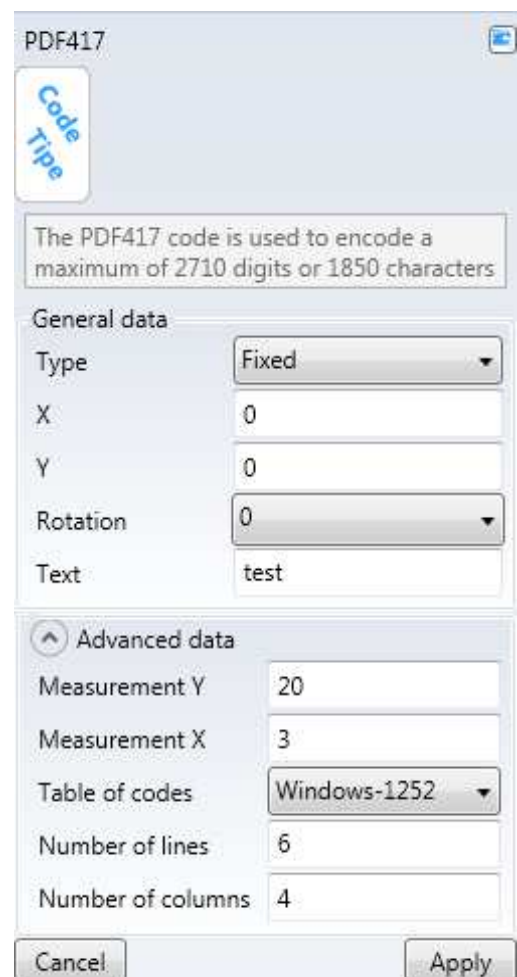
Use the "PDF417" command in the "Home" menu → "Elements" → "Barcode 2D" to create a PDF417 code.



A list with the PDF417 code attributes is displayed. The characteristic data of the PDF417 code are:

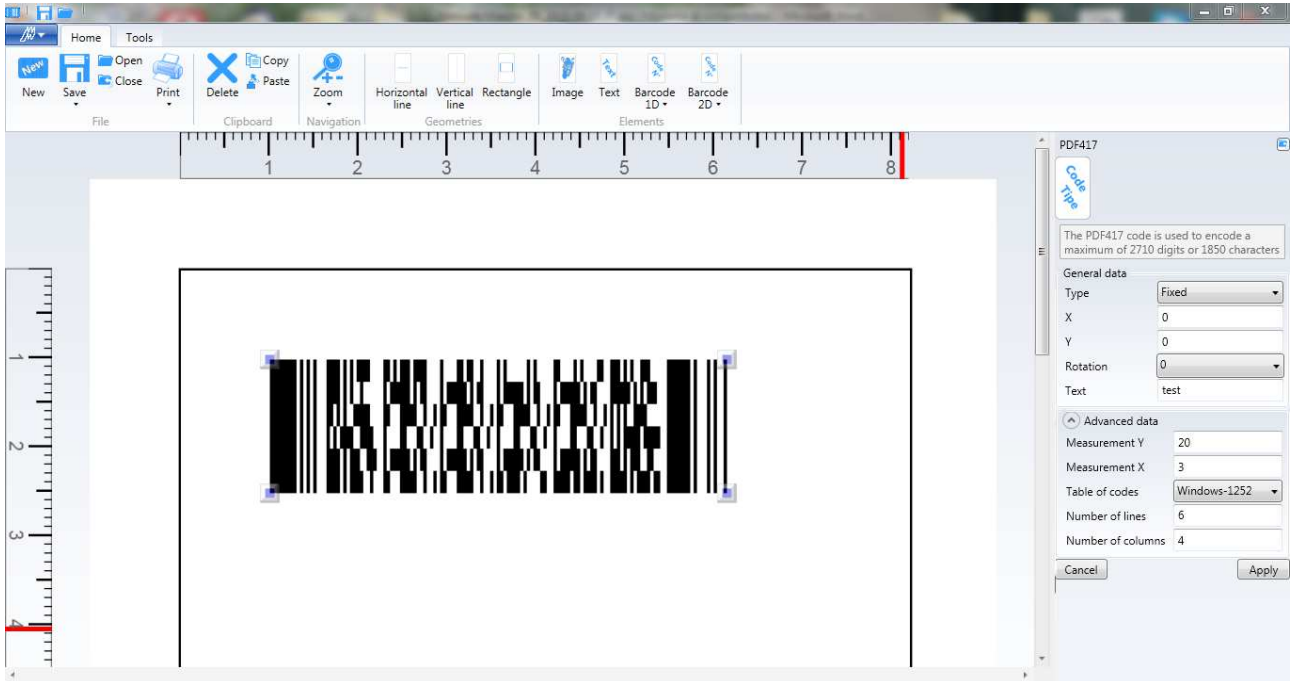
- Type: fixed, parametric, date or time (same as for the "Text" element)
- X → initial X coordinate
- Y → initial Y coordinate
- Rotation → Datamatrix code rotation angle
- Text → text
- Measurement Y → adimensional code vertical measurement
- Measurement X → adimensional code horizontal measurement
- Codes table → page of the codes used to encode the text
- Lines number → number of lines (the value must be between 3 and 90)
- Columns number → number of columns (the value must be between 1 and 30)

The other measurements are expressed in the unit of measurement defined in the options.



The PDF417 code is used to encode a maximum of 2710 digits or 1850 characters.

Press "Apply" to add the PDF417 code to the label. To restore the initial values press "Cancel". Select the PDF417 code graphic object in the label to display the list of attributes. They can be changed and the changes rendered effective by pressing "Apply". The element can be moved also by selecting it and dragging it with the mouse. The element can be resized also by selecting and dragging the end points with the mouse.

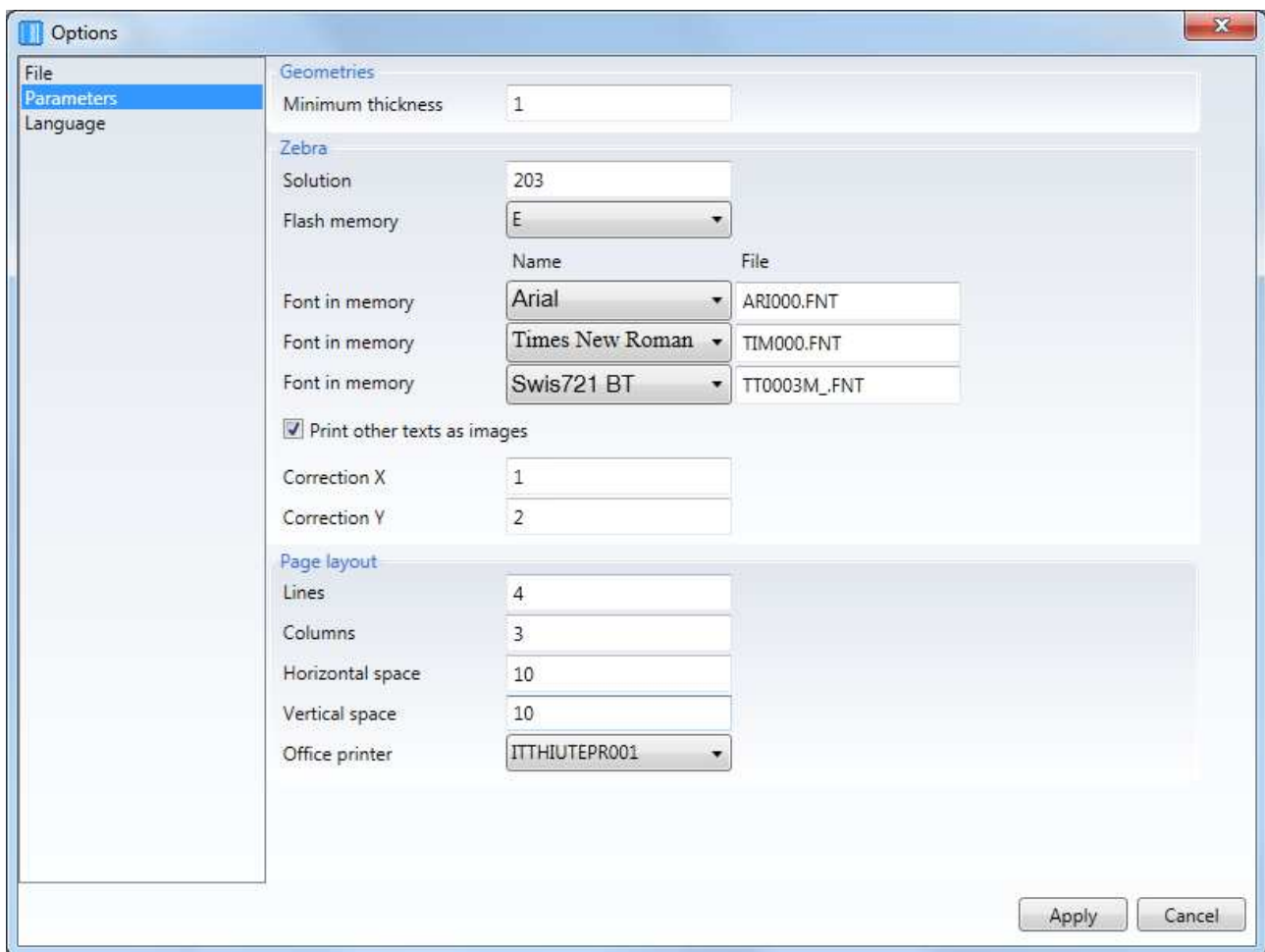


## 9.9 Options

Use the “Options” command in the “Tools” menu → “Options”.



The “Files”, “Parameters” and “Language” sheets are displayed in the window.

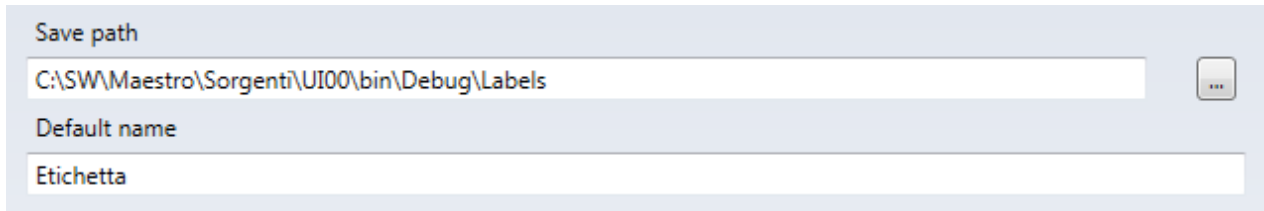




## 9.9.1 Files

In the "Files" sheet you can specify:

- The default path from which to open and save the labels
- The default name to associate to a new label



The screenshot shows a configuration panel with two input fields. The first field is labeled "Save path" and contains the text "C:\SW\Maestro\Sorgenti\UI00\bin\Debug\Labels". To the right of this field is a small button with three dots. The second field is labeled "Default name" and contains the text "Etichetta".

## 9.9.2 Parameters

### 9.9.2.1 Geometries



The screenshot shows a configuration panel titled "Geometries". It contains a single input field labeled "Minimum thickness" with the value "1".

- Minimum thickness → minimum thickness of the horizontal lines, vertical lines, rectangles

The measurement is expressed in the unit of measurement set in the options.

### 9.9.2.2 Zebra

Zebra printer parameters



The screenshot shows a configuration panel titled "Zebra". It contains several parameters:

- Solution:** 203
- Flash memory:** E
- Font in memory:** A table with two columns: "Name" and "File".

Name	File
Arial	ARI000.FNT
Times New Roman	TIM000.FNT
Swis721 BT	TT0003M_.FNT
- Print other texts as images
- Correction X:** 1
- Correction Y:** 2

- Resolution → print density expressed in dpi (dots per inch), **the printers of the GC420 family (GC420d and GC420t models) have resolutions equal to 203**
- Flash memory → printer rom memory used to save the fonts
- Font in memory → fonts that have been downloaded in the printer with the reference names (up to 3 fonts)
- Print other texts as images → **if the option is enabled, the texts whose fonts have NOT been downloaded in the zebra printer with the relative procedure, are printed as images but it is NOT guaranteed that the print will match the image preview. If the option is disabled the texts whose fonts have NOT been downloaded in the zebra printer with the relative procedure will NOT be printed**
- Correction X → correction factor in X in the label calibration (a positive value corresponds to a movement to the right, the measurement is expressed in the unit of measurement set in the options).
- Correction Y → correction factor in Y in the label calibration (a positive value corresponds to a movement downwards, the measurement is expressed in the unit of measurement set in the options)

### 9.9.2.3 Page layout

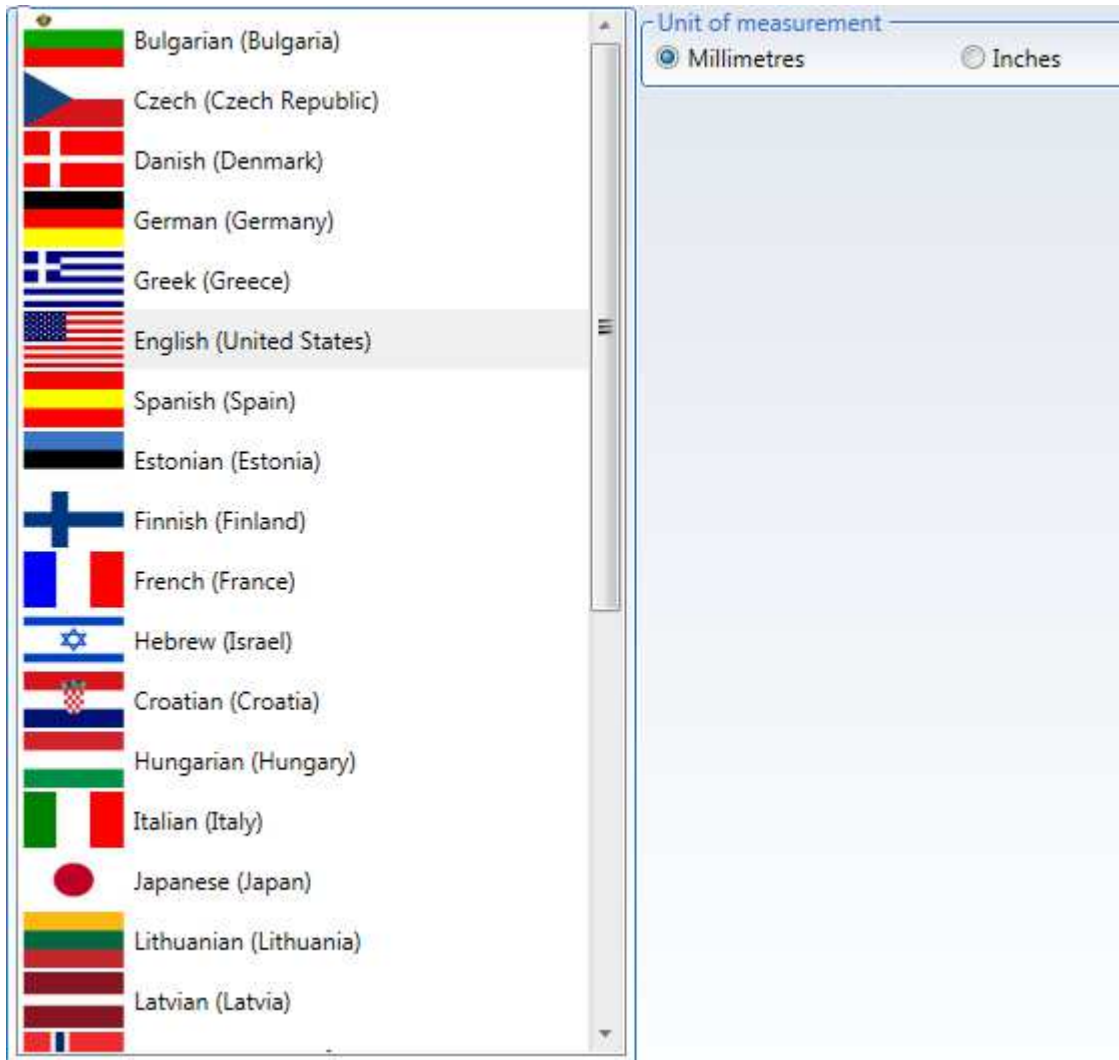
These parameters indicate how to layout the labels matrix in the sheet to print in the office printer (in the final labels positioning in the sheet take into account the top and left margins of the printer).

Page layout	
Lines	4
Columns	3
Horizontal space	10
Vertical space	10
Office printer	ITTHIUTEPR001

- Lines → number of lines of labels in the sheet being printed
- Columns → number of columns of labels in the sheet being printed
- Horizontal space → horizontal gap between labels
- Vertical space → vertical gap between labels
- Office printer → printer selection (if there is no value, at the first printing operation you will be asked to select a printer and the one selected will become the value of the parameter also for the subsequent times)

### 9.9.3 Language

The language of the application and the unit of measurement to use (millimetres or inches) are set in the "Language" sheet. The unit of measurement selected will be used for all the measurements of the label and the objects.



## 9.10 Zebra printer

### 9.10.1 Installation

The Zebra printer must be a printer of the GC420 family (GC420d or GC420t model).

The following window is displayed with the automatic start from the CD ROM. If not manually execute the “RunCD.exe” from the CD.

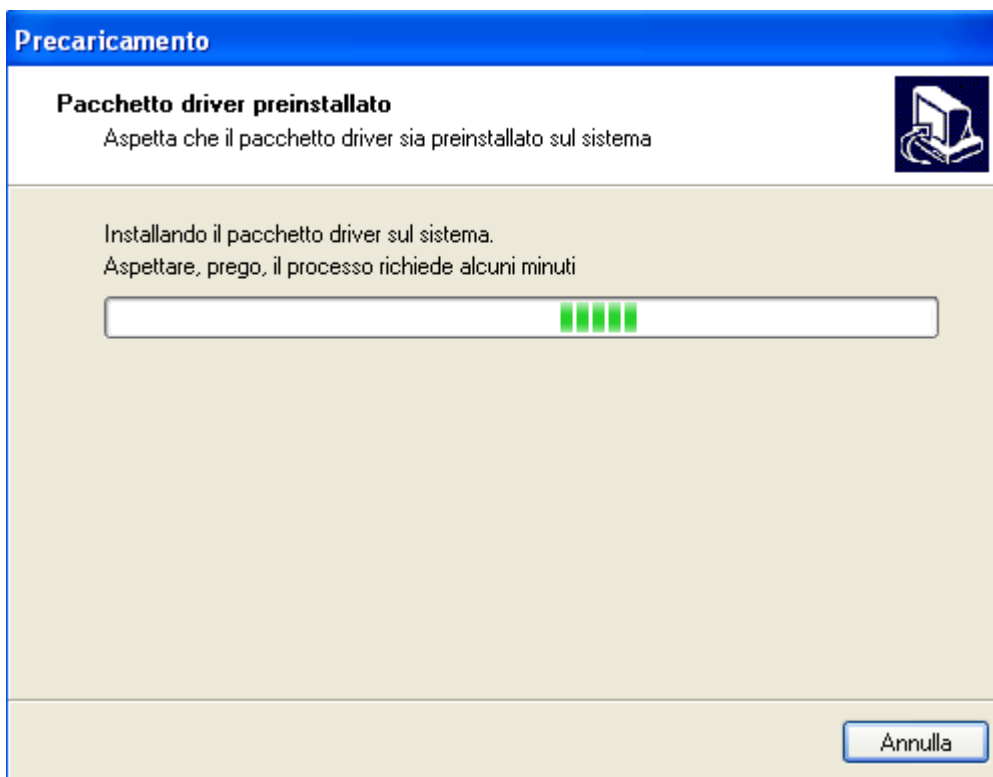


Select the printer model.

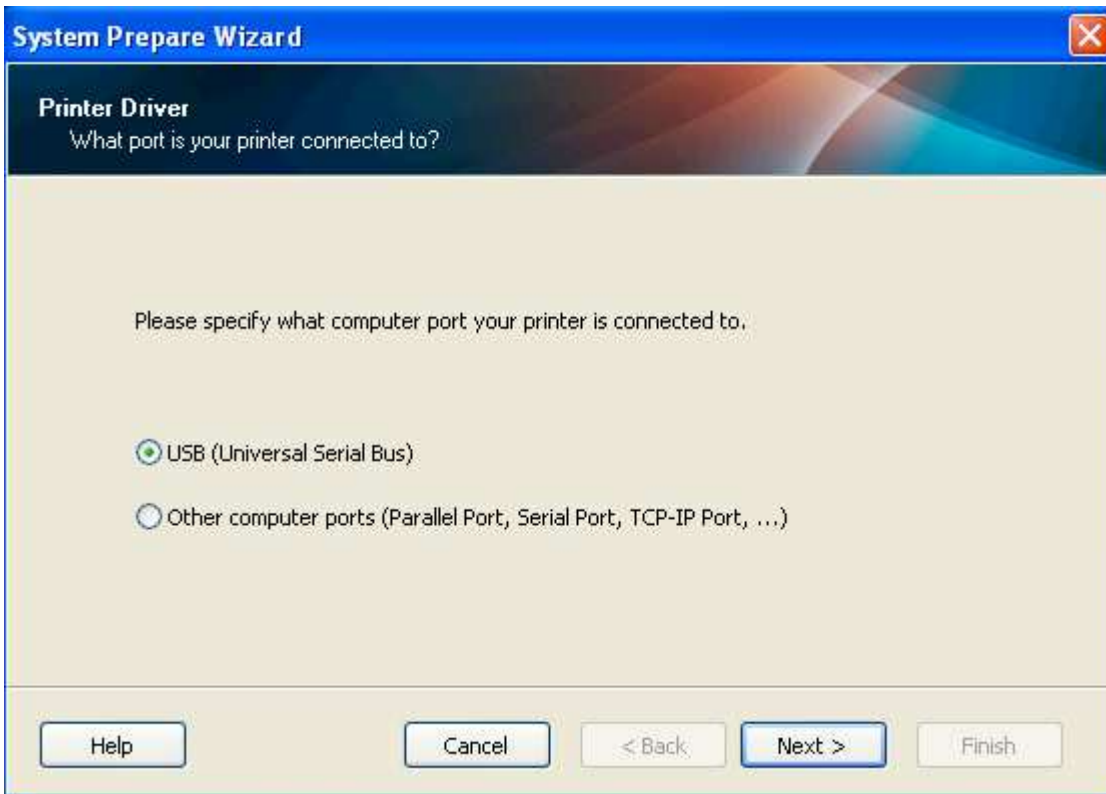
In the menu on the left select “Install printer” and in the menu on the right “Execute Zebra Setup Utilities”.



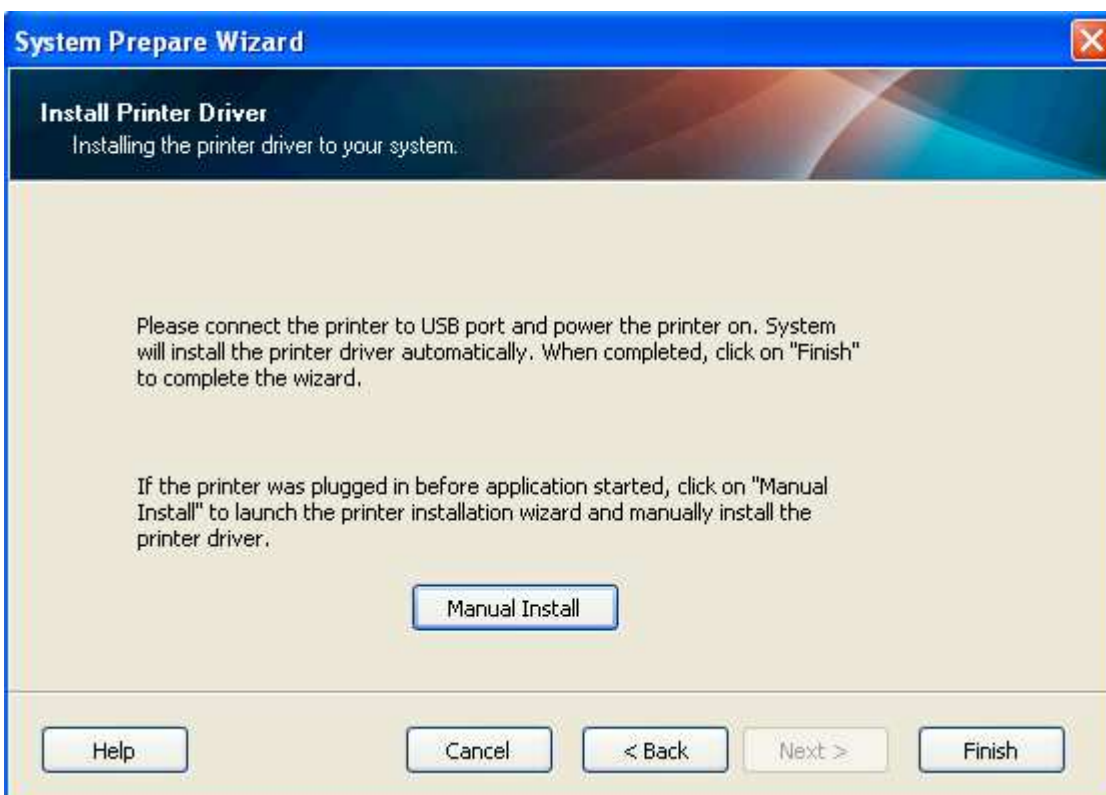
Press "Next".



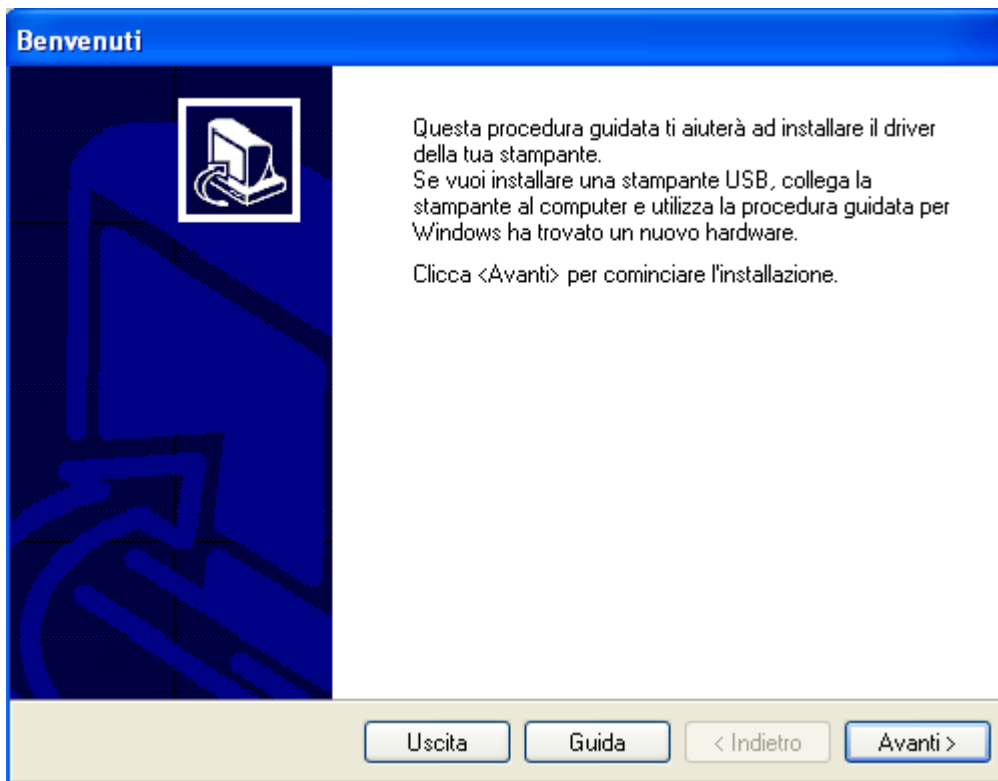
Wait.



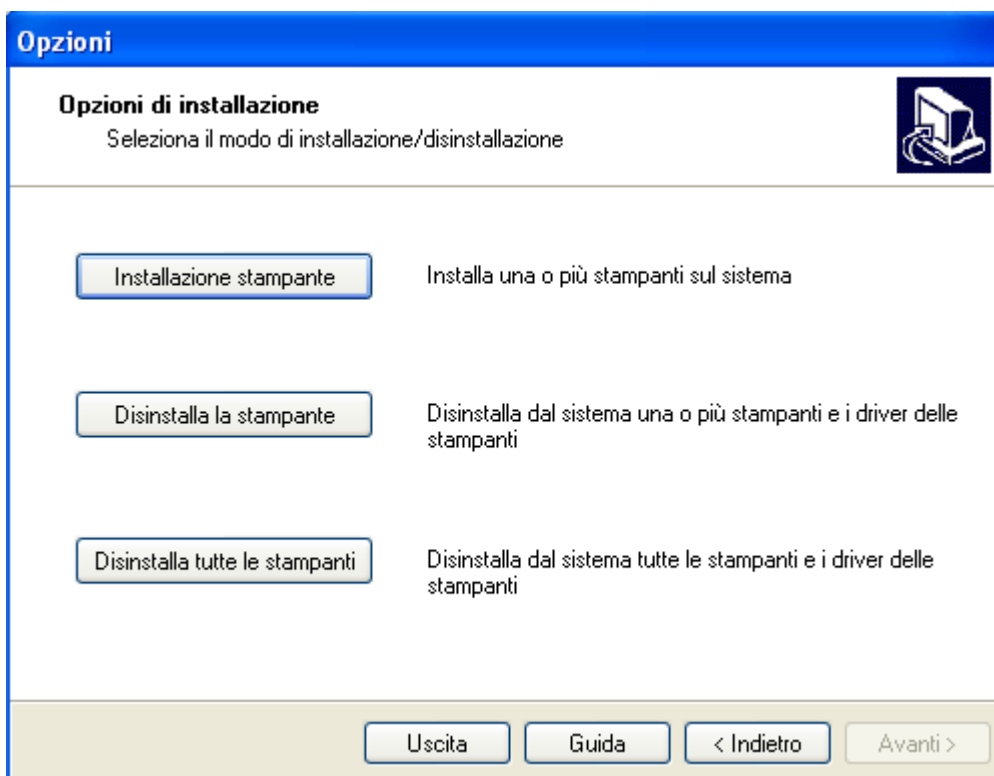
Select the printer/computer connection mode and press “Next”.



Press “Manual Install”.

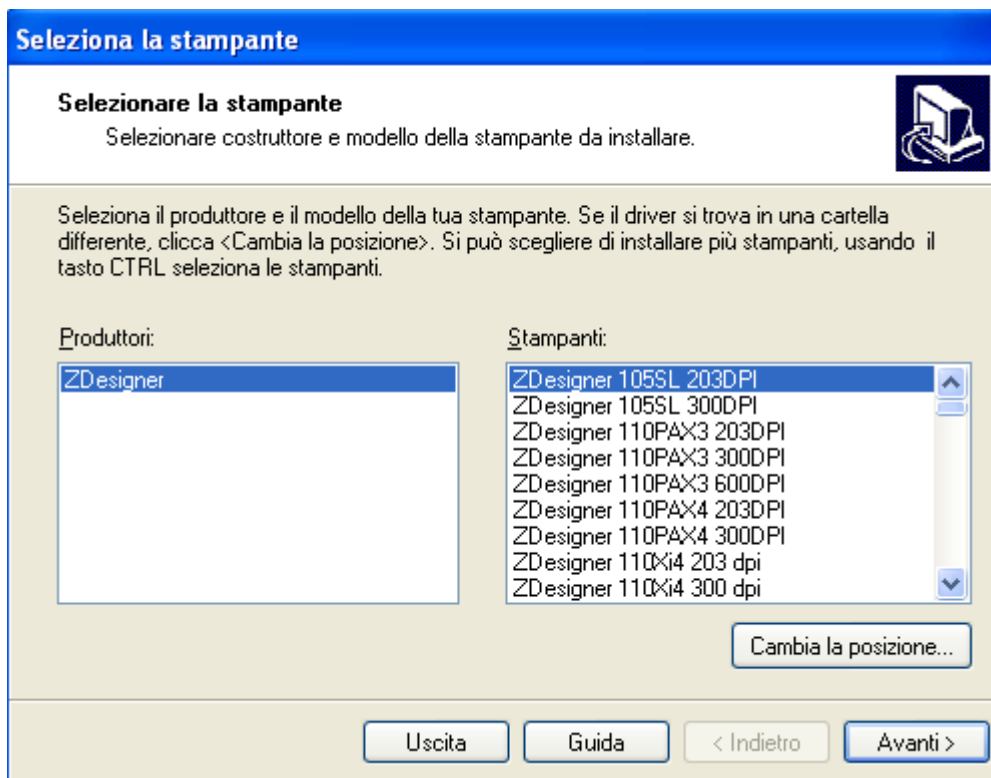


Press "Next".

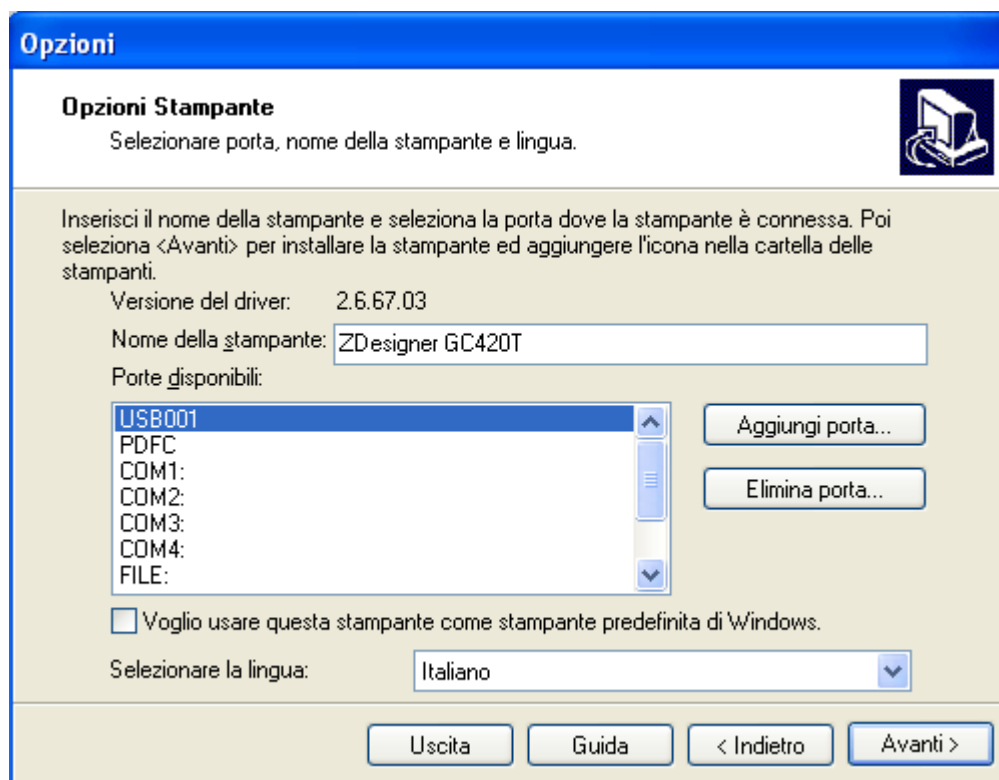


Press "Printer installation".

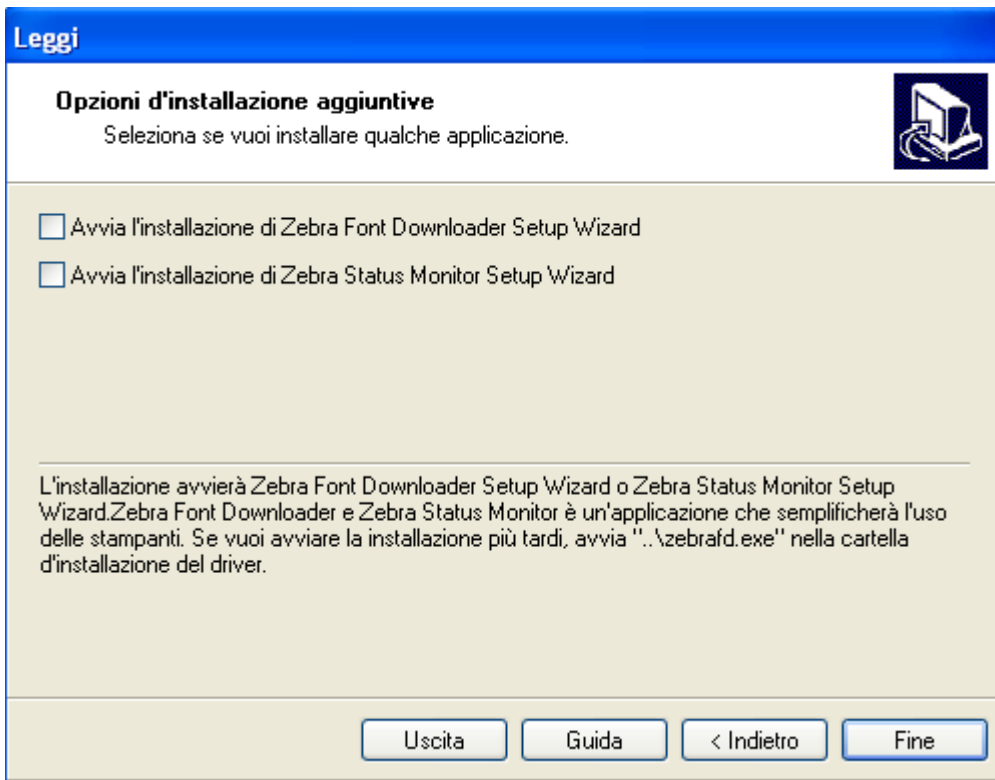




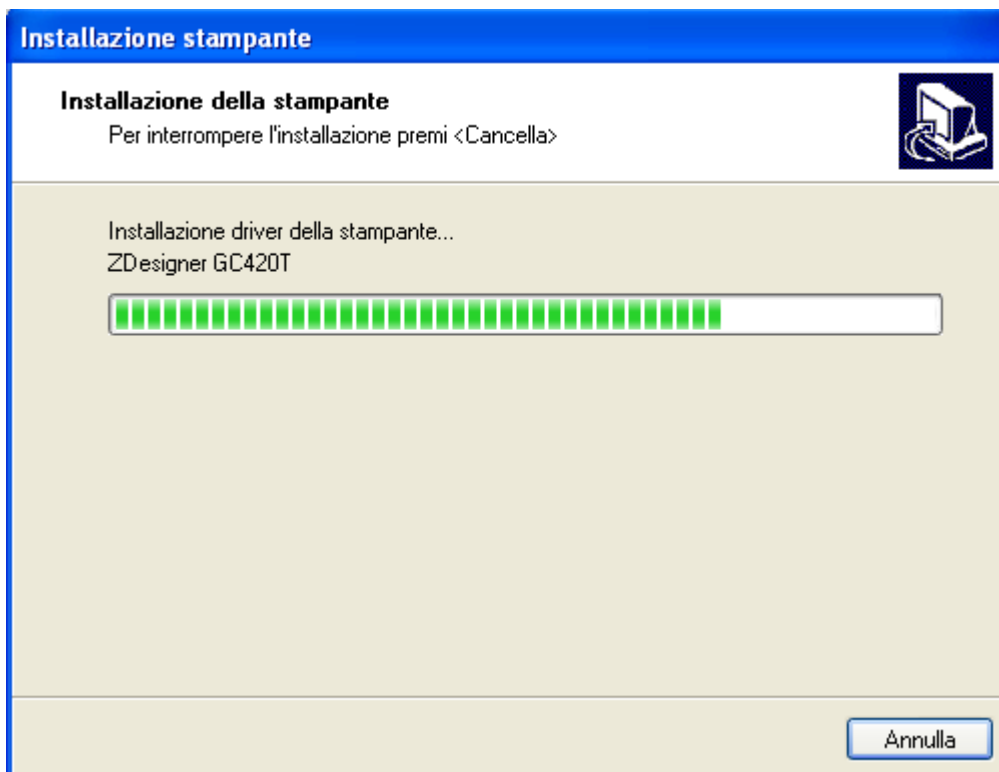
In the list on the right select the printer model and press “Next”.



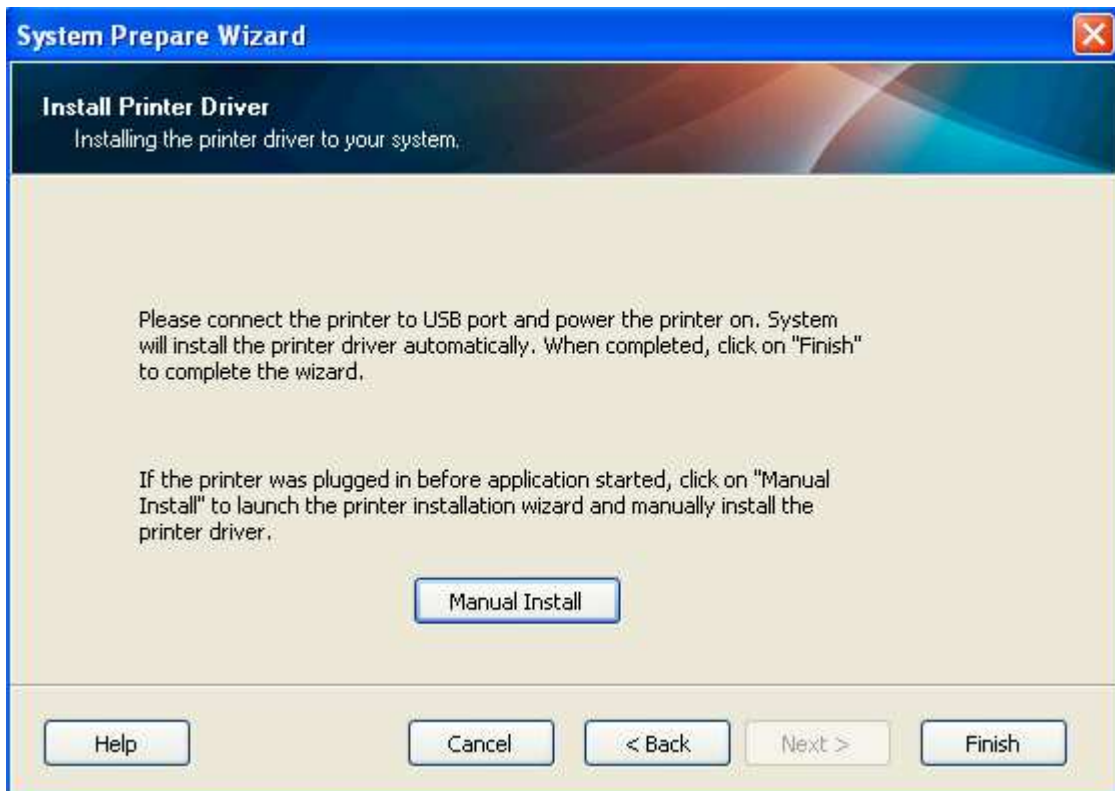
In the list select the printer/computer connection port and press “Next”.



Deselect the additional installation options suggested and press “End”.



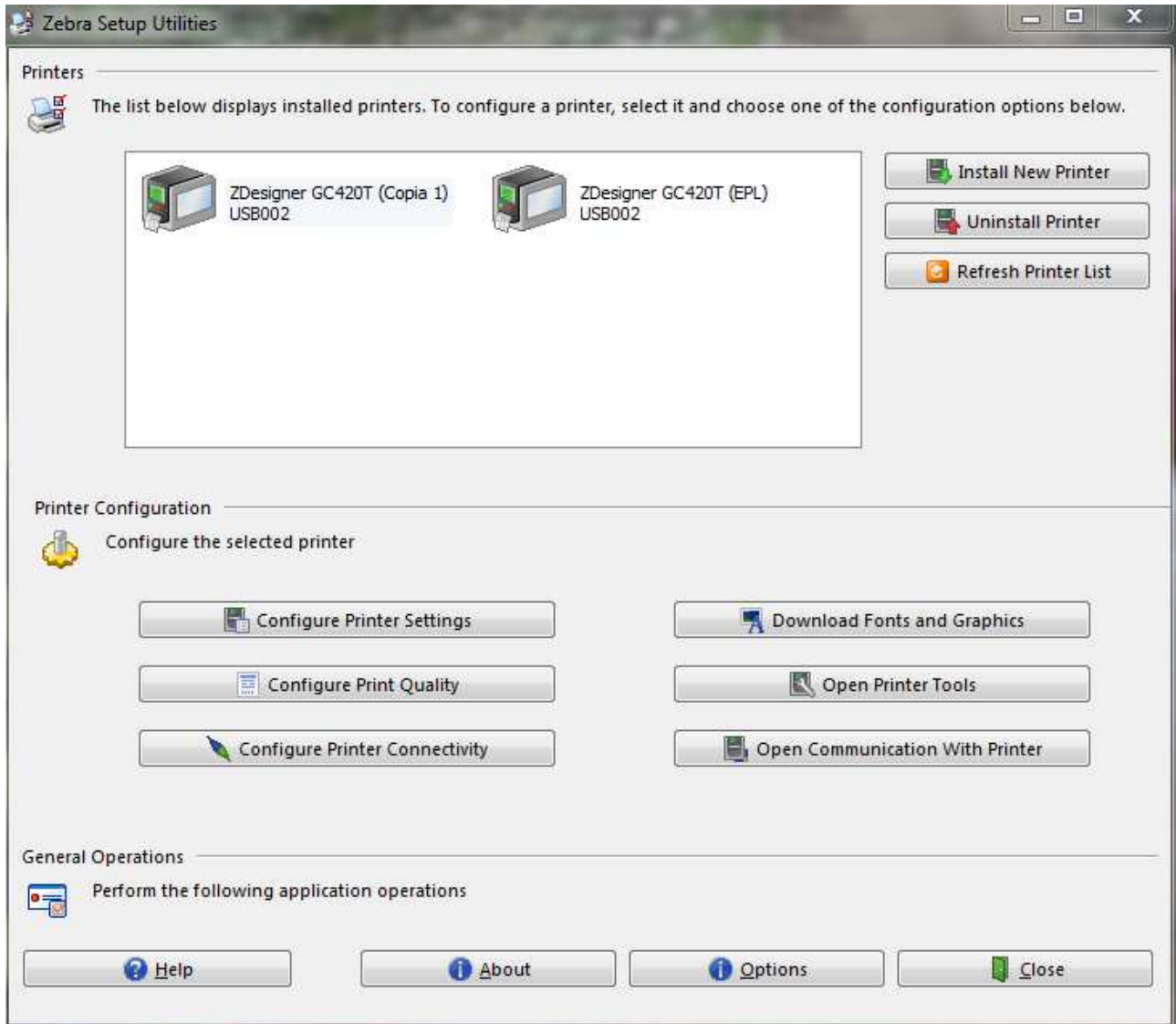
Wait.



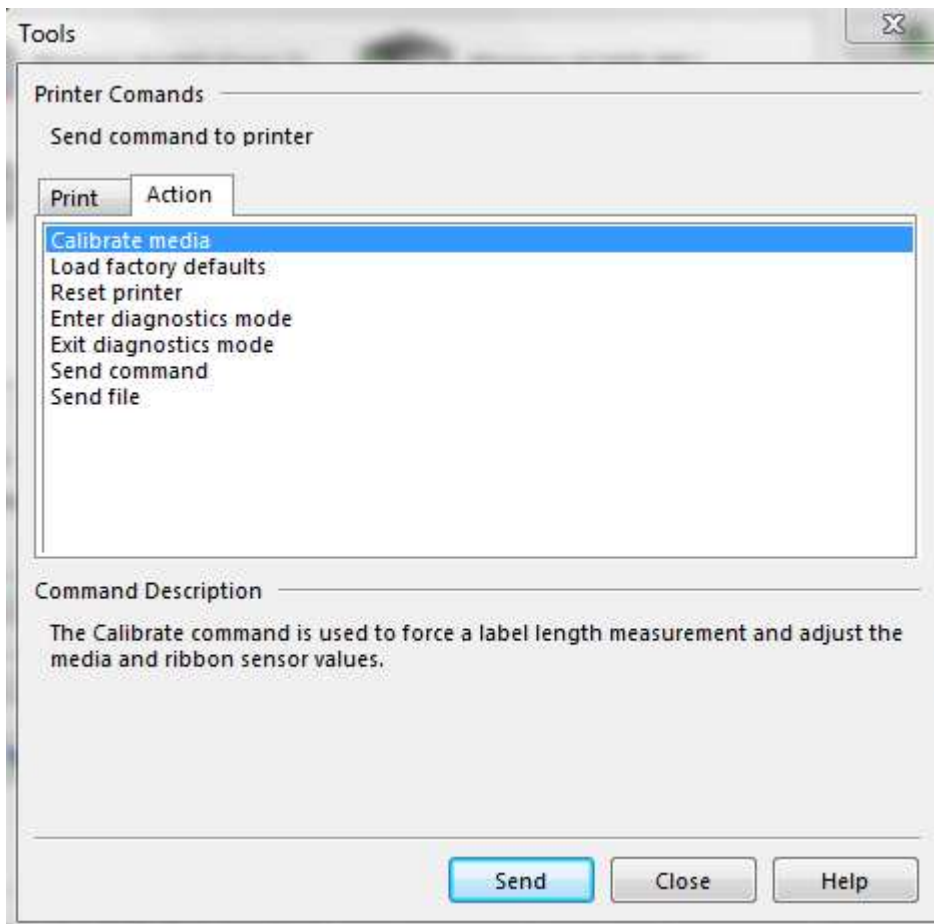
Press "End".

## 9.10.2 Calibration

Once the zebra printer is installed on the PC, you must carry out the calibration in order to calibrate the labels correctly.



In the “Zebra Setup Utilities” program press “Open Printer Tools”.



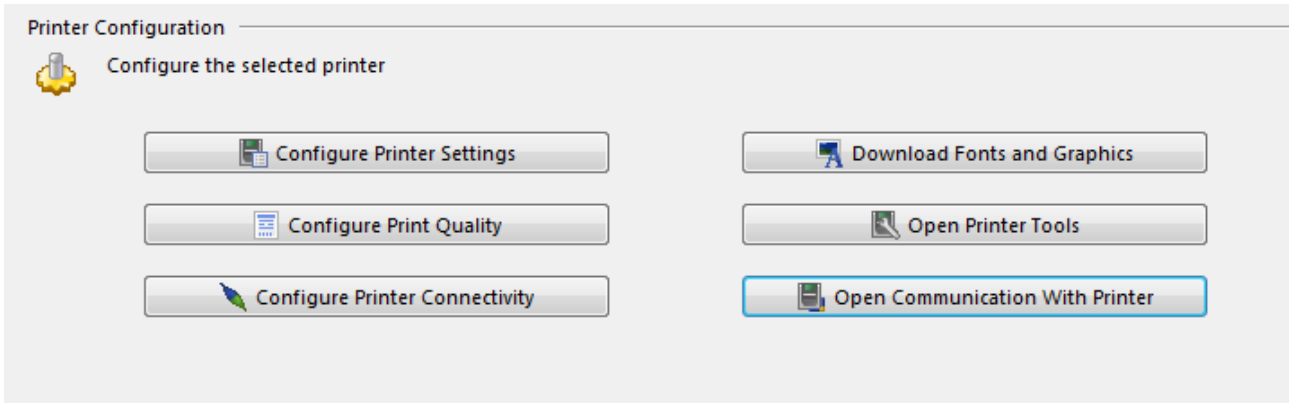
In the window displayed, in the “Action” sheet, select “Calibrate media” and press “Send”.

Wait for the printer to complete the operation.

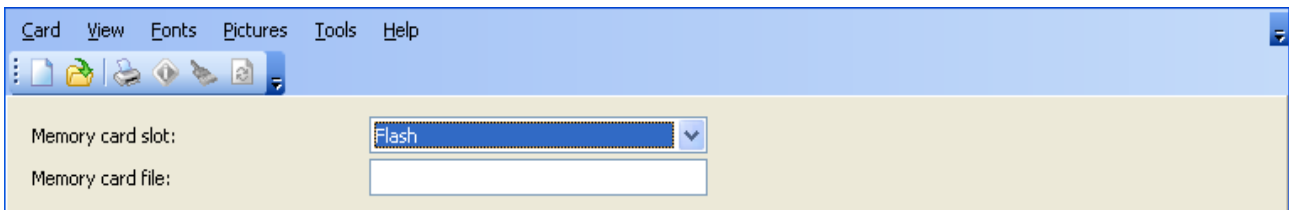
### 9.10.3 Download font

To use fonts to print labels they must be converted and downloaded in the Zebra printer memory.

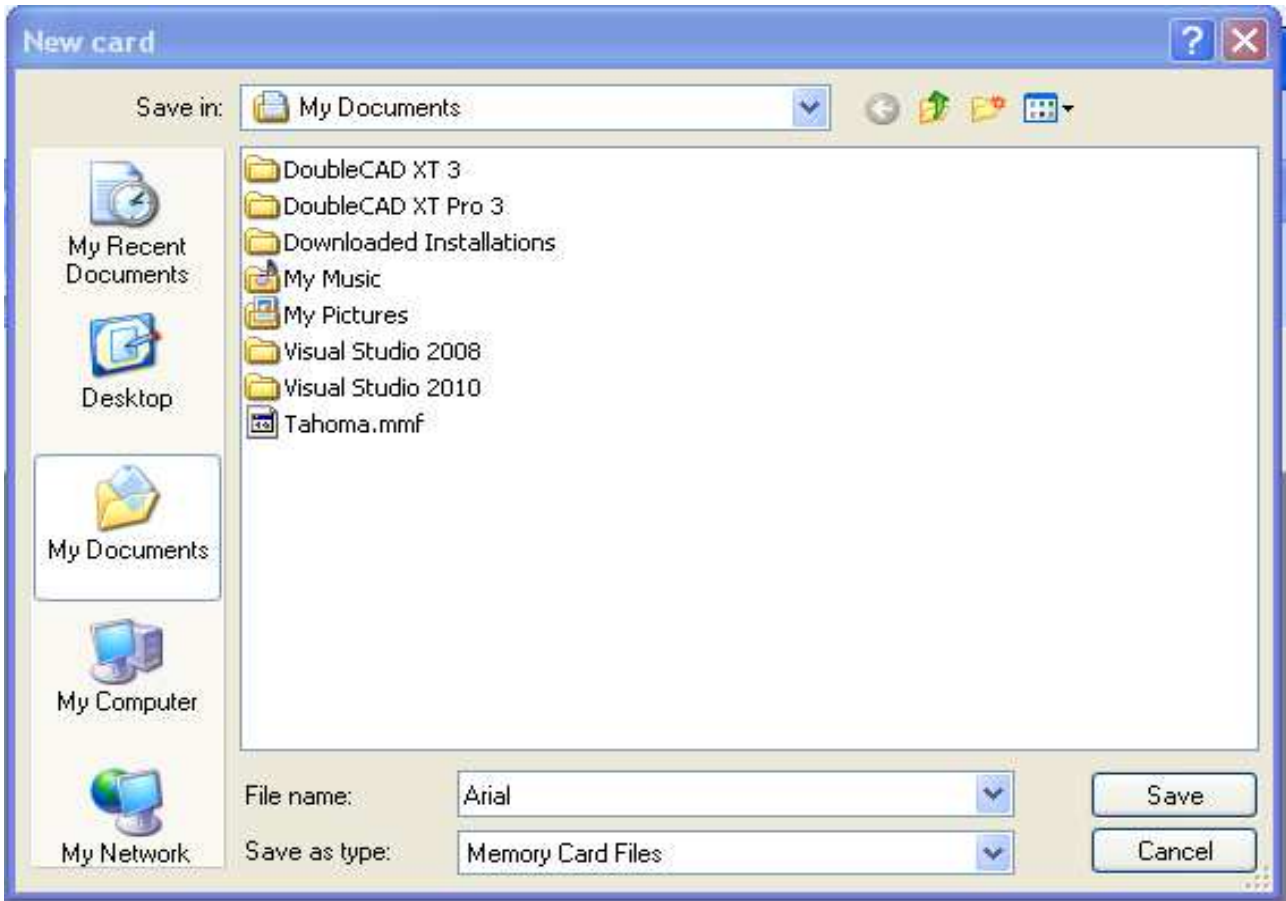
In the program Zebra Setup Utilities, select the program “Download Fonts and Graphics”:



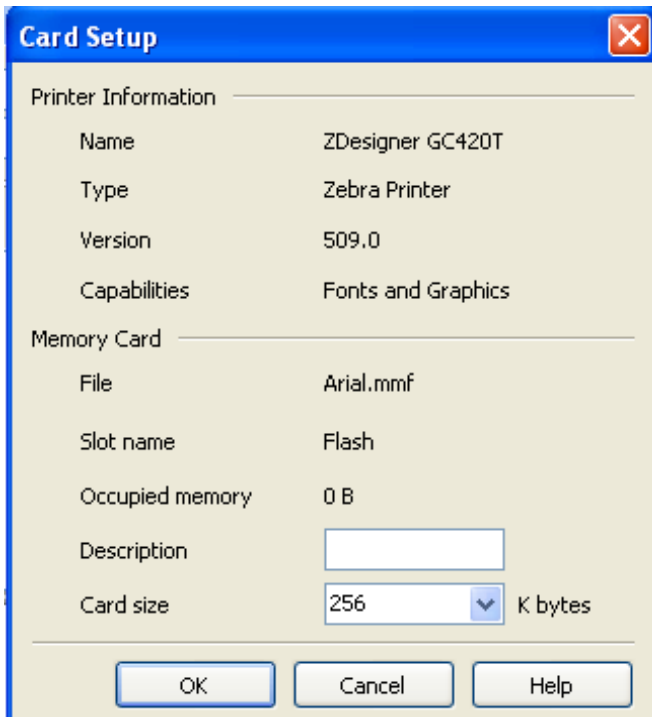
In the drop down menu “Slot for memory board” select “Flash”, which corresponds to the printer memory device used to save the type of character.



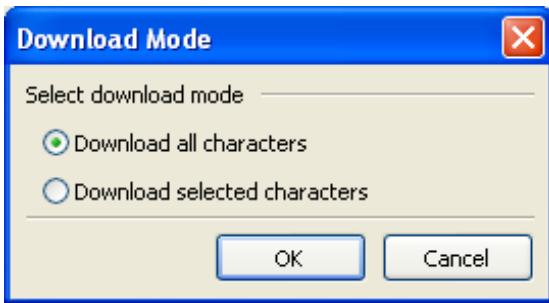
Now create a new configuration file for the memory sheet. Select a new icon. You can create a new file or select an existing one in the local hard disc. For example, create a new configuration file for the "Arial" memory sheet:



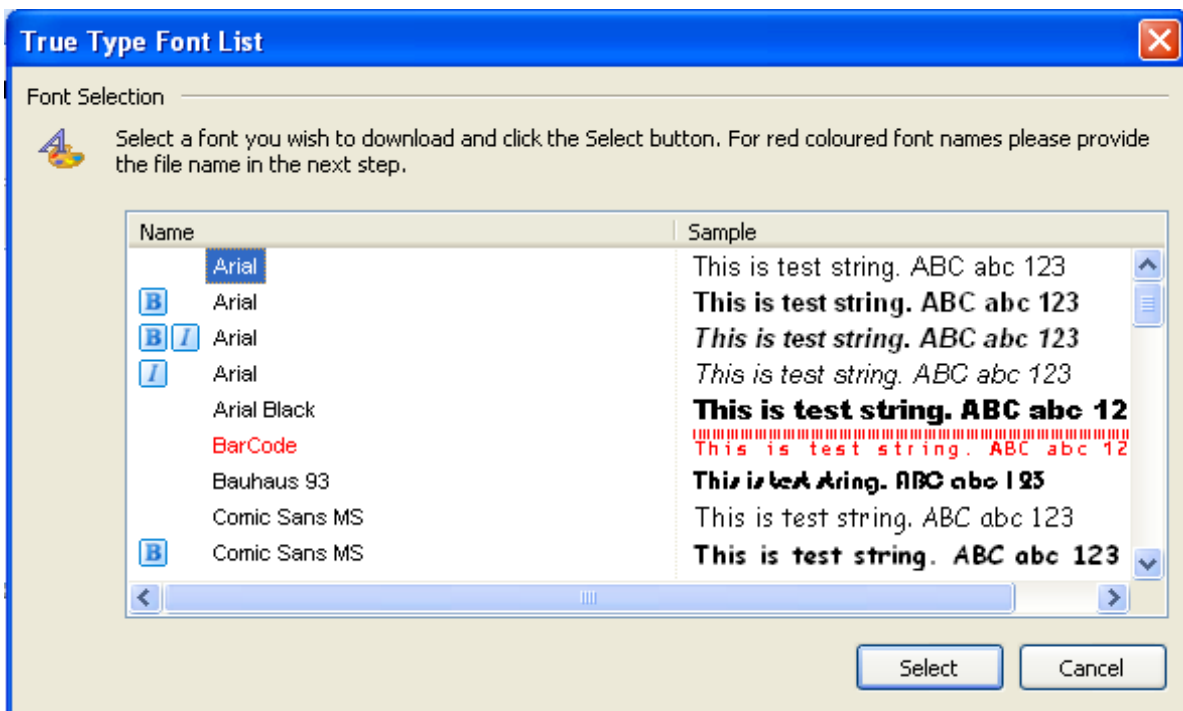
You can enter a description for the memory sheet configuration file and select the size of the paper. This depends on the type of character selected and it must be sufficient to contain it. It can be selected in the interval from 256Kbyte to 131072Kbyte.



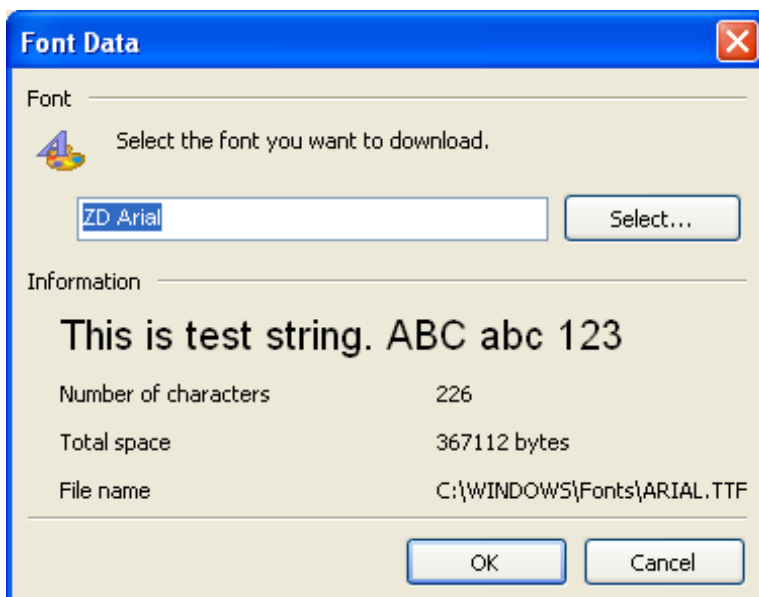
Then select "Fonts" → "New" then select to load all characters.



Select the type of character and then click on "Select".

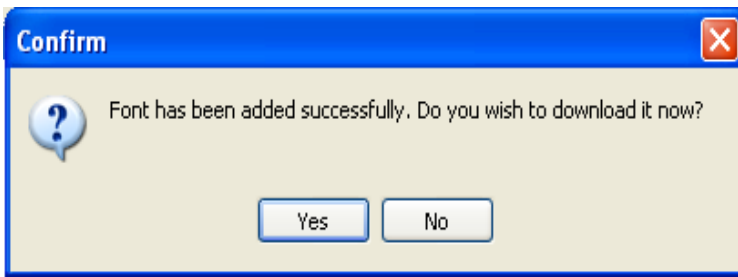


In the font data preview click on "OK":

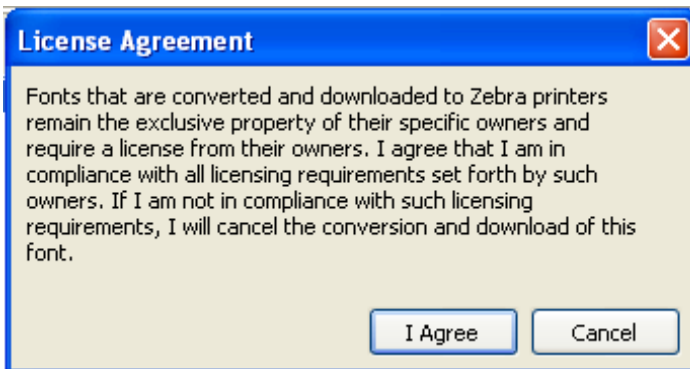




Now you can download the font in the printer memory:



You must agree with the licence contract for the type of character to download it in the printer.

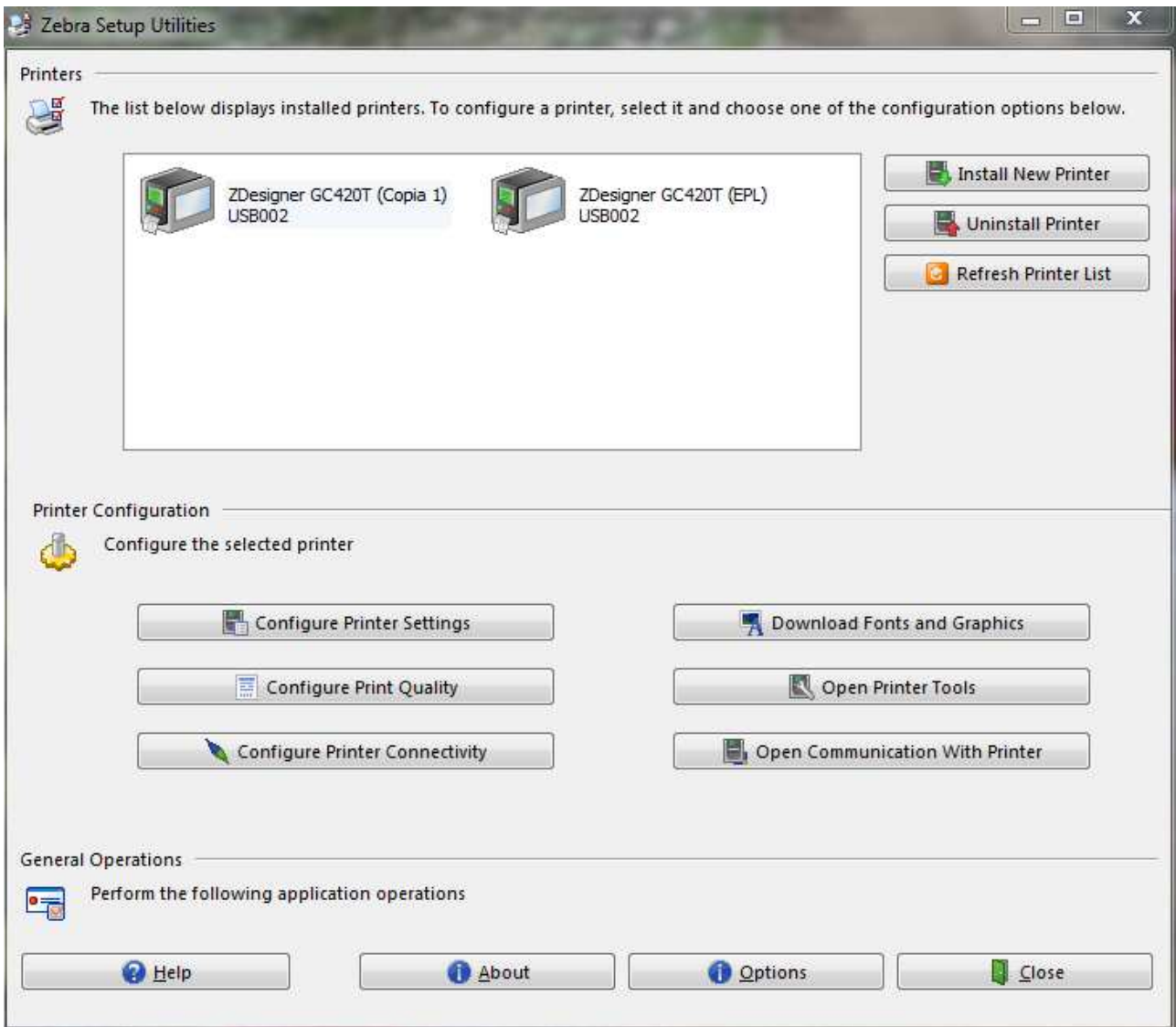


If there is the "Insufficient free space in the memory board to download" error, go back to "Paper" → "Memory sheet information" and increase the size before repeating the procedure.

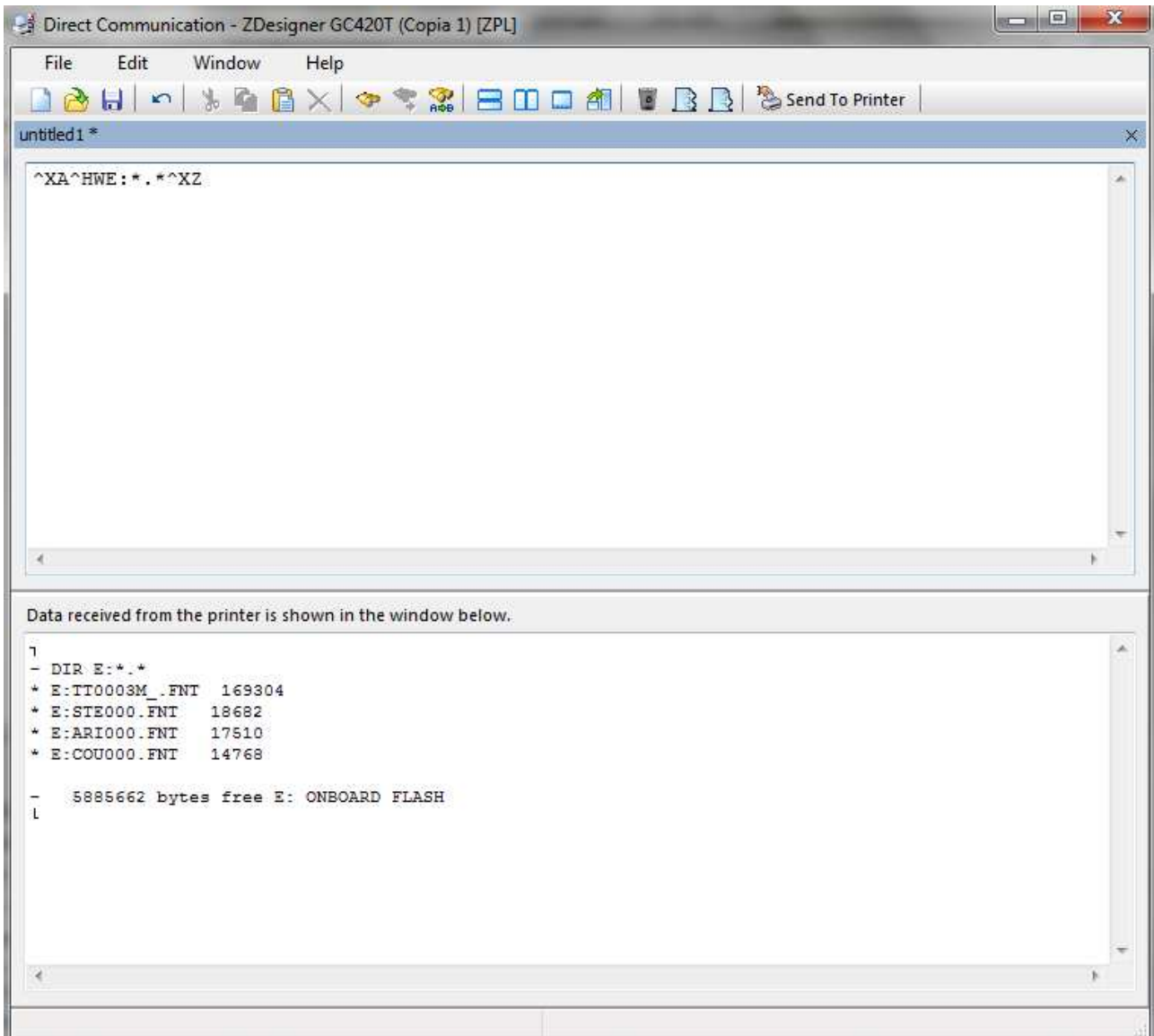
## 9.10.4 Memory

The Zebra printer is fitted with two memories: one eeprom "E" flash memory used to contain the fonts and a ram "R" memory to temporarily save the graphic objects. It may be useful to poll the content of the Zebra printer memories.

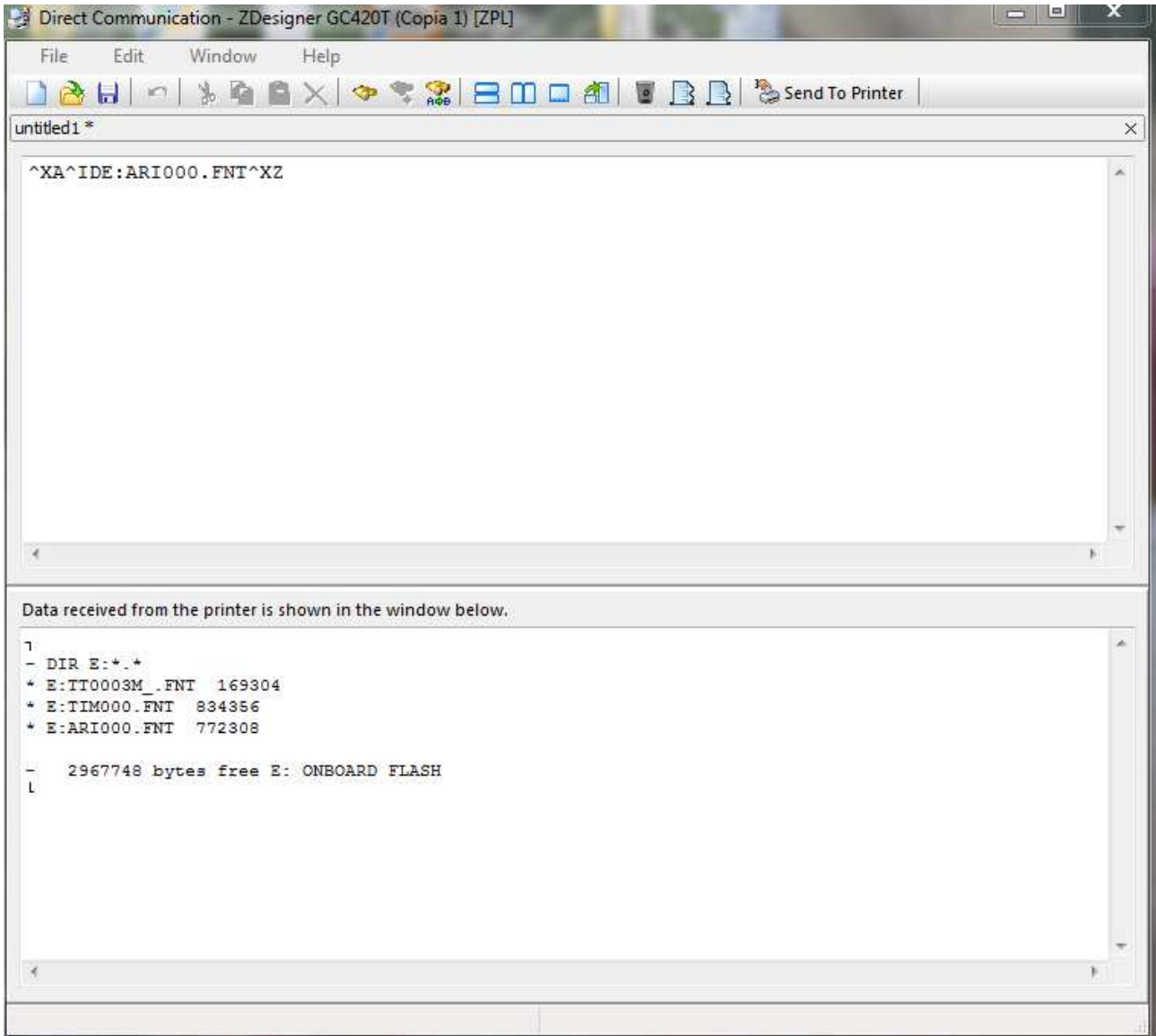
In the program "Zebra Setup Utilities", select the program "Open Communication With Printer":



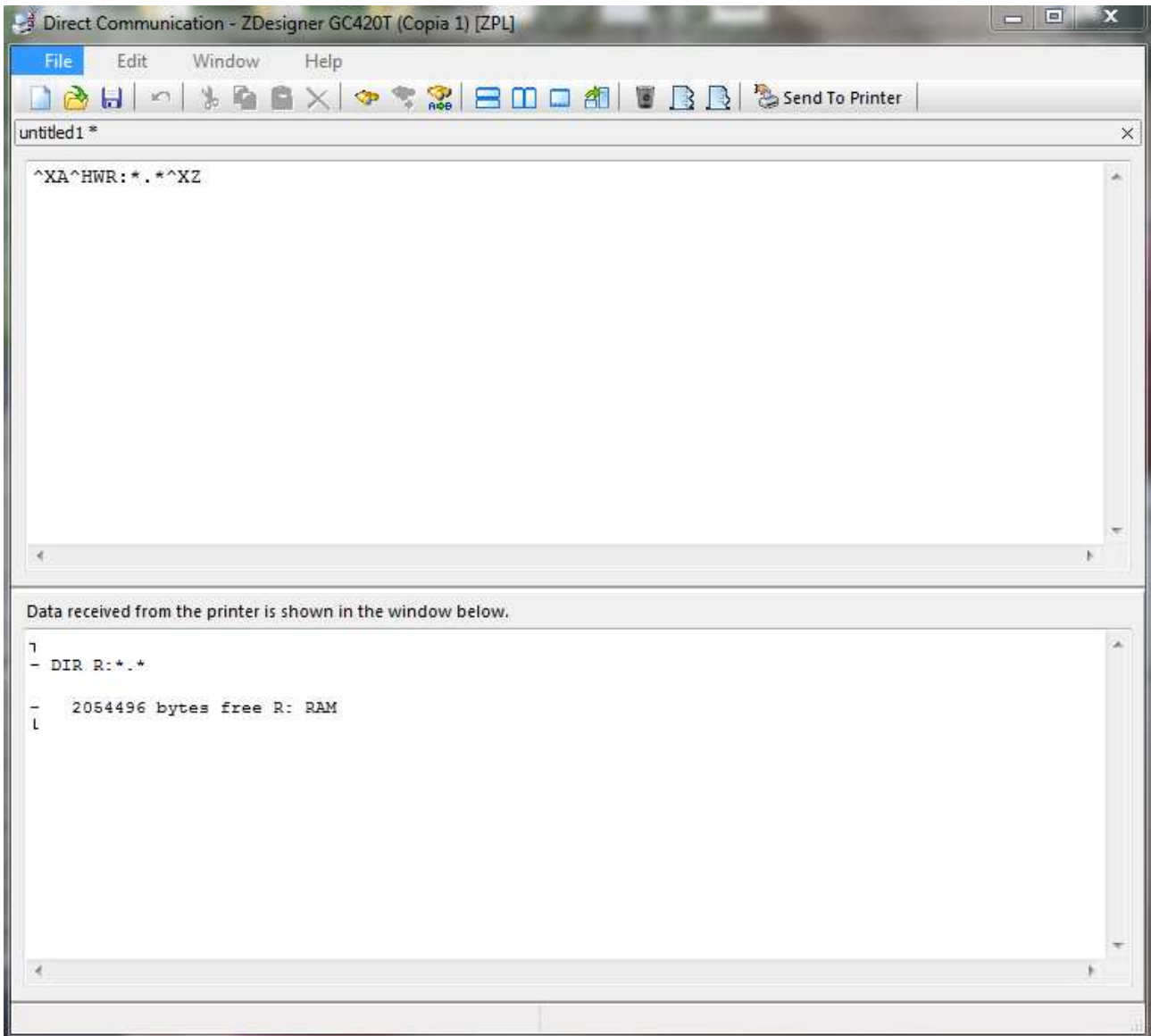
In the window displayed you can write the poll commands in the top box and the results are shown in the bottom box. To display the fonts saved in the flash memory "E" type the command "`^XA^HWE.*:*^XZ`" and press "Send To Printer".



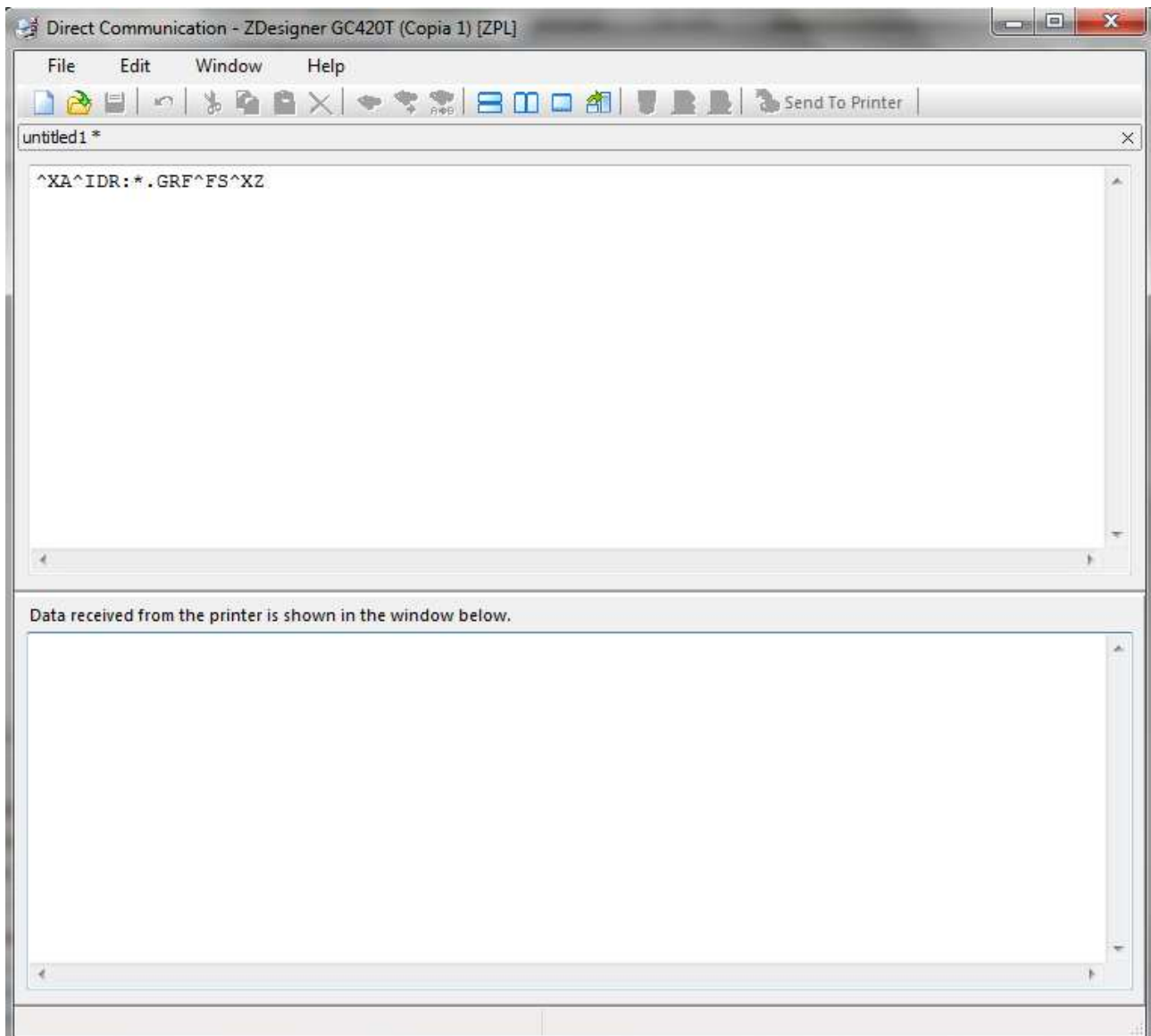
To delete a font saved in the ram memory “E”, for example the “Arial” font, type the “^XA^IDE:ARI000.FNT ^XZ” command and press “Send To Printer”.



To display the graphic objects saved in the ram memory “R” type the command “^XA^HWR:\*. \*^XZ” and press “Send To Printer”.



To delete all the graphic objects saved in the ram memory "R" type the command `^XA^IDR:*.GRF^XZ` and press "Send To Printer".



## 9.11 AT automatic printer

### 9.11.1 Notes

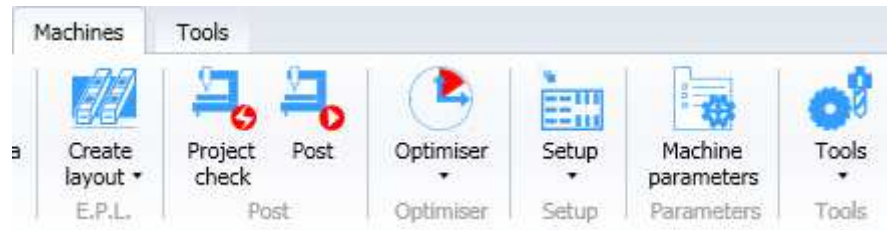
For the AT automatic printer use the labels editor supplied with the labeller software. To generate the correct label position use the Maestro "Label editor" to create an empty label with the same dimensions of the label that will be applied by the automatic labeller and select it during nesting.

The maximum number of parametric fields that can be used with the AT automatic printer is 14.

# 10 Tool Data-Base

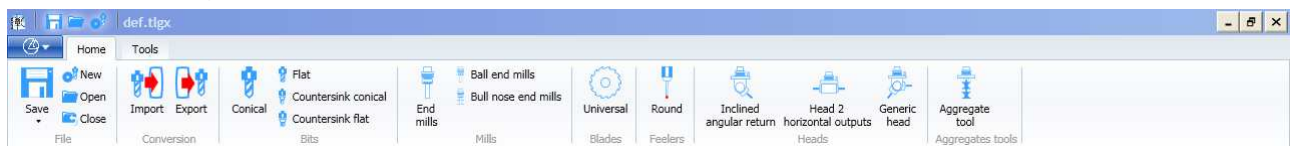
The tool data-base includes all necessary information to choose the tools for the working processes as it has been established.

In order to manage the tool data-base you use the "ToolManager.exe" , launched by clicking on hte icon "Tools" of the menu "Machines".



Tool Manager can be also launched from the installation directory by clicking on the file "ToolManager.exe" .

The Tool Manager Interface is shown as follows:



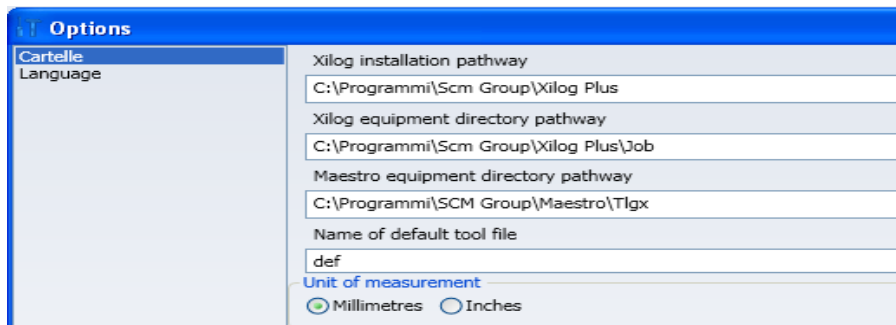
## 10.1 Default for tool Data-Base

It is possible to change the Data-Base defaults by clicking on the menu "Options" in the folder "Tools"



### 10.1.1 Default "Name Tool Data-Base "

In order to modify the default name of a new tool Data-Base:  
Click on the icon "Opzioni"  
Fill the field "Name of the default tool file".



### 10.1.2 Default "Folder for tool Data-Base"

In order to modify the folder name where toll data-base are open/saved:  
Click on the icon "Opzioni"  
Fill the field "Path of the Tlgx export directory"

### 10.1.3 Default "Folder for Import/Export toll Data-Base"

In order to modify the default name in which tool Data-Base are imported exported:  
Click on the icon "Opzioni"  
Fill the field "Path of the Xilog Bin directory".



## 10.2 Tool Data -Base creation

In order to create a tool data-base, you use the command " in the menu "Home" "File".



The data-base will be empty but structured with following categories:

- + Boring bits
- + Milling cutters
- + Blades
- + Contact tools
- + Heads
- + Aggregates

## 10.3 Tool data-base opening

In order to open a tool data base, click on the icon "Open".

A window will appear where you can select a Tlgx file.

The default folder will be defined as described in paragraph 10.1.2. Default "Folder for tool Data-Base"

Then select a Tlgx file and the button open

Or click twice on the Tlgx file.

If a tool data-base has been already open, the option "Tool data-base closing" would be automatically performed.

## 10.4 Closing tool data-base



In order to close an open tool data-base, click on the icon "Close".

If the tool data-base has not yet been saved, following question will appear:

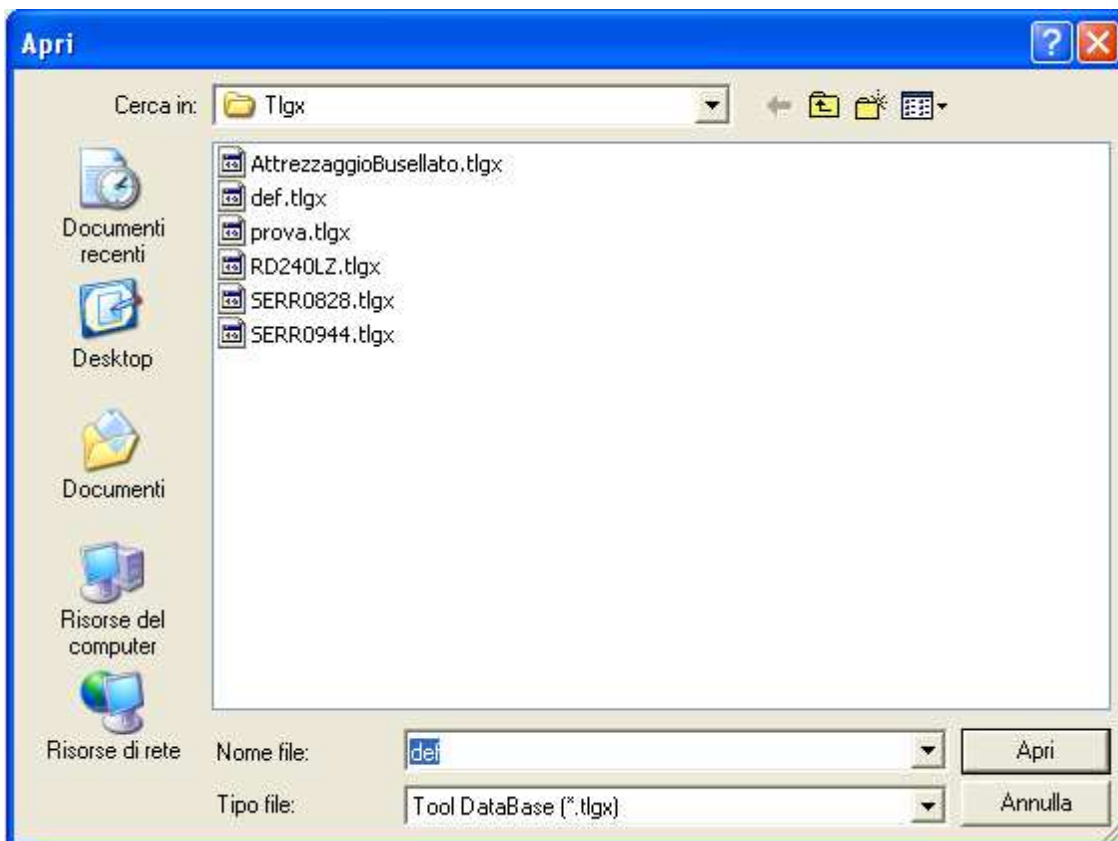


By clicking on "Yes", the option "Saving tool data-base" will be automatically performed.

## 10.5 Saving tool data-base



In order to save an open tool data-base, click on the icon "Save"  
If the tool data-base had already been named, no question would be made.  
If the project was already created, the option "Save con nome" would be performed.



The default folder proposed for the saving is set as described in paragraph 10.1.2.. Default "Folder for tool Data-Base"

You can specify the data-base name in the folder "File Name".  
then click on "Save".

## 10.6 Tool data-base import from Xilog



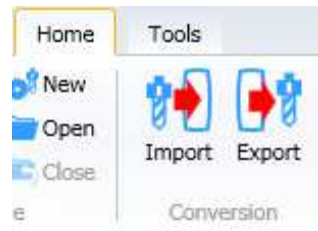
In order to import a tool data-base from Xilog, use the command "import" in the menu "Home" "Conversion".



The default folder proposed to import a tlg file is set as in paragraph 10.1.3 Default Folder Import/Export tool Data-Base.

You can also specify the tool data-base Xilog (tlg) in the field "File name:" then click on "Open".

## 10.7 Tool data-base export to Xilog



In order to export a tool data-base to Xilog (tlg) in the menu "Home" "Conversion".

select the file. Tlgx to be exported from the dialogue window and click on the button "Open".

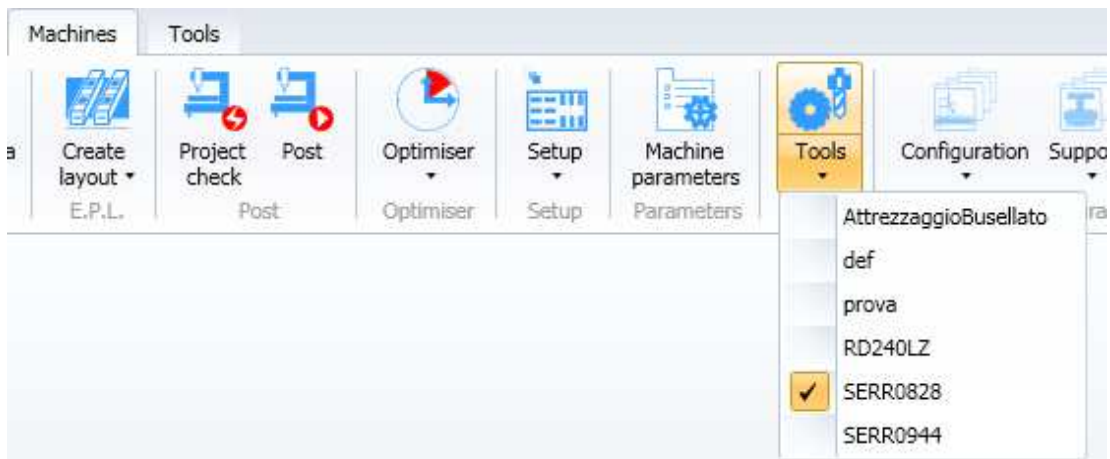
Following message will appear to confirm the export:



## 10.8 Activate Tool Data-Base

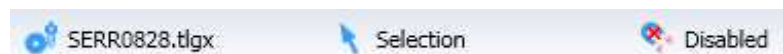
In order to insert any working process on a sole project, you have to select a tool data-base to be activated since the working processes are described by combining a tool and a 2D geometry.

In order to activate a tool data-base, use the menu "Machines" "Tools".  
By clicking on the button "Tools"; a list of available data-base will appear in the "default" folder.



The tool data-base marked on the left is the active one.  
In order to activate another one, select it with the left button of the mouse.

You find the information about the tool data-base is to be found also in the lower workpiece to the right of Maestro interface.



## 10.9 Tools

The tools are the object used for working processes.

In order to use them, you need to know all geometric and technological information of every tool.

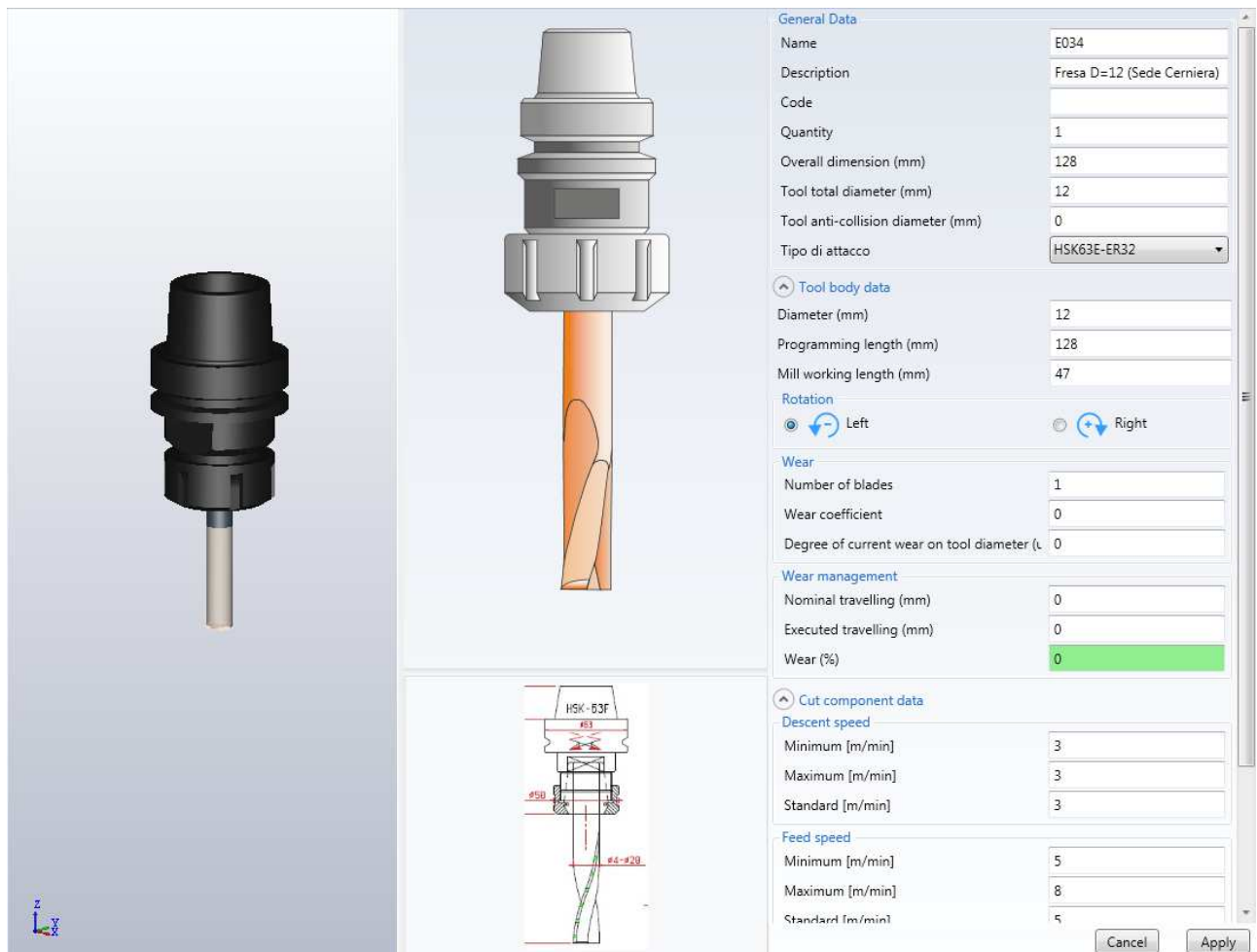
The information collected must be inserted in a tool Data-Base

### 10.9.1 Tool introduction

In order to insert a new tool in the data-base, you use one of the commands in the following menu:



The interface that appears is the following:



The image is divided into four zones:

- In the left area, you can see a picture of the tool
- In the upper central area, you can see the picture of the type of tool
- In the lower central area, you can import an image or a drawing of the tool
- In the area to the right, you can enter the tool data.

An image or a drawing of the tool can be added by clicking in the lower central area where the following image is initially seen:



In the area to the right, rather, the tool data is entered:

- The data at the top is the "General data" and is common to all tool types
- The subsequent data varies according to the type of tool.

After selecting the command about the tool you have to fill the fields in the menu "General Data".

General Data	
Name	E034
Description	Fresa D=12 (Sede Cerniera)
Code	
Quantity	1
Overall dimension (mm)	128
Tool total diameter (mm)	12
Tool anti-collision diameter (mm)	0
Tipo di attacco	HSK63E-ER32

These data are common for all tools, for every category will be necessary to insert the data related to the tool type.

The code used during machining must be entered in "Name" field. We recommend using the historical code used on Xilog "E001" "E002" .... Etc. for congruence, even if not essential.

The main mill features can be reported in the description field to immediately understand which tool it is about.

The tool producer code or one chosen by the customer can be entered in the Code field.

The quantity represents the number of identical tools that are available. This figure is important when setting up a boring machine

The remaining main data three values represent mill's clearance.

The "Type of taper" represents the part of the tool that lies between the spindle and the actual tool.

The geometric data of the tool is always accompanied by a dynamic image that varies upon moving the mouse over the various fields to be filled out order to render the value that must be entered more comprehensible.



The image shown in the left area is updated according to the values set each time the "Apply" button is clicked

The image is made up of four significant elements. From top to bottom are:

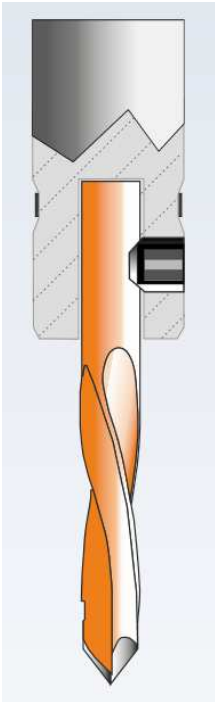
- The type of attack (cone HSH of black color)
- The part of the non-cutting tool (tang of gray color)
- The part of the cutting tool (other tool)
- A dashed circle that represents the programming diameter that will follow the machining geometry  
This circle in the end mills will always be in the lower part of the milling cutter while in the shaped milling cutters it will be possible to find in any position and not necessarily coinciding with an edge of the milling cutter.



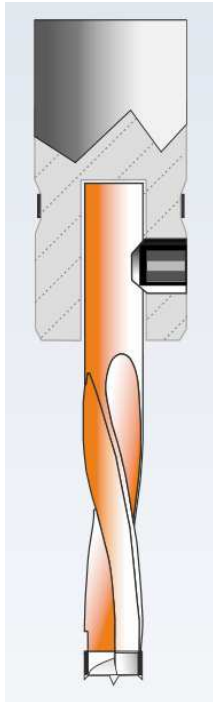
### 10.9.1.1 Bits

Bits drill are divided into following categories:

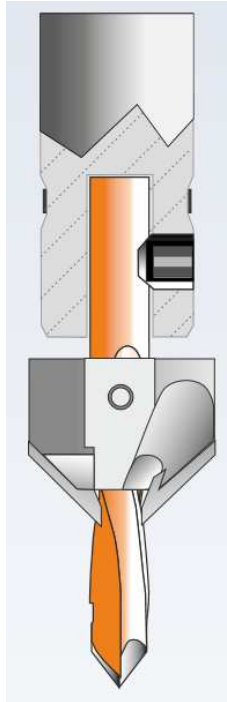
Lance Bit



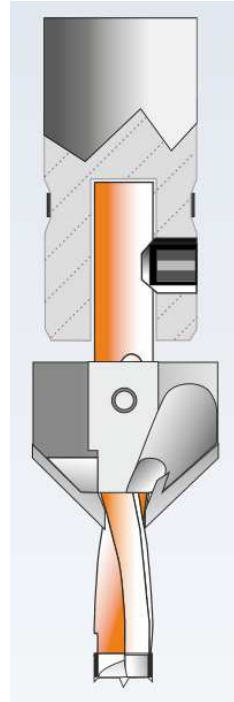
Flared Bit



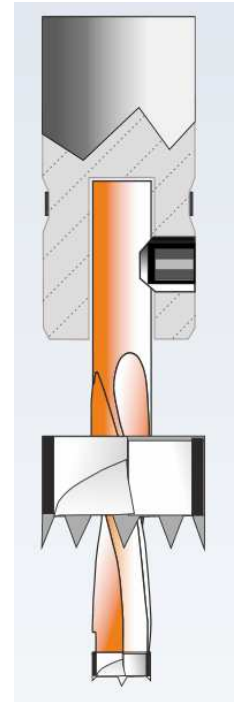
Lance Bit  
Flared



Plain Bit  
Flared



Flat bit  
Blitz



In order to insert a Bit in the data-base, click on the command related to the wished head in the menu "Boring bits" and fill all fields in order to describe the point shape and use properties (speed, rotation number and so on)

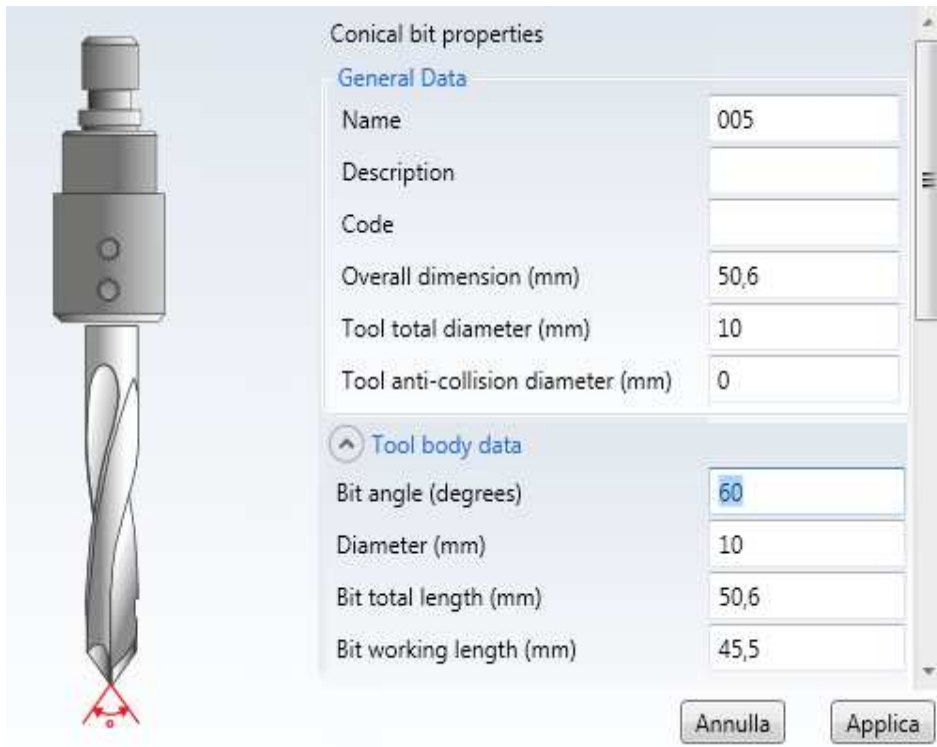
See description and dynamic bitmaps appearing on some values when going over them with the mouse for individual parameters' meaning.

**NOTE:**

For the lance tools the boring cycle is affected by the panel thickness and the angle set in the relative "Tool angle (degrees)" field.

Set the correct angle to achieve a correct boring cycle.

As an alternative to the tip angle, the height of the cone can be inserted.



### 10.9.1.1.1 Tools wear

For all tools, the wear condition can only be seen if there is a quantity of 1.

To do so, set:

"Maximum number of uses" for tips

"Nominal distance (mm)" for cutters or blades

Panel Mac will record the number of uses for drills and the path taken for blades and cutters.

The data identified will be entered into the effective use field and percentage wear field.

The wear range will be green, yellow or red depending on the state of wear.

Wear management	
Maximum number of uses	100
Number of uses executed	0
Wear (%)	0

## 10.9.1.2 Milling cutters

Cutters are divided into following categories:

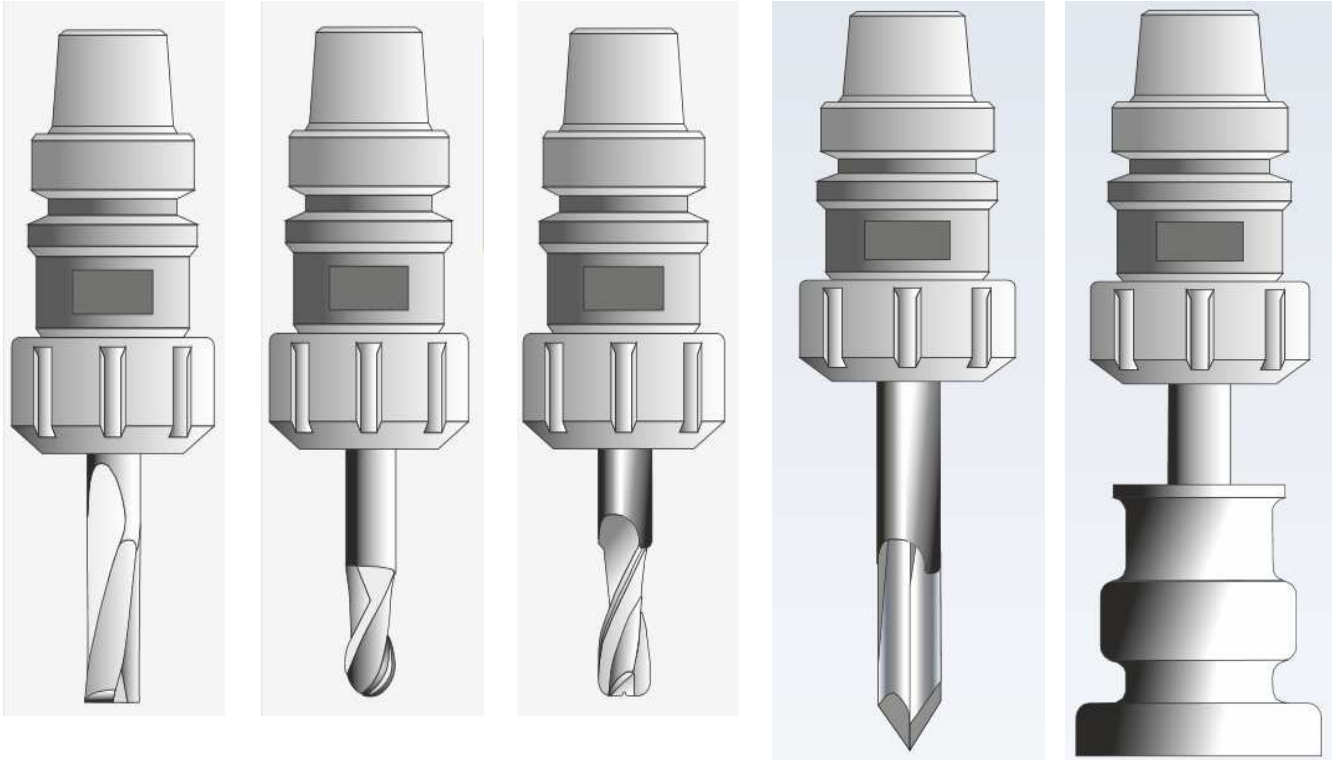
Cylinder cutter

Spheric Cutter

Toroidal Cutter

Conical Cutter

Shaped Cutter



In order to insert a cutter in the data-base, click on the cutter type in the menu "Milling cutters" and fill all fields to describe the cutter shape and use properties (speed, rotation number and so on).

See description and dynamic bitmaps appearing on some values when going over them with the mouse for individual parameters' meaning.

In this category, there are also cutters with specialised uses:

- Sanders
- Edge scraper
- Glue scraper
- Trimmers

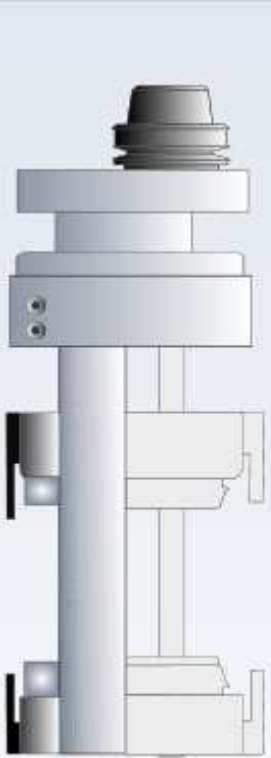
### 10.9.1.2.1 Sanders

This type of tool is used to increase the finish of a machining process by using an abrasive sheet.

### 10.9.1.2.2 Edge scraper

The EdgeScraper tool is used during the edging process to scrape the edge. It consists of a pair of knives that scrape the edge applied to a profile. The EdgeScraper is loaded on a horizontal double output head and is always paired with the trimmer tool.

The tool General Data area lists the machining data as well as the tool encumbrance data:



General Data	
Name	Edge scraper
Description	
Code	
Quantity	1
Overall dimension (mm)	120
Tool total diameter (mm)	30
Tool anti-collision diameter (mm)	0
Work height	
Maximum machining height (mm)	85
Minimum machining height (mm)	0
Times	
Tool opening time (sec)	2
Tool closure time (sec)	2
Edges	
Minimum edge thickness (mm)	0
Maximum edge thickness (mm)	1

The Edge Scraper operation is defined with the Track feeling and Time parameters: the tool rests on the panel being scraped with the tool Horizontal and Vertical Track Feeling, then it closes on the panel in a time defined in the Tool Closure field. At the end of the machining the tool, before exiting the panel, will open by a time defined in the Tool Opening field.

The Edge fields define the scraper operation interval, that is the minimum and maximum edge thickness that the tool is able to scrape.

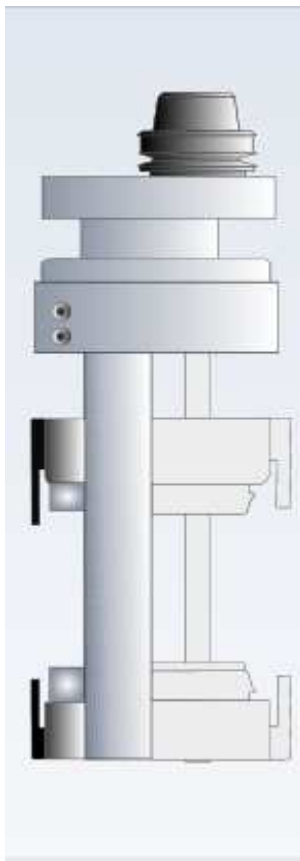
### 10.9.1.2.3 Glue scraper

The GlueScraper tool is used during the edging process to scrape the adhesive used to glue the edge.

It consists of a pair of small blades that scrape the excess glue after an edging.

The GlueScraper is loaded on a horizontal double output head and is always paired with the trimmer tool.

The tool General Data area lists the machining data as well as the tool encumbrance data:



General Data	
Name	Glue scraper
Description	
Code	
Quantity	1
Overall dimension (mm)	120
Tool total diameter (mm)	30
Tool anti-collision diameter (mm)	0
Work height	
Maximum machining height (mm)	85
Minimum machining height (mm)	0
Times	
Tool opening time (sec)	2
Tool closure time (sec)	2
Tool body data	
Scraper diameter (mm)	10
Scraper useful length (mm)	60

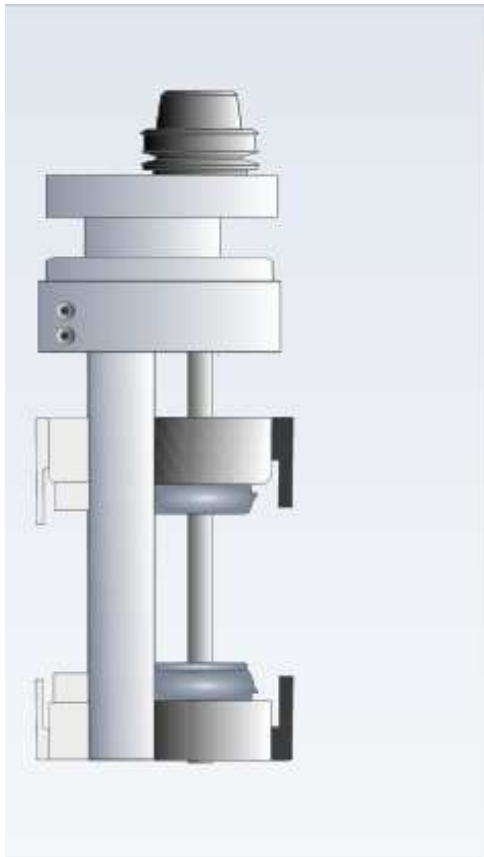
The Glue Scraper operation is defined with the Track feeling and Time parameters: the tool rests on the panel being scraped with the tool Horizontal and Vertical Track Feeling, then it closes on the panel in a time defined in the Tool Closure field.

At the end of the machining the tool, before exiting the panel, will open by a time defined in the Tool Opening field.

#### 10.9.1.2.4 Trimmers

The trimmer tool is used during the edge banding process to finish the edge. It consists of a pair of cutters that trim the edge applied to a profile after the edge banding and overhangs from the upper and lower side of the panel. The trimmer is loaded on a horizontal double output head and is always paired with the scraper tool.

The tool General Data area lists the machining data as well as the tool encumbrance data:



General Data	
Name	Trimmer
Description	
Code	
Quantity	1
Overall dimension (mm)	120
Tool total diameter (mm)	30
Tool anti-collision diameter (mm)	0
Work height	
Maximum machining height (mm)	85
Minimum machining height (mm)	0
Times	
Tool opening time (sec)	2
Tool closure time (sec)	2
Edges	
Minimum edge thickness (mm)	0
Maximum edge thickness (mm)	1

The trimmer operation is defined with the Track feeling and Time parameters: the tool rests on the panel being trimmed with the tool Horizontal and Vertical Track Feeling, then it closes on the panel in a time defined in the Tool Closure field.

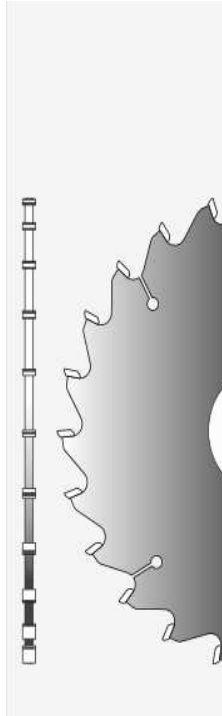
At the end of the machining the tool, before exiting the panel, will open by a time defined in the Tool Opening field.

The Edge fields define the trimmer operation interval, that is the minimum and maximum edge thickness that the tool is able to trim.



### 10.9.1.3 Blades

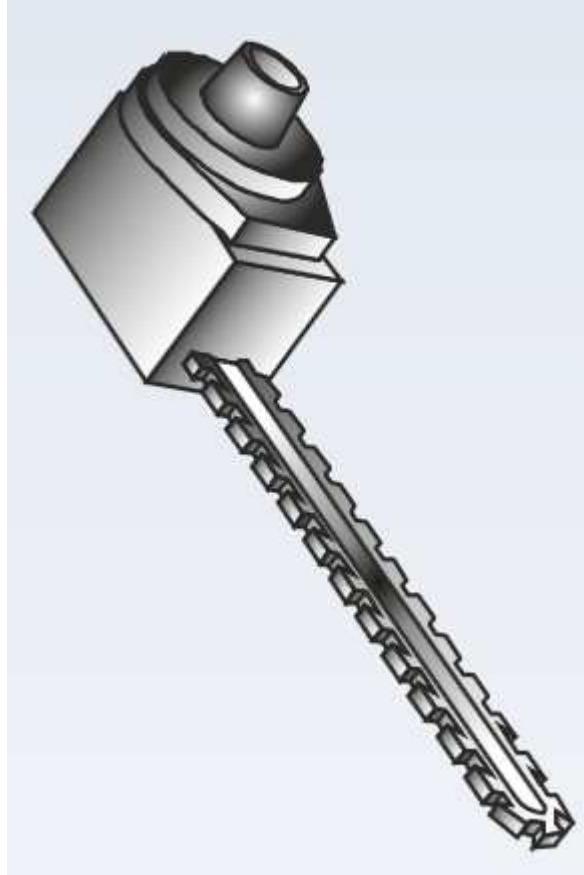
Blades are all contained in the category "Blades".



In order to insert a blade in the data-base, click on the command "Universals" in the menu "Blades" and fill all fields to describe the blade shape and use properties (speed, rotation number and so on)

#### 10.9.1.4 Mortises

Mortises are used to make sharp-edged eyelets.



The data to be inserted is very particular and different from the other types of tools

See description and dynamic bitmaps appearing on some values when going over them with the mouse for individual parameters' meaning.

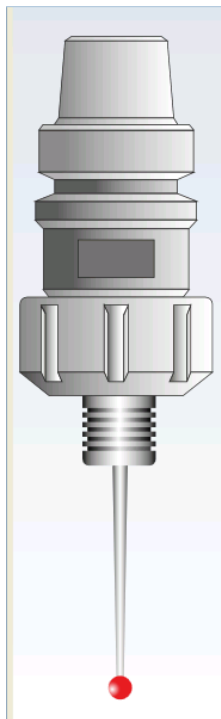
### 10.9.1.5 Contact tools

This category of tools does not undertake removals and is subdivided into:

- Probes
- Pressure rollers

#### 10.9.1.5.1 Probes

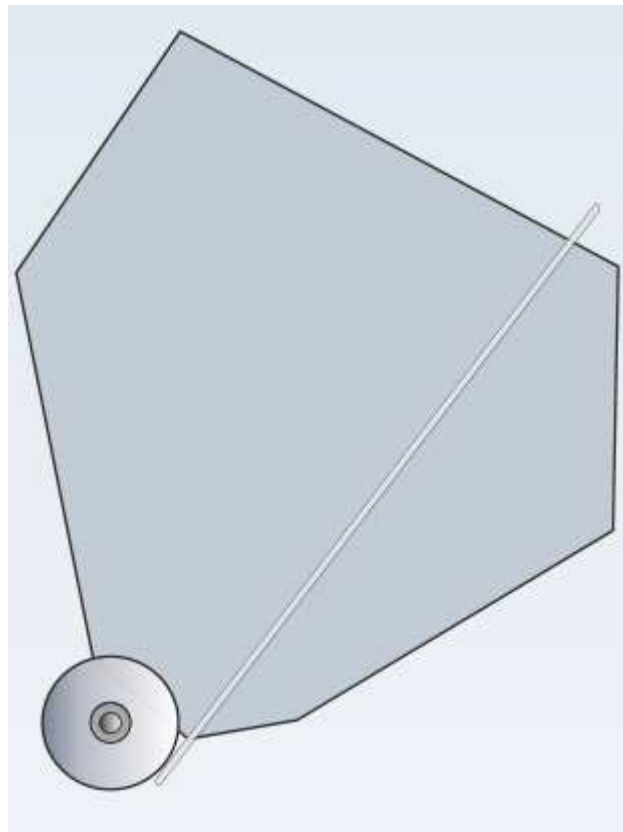
Probes are all included in the field "Tools for contact".



In order to insert a probe in the data-base, click on the command " sphere-shaped" in the menu "Probes" and fill all fields to describe the probe shape and use properties (speed and so on).

### 10.9.1.5.2 Pressure rollers

These components are used to press the edges on the side of the panel.



To insert a new pressure roller, enter the category of Contact tools Pressure rollers, then with a right click on the mouse, select "New pressure roller"

See description and dynamic bitmaps appearing on some values when going over them with the mouse for individual parameters' meaning.

### 10.9.1.6 Special tools

This category includes devices that are not tools but are assembled on spindles.

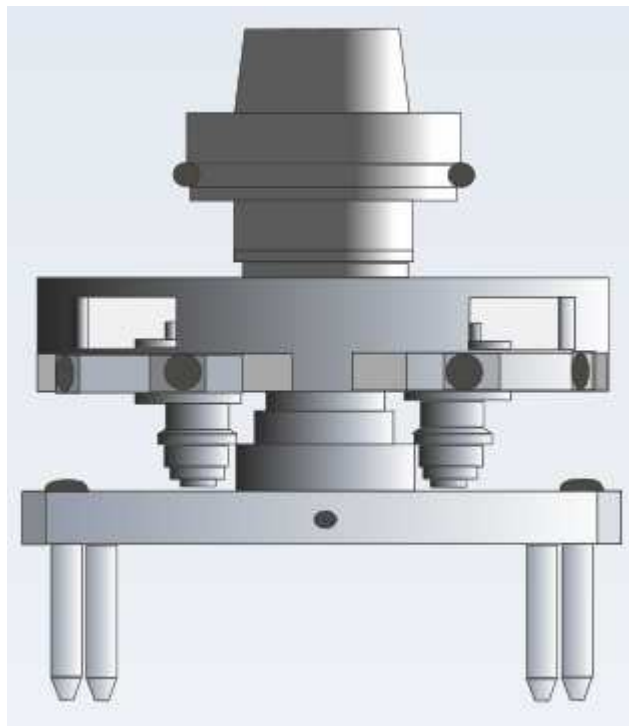
The types in this category are:

- Manipulators
- Blowers

#### 10.9.1.6.1 Manipulators

A manipulator is a device used to move the suction cups on certain types of machines.

To insert a new Manipulator, click on menu "Manipulator tool" from the "Special tools" menu then fill in all relevant fields that appear to describe the shape and characteristics of use.



See description and dynamic bitmaps appearing on some values when going over them with the mouse for individual parameters' meaning.

### 10.9.1.6.2 Blowers

A Blower is a device used to clean certain areas of the machine or parts of the piece.

To insert a new Blower, click on menu "Blower tool" from the "Special tools" menu then fill in all relevant fields that appear to describe the shape and characteristics of use.

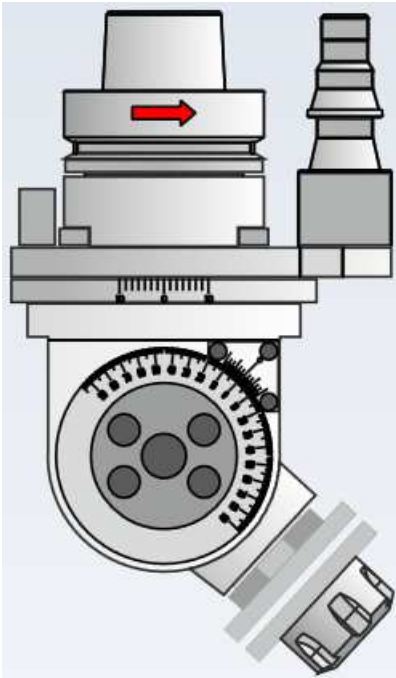


See description and dynamic bitmaps appearing on some values when going over them with the mouse for individual parameters' meaning.

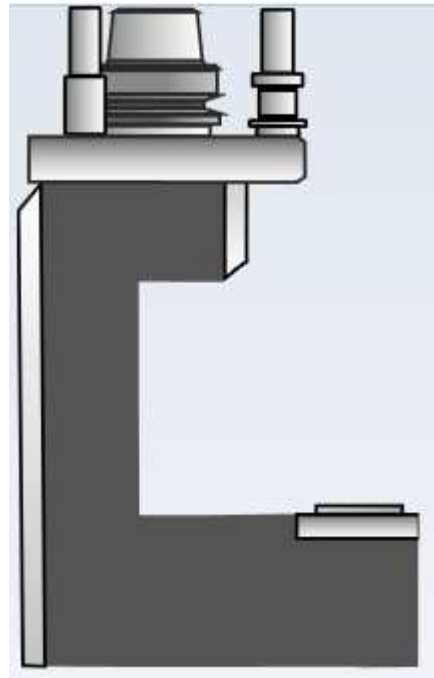
### 10.9.1.7 Heads

The heads are divided into the following categories:

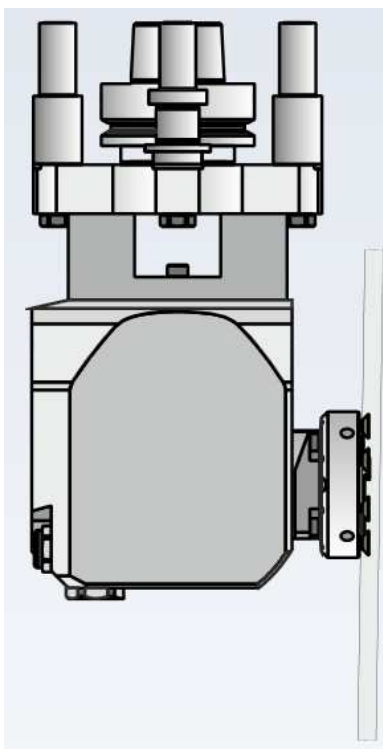
Inclined angular head



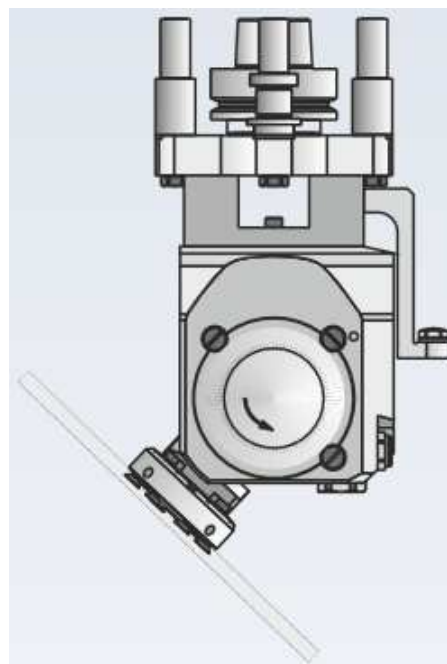
Weldon



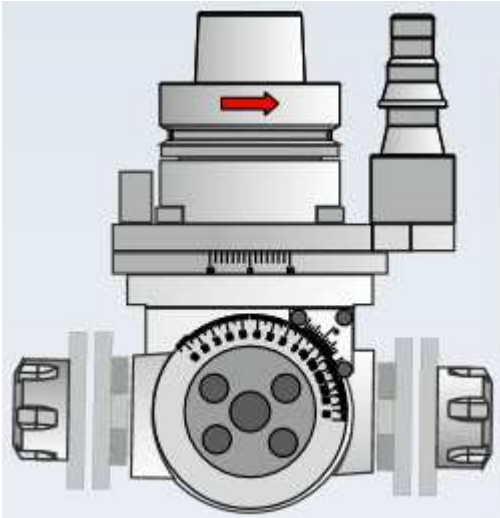
End cutting unit



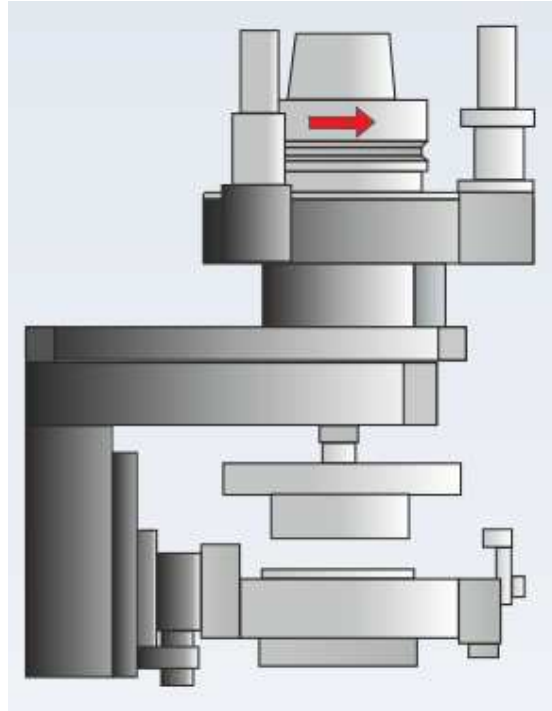
Qinx head



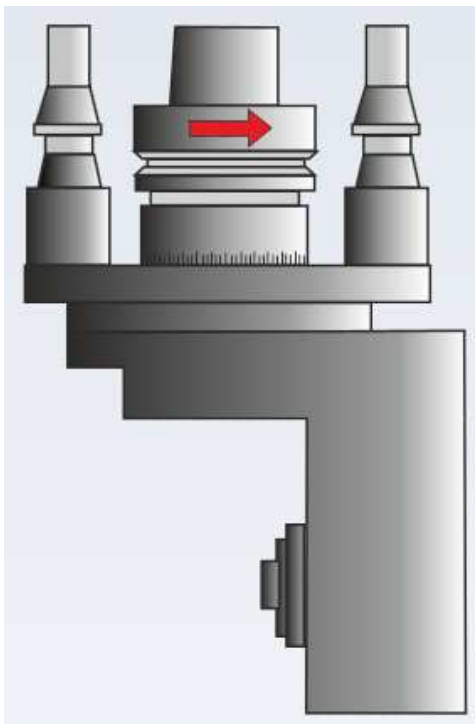
Head 2 horizontal outlets



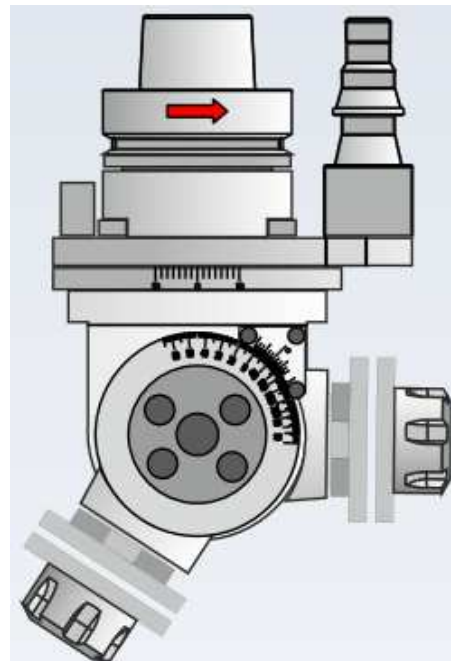
Lower lateral copier



Edge cleaning head

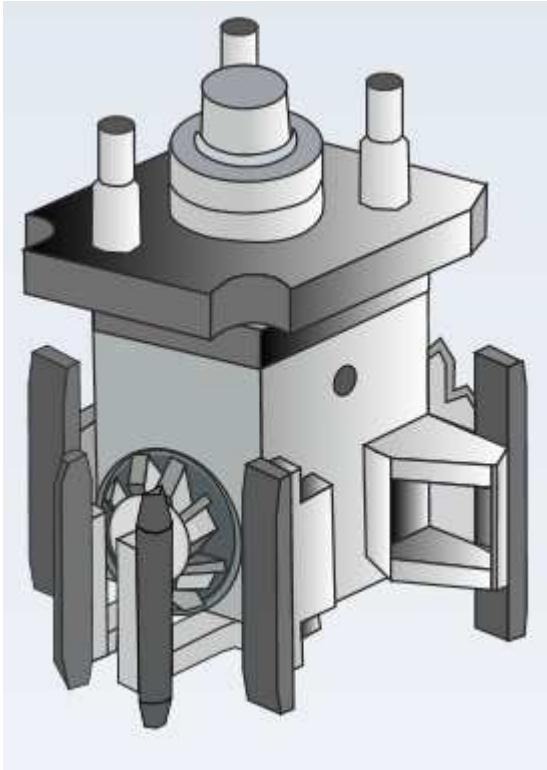


Generic Head

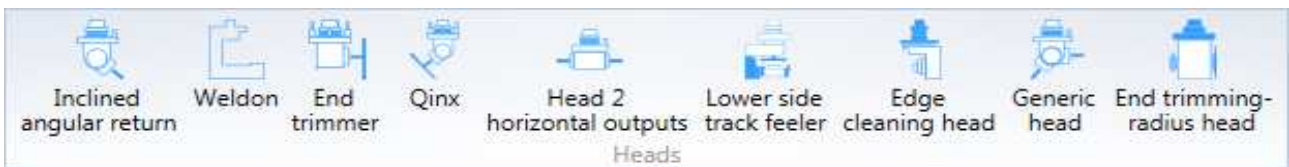




## End cutting unit-rebater head



In order to insert a head in the data base, click on the command related to the head type in the menu "Heads" and fill all fields to describe the head shape and use properties (speed, rotation number and so on).



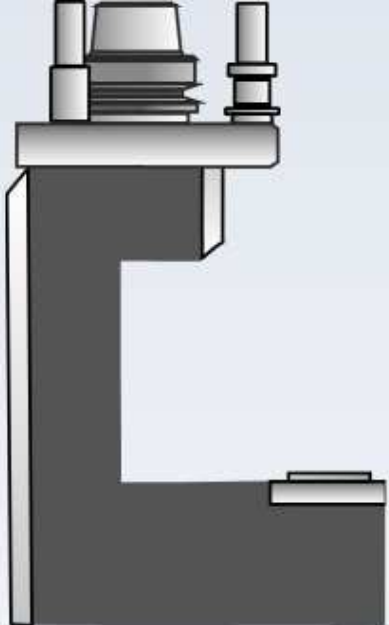
Each head has at least one Spindle on which a previously described tool can be assembled.

See description and dynamic bitmaps appearing on some values when going over them with the mouse for individual parameters' meaning.

### 10.9.1.7.1 Weldon

Of all the cartridges, the Weldon is the only one that can work on the lower face. It is therefore the one that is most at risk of collision with the clamping devices underneath the workpiece.

To avoid collisions, it is very important to configure all head parameters correctly:



Property of the Weldon head

**General Data**

Name	Weldon
Description	Testina che fora da sotto
Code	
Quantity	1
Speed reduction ratio	1
ID key	1

**Parameters**

Overall size (l) (mm)	248,5
Length X positive (mm)	173,5
Length X negative (mm)	50,5
Width Y positive (mm)	42,5
Width Y negative (mm)	42,5
Lateral encumbrance (mm)	173,5

**Technological data**

**Spindle - E203**

**Spindle body**

X offset (mm)	132,5
Y offset (mm)	0
Z offset (mm)	193,5
Angle A	180
Pin angle	0
Spindle diameter (mm)	65
Spindle length (mm)	55
Total length (spindle + tool) (mm)	100

In particular, the parameters that define the head size with respect to the axis of the operating unit: Length X Positive, Length X Negative, Width Y Positive Negative Y Width and Lateral Dimension

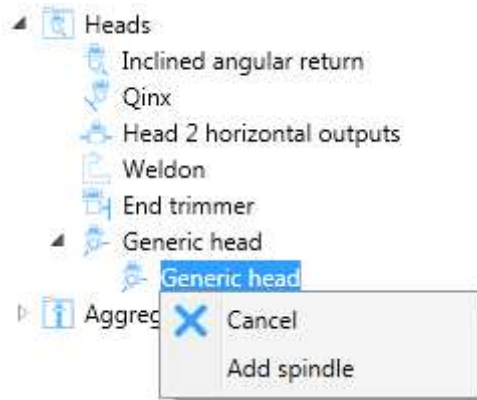
The head oriented with "Pivot angle" at 0 is seen as in the image above, so the dimensions "Positive X Length", "Side Dimension" and "X Offset" are those found on the right of the spindle axis.

### 10.9.1.7.2 General Head

This type of head can be configured with more than one spindle

Spindles can be added to the head of generic heads.

Do this by right-clicking the mouse on the Generic head's name and choose "Add spindle" from the appearing context menu.



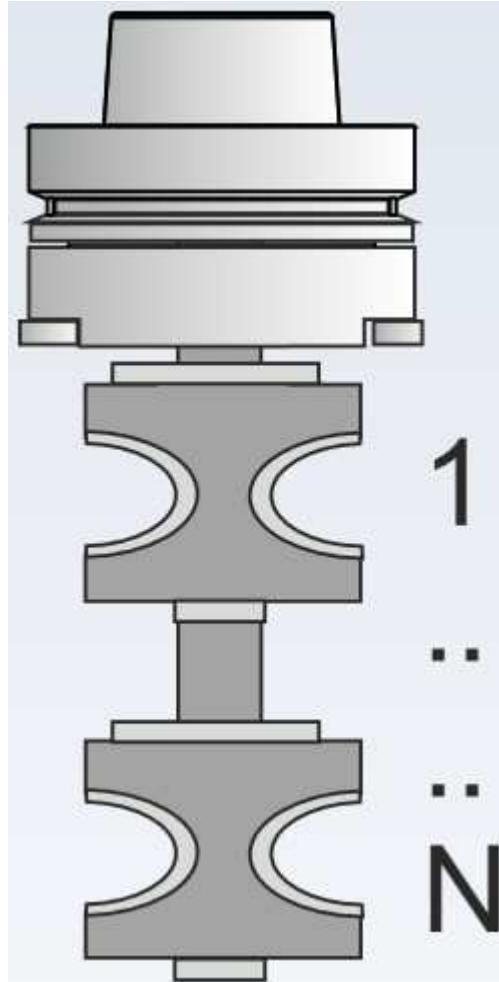
Fill-in new added spindle data.

Proceed to complete defining Generic head.

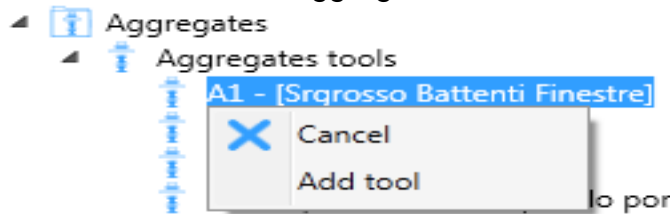
### 10.9.1.8 Aggregates tools

Define an aggregate tool when using a "physical tool" in programming with different tools diameters or useful lengths.

The aggregate tool corresponds to "physical tool" having more virtual tools.



Click on "Aggregate tool" control in "Aggregate tools" menu and fill-in Tool 1 field with previously described tool name, to insert an Aggregate tool in the database. Right-click the mouse on the aggregate name and choose "Add tool" from the appearing context menu to add another tool to the aggregate.



Fill in Tool 2 with the name of another tool. Proceed to complete aggregate definition.

## 10.9.2 Modifying tolls

In order to modify a tool:

- Search the tool in the type folders:
- choose the tool
- modify the values
- confirm the modification by clicking on the button "Apply"

In order to confirm the tool data-base modification, the tool data-base saving must be performed (see paragraph 10.5 )

## 10.9.3 Tool removal

In order to remove a tool data-base:

- Search the tool in the type folders:
- choose the tool
- click with the right button on the tool name
- click on the command "Remove" in the contextual menu

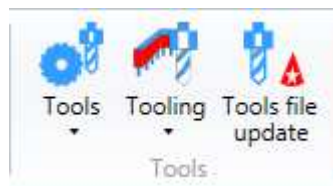
In order to confirm the tool data-base modification, the tool data-base saving must be performed (see paragraph 10.5 )

# 11 Tool equipment

Tool equipment establishes which tools are assembled on the operator groups that do not have a change tool.

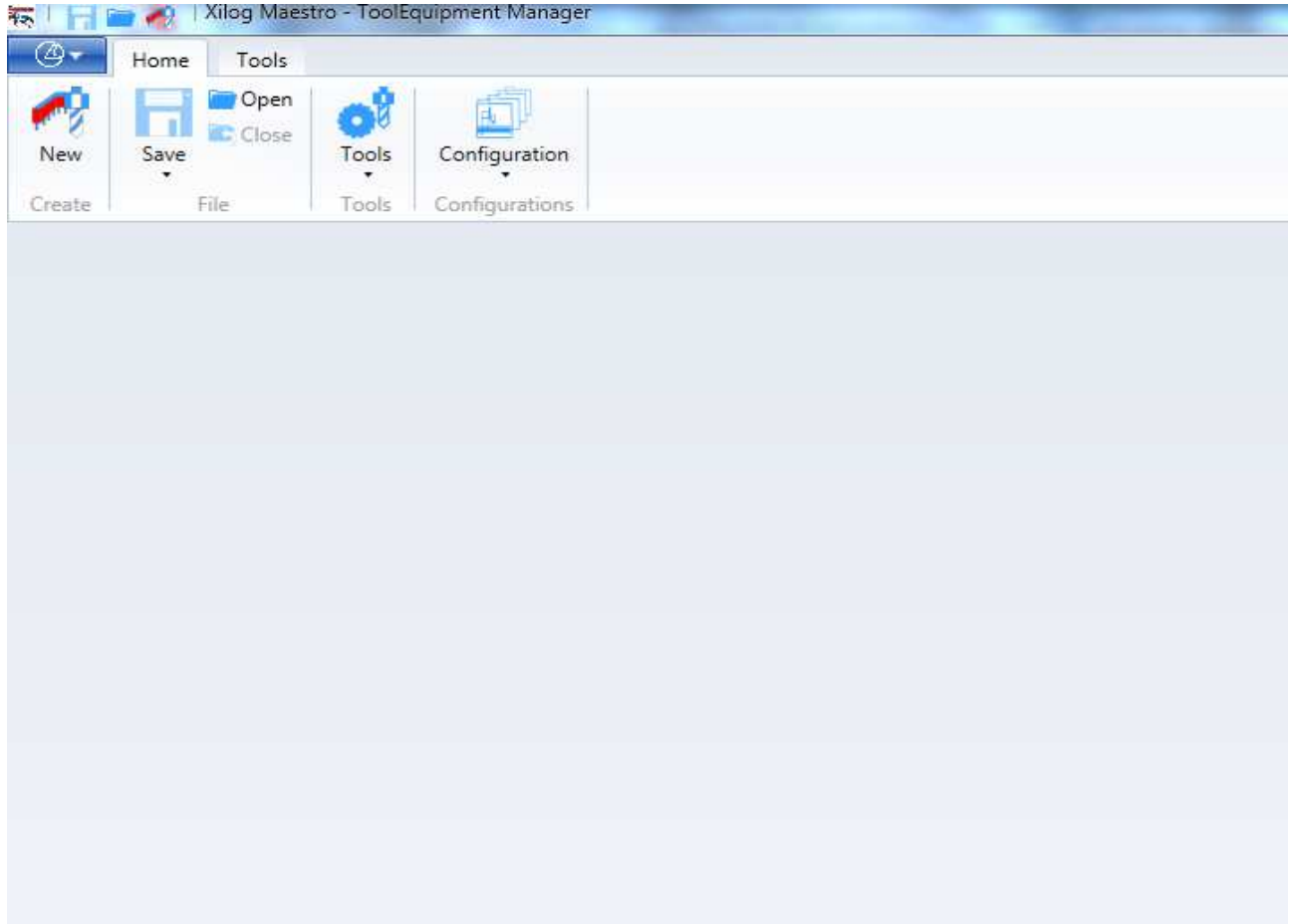
On such operator groups (Borings, Spindles without change tool and the like), tools are to be assembled manually: therefore, the operator must store in the tool equipment section the tools assembled on the spindles.

Tool equipment is managed through the application "ToolEquipmentManager.exe" that can be automatically launched by clicking on the icon "Tool Equipment" of the "Machines" - "Tools" menu.



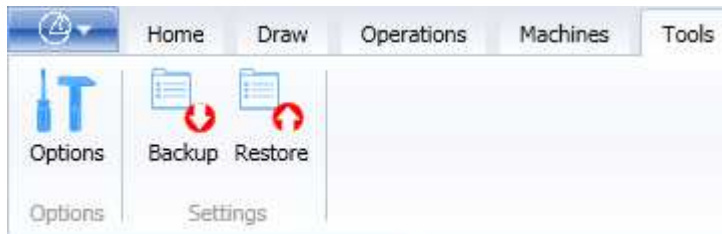
ToolEquipmentManager.exe can also be launched autonomously from the installation directory by clicking on the "ToolEquipmentManager.exe" file.

The Tool equipment interface looks like this:



## 11.1 Tool Equipment default

You can change the tooling defaults by clicking on the "Options" menu in the "Tools" folder of the Master interface.



In order to change the name of the tools default, click on the file "Preferences" and fill in the field "Default tools in spindles tooling file"



## 11.2 Tool Equipment creation

To create the Tool Equipment ,use the “New” command of the "Home" menu, option "Create"



## 11.3 Open Tool Equipment

To open an existing Tool Equipment ,use the “Open” command of the "Home" menu, option "File"



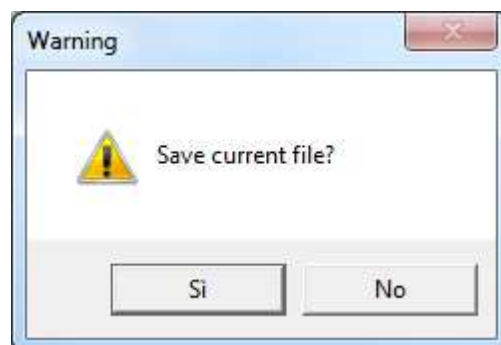
A window will be displayed from which you can select a file with extension .atr

## 11.4 Close Tool Equipment

To close the Tool Equipment ,use the “Close” command of the "Home" menu, option "File"



If the tool equipment database has not been saved yet the following question is displayed:



Clicking “Yes” automatically recalls the “Save Tool Equipment"

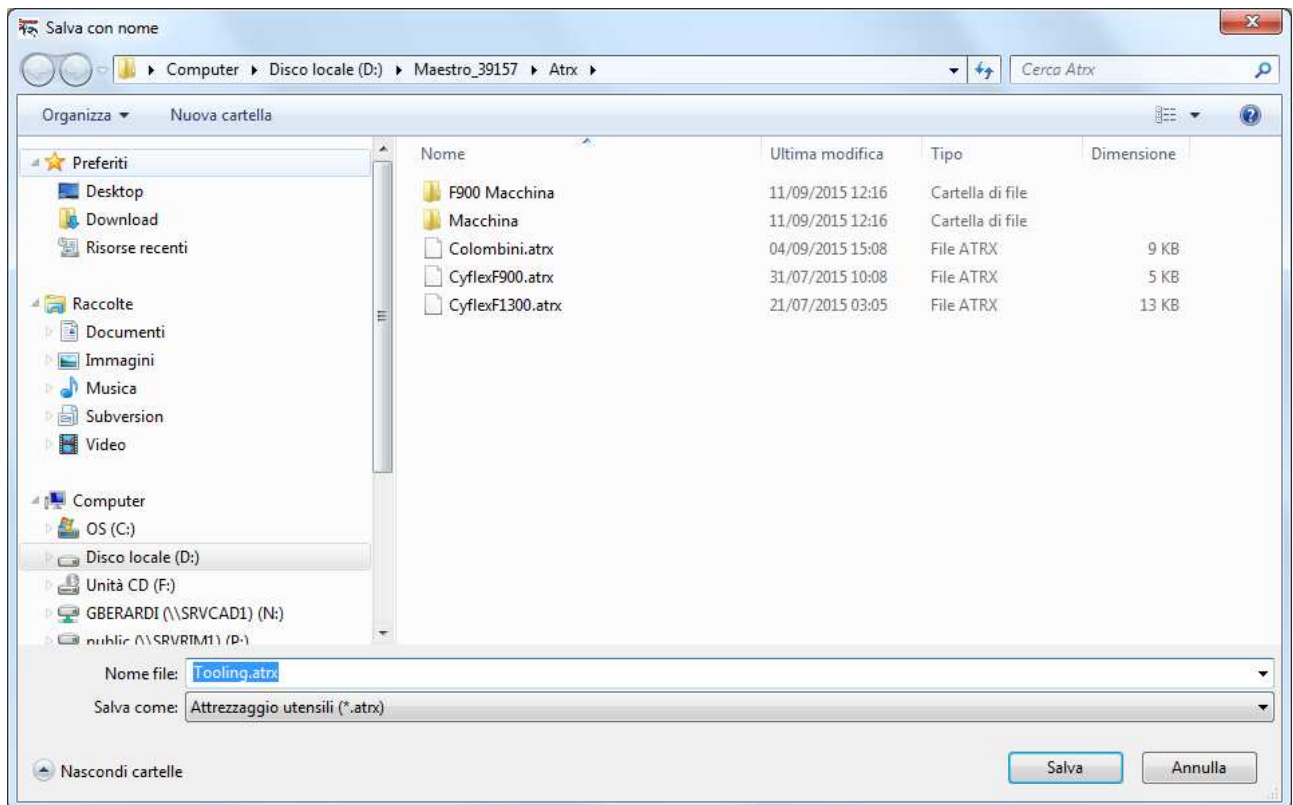


## 11.5 Save Tool Equipment

To save a Tool Equipment file, click on “Save” command of the "Home" menu, option "File".



If the Tool Equipment file already had a name, no question will be asked.  
If the Tool Equipment had been created, "Save with name" function will be recalled

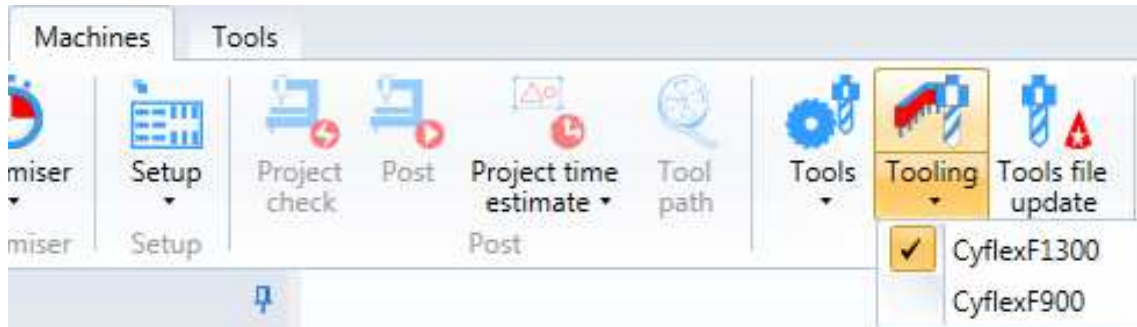


The Tool Equipment name can be specified in the “File name:” box then click on “Save”.

N.B.: The folder where all .atrx files are saved is always the subfolder "Atrx" of the Maestro installation file.

## 11.6 Tool Equipment activation

To activate a tool equipment file use the "Machines" "Tools" - "Equipment" menu. By clicking on the arrow placed under "Tool Equipment", the list of the available tool equipment files will appear.



The tool equipment with a check on its left is the one that is active at that moment. To activate another database simply select it with the left hand button of the mouse.

## 11.7 Tool equipment editor

In order to match Tool with Spindle, select :

- A configuration file (.cfgx)
- A tools data-base ( .tlgx)

### 11.7.1 Configuration selection

Such selection identifies the operator groups available.

Selection is done by clicking on one of the configurations contained in the "Machines - Configuration" menu



### 11.7.2 Tools database selection

This selection allows the identification of the tools database you want to use to equip the machine.

Selection is done by clicking on one of the configurations contained in the "Machines - Tools" menu



### 11.7.3 Machine Selection

If you have selected a configuration containing more than one machine, you can select the machine you wish to equip.

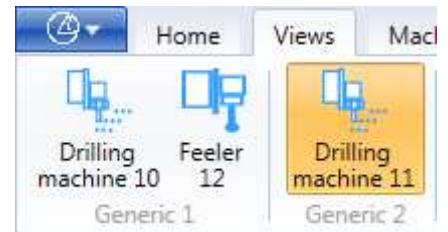
To do this, select the machine visible in the pull-down menu that appears in the "Machines" menu.



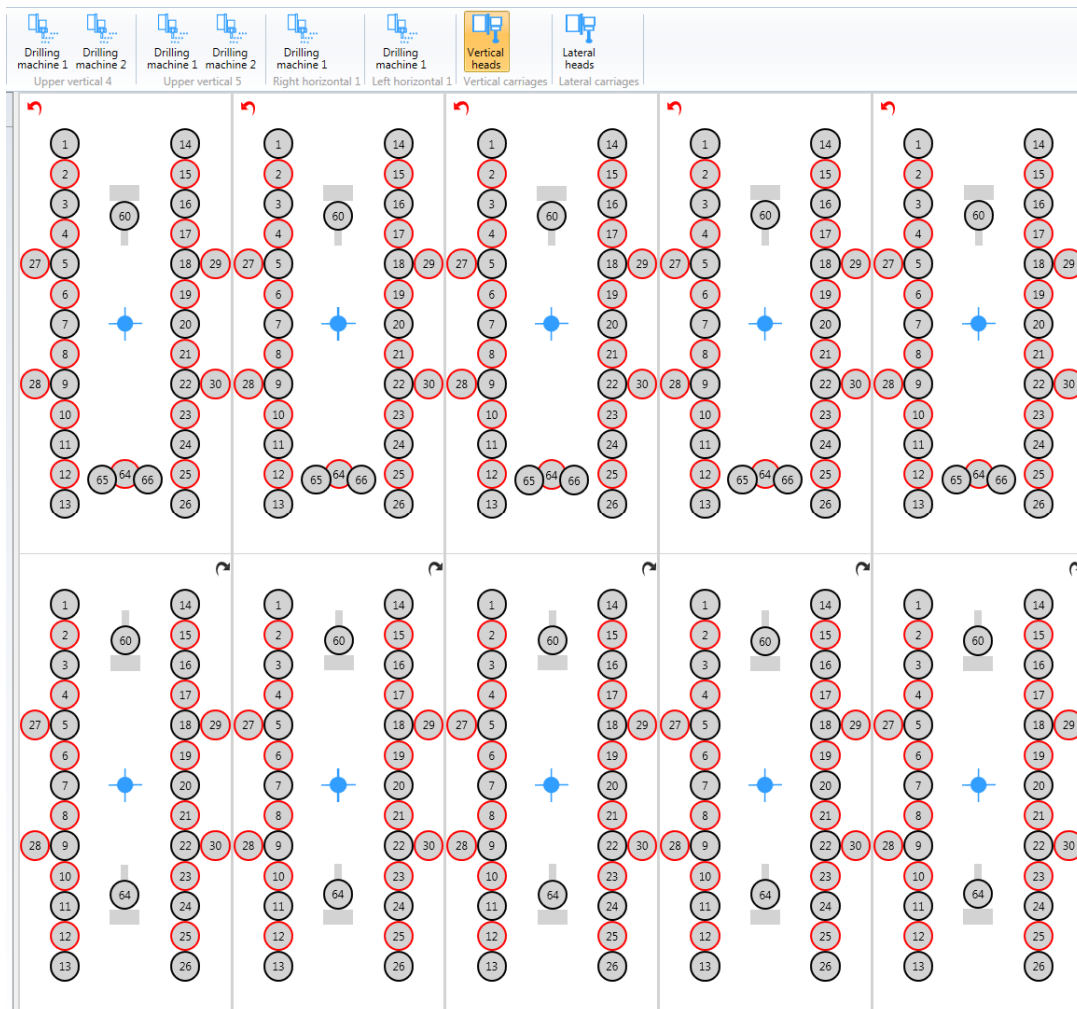
### 11.7.4 Operator Group Selection

If the machine is equipped with more than one Operator Group, you can select the group you want to equip.

To do this, select the operator group visible in the menu "Views".



Alternatively, a series of operator groups of the same type can be displayed simultaneously, e. g. all "Vertical Heads" or all "Side Heads". Obviously, this representation will depend on the machine configuration.



## 11.7.5 Association Tool-Spindle

In order to match a tool with a spindle, use the following interface :

Spindle	
Carriage	Generic 1
Head	Drilling machine 10
Number	10
Description	
X offset	-160
Y offset	-32
Z offset	0
R offset	0
Inclination	0
Table	Upper
Type of tool	Tool
Max length	55
Max diameter	0
Tool	
Rotation	Not defined

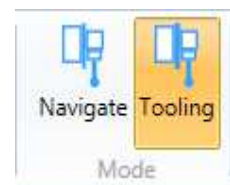
End mill	
Name	D10 L124,7
Description	
Quantity available	3
Total length	124,7
Diameter dimensions	10
Diameter	10
Working length	25
Rotation	Right
Total quantity	3

Where :

- Displays the characteristics of the object type selected.
- On the right there are the tools of the tools database
- In the middle there are the spindles of the selected operator group with the tools that have already been assigned to the spindles.

In order to match a tool with a spindle :

- Activate "Instrument Mode".
- Select a tool (on the right)
- Clicking on all the spindles on which the selected tool is to be assembled. (The spindle number will be replaced with the diameter of the mounted tool.)



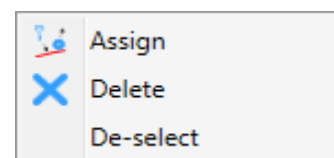
## 11.7.6 Tool removal from the spindle

In order to remove a tool from a spindle :

- Select a spindle and click on "Remove"

"Remove" can be obtained:

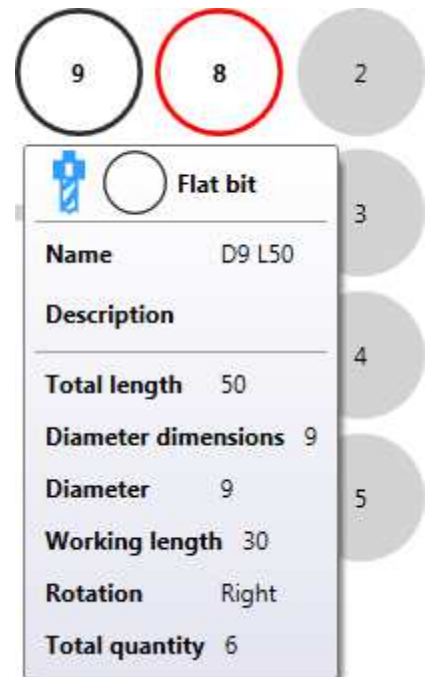
- By right-clicking on a spindle



## 11.7.7 Viewing Tool Information

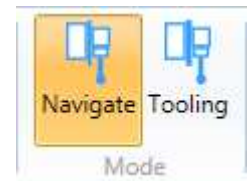
Regardless of the selected mode, it is always possible to have information about the tool mounted on a spindle by simply moving the Mouse onto the tool.  
A Tool-Tip with complete tool information appears.

The colour of the circle representing the tool is provided as additional information:  
The Black Circle represents a rotation right  
The Red Circle represents a rotation left



## 11.7.8 Spindle information display

With "Navigate Mode", you can check the information for each spindle.



By clicking on each spindle in the central area, the information of the selected spindle will appear in the left area.

The selected spindle graphics will be displayed:  
Yellow for empty spindles  
- Green for chucks with a tool.

Spindle	
Carriage	Generic 1
Head	Drilling machine 10
Number	9
Description	
X offset	-128
Y offset	-32
Z offset	0
R offset	0
Inclination	0
Table	Upper
Type of tool	Tool
Max length	55
Max diameter	0
Tool	D5 L50
Rotation	Not defined

## 11.7.9 Displaying Equipment

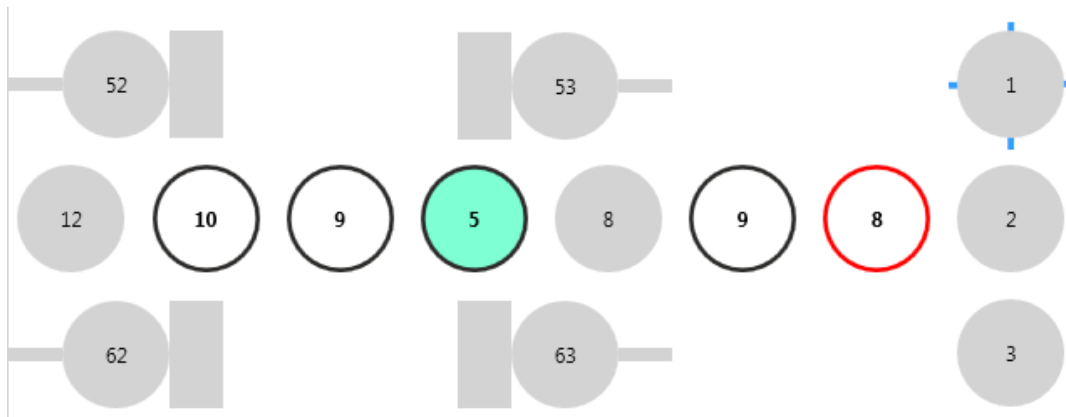


The tooling in the central area can be displayed in three modes:

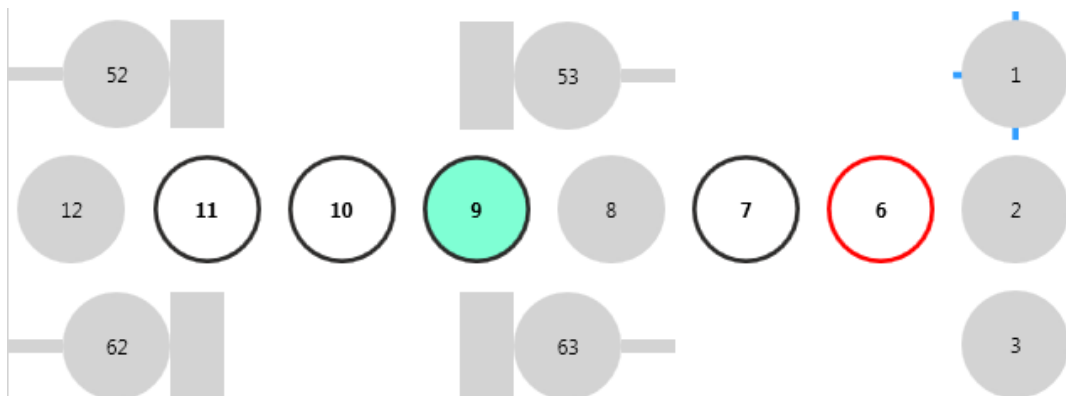
1. Complete
2. Spindles
3. Tools

The "Full View" displays a number on each spindle it represents:

- Spindle name (when the spindle is empty)
- The diameter of the tool (in the case of drill bits)
- The name of the tool (if not a drill bit)

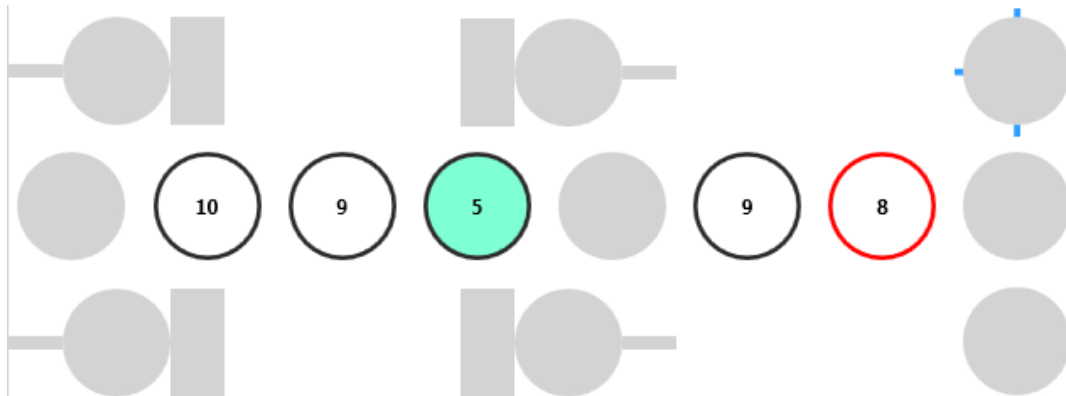


The "Spindle view" displays a number on each spindle representing the spindle name.



In "Tool Viewing" only the numbers they represent will be displayed;

- The diameter of the tool (in the case of drill bits)
- The name of the tool (if not a drill bit)



## 11.8 Print fixturing data

To print the tooling, use the "Print" command in the "Home-File" menu.



The result of printing is a file that:

On the first page shows a series of information related to the file name, the type of measurement unit used and the date of printing, as well as a series of explanatory legends on the document:

### TOOLING REPORT

NAME: C:\SW\Maestro\Sorgenti\UI00\bin\Debug\Atrx\Tooling-test.atrx

UNIT OF MEASUREMENT: millimetres

DATE: 2/13/2018 9:27:43 AM

#### DRILL LEGEND

L = Flat tipped

LS = Flat tipped countersunk

PS = Flat countersunk

PB = bliz flat

CW = Right rotation

**CCW** = Left rotation

#### ROUTER LEGEND

T = Toroidal

S = Spherical

C = Slot mill

L = Sander



The following is a list of the drill bits or cutters required for mounting (with the necessary number for each tool).

### LIST OF NECESSARY DRILLS

Name	Qty	Type	Rot.	Diam.	Len.	Diam.Se.	Leng.Se.
D8 L50	1	P	<b>CCW</b>	8	30	-	-
D9 L50	2	P	CW	9	30	-	-
D5 L50	1	P	CW	5	30	-	-
D10 L50	1	P	CW	10	30	-	-

Finally, there is the assembly diagram for the machine listing the spindles of each head with the tool to be mounted.

### TOOLINGS

#### 1. CYFLEXF1300

##### UNIT 1

TESTA NUMERO 10						
Spindle	Name		Type	Diam.	Len.	Rot.
52			-	-	-	-
53	-		-	-	-	-
62	-		-	-	-	-
63	-		-	-	-	-
64	-		-	-	-	-
65	-		-	-	-	-
1	-		-	-	-	-
2	-		-	-	-	-
3	-		-	-	-	-
4	-		-	-	-	-
5	-		-	-	-	-
6	D8 L50		P	8	30	<b>CCW</b>
7	D9 L50		P	9	30	CW
8	-		-	-	-	-
9	D5 L50		P	5	30	CW
10	D9 L50		P	9	30	CW
11	D10 L50		P	10	30	CW
12	-		-	-	-	-
66	-		-	-	-	-

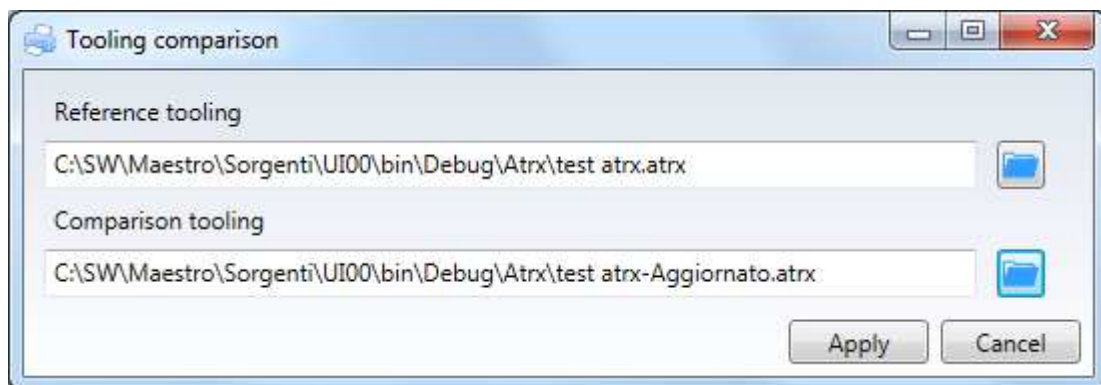
## 11.9 Comparing tooling

After changing an existing tooling, a comparison could be used to evaluate the differences with the previous tooling.

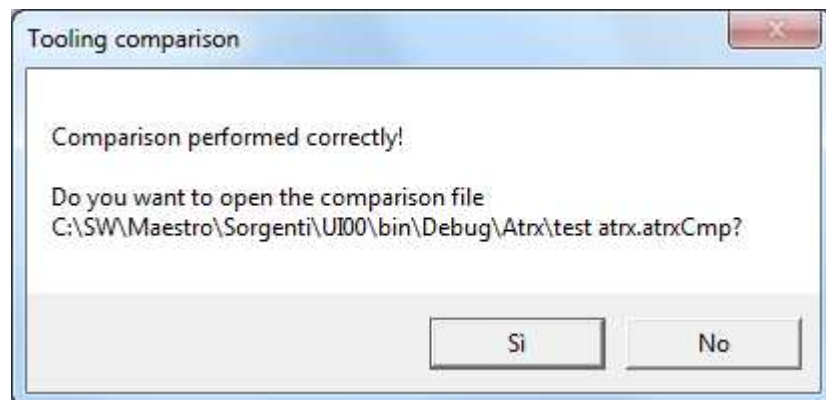
To compare the equipment, use the "Print" command in the "Home-File" menu.



Selecting the command opens a window in which 2 tools must be specified, the current and final one, i. e., the one you want to load into the machine.



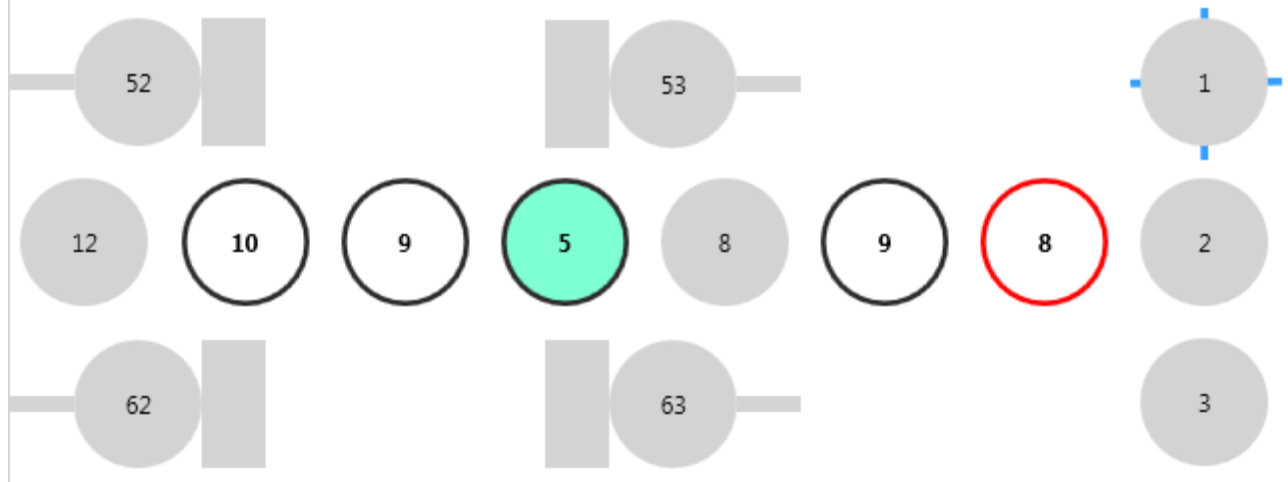
When you select Apply, a comparison file with the same name is generated of the current tooling with a new extension. atrxCmp.



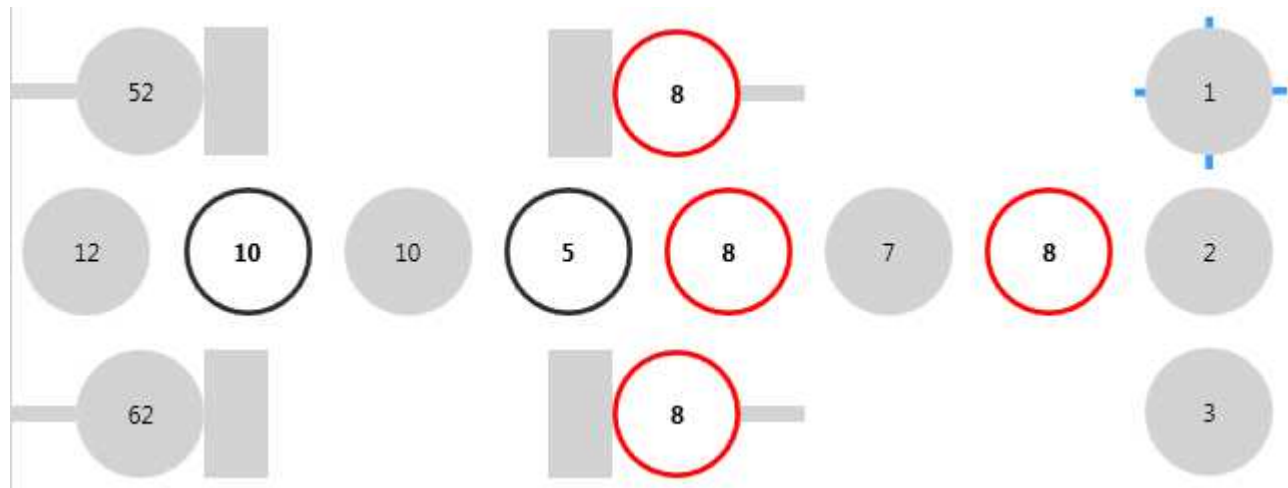
By clicking on "Yes", the comparison file is displayed.

Example:

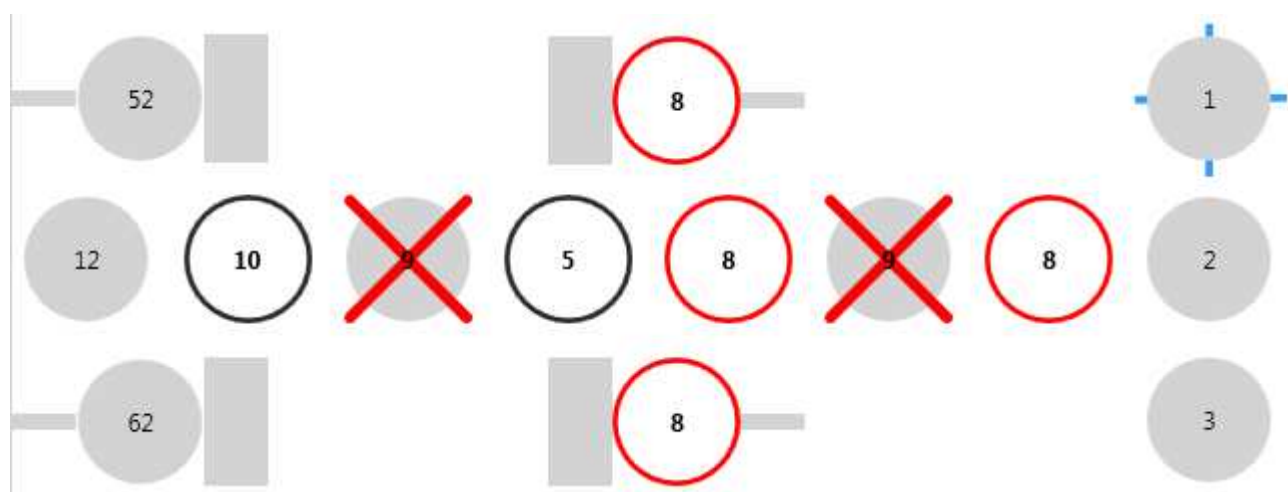
Previous file: atrx. atrx. atrx test



Existing file: atrx test - Updated. atrx test.



Comparison files: atrx. atrxCmp test



In the diagram, the tools to be removed are represented by a red cross.  
 The. atrxCmp file can also be printed, the result is similar to that of tooling printing with some important differences.

- On the start page there is a new legend about the tools to be mounted/disassembled.

## TOOLING COMPARISON REPORT

INITIAL TOOLING C:\SW\Maestro\Sorgenti\UI00\bin\Debug\Atrx\test atrx.atrxCmp

UNIT OF MEASUREMENT: millimetres

DATE: 2/13/2018 11:38:37 AM




### DRILL LEGEND

- L = Flat tipped
- LS = Flat tipped countersunk
- PS = Flat countersunk
- PB = bliz flat
- CW = Right rotation
- CCW** = Left rotation

### ROUTER LEGEND

- T = Toroidal
- S = Spherical
- C = Slot mill
- L = Sander

### SPINDLES LEGEND

-  Tool to remove
-  Tool to fit
-  Tool already fitted

- The pattern of the heads shows only the spindles in which a tool has to be removed or mounted.

### LIST OF DRILLS TO FIT

Name	Qty	Type	Rot.	Scoring margin	Len.	Diam.Se.	Leng.Se.
D8 L50	3	P	<b>CCW</b>	8	30	-	-

- The list of drills or cutters required are only those that must be mounted/replaced in the tooling

## TOOLINGS

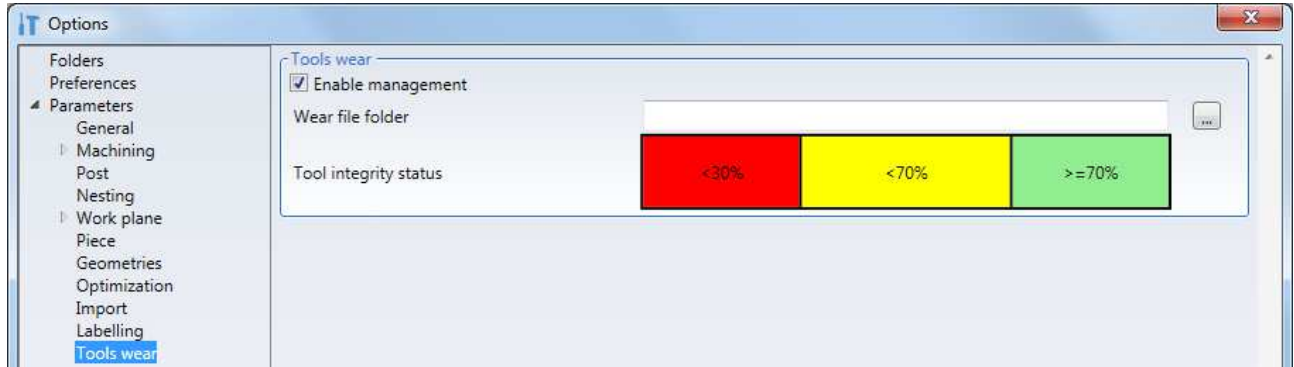
### 1. CYFLEXF1300

#### UNIT 1

TESTA NUMERO 10						
Spindle	Status	Name	Type	Scoring margin	Len.	Rot.
53	Assemble	D8 L50	P	8	30	CCW
63	Assemble	D8 L50	P	8	30	CCW
7	Remove	D9 L50	-	-	-	-
8	Assemble	D8 L50	P	8	30	CCW
10	Remove	D9 L50	-	-	-	-

## 11.10 Tools wear

Tool wear management must be enabled in the visible screen with "Tools-Options-Parameters-User-wear tools" by placing a flag under "Enable management".



The wear status will be displayed in Yellow and Green Red to immediately see if the tools should be replaced.

The representation is what you see in the options:

Green Tool with less than 30% wear (new)

Yellow: Tool with wear ranging from 30 to 70% (still usable)

Red: Tools with wear greater than 70% (to be replaced)

Note that the image shows the state of the tool's integrity and not its wear, even if the concept is the same.

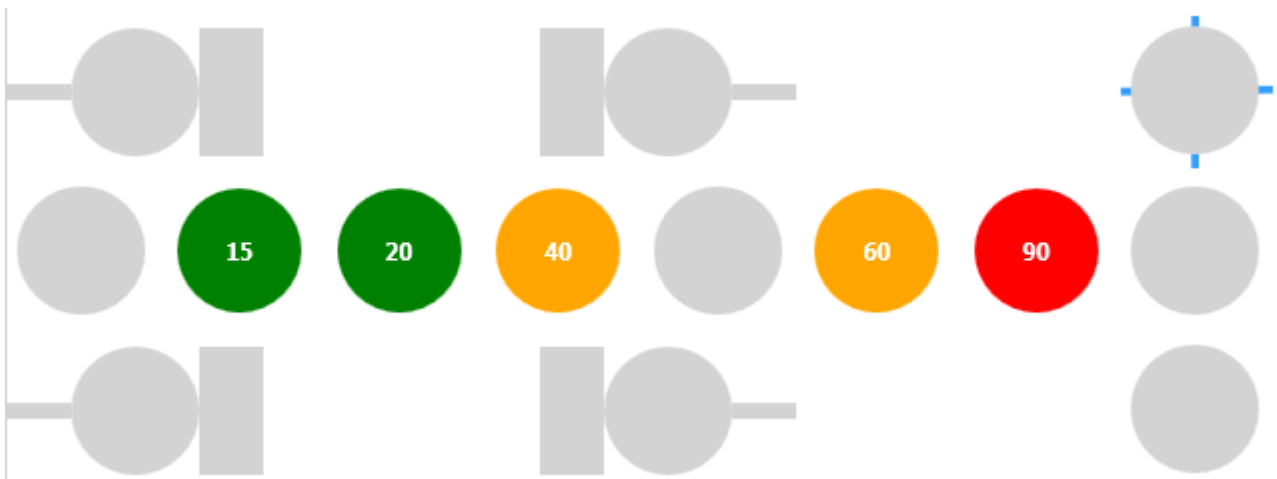


Tool wear is controlled by the following controls:

### 11.10.1 Tool wear display

The display of the wear status is obtained with the "Display" command on the "Wear" menu.

The graphical representation of the wear status is represented by the colours Green-Yellow-Red, with a number representing the percentage of use.



## 11.10.2 Tool wear update

The display of the wear status may be misaligned compared to effective use as the machine operated after the wear status display.

The "Update" command will be used to read the wear data rewritten by PanelMac.

## 11.10.3 Reset wear.

When inserting one or more new tools, the degree of wear must be reduced to zero. To do this, use the "Reset selection" and "Reset all" commands.

With the "Reset selection" command active, you can reset the wear state of all the tools selected subsequently.

By clicking on "Reset everything", the following message will be displayed:



Clicking on "yes" will reset the wear conditions of all tools in the active tooling.

## 11.10.4 Wear report

If the tooling contains wear information, a printout is possible.

Selecting the Report button opens a window in which you can choose the type of report:

- for all tools in the setup
- only for tools with a wear value higher than a certain threshold.



The tool wear report file has the following key:

## TOOLS WEAR REPORT

NAME:

UNIT OF MEASUREMENT: millimetres

DATE: 2/13/2018 3:30:49 PM

### DRILL LEGEND

L = Flat tipped

LS = Flat tipped countersunk

PS = Flat countersunk

PB = bliz flat

CW = Right rotation

**CCW** = Left rotation

### ROUTER LEGEND

T = Toroidal

S = Spherical

C = Slot mill

L = Sander

### WEAR LEGEND

	Maximum
	Medium
	Minimum

Listed below are all the orders placed with their progress.

## TOOLINGS

### 1. CYFLEXF1300

#### UNIT 1

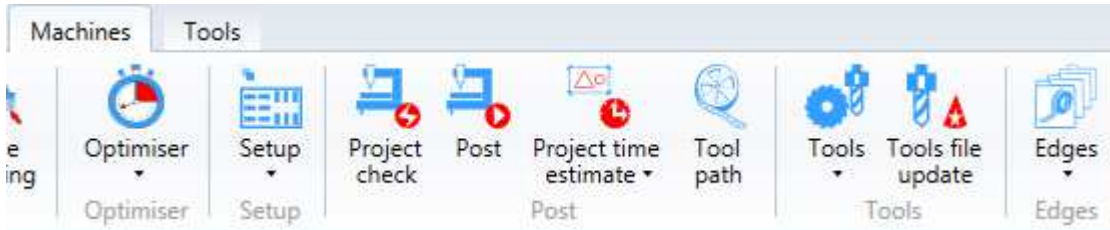
TESTA NUMERO 10						
Spindle	Name	Type	Diam.	Len.	Rot.	Wear
6	D8 L50	P	8	30	<b>CCW</b>	<b>91%</b>
7	D9 L50	P	9	30	CW	<b>60%</b>
9	D5 L50	P	5	30	CW	<b>40%</b>
10	D9 L50	P	9	30	CW	<b>35%</b>
11	D10 L50	P	10	30	CW	<b>15%</b>



## 12 Edges Database(EdgeManager)

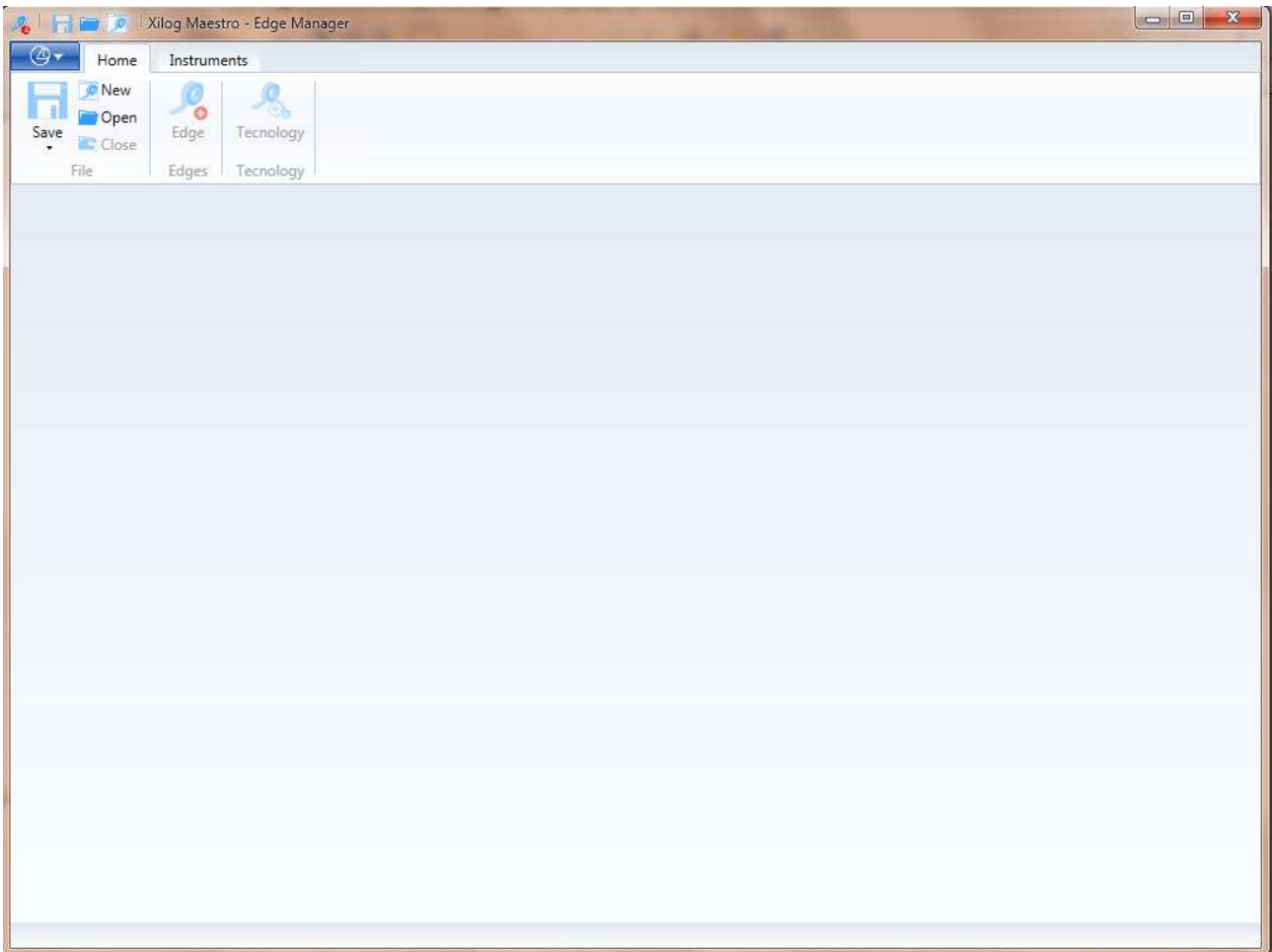
The edges database (EdgeManager) contains all the information necessary to define the edges used for the edge banding operations.

The “EdgeManager.exe” application is used to manage the edges database and it is launched automatically by clicking on “Edges” in the “Machines” menu.



EdgeManager can also be launched autonomously from the Maestro installation directory by clicking on the “EdgeManager.exe” file.

The EdgeManager main window looks like this:



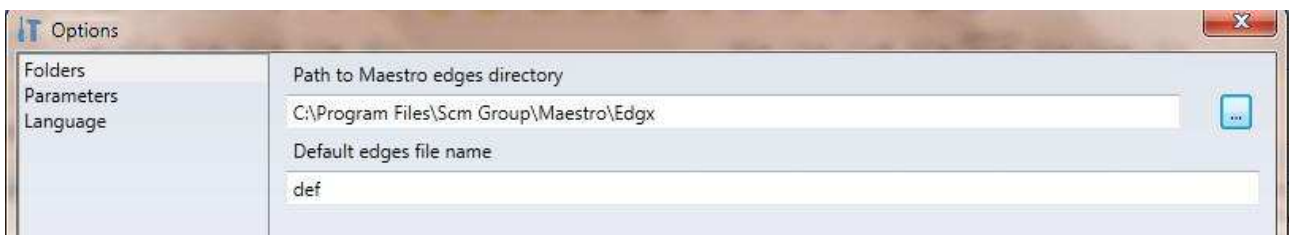
## 12.1 Edges Database Default

The tools database defaults can be changed by clicking on the “Options” menu of the “Tools” folder.



### 12.1.1 “Edges Database Name” Default

To change the default name of a new Edges database proceed as follows:  
Click on “Options”  
Fill in the “Default edges file name” field



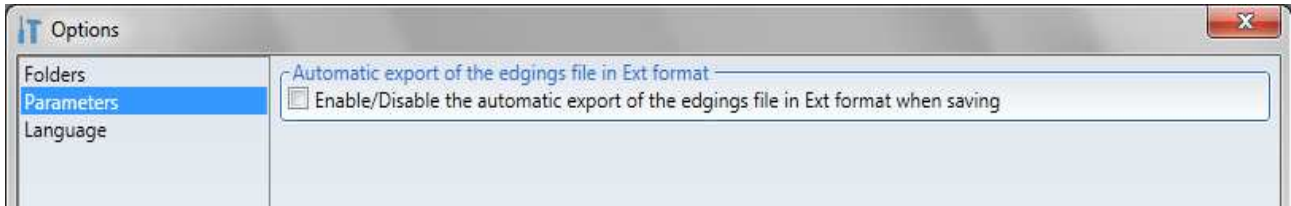
### 12.1.2 “Edges Database Folder” Default

To change the name of the folder that the edges database are opened and saved:  
Click on “Options”  
Select the “Folders” node on the left tree  
Fill in the “Maestro edges directory path” field

### 12.1.3 Automatic Export of the Edges Database in Ext. format

In this options page you can enable/disable the automatic export of the edge database in Ext format (function necessary to interface with Genio).

To enable/disable the automatic export of the edges database in the Ext format proceed as follows:



Click on "Options"

Select the "Parameters" node on the left tree

Select "Enable/disable automatic export of edges file in Ext format when saving"

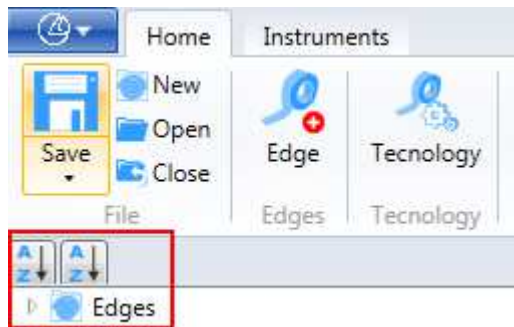
Fill in the "Ext file export folder path" field

## 12.2 Edges Database Creation

To create an edges database use the “New” command in the “Home” “File” menu.



The database will appear empty but already structured as follows:



## 12.3 Open Edges Database

To open an existing edges database click on “Open”.

A window will be displayed from which you can select a file with extension .Edgx.

The default folder will be the one defined, as described in section

12.1.2 “Edge Database Folder” Default

Then select an .Edgx file and then open

Or double click on the .Edgx file.

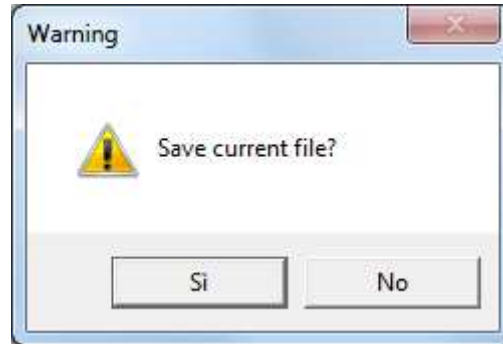
If there was an edges database already open, the “Close Edges Database” function would be performed automatically.

## 12.4 Close Edges Database

To close an open edges database click on “Close”.



If the edges database has not been saved yet the following question is displayed:



Clicking "Yes" automatically recalls the "Save Edges Database" function.

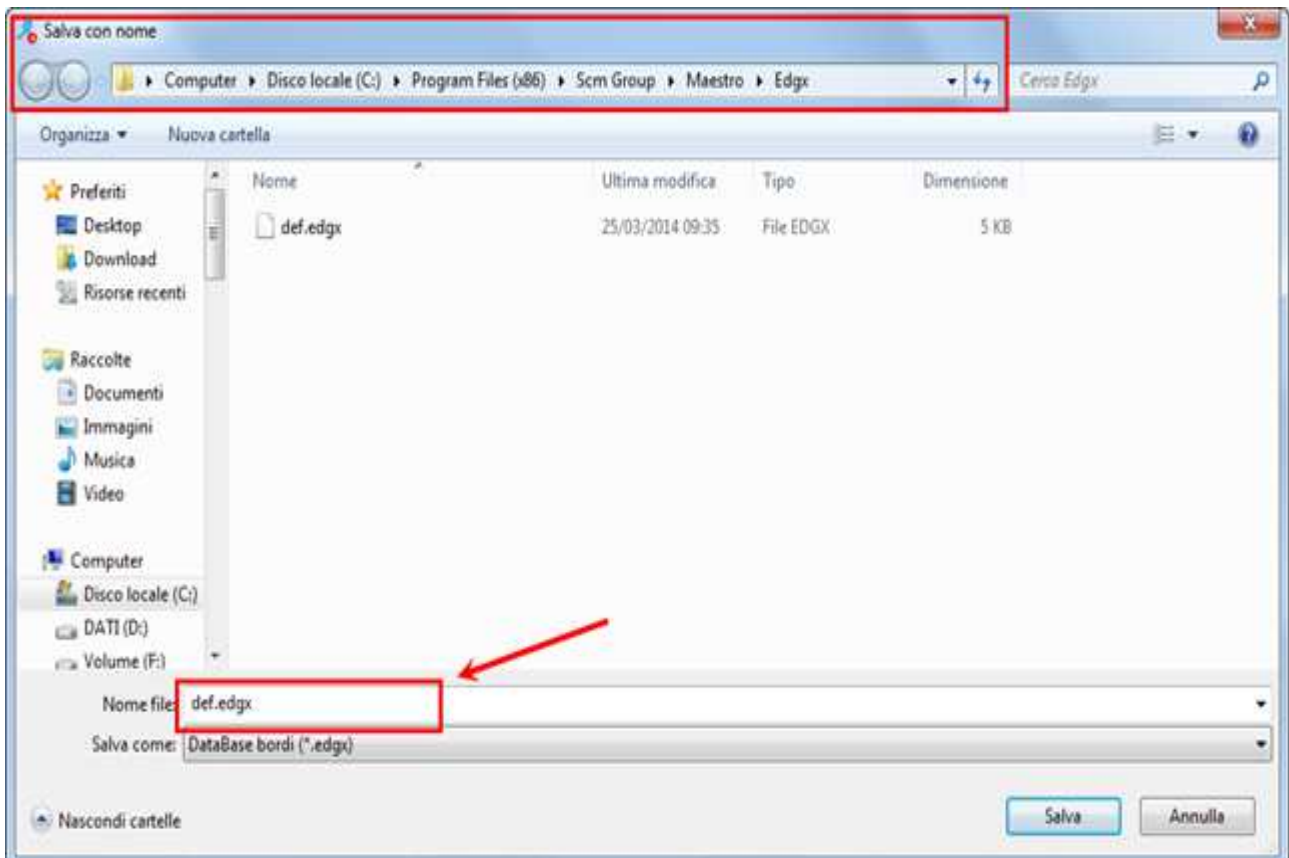
## 12.5 Save Edges Database

To save an open edges database click on "Save".



If the edges database already had a name, no question will be asked.

If the Database was just created the "Save with name" function will be recalled.



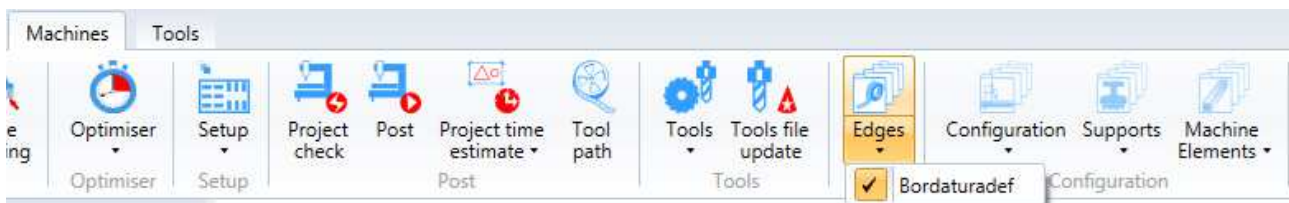
The default folder that will be suggested is the one defined, as described in section 12.1.2 “Edges Database Folder” default

The edges database name can be specified in the “File name:” box then click on “Save”.

## 12.6 Activate Edges Database

To enter an edge banding operation in a project, select an edges database to activate it, as the edge banding operations are described by combining an edge to a 2D geometry.

To activate an edges database use the “Machines” “ Tools” menu. Click on the “Edges” key to display the list of edges databases available in the “default” folder.



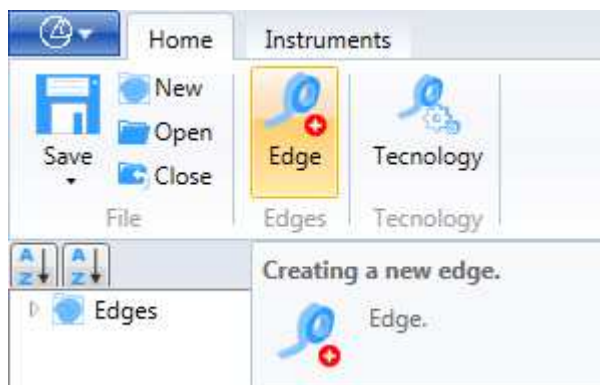
The edges database with a check at its left is the one active. To activate another database simply select it with the left hand button of the mouse.

## 12.7 Edges

The edges are the objects used for the edge banding operations. In order to use them you must know the technological information of each edge. The information collected must be entered in the Edges Database.

### 12.7.1 Entering Edges

To enter a new edge in the database use the command in the “Home” menu



After having selected the enter a new edge command, fill in the fields in the "General Data" and "Edge Data" menu:

This data will be the same for all edges. Then you must enter for each edge the specific “machining technology” data and the “close” data.

Edge properties	
<b>General data</b>	
Name	Edge
Description	
<b>Edge data</b>	
Thickness (mm)	1
Height (mm)	1
Flexibility	1
Material	
Pressione del contro rullo colla	Low pressure
Type of shearer cut	Low pressure
Colour	Black

### 12.7.1.1 Edge name

The “Name” field must be filled with the code which will be used during machining.

The description field can contain the main edge specifications in order to immediately see the type of edge.

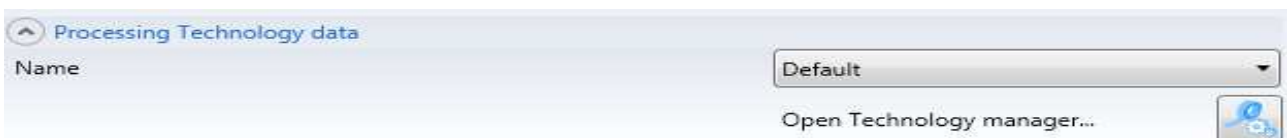
### 12.7.1.2 Edge data

In the Edges data section enter the following:

- Edge Thickness
- Edge Height
- Flexibility: numerical value between 1 and 9; 1 indicates maximum flexibility, 9 minimum flexibility
- Material: this field is used to describe the type of material used for the edge
- Pressure of the glue counter roller: pressure mode of the glue counter roller for this type of edge. High or Low pressure of the glue counter roller
- Type of cutting slice: cutting slice mode for this type of edge; High or Low cutting pressure
- Colour: the edge colour can be specified to make it easier to identify it

### 12.7.1.3 Machining Technological Data

When a new edge is added, it is assigned a default machining technology. The technology contains the edge data that affect the edge banding operation.



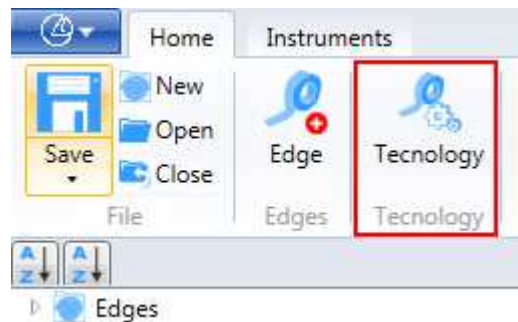
By selecting the drop down menu you can associate one of the machining technologies in the database to the edge.



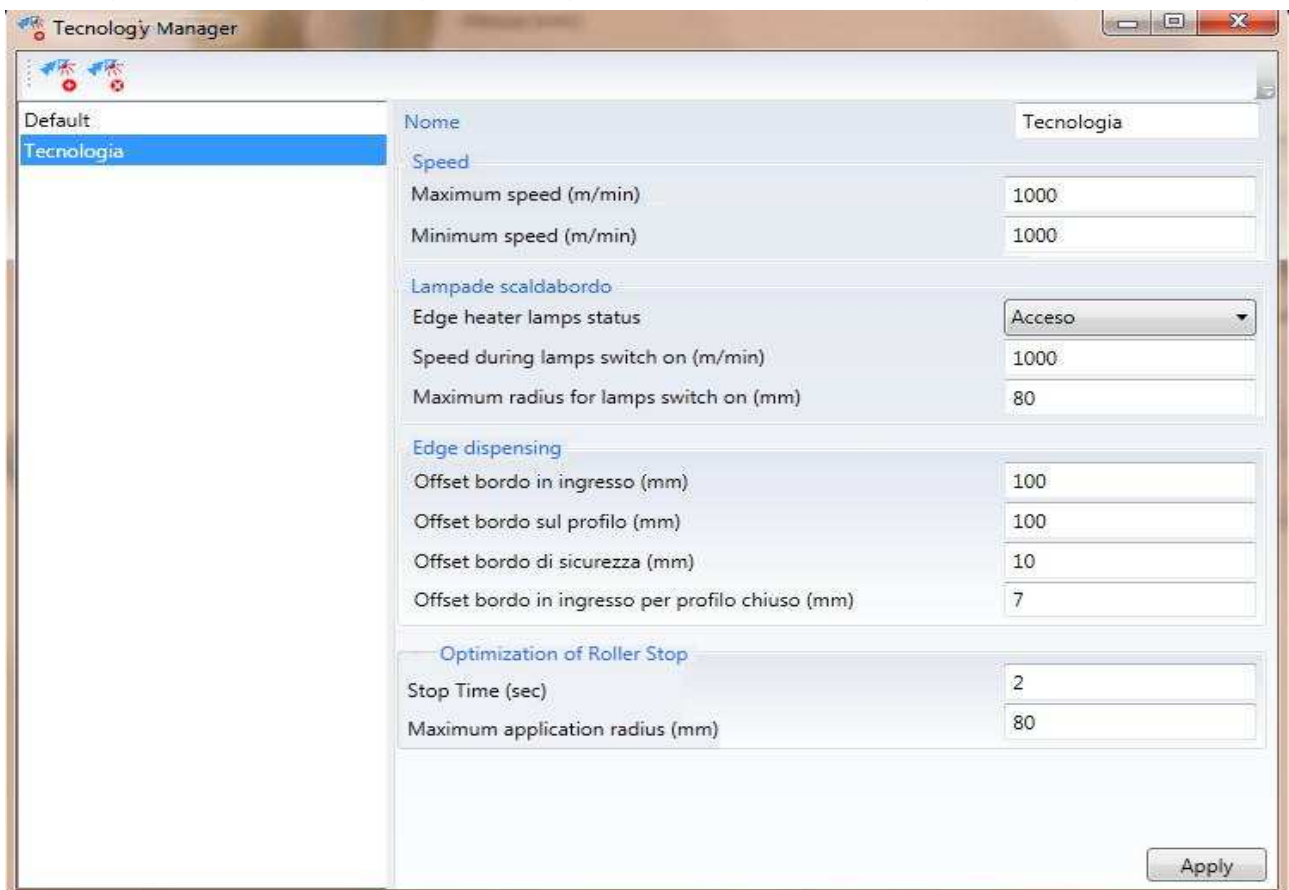


## 12.8 Technologies Manager

The Technologies Master is used to display, create, change and delete the machining technologies. The Technologies Manager is opened with the “Open Technologies Manager...” key, under the technology selection menu or with the “Open Technologies Manager” key on the Home menu:



Selecting the “Open Technologies Manager” opens the “Technologies Manager” window:



On the left of this window the technologies in the database are displayed, while the technology parameters are displayed on the right. When a technology is selected on the left of the window, the fields on the right are updated.

The fields on the right can be changed for each technology, except for the default technology; this is the standard technology used to create an edge. In a second stage the new edge can be associated to a technology already in the database or create a new one with the Technologies Manager.

To add a technology to the list of technologies use the “Add Technology” command in the window menu. (“+” icon).

Similarly to remove a technology from the list of technologies use the “Remove Technology” command in the window menu. (“X” icon).  
The default technology cannot be deleted.



The technology data are:

### 12.8.1 Name

- Technology name

### 12.8.2 Speed

- Speed: standard edge banding speed
- Speed: minimum edge banding speed

The following parameters are used for the speed optimizer:

- Minimum application radius: minimum radius of the profile on which the optimizer applies the speed attribute.
- Maximum application radius: maximum radius of the profile on which the optimizer applies the speed attribute.

Speed	
Standard speed [m/min]	10
Minimum speed [m/min]	1
Speed optimizer	
Minimum application radius (mm)	10
Maximum application radius (mm)	100

### 12.8.3 Edge heating lamps

- Status of the edge heating lamps when the edge banding starts: on/off.
- Speed during the lamps switch on
- Maximum radius for the lamps switch on: maximum curvature radius of the trajectory above which the edge heating lamps cannot be switched on.
- Power of the lamps expressed in percentage.

Edge Heating Lamps	
Edge heater lamps status	Off
Speed during lamps switch on [m/min]	6
Maximum radius for lamps switch on (mm)	150
Lamps power (%)	52

### 12.8.4 Edge dispensing

Extra dispensing during the head downstroke from the magazine at the first approach point of the profile being edge banded.

Open profile:

- Extra dispensing of the edge during the approach section
- Extra dispensing of the edge on the profile being edge banded

Closed profile:

- Extra dispensing of the edge during the approach section
- Extra dispensing of the edge on the profile being edge banded

Edge dispensing	
Extra dispensing in downstroke (mm)	5
<b>Profile open</b>	
In infeed (mm)	50
On profile (mm)	100
<b>Profile closed</b>	
In infeed (mm)	20
On profile (mm)	0

## 12.8.5 Roller stop optimization

- Stop time: Stop time of the edge banding head along the profile.
- Maximum application radius: maximum curvature radius of the trajectory above which the roller stop cannot be optimised.

Optimization of Roller Stop	
Stop Time (sec)	<input type="text" value="0.5"/>
Maximum application radius (mm)	<input type="text" value="80"/>

## 12.8.6 Stoptime of the edge bander in infeed and outfeed to the profile

Open profile:

- Edge bander stop time on the first point of the profile being edge banded.
- Edge bander stop time on the last point of the profile being edge banded.

Closed profile:

- Edge bander stop time on the first point of the closed profile being edge banded.
- Edge bander stop time on the last point of the closed profile being edge banded.

Edge bander stop time	
<b>Profile open</b>	
In infeed (sec)	<input type="text" value="0"/>
In outfeed (sec)	<input type="text" value="0"/>
<b>Profile closed</b>	
In infeed (sec)	<input type="text" value="0"/>
In outfeed (sec)	<input type="text" value="0"/>

## 12.8.7 Edge in closure compensation

Defines the compensation in the closure of an edge banding. The reference is the type of geometry of the final section of a closed edge banding.

- Linear section: compensation.
- Arc section: based on the concavity of the section (concave arc, convex arc) a table is defined that associates a compensation to the curvature radius of the section. The sections that have a curvature arc within an interval in the table will have a compensation extracted from the interpolation of the compensation data of that interval.

Edge in closure compensation

Linear Section (mm)

Arc Section

Arc radius	Compensation
100	1

## 12.8.8 Edge banding early-late

Solo per profili aperti:

- Posticipo ingresso della bordatura:
- Anticipo uscita della bordatura

Edge banding advance-delay

Profile open \_\_\_\_\_

Edge banding infeed delay

Edge banding outfeed advance

## 13 Post-Processor

Projects performed with Maestro can be exported in xxl or pgm files as input data for the interface "Xilog"

### 13.1 Create .xxl and .pgm files

To create .xxl and .pgm files from the Maestro Post-Processor, proceed as follows:

- 1) Install Xilog
- 2) Set the output folder
- 3) Run the Post-Processor

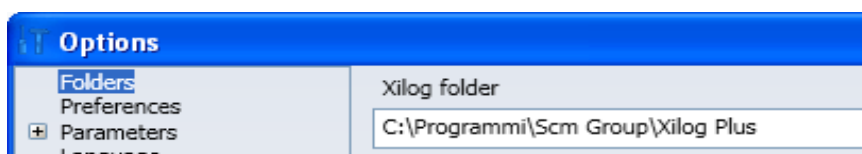
#### 13.1.1 Xilog installation

In order to translate project into files compatible with Xilog (xxl or pgm) it is necessary that Xilog is installed.

In order to choose the field for Xilog installation, click on the icon "Options" in the field "Tools"



Write the folder name in the field "Xilog folder"

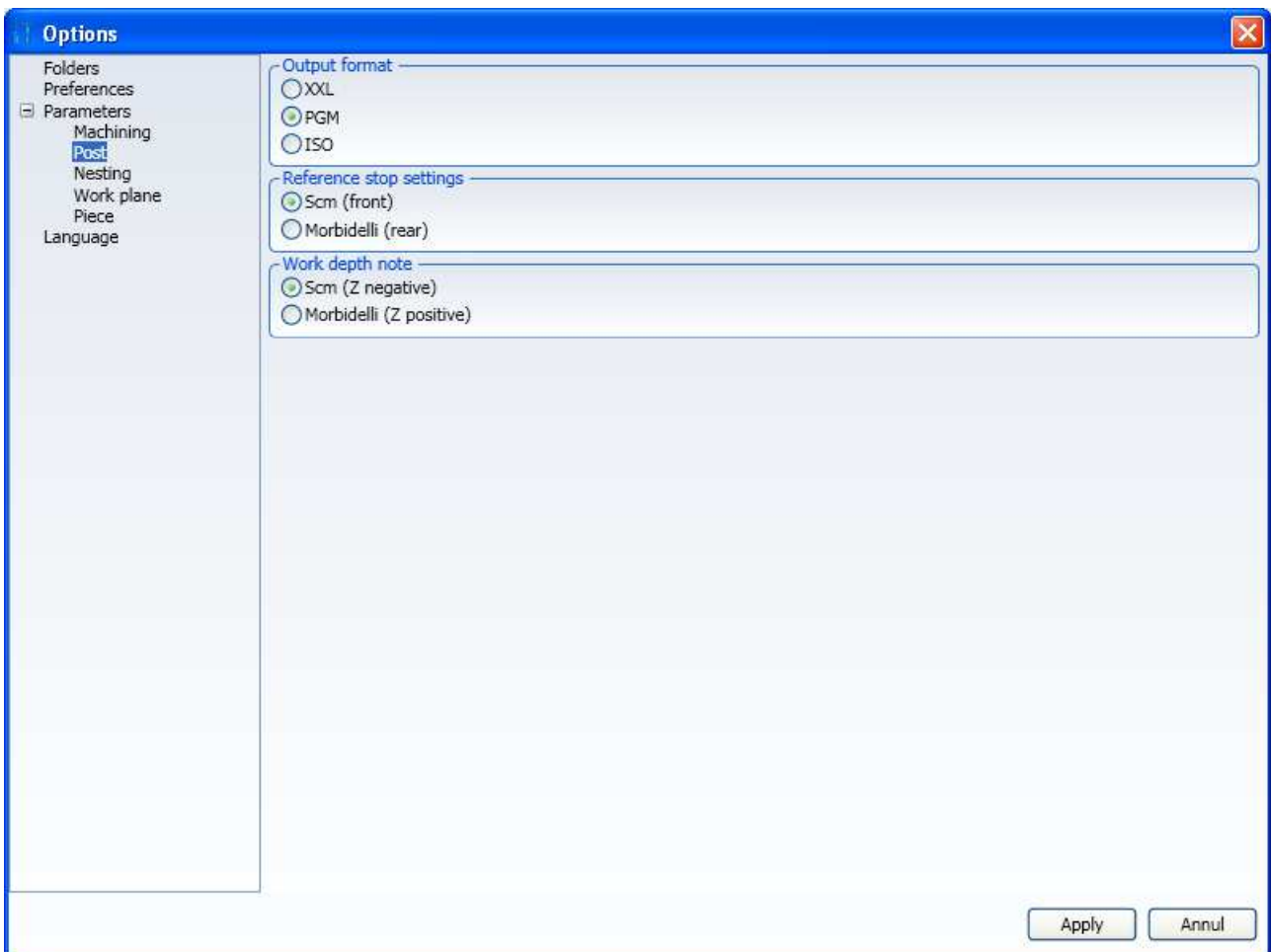


## 13.1.2 Set output folder

To set the Maestro Post-Processor output folder use the "Options" command in the "Tools" folder.



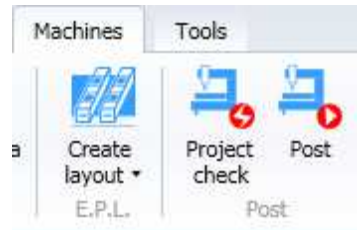
If you select the folder "Post", following screen will appear:



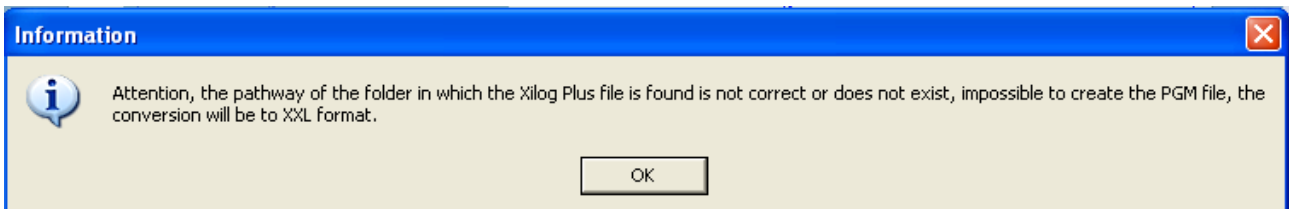
The "Output Folder" is the folder where XXL or PGM projects will be written as selected in "Output Format"

### 13.1.3 Execution Post-Processor

In order to translate the open project in a file compatible with Xilog (xxl or pgm) use the command "Post" in the menu "Machines".

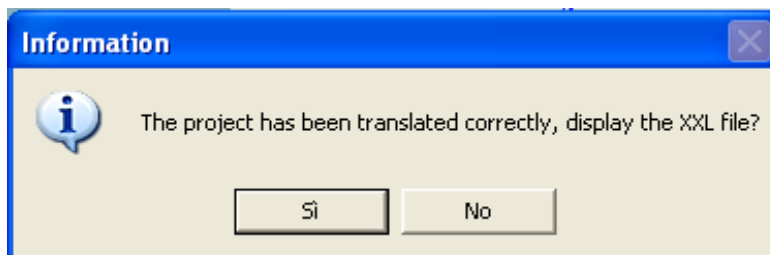


If this message appears:



You have to insert the correct path where Xilog has been inserted (see paragraph 13.1.1)

If the translation is well performed, following message will appear:

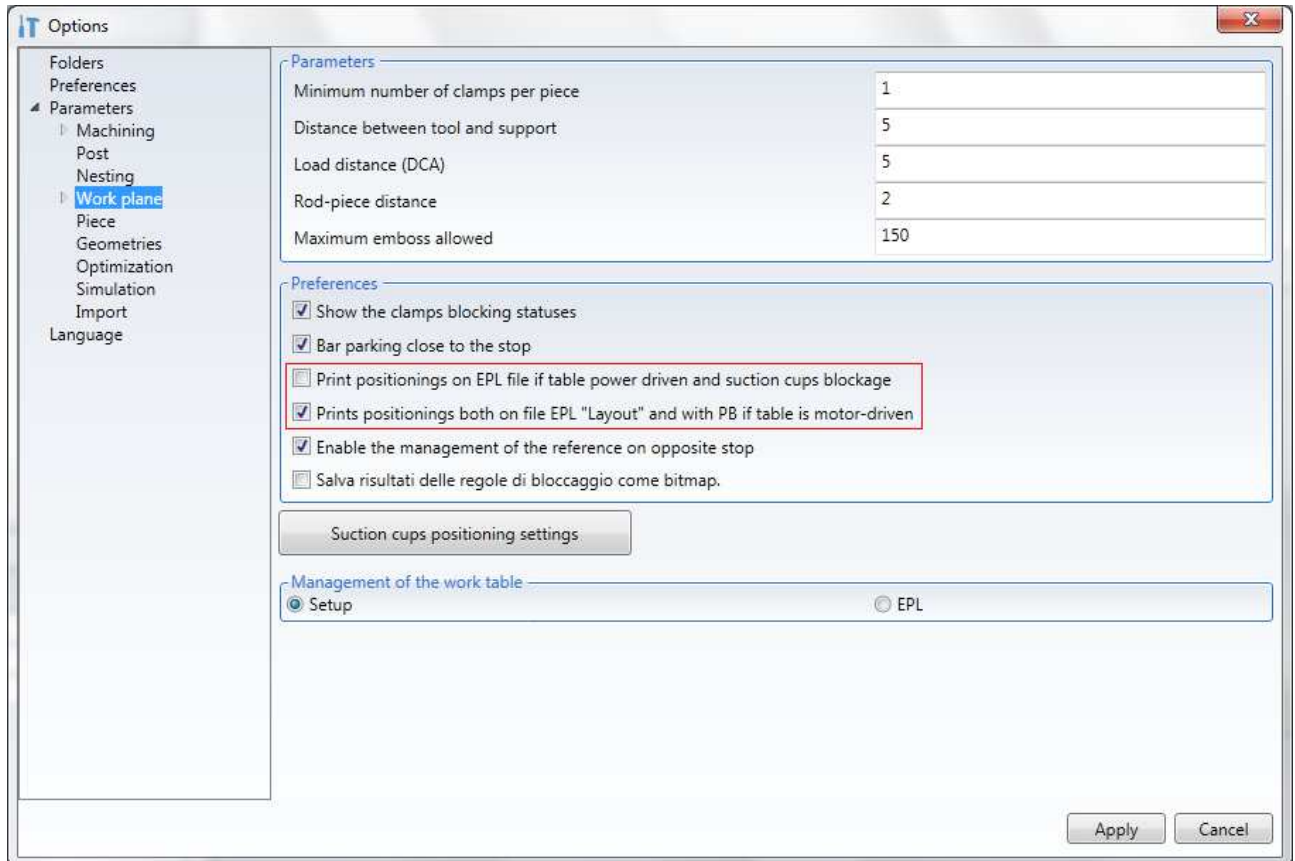


By clicking on "Yes", the program for Xilog will be displayed.



## 13.2 Create .epi files

To generate the EPL file(Worktable Editor) during the Post-Processor select one of the two items in the “Options-Parameters\_Worktable” menu.



By selecting:

“Print positionings on EPL file with motor driven table and suction cups blockage”

When executing the Post-Processor an EPL file will be produced as well as the .xml and .pgm files.

Normally only the manual tables produce an EPL file.

For motor driven tables the .epi file produced will contain only the bars and supports positioning of the first phase in Arrange.

By selecting:

“Print positionings on “Layout” EPL file and with PB with motor driven table”

When executing the Post-Processor the .xml and .pgm files will also include PB instructions and the EPL file will also be produced.

# 14 Appendix

## 14.1 Sizes used by Maestro

Maestro uses several files with following sizes:

csv	Workpieces for nesting and mixx definition
epl	Instructions to set clamps/suction pads
lbdx	machine elements library
lbsx	supports library
mixx	File for mix of workpieces
nstx	Nesting projects
nstrptx	Nesting report file
pgm	Files exported for Xilog
pgmx	Projects
settings	Save the configuration parameters
tlgx	Data-base Tools
xcs	Script Files
xml	Instructions to set parameters/geometries
xsp	Sub-programs
xxl	Files exported for Xilog
pgm	Files exported for Xilog