Xilog Plus

for Bore-Milling Machines of the SCM Group

User and Programming Guide for the Xilog Plus Editor - v. 1.11

v. 3.2 – September 2005

code 0000571270D

SCM GROUP

This manual has been designed to provide information relative to the use of Bore-Milling machines manufactured by the SCM Group.

The information contained in this manual is the property of the SCM Group and may not be reproduced or disclosed without authorization.

SCM Group cannot be held liable for incorrect use of the information contained in the manual.

Editored by Gedux Software & Services - www.gedux.com

Contents

Index of Instructions

Index

1. Installation and Start

The Xilog Plus editor can be installed on a personal computer IBM® or compatible, that has the requisites listed below:

Minimum hardware requirements:

- CPU Intel® Pentium® or better;
- at least 128 MB Ram;
- drive for floppy disk 3” 1/2, 1.44 MB;
- drive for CD-ROM;
- at least 500 MB of free space on hard disk;
- 2 serial ports;
- 1 parallel port;
- SVGA video card;
- color monitor with a resolution of at least 800x600 pixels

Minimum software requirements:

- Microsoft** Windows 98 SE operative system or subsequent versions.

(**) Windows 98 SE is a brand name of the Microsoft™ Corp.

WARNING

Current system session must be closed before turning off the machine

To install Xilog Plus editor, follow these instructions:

1. turn on the computer;
2. insert the installation CD-ROM into the drive;
3. open (by double clicking the mouse) *My Computer*;
4. open the CD-ROM for Xilog Plus;
5. start (always by double clicking the mouse) the *Setup.exe* file.
6. following the installation instructions.

During the installation procedure the program will ask you to select the type of communications port from amongst the following options:

- **None (PC Office).** To be selected if the calculator is not connected to the Numeric Control (N.C.), but is installed on and office computer.
- **RS232 Com 1 (Serial)** and **RS232 Com 2 (Serial).** If the calculator is connected to the N.C. it is necessary to indicate which of the two serial ports (Com 1 or Com 2) is used for the connection.

To start Xilog Plus editor:

1. click on the **START** button;
2. select *All programs*;
3. select the folder *Scm Group*;
4. click on *Xilog Plus*.

To edit the current language:

1. start Xilog Plus;
2. click on the **OPTIONS/INTERNATIONAL SETTINGS** menu;
3. select the flag for the country desired by repeatedly clicking on the **LANGUAGE** button;
4. click on the **OK** button and perform the operations requested by the Xilog Plus editor to make the change effective.

2. Configuration

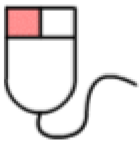
Xilog Plus is ready configured to operate with your machine. However, in some cases you may need to adjust some configuration files.

IMPORTANT!

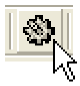
Most configuration files are configured by the manufacturer's technicians. Changing them may cause serious machine malfunctions!

► Open the configuration file editor.

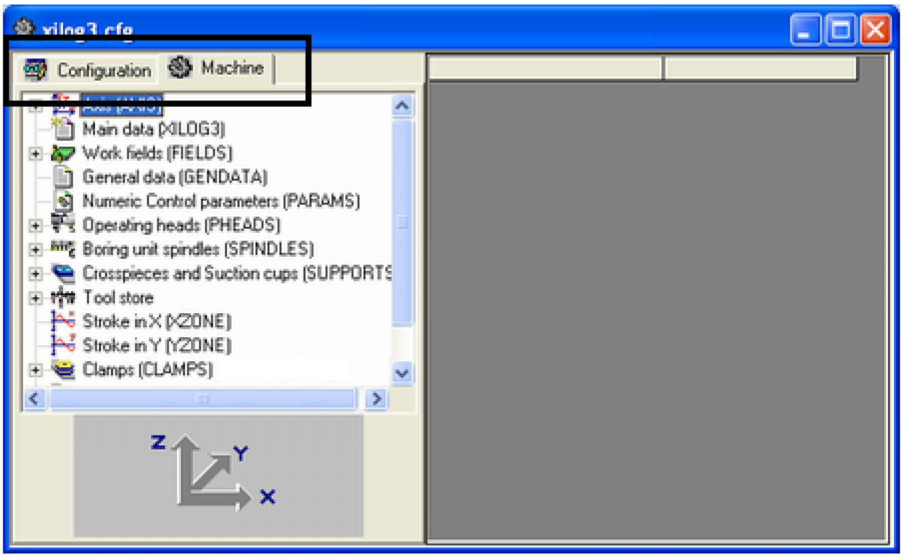
Click on the **FILE/CONFIGURING MACHINE** menu...



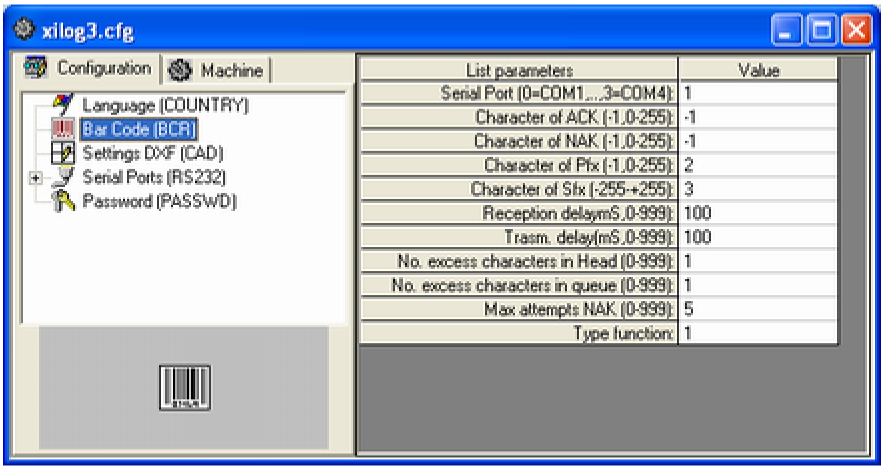
... or on the button.



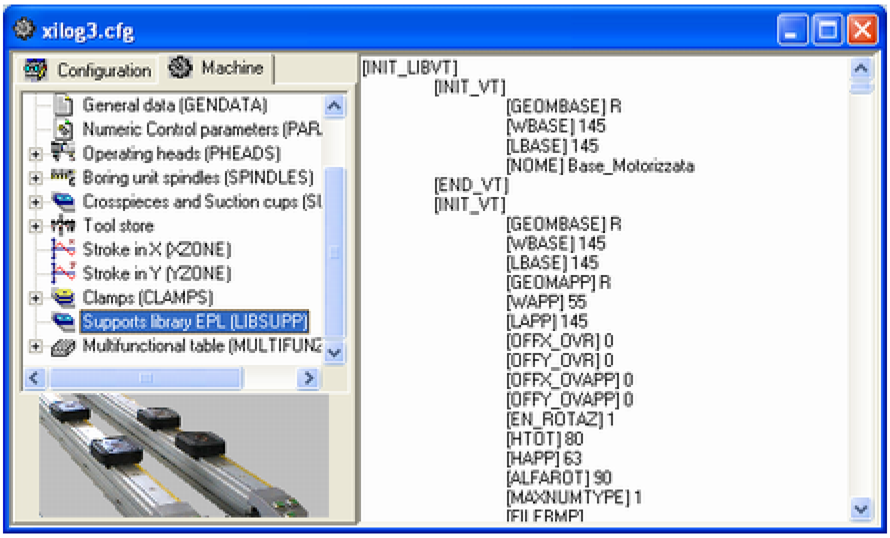
The configuration files are grouped in two sections: configuration (contains software configuration files) and machine (contains machine configurations files, normally set by the manufacturer).



A template with parameters is sometimes available to change the files:



In other cases, the files are simple text files that can be freely edited:



In the latter case, you can edit the files directly with any text editor, outside Xilog Plus. The configuration files have the extension .CFG and are in the folder ../Xilog Plus/Cfg. In the configuration parameters editor their name appears in brackets after a descriptive title, without an extension (e.g.: in the editor file Libsupp.cfg appears as “Supports library EPL (LIBSUPP)”).

3. Introduction to Programming

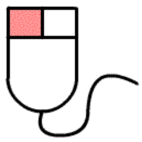
The Xilog Plus editor allows you to create five types of files:

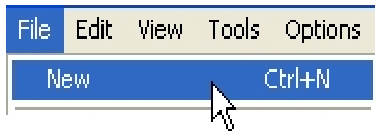
- **Program**. A list of instructions (profiles, boring, movements) that the machine must perform to implement the work.
- **Mix of programs**. A list that recalls a series of programs to be carried out in sequence.
- **Tooling**. List of the tools used for machining the pieces.
- **Program ISO.** A program written in ISO language.
- **Multiple program**. A list of programs associated on the same machine table.

► Creating a new file.

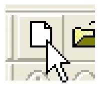
1

To create a new file click on **FILE/NEW** from the menu…



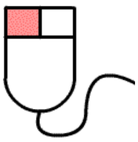


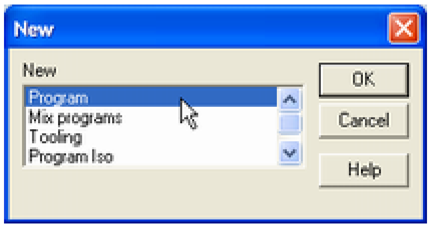
… or on the button.



2

A window will appear from which it is possible to select the type of file. Select the type of file and click on the **OK** button.

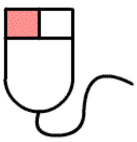


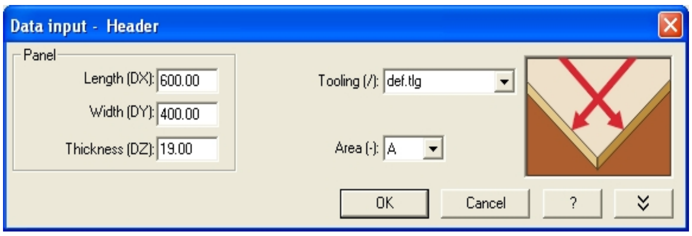


3

If you are creating a *Xilog Plus program* it will ask that first of all you set the *Header*.

Click on the **OK** button to create a program.

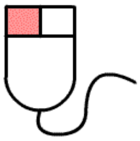


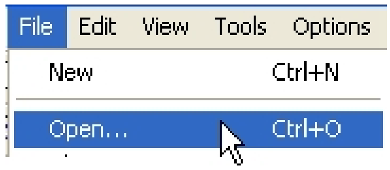


► Opening an existing file.

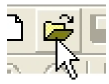
1

To open an existing file click on **FILE / OPEN** from the menu…



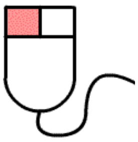


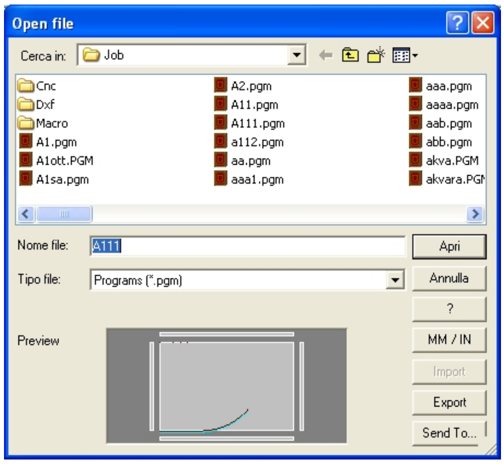
… or the button.



2

A window will appear from which to open the file. Using the mouse, select the file to open and click on the **OPEN** button.

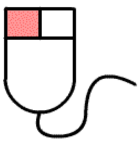


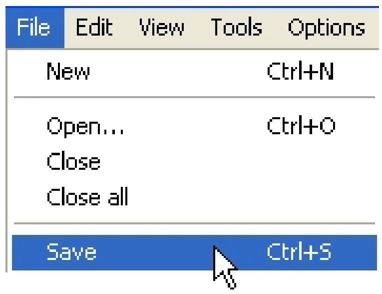


► Saving a file.

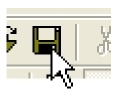
1

o save a file in the computer's memory click on **FILE/SAVE** from the menu...





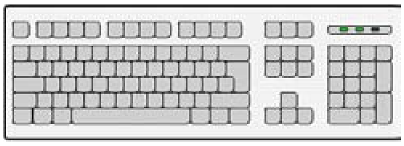
… or on the button.

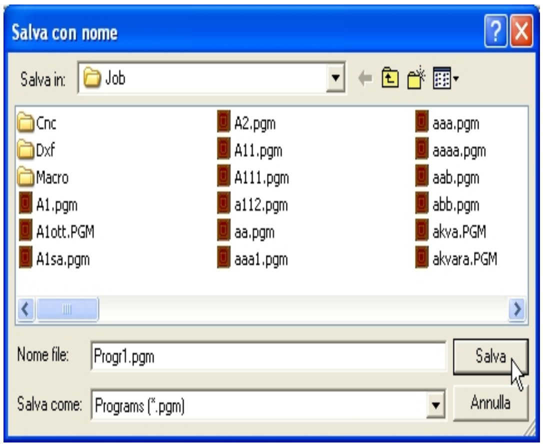


2

A window will appear for saving the file.

In the file name box give the file a name and click on the **SAVE** button.

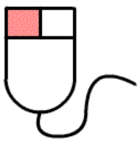


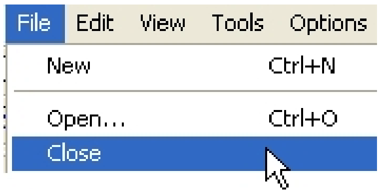


Xilog files are normally saved in the Job folder. You can select other folders or storage supports (a floppy disk for example) from the ***SAVE IN*** *drop-down menu.*

► Closing a file.

To close a file click on **FILE / CLOSE** form the menu





4. Basic Programming

4.1 Tooling

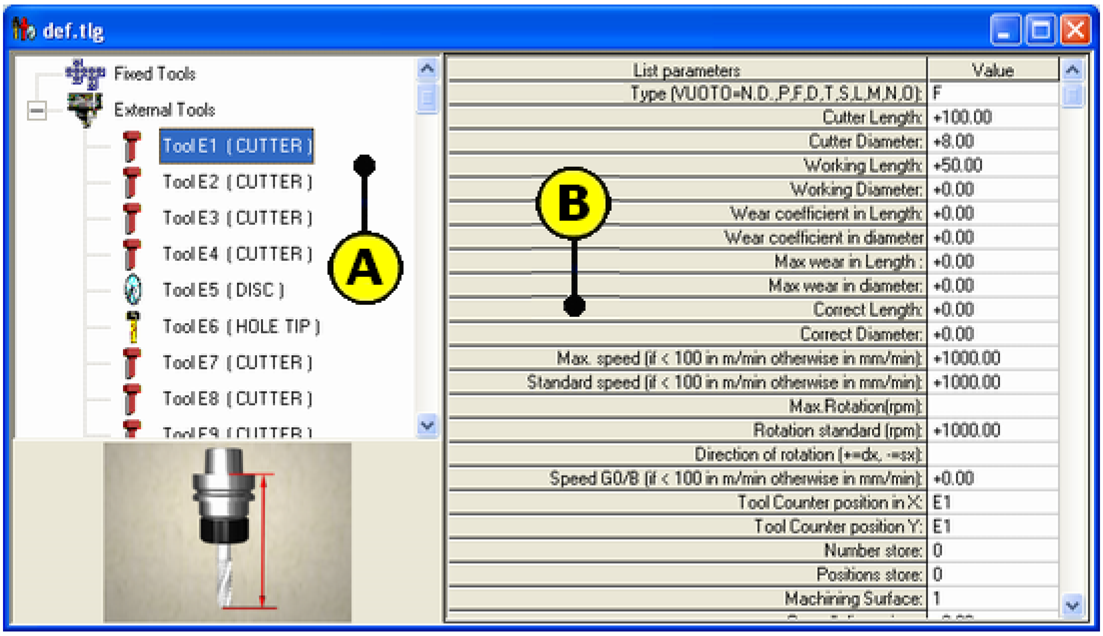
4.1.1 Tooling Editor

Tooling contains the parameters for all tools available on the Xilog Plus machine and makes it possible to create and store various tooling files that can be utilized by indicating the name in the program header.
When a program is run, the established tooling file must be activated by selecting it from Xilog Plus Machine Panel (see: Machine Panel Manual) and ensure that the tools mounted on the machine correspond to those described in the active tooling file. The tooling selected is called *active* or *current tooling.*
Standard machine tooling, that also corresponds to current standard tooling, is the file **def.tlg** which **must always be present** in the main tooling folder.....\XILOG PLUS\JOB.

WARNING

The characteristics of the tools mounted on the machine must match with the characteristics described in the current tooling file at all times.
A tooling file can be enabled as active tooling only if it is contained in the main folder and if its name does not exceed 12 characters (including the extension .tlg).
During the execution of a program, the file set in the program must correspond with the active file.

The tooling editor has two main areas:



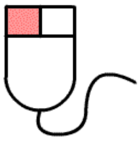
E) Area for selecting a tool.
F) Window for editing parameters relative to the selected tool.

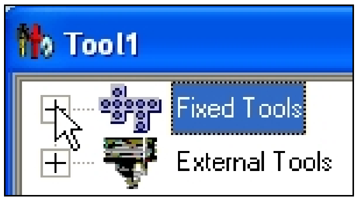
► Configuring a tool.

1

Click on the square marked with a +, to the left of a group of tools.

The list will open showing each individual tool. The + sign will change to a – sign. Click on the square again to close the list.

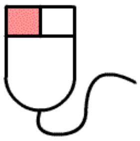


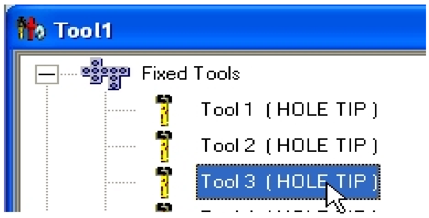


2

Select a tool from the list.

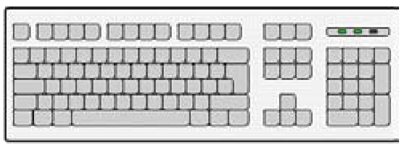
A list of parameters for the tool selected will appear in area B. If the tool is NOT STATED, only the "Type" box will appear: in this case enter one of the values indicated and confirm by pressing the ***[ENTER]*** *key.*

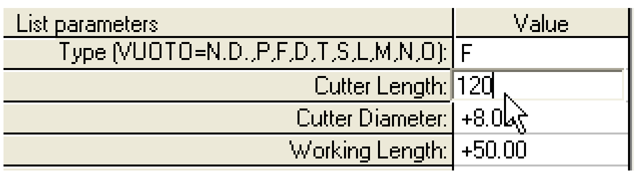




3

Insert or edit the tool parameters in the fields of the list that appears in area B. Each parameter entered must be confirmed by pressing the **[ENTER]** key.





The tool configurations can also be easily copied by dragging them with the mouse. For example, by dragging the icon for fixed tool 1 over the icon for fixed tool 2, the parameters for tool 1 will be copied automatically into the configuration table for tool 2.
The tools are divided into:

- **Fixed**. Fixed tools are those mounted on the boring head of the machine. With the tooling file it is possible to describe up to96 fixed tools, numbered from 1 to 96. The number and characteristics of the available fixed tools depends on the configuration of the boring head mounted on the machine and are set by the manufacturer in the configuration file spindles.cfg.
- **External**. External tools are those found in the machine store for tool change. With the tooling file it is possible to describe up to 95 external tools, numbered from E1 to E96.

Based on their Type, both fixed and external tools can be classified into six categories:

- tools type P (hole tip);
- tools type F (fluted mill);
- tools type D (disk mill);
- tools type T (tracer);
- tools type S (special tool);
- tools type L (pads).

For types M (head with shavings conveyor flight), N (radial floating head), O (head with air blower) see: Special Heads Benz, in Configuration Manual for the Heads.

The category to which the tool belongs is indicated in the “Type” field. In the case of fixed tools, the Type field is present in the configuration file spindles.cfg and it has already been programmed by the machine manufacturer. In the case of external tools, the Type field, present in the tooling file, must be set according to the specific requirements of the operator.
A tool (from 1 to 96 and from E1 to E96) is considered configured if its “type” has been defined. For the Xilog Plus editor all configured tools are considered valid (and therefore interprets their tooling characteristics, no matter what they are).

NOTE. The speed value is interpreted by Xilog Plus as expressed in meters per minute, if less than 100; millimeters per minute, if equal to or greater than 100.

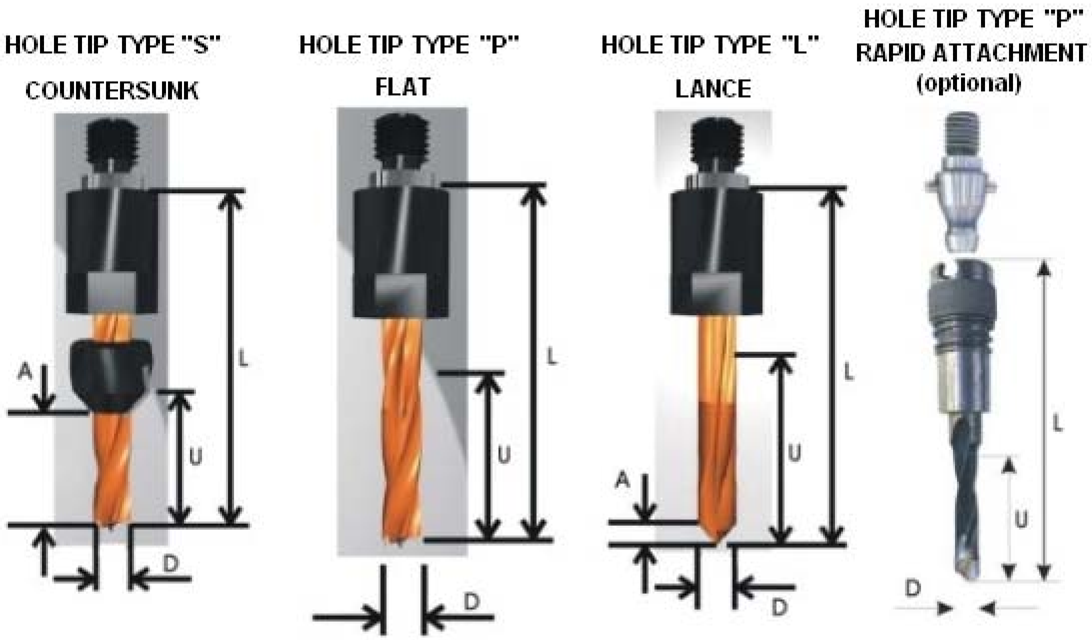
NOTE. In the tooling table of each tool the “Masked tool change not permitted” (default value=NO) parameter, is triggered each time the tool is used, regardless of the machining operation. This way, even tool aggregates, in which several tools (o parts of tools) are used for roughing and others for finishing, can be managed

WARNING

If the type of tool is set, the tool is considered, for all intents and purposes, as present on the machine; in this case, all the other parameters must also be entered correctly.

4.1.2 Tool Configuration

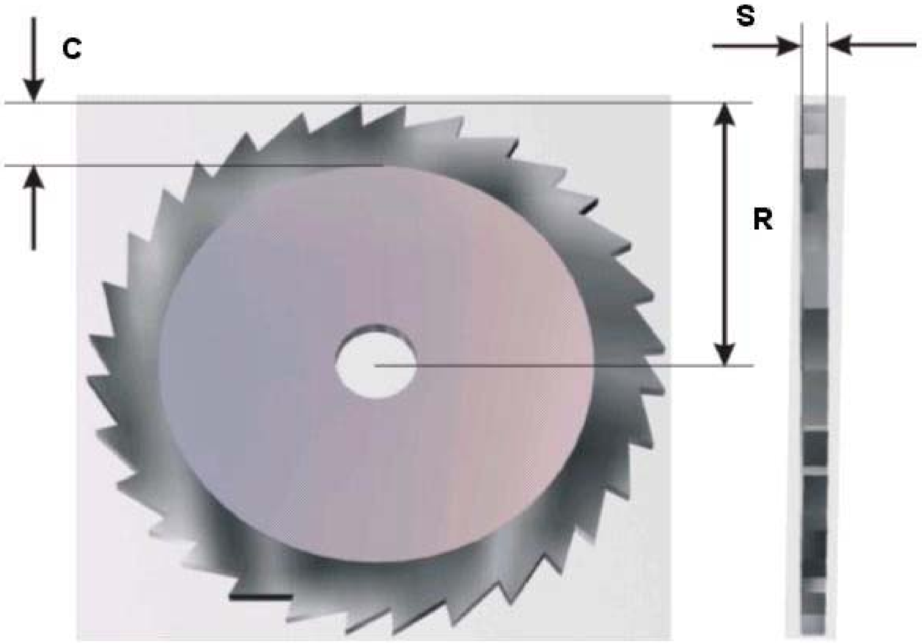
4.1.2.1 Fixed Tools: Example with Hole Tips



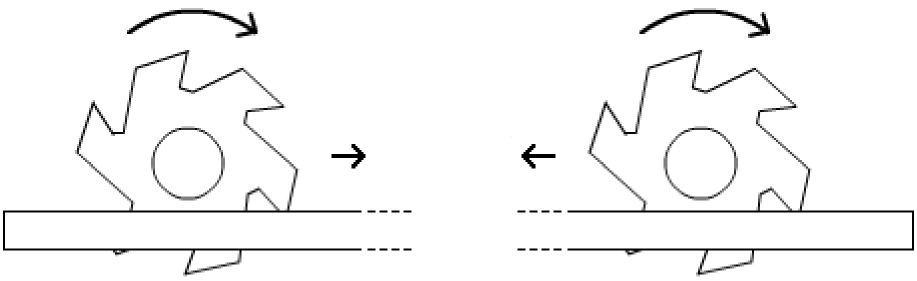
Tip length	Tool length (see figure, L).
Tip diameter	Tool diameter (see figure, D).
Working length	Maximum thickness tool can machine (see figure, U).
Working diameter	Tool diameter (see figure, D). Must coincide with the "Tip Diameter"; if the value is 0, the "Tip Diameter" is the one that counts. For a bit mounted as an *external tool,* it represents the diameter of the bore; if the value is 0, the "Tip Diameter" also represents the bore diameter.
Type tip	Type of tool to be configured: S = tip with countersink P = flat tip L = flat tipped tip
Height countersink	Distance from the lower tip of the tip to the beginning of the countersink (see figure, A).
Wear coefficient in lenght	Not enabled.
Wear coefficient in diameter	Not enabled.
Max. wear in length	Not enabled.
Max. wear in diameter	Not enabled.
Correct length	Not enabled.
Correct diameter	Not enabled.
Max. speed	Parameter invalid for the boring unit.
Standard speed	Parameter invalid for the boring unit.
Max. rotation	The boring unit has a fixed speed (this parameter makes no

	difference).
Rotation standard	The boring unit has a fixed speed (this parameter makes no difference).
Direction of rotation	The boring unit has a fixed rotation (leave empty).
Speed G0/B	Speed at which hole is made.
Windowed tool change not allowed	Change of tool on main spindle while the drilling machine is operating (default value=NO).
Tool counter position in X	Number of the tool to be used for mirrored boring in X.
Tool counter position in Y	Number of the tool to be used for mirrored boring in Y.

4.1.2.2 Fixed Tools: Example with Blade / Disk



Disc radius	Radius of the disc (see figure, R).
Working crown	Maximum thickness that the tool can machine (see figure, C).
Blade thickness	Thickness of the blade (see figure, S).
Machining direction	- **Pro-feed**: the blade moves forward, pushing the teeth downwards. - **Counter-feed**: the blade moves forward, pushing the teeth upwards. Set the machining direction with the + and - values. The resulting machining direction also depends on the direction in which the blade is fitted and the transmission mechanical conditions. See: Appendix L - Blade cuts management.



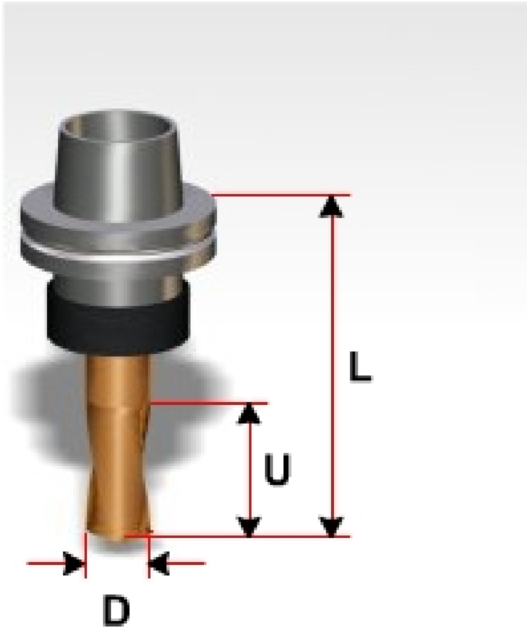
Pro-feed Counter-feed

Wear coefficient of disc in radius	Not enabled.
Wear coefficient of disc in thickness	Not enabled.
Max. wear on radius	Not enabled.
Max. wear on thickness	Not enabled.

Correct radius	Not enabled.
Correct thickness	Not enabled.
Max. speed	Tool maximum operating speed.
Standard speed	Tool operating speed. This value cannot exceed the maximum speed. You can set this parameter when programming the work piece.
Max. rotation	Maximum number of turns at which the tool can rotate (in turns per minute). **Warning!** Check the speed engraved on the head. To modify this parameter, a password must be inserted: contact service department for assistance.
Rotation standard	Tool rpm during use. This value cannot exceed the maximum rotation. You can set this parameter when programming the work piece.
Direction of rotation	Tool direction of rotation: + clockwise (right-hand tool) - anti-clockwise (left-hand tool)
Speed G0/B	Descent speed in the section which goes from the safety dimension to the machining dimension. You can set this parameter when programming the work piece with G0 or XG0.
Windowed tool change not allowed	Change of tool on main spindle while the drilling machine is operating (default value=NO).
Tool counter position in X	Number of the tool to be used for mirrored routing in X.
Tool counter position in Y	Number of the tool to be used for mirrored routing in Y.

4.1.2.3 External Tools

4.1.2.3.1 Example of a Type F Cylindrical Tool

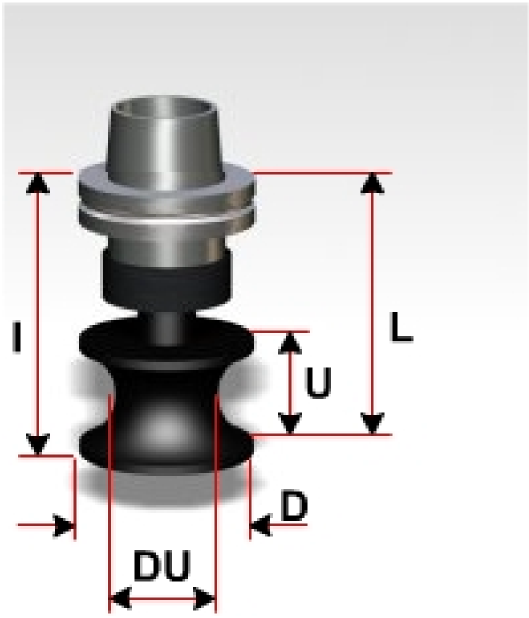


Cutter length	Tool length; is used in programming to calculate the quota in Z (see figure, L).
Cutter diameter	Maximum tool diameter (see figure, D).
Working length	Maximum thickness machined by the tool (see figure, U).
Working diameter	Diameter of tool used to calculate compensation (see figure, D). In this case, equal to the cutter diameter, if the value is 0, the "Cutter diameter" counts.
Wear coefficient in lenght	Not enabled.
Wear coefficient in diameter	Not enabled.
Max. wear in length	Not enabled.
Max. wear in diameter	Not enabled.
Correct length	Not enabled.
Correct diameter	Not enabled.
Max. speed	Tool maximum operative speed.
Standard speed	Tool operative speed. This value cannot exceed the maximum speed. You can set this parameter when programming the work piece.
Max. rotation	Maximum number of turns at which the tool can rotate (in turns per minute). **Warning!** Check the speed engraved on the head. To

	modify this parameter, a password must be inserted: contact service department for assistance.
Rotation standard	Tool rpm during use. This value cannot exceed the maximum rotation. You can set this parameter when programming the work piece.
Direction of rotation	Tool direction of rotation: + clockwise (right-hand tool) - anti-clockwise (left-hand tool)
Speed G0/B	Descent speed in the section which goes from the safety dimension to the machining dimension. You can set this parameter when programming the work piece with G0 or XG0.
Windowed tool change not allowed	Change of tool on main spindle while the drilling machine is operating (default value=NO).
Tool counter position in X	Number of the tool to be used for mirrored routing in X.
Tool counter position in Y	Number of the tool to be used for mirrored routing in Y
Number store	Number of the magazine containing the tool. Must be 0 if the machine has only one store. Not enabled for NWT machines.
Positions store	Position of the tool in the magazine. Not enabled for NWT machines.
Machining surface	Face on which the tool carries out machining.
Overall dimensions	Tool length from the taper to the opposite end.
Code	Tool identification code, if present (max. 11 characters).
Comment	Comment, if present (max. 40 characters).

N.B. Any parameters not indicated in the example are not used for vertical tools.

4.1.2.3.2 Example of a Type F Shaped Tool

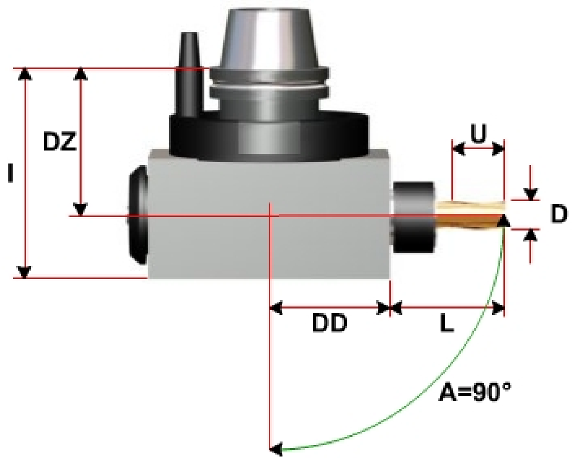


Cutter length	Tool length; is used in programming to calculate the quota in Z (see figure, L).
Cutter diameter	Maximum tool diameter (see figure, D).
Working length	Maximum thickness machined by the tool (see figure, U).
Working diameter	Diameter of tool used to calculate compensation (see figure, DU). If the value is 0, the “Cutter diameter” counts.
Wear coefficient in lenght	Not enabled.
Wear coefficient in diameter	Not enabled.
Max. wear in length	Not enabled.
Max. wear in diameter	Not enabled.
Correct length	Not enabled.
Correct diameter	Not enabled.
Max. speed	Maximum tool machining speed.
Standard speed	Tool machining speed. This value cannot exceed the maximum speed. You can set this parameter when programming the work piece.
Max. rotation	Maximum number of turns at which the tool can rotate (in turns per minute). **Warning!** Check the speed engraved on the head. To modify this parameter, a password must be inserted: contact service department for assistance.
Rotation standard	Tool rpm during use. This value cannot exceed the maximum rotation. You can set this parameter when programming the work

	piece.
Direction of rotation	Tool direction of rotation: + clockwise (right-hand tool) - anti-clockwise (left-hand tool)
Speed G0/B	Descent speed in the section which goes from the safety dimension to the work dimension, (in m/min or mm/min. You can set this parameter when programming the work piece with G0 or XG0.
Windowed tool change not allowed	Change of tool on main spindle while the drilling machine is operating (default value=NO).
Tool counter position in X	Number of the tool to be used for mirrored routing in Y.
Tool counter position in Y	Number of the tool to be used for mirrored routing in X.
Number store	Number of the magazine containing the tool. Must be 0 if the machine has only one store. Not enabled for NWT machines.
Positions store	Position of the tool in the magazine. Not enabled for NWT machines.
Machining surface	Face on which the tool carries out machining; if 0 (zero) it can machine all faces and inclined surfaces (for angular heads and vector axes).
Overall dimensions	Tool length that starts from the taper to the opposite end (see figure, I).
Code	Tool identification code, if present (max. 11 characters).
Comment	Comment, if present (max. 40 characters).

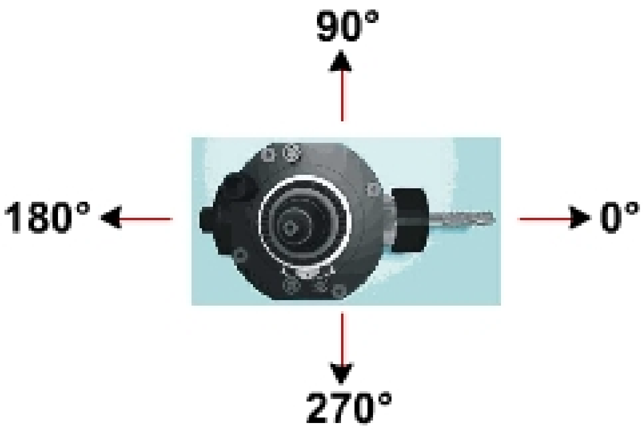
N.B. Any parameters not indicated in the example are not used for vertical tools.

4.1.2.3.3 Example of a Type F Tool Mounted on an Angular Offset



Cutter length	Tool length (see figure, L).
Cutter diameter	Maximum tool diameter (see figure, D).
Working length	Maximum thickness machined by the tool (see figure, U).
Working diameter	Diameter of tool used to calculate compensation (see figure, D). If the value is 0, the “Cutter diameter” counts.
Wear coefficient in lenght	Not enabled.
Wear coefficient in diameter	Not enabled.
Max. wear in length	Not enabled.
Max. wear in diameter	Not enabled.
Correct length	Not enabled.
Correct diameter	Not enabled.
Max. speed	Maximum tool machining speed.
Standard speed	Tool machining speed. This value cannot exceed the maximum speed. You can set this parameter when programming the work piece.
Max. rotation	Maximum number of turns at which the tool can rotate (in turns per minute). **Warning!** Check the speed engraved on the head. To modify this parameter, a password must be inserted: contact service department for assistance.
Rotation standard	Tool rpm during use. This value cannot exceed the maximum rotation. You can set this parameter when programming thework piece.
Direction of rotation	Electro-spindle direction of rotation: + clockwise (right-hand tool)

	- anti-clockwise (left-hand tool)
Speed G0/B	Descent speed in the section which goes from the safety dimension to the work dimension, (in m/min or mm/min. You can set this parameter when programming the work piece with G0 or XG0.
Windowed tool change not allowed	Change of tool on main spindle while the drilling machine is operating (default value=NO).
Tool counter position in X	Number of the tool to be used for mirrored routing in X.
Tool counter position in Y	Number of the tool to be used for mirrored routing in Y.
Number store	Number of the magazine containing the tool. Must be 0 if the machine has only one store. Not enabled for NWT machines.
Positions store	5 or 10 tool position in magazine (angular heads have fixed position). Not enabled for NWT machines.
Machining surface	Face on which the tool carries out machining; if 0 (zero) it can machine all faces and inclined surfaces (for angular heads and vector axes).
Overall dimensions	Tool length that starts from the taper to the opposite end (see figure, I).
Offset X	Must be 0.
Offset Y	Must be 0.
Distance Z	Length that starts at the taper and goes to the joint of the tool rotation axis (see figure, DZ).
Offset R	Tool angle (with vector axis=0, if present) compared to trigonometric zero (see figure below).



Distance D	Distance from the rotation axis of the electro-spindle to the end of the head unit (see figure, DD).
Angle A (B for type	Angle between the perpendicular of the surface and the rotation

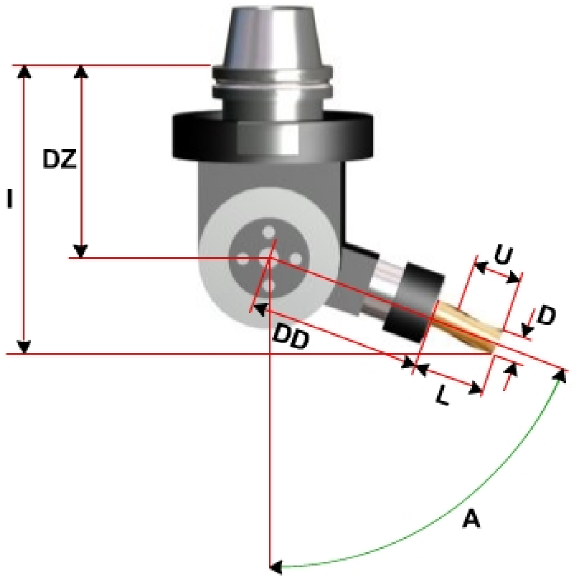
"D")	axis of the tool (see figure, A).
Number total	Number used to identify several tools mounted on the same angular head.

WARNING

Only for TOOL ROOM type magazines, and if all fields "T3(0=NO)" of the type D – RAPID magazine even posts are programmed (in the configuration file storepos.cfg) with the value -1, the value -2 means that this tool is automatically considered as type 2 (dimD=2).
It will therefore not be automatically unloaded from the RAPID.

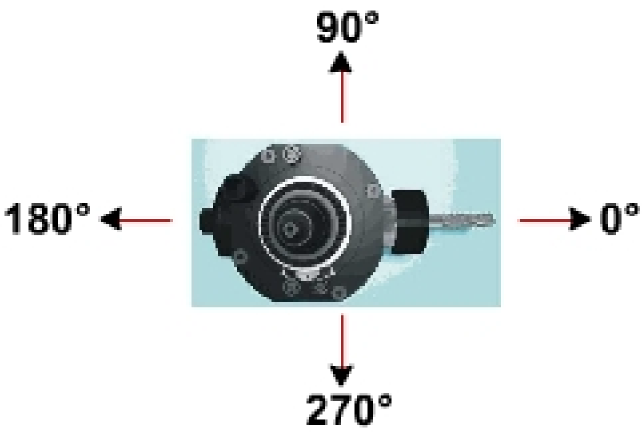
Code	Tool identification code, if present (max. 11 characters).
Comment	Comment, if present (max. 40 characters).

4.1.2.3.4 Example of a Type F Tool Mounted on an Inclined Angular Offset



Cutter length	Length (see figure, L).
Cutter diameter	Maximum tool diameter (see figure, D).
Working length	Maximum thickness machined by the tool (see figure, U).
Working diameter	Diameter of tool used to calculate compensation (see figure, D). If the value is 0, the "Cutter Diameter" counts.
Wear coefficient in lenght	Not enabled.
Wear coefficient in diameter	Not enabled.
Max. wear in length	Not enabled.
Max. wear in diameter	Not enabled.
Correct length	Not enabled.
Correct diameter	Not enabled.
Max. speed	Maximum tool machining speed.
Standard speed	Tool machining speed. This value cannot exceed the maximum speed. You can set this parameter when programming the work piece.
Max. rotation	Maximum number of turns at which the tool can rotate (in turns per minute). **Warning!** Check the speed engraved on the head. To modify this parameter, a password must be inserted: contact service department for assistance.
Rotation standard	Tool rpm during use. This value cannot exceed the maximum rotation. You can set this parameter when programming the work

	piece.
Direction of rotation	Tool direction of rotation: + clockwise (right-hand tool) - anti-clockwise (left-hand tool)
Speed G0/B	Descent speed in the section which goes from the safety dimension to the work dimension, (in m/min or mm/min. You can set this parameter when programming the work piece with G0 or XG0.
Windowed tool change not allowed	Change of tool on main spindle while the drilling machine is operating (default value=NO).
Tool counter position in X	Number of the tool to be used for mirrored routing in X.
Tool counter position in Y	Number of the tool to be used for mirrored routing in Y.
Number store	Number of the magazine containing the tool. Must be 0 if the machine has only one store. Not enabled for NWT machines.
Positions store	5 or 10; position of the tool in the store (the angular offsets have a fixed position). Not enabled for NWT machines.
Machining surface	Face on which the tool carries out machining; if 0 (zero) it can machine all faces and inclined surfaces (for angular heads and vector axes).
Overall dimensions	Length that starts from the taper to the opposite end (see figure, I).
Offset X	Must be 0.
Offset Y	Must be 0.
Distance Z	Length that starts at the taper and goes to the joint of the tool rotation axis (see figure, DZ).
Offset R	Tool angle (with vector axis=0, if present) compared to trigonometric zero (see figure below).



Distance D	Distance from the rotation axis of the electro-spindle to the end of

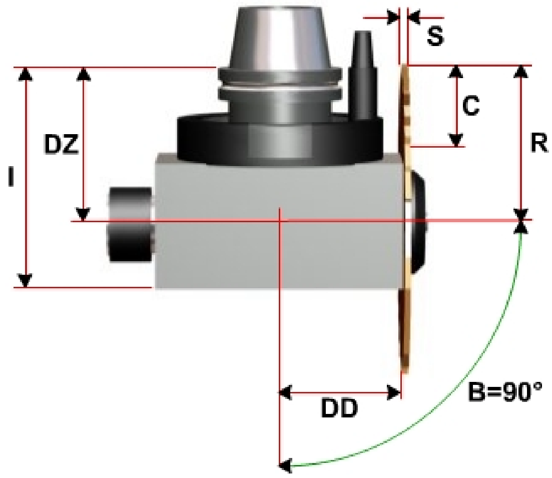
	the head unit (see figure, DD).
Angle A (B for type "D")	Angle between the perpendicular of the surface and the rotation axis of the tool (see figure A).
Number total	Number used to identify several tools mounted on the same angular head.

WARNING

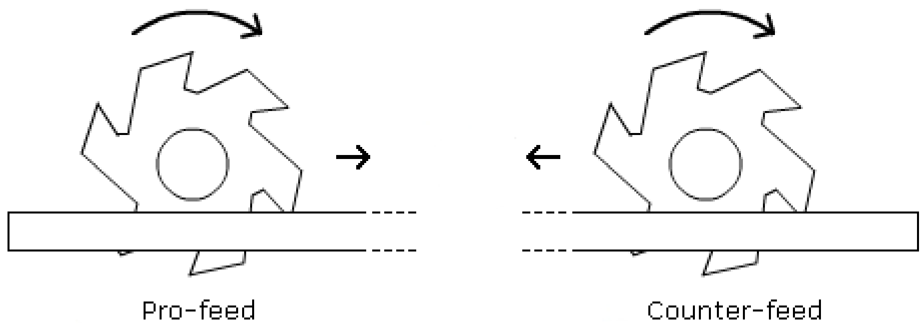
Only for TOOL ROOM type magazines, and if all fields "T3(0=NO)" of the type D – RAPID magazine even posts are programmed (in the configuration file storepos.cfg) with the value -1, the value -2 means that this tool is automatically considered as type 2 (dimD=2).
It will therefore not be automatically unloaded from the RAPID.

Code	Tool identification code, if present (max. 11 characters).
Comment	Comment, if present (max. 40 characters) .

4.1.2.3.5 Example of a Type D Tool Mounted on an Angular Offset

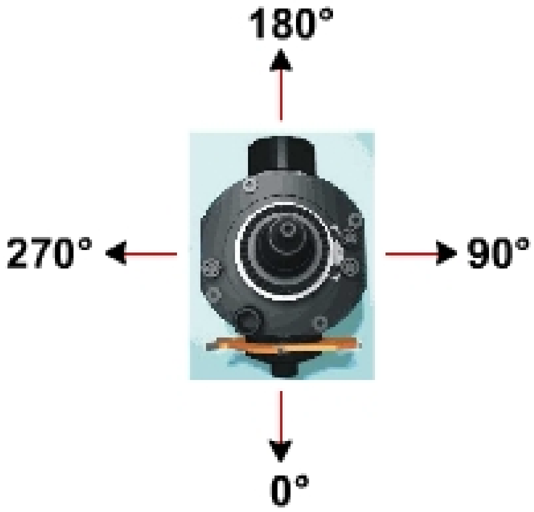


Disk radius	Disk radius (see figure, R).
Working crown	Maximum thickness that the tool can machine (see figure, C).
Blade thickness	Blade thickness (see figure, S).
Machining direction	- **Pro-feed**: the blade moves forward, pushing the teeth downwards. - **Counter-feed**: the blade moves forward, pushing the teeth upwards. Set the machining direction with the + and - values. The resulting machining direction also depends on the direction in which the blade is fitted and the transmission mechanical conditions. See: Appendix L - Blade cuts management.



Wear coefficient of disc in radius	Not enabled.
Wear coefficient of disc in thickness	Not enabled.
Max. wear on radius	Not enabled.
Max. wear on thickness	Not enabled.
Correct radius	Not enabled.

Correct thickness	Not enabled.
Max. speed	Maximum tool machining speed.
Standard speed	Tool machining speed. This value cannot exceed the maximum speed. You can set this parameter when programming the work piece.
Max. rotation	Maximum number of turns at which the tool can rotate (in turns per minute). **Warning!** Check the speed engraved on the head. To modify this parameter, a password must be inserted: contact service department for assistance.
Rotation standard	Tool rpm during use. This value cannot exceed the maximum rotation. You can set this parameter when programming the work piece.
Direction of rotation	Tool direction of rotation: + clockwise (right-hand tool) - anti-clockwise (left-hand tool)
Speed G0/B	Descent speed in the section which goes from the safety dimension to the work dimension. You can set this parameter when programming the work piece with G0 or XG0.
Windowed tool change not allowed	Change of tool on main spindle while the drilling machine is operating (default value=NO).
Tool counter position in X	Number of the tool to be used for mirrored routing in X
Tool counter position in Y	Number of the tool to be used for mirrored routing in Y
Number store	Number of the magazine containing the tool. Must be 0 if the machine has only one store. Not enabled for NWT machines.
Positions store	5 o 10; position of the tool in the store (the angular offsets have affixed position). Not enabled for NWT machines.
Machining surface	Face on which the tool carries out machining; if 0 (zero) it can machine all faces and inclined surfaces (for angular heads and vector axes).
Overall dimensions	Length that goes from the taper to the opposite end (see figure I).
Offset X	Must be 0.
Offset Y	Must be 0.
Distance Z	Length that starts at the taper and goes to the joint of the tool rotation axis (see figure, DZ).
Offset R	Tool angle (with vector axis=0, if present) compared to trigonometric zero (see figure below).



WARNING

The zero for offset R is different than the one for type F tools.

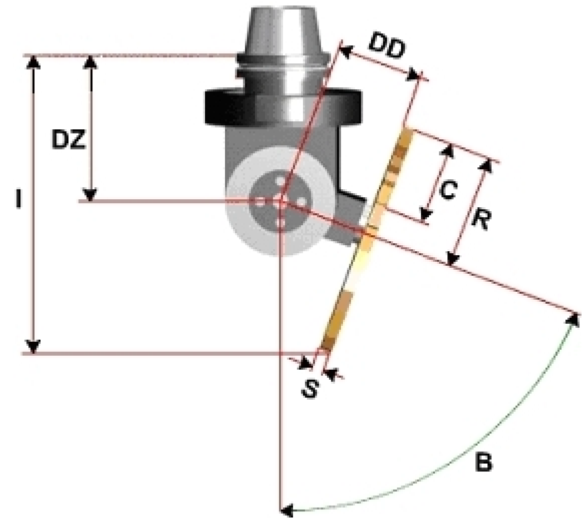
Distance D	Distance from the rotation axis of the electro-spindle to the blade (see figure, DD).
Angle A (B for type "D")	Angle between the perpendicular of the surface and the rotation axis of the tool (see figure, B).
Number total	Number used to identify several tools mounted on the same angular head.

WARNING

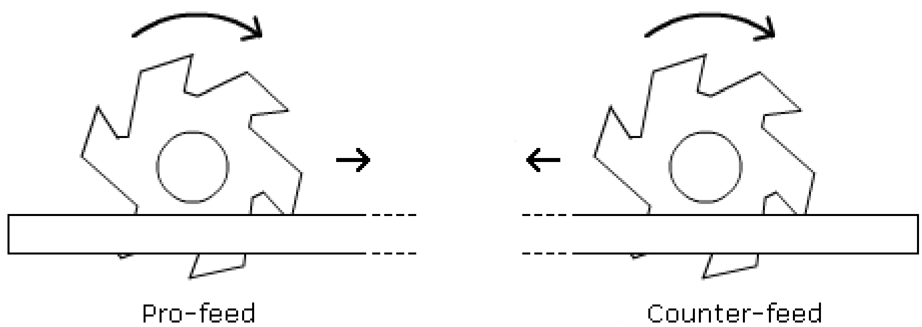
Only for TOOL ROOM type magazines, and if all fields "T3(0=NO)" of the type D – RAPID magazine even posts are programmed (in the configuration file storepos.cfg) with the value -1, the value -2 means that this tool is automatically considered as type 2 (dimD=2).
It will therefore not be automatically unloaded from the RAPID.

Code	Tool identification code, if present (max. 11 characters).
Comment	Comment, if present (max. 40 characters).

4.1.2.3.6 Example of a Type D Tool Mounted on an Inclined Angular Offset

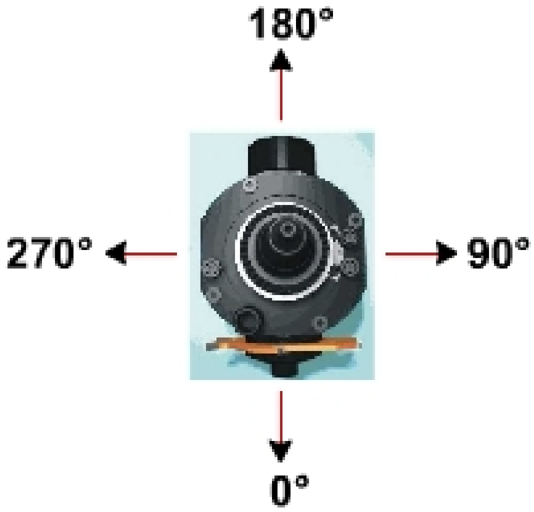


Disc radius	Disc radius (see figure, R).
Working crown	Maximum thickness that the tool can machine (see figure, C).
Blade thickness	Blade thickness (see figure, S).
Machining direction	- **Pro-feed**: the blade moves forward, pushing the teeth downwards. - **Counter-feed**: the blade moves forward, pushing the teeth upwards. Set the machining direction with the + and - values. The resulting machining direction also depends on the direction in which the blade is fitted and the transmission mechanical conditions. See: Appendix L - Blade cuts management.



Wear coefficient of disc in radius	Not enabled.
Wear coefficient of disc in thickness	Not enabled.
Max. wear on radius	Not enabled.
Max. wear on thickness	Not enabled.
Correct radius	Not enabled.

Correct thickness	Not enabled.
Max. speed	Maximum tool machining speed.
Standard speed	Tool machining speed. This value cannot exceed the maximum speed. You can set this parameter when programming the work piece.
Max. rotation	Maximum number of turns at which the tool can rotate (in turns per minute). **Warning!** Check the speed engraved on the head. To modify this parameter, a password must be inserted: contact service department for assistance.
Rotation standard	Tool rpm during use. This value cannot exceed the maximum rotation. You can set this parameter when programming the work piece.
Direction of rotation	Tool direction of rotation: + clockwise (right-hand tool) - anti-clockwise (left-hand tool)
Speed G0/B	Descent speed n the section which goes from the safety dimension to the work dimension. You can set this parameter when programming the work piece with G0 or XG0.
Windowed tool change not allowed	Change of tool on main spindle while the drilling machine is operating (default value=NO).
Tool counter position in X	Number of the tool to be used for mirrored routing in X.
Tool counter position in Y	Number of the tool to be used for mirrored routing in Y.
Number store	Number of the magazine containing the tool. Must be 0 if the machine has only one store. Not enabled for NWT machines.
Positions store	5 or 10 tool position in magazine (angular heads have fixed position). Not enabled for NWT machines.
Machining surface	Face on which the tool carries out machining; if 0 (zero) it can machine all faces and inclined surfaces (for angular heads and vector axes).
Overall dimensions	Length that goes from the taper to the opposite end (see figure I).
Offset X	Must be 0.
Offset Y	Must be 0.
Distance Z	Length that starts at the taper and goes to the joint of the tool rotation axis (see figure, DZ).
Offset R	Tool angle (with vector axis=0, if present) compared to trigonometric zero (see figure below).



WARNING

The zero for offset R is different from the one for type F tools.

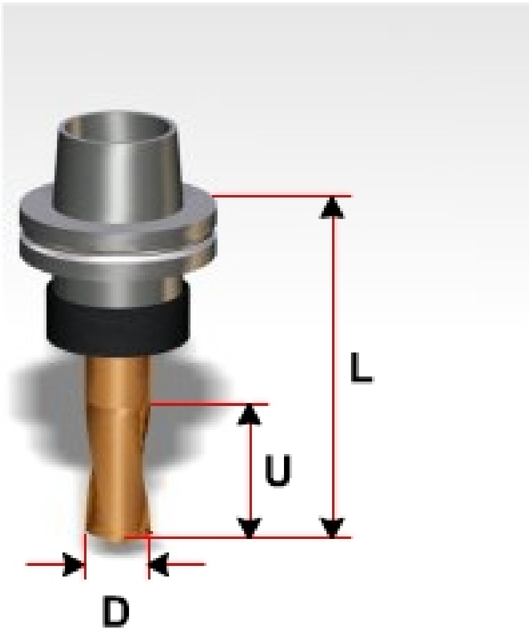
Distance D	Distance from the rotation axis of the electro-spindle to the blade (see figure, DD).
Angle A (B for type "D")	Angle between the perpendicular of the surface and the rotation axis of the tool (see figure, B).
Number total	Number used to identify several tools mounted on the same angular head.

WARNING

Only for TOOL ROOM type magazines, and if all fields "T3(0=NO)" of the type D – RAPID magazine even posts are programmed (in the configuration file storepos.cfg) with the value -1, the value -2 means that this tool is automatically considered as type 2 (dimD=2).
It will therefore not be automatically unloaded from the RAPID.

Code	Tool identification code, if present (max. 11 characters).
Comment	Comment, if present (max. 40 characters).

4.1.2.3.7 Example of a Type L Tool



Pad length	Tool length; is used in programming to calculate the quota in Z (see figure, L).
Pad diameter	Maximum tool diameter (see figure, D).
Working length	Maximum thickness machined by the tool (see figure, U).
Working diameter	Diameter of tool used to calculate compensation (see figure, D). In this case, equal to the pad diameter, if the value is 0, the “Pad diameter” counts.
Max. speed	Tool maximum operative speed.
Standard speed	Tool operative speed. This value cannot exceed the maximum speed. You can set this parameter when programming the work piece.
Max. rotation	Maximum number of turns at which the tool can rotate (in turns per minute). **Warning!** Check the speed engraved on the head. To modify this parameter, a password must be inserted: contact service department for assistance.
Rotation standard	Tool rpm during use. This value cannot exceed the maximum rotation. You can set this parameter when programming the work piece.
Direction of rotation	Tool direction of rotation: + clockwise (right-hand tool) - anti-clockwise (left-hand tool)
Speed G0/B	Descent speed in the section which goes from the safety dimension to the machining dimension. You can set this parameter when programming the work piece with G0 or XG0.
Windowed tool change not allowed	Change of tool on main spindle while the drilling machine is operating (default value=NO).

Tool counter position in X	Number of the tool to be used for mirrored routing in X.
Tool counter position in Y	Number of the tool to be used for mirrored routing in Y
Number store	Number of the magazine containing the tool. Must be 0 if the machine has only one store. Not enabled for NWT machines.
Positions store	Position of the tool in the magazine. Not enabled for NWT machines.
Machining surface	Face on which the tool carries out machining.
Overall dimensions	Tool length from the taper to the opposite end.
Offset X	See: 4.1.2.3.3, 4.1.2.3.4.
Offset Y	See: 4.1.2.3.3, 4.1.2.3.4.
Distance Z	See: 4.1.2.3.3, 4.1.2.3.4.
Offset R	See: 4.1.2.3.3, 4.1.2.3.4.
Distance D	See: 4.1.2.3.3, 4.1.2.3.4.
Angle A (B for type “D”)	See: 4.1.2.3.3, 4.1.2.3.4.
Number total	See: 4.1.2.3.3, 4.1.2.3.4.
Code	Tool identification code, if present (max. 11 characters).
Comment	Comment, if present (max. 40 characters).

4.1.2.4 Notes

Speed G0/B
If the tool is used during boring, this is the default value for execution of the hole (only if not programmed in the V field, see instructions B and BR). If the tool is used during milling, this is the speed at which the tool enters the wood (only if greater than zero; otherwise the entry speed is the same as the feed speed). The entrance speed can also be set in the field V of the GO/G0R instruction.
Max. speed
Only valid if the tool is used for milling. If greater that zero, it corresponds to the maximum speed for tool feed.
Standard speed
Only valid is the tool is used for milling. If greater than zero, it is the standard tool feed speed (if not programmed in field V of the G0 instruction). It cannot be greater that the "Maximum speed".

NOTES These field are used only if their value is greater than zero. In this case function is as follows:

- **Boring without V parameter**: the lower value of "Speed G0/B" and "Max. speed" is used.
- **Boring with V parameter:** the lower value of "V" and "Max. speed" is used.
- **Routing without V parameter in the instruction G0/G0R**: to feed the piece the lower value of "Speed G0/B" and "Max. speed" is used.
- **Routing without V parameter in the instructions following G0/G0R**: to feed the piece the lower value of "Standard speed**"** and "Max. speed" is used.
- **Routing with V parameter in the instruction G0/G0R**: to feed the piece the lower value of "V" and "Max. speed" is used.
- **Routing with V parameter in the instructions following G0/G0R**: to feed the piece the lower value of "V" and "Max. speed" is used.

Max. rotation. Not to be programmed for fixed tools. For external tools, it corresponds to the speed of the spindle set during milling and boring phase only if inferior to the speed programmed in the S field.

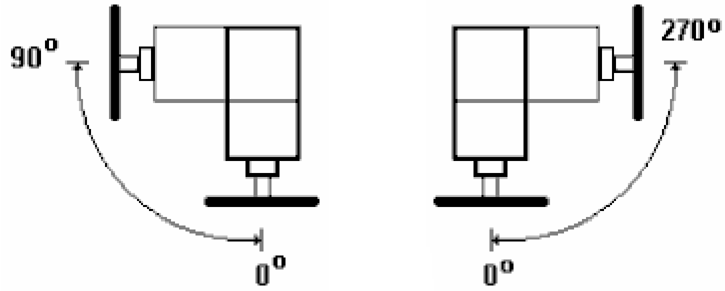
4.1.2.5 Blade Unit

The Blade Unit can be configured in any position of the configuration file pheads.cfg between 4 and 11. The Blade Unit is configured by entering 6 in the "Actuator" field; The configuration of the tools mounted on the Blade Unit is similar to the configuration of external tools.

No rotation

In the configuration file pheads.cfg, the "Limitswitch+ (positive) of Vector**"** field relative to the Blade Unit must be set to -1; The tool installed can be associated to any configuration from E1 to E96.

Pneumatic rotation



In the configuration file pheads.cfg, the “Limitswitch+ (positive) of Vector” field relative to the Blade Unit must be set to -90 two different tools between E1 and E96 must be configured with the convention that the odd numbered tools make the unit rotate, the even ones do not. All of the tools must have the same set number; the tooling "Offset R" parameter must be configured correctly (see: examples of tool configuration on angle heads).

Motorized Rotation (Vector)

In the configuration file pheads.cfg the “Limitswitch+ (positive) of Vector” field relative to the Blade Unit must be greater or equal to zero; the tool associated to the unit can be any one of those between E1 and E96.

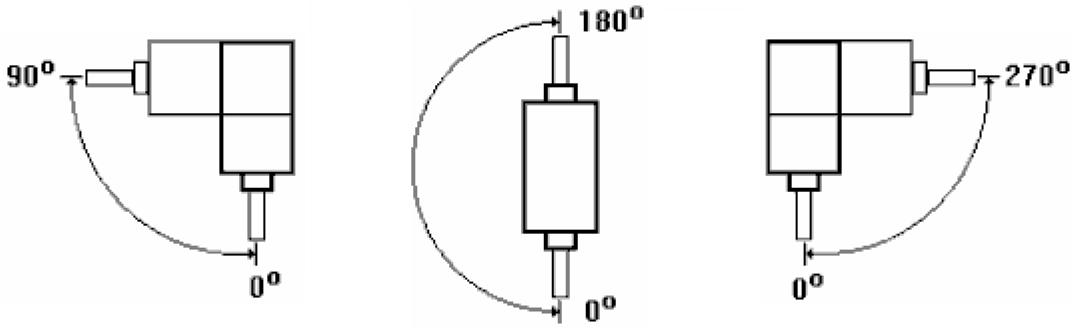
4.1.2.6 Lock Mortising Unit

The Lock Mortising Unit can be configured in any position of the configuration file pheads.cfg between 4 and 11. The Lock Mortising Unit is configured by entering 5 in the “Actuator” field of the configuration file pheads.cfg and can accommodate one (in the *single overhanging* unit) or two tools (in the *double overhanging* unit). The configuration of the tools mounted on the Lock Mortising Unit is similar to the configuration of external tools.

No rotation

In the configuration file pheads.cfg, the “Limitswitch+ (positive) of Vector” field relative to the Lock Mortising Unit must be set to -1; the tools installed can be associated to any configuration from E1 to E96.

Pneumatic rotation



In the configuration file pheads.cfg, the “Limitswitch+ (positive) of Vector” field must be set to -90; two (in the *single overhanging* unit) or four different tools (in the *double overhanging* unit) must be selected between E1 and E96 with the convention that the odd numbered tools make the unit rotate, the even ones do not. All of the tools must have the same set number; the tooling "Offset R" parameter must be configured correctly (see: examples of tool configuration on angle heads).

Motorized Rotation (Vector)

In the configuration file pheads.cfg, the "Limitswitch+ (positive) of Vector" field must be greater or equal to zero; the tools associated to the unit must be selected from those between E1 and E96.

4.1.2.7 Vertical Unit

The Vertical Unit can be configured in any position of the configuration file pheads.cfg between 4 and 11.The Vertical Unit is configured by entering 7 in the "Actuator" field of the configuration file pheads.cfg; the configuration of the tools mounted on the Vertical Unit is similar to the configuration of external tools. The tools installed can be associated to any configuration from E1 to E96.

4.1.2.8 Borer on Independent Axis

To configure the Borer on independent axis write 1 in the number 1 head (borer) "Actuator" field of the configuration file pheads.cfg.

4.1.2.9 Managing a "Large Tool"

Automatic management

The following modifications are necessary for the management of a "large tool" (for example an angular head with a 300 mm blade), which is, however, compatible with the tool changers on the machine, and with automatic management using automatic tool changes.

1. Configuration file:

- gendata.cfg: the axis Z distance from the table must be detected by the pneumatic movement of the high reference head (spindle).
- xilog3.cfg: the origin in Z must be detected by the pneumatic movement of the high reference head (spindle).
- pheads.cfg: in reference head 3 (spindle), insert the true stroke of the piston, in mm, in the parameter "Set up 0 (Axis Z mm)" and "Set up 1 (Axis Z mm)".
- cfg.cfg: insert a new key to manage the new ascent command of the main head so that it is always FH.

```
$H 03_UPFH
E10015=0
M 81
E10015=1
$
```

- axis.cfg: enable the axis travel stop control.

2. Program PGM:

Insert the SETDONTCARE=1 instruction at the beginning of each profile in which the angle head with 300 mm blade is to be used. In this case, the safety distances from the piece are no longer observed.

3. %8086 (Tool Changer Program; Record 240-142):

Enable the variable [UTBIG]=1 (management UT>300 mm large)

Management with manual loading on the spindle

If the "large tool" is not compatible with the tool changers on the machine, due to the particular dimensions (diameter) or volumes (for example, in the case of several angle heads), automatic management is not a possibility. In these cases, the operator can manually change the tool in MDI mode (see: Machine Panel Manual), using the codes M 51 (enable main spindle tool release) and M 171 (memorise tool number present on spindle; to be executed only after a M51). For the use of these codes see: machine diagnostics.

For a manual tool change, several precautions must be taken:

- The operation is allowed only on the main spindle of the machine and is not otherwise possible on any supplementary spindles;
- A special piece program must be generated, which uses only one tool (that introduced manually in MDI mode), to prevent the machine from going into block before an automatic tool change that is incompatible with the manual tool change. Additional machining operations with other tools on the same piece must therefore be inserted in a distinct program to be executed afterwards with the normal automatically managed tool change procedures.

4.1.3 Fields of Significance for the Boring Optimizer

Only type P (hole tip) tools are considered by the boring optimization algorithm. Since it is carried out in a single operation (with just one instruction), a series of borings must consist of elements having the same depth, the same execution speed and the same machining surface.
To generate an instruction comprising this series of borings, the optimizer proceeds as follows:

- it only considers the tips mounted on the spindles active in the set machining surface;
- it disables all bits having "Working length" less than the set depth;
- only analyzes tools with the same "Tip length".".

In order to be capable of performing a given boring, a tip must meet the following conditions:

- the "Tip diameter" must coincide with the boring diameter (remember that in case of fixed tools, the "Working diameter" is the same as the "Tip diameter");

- The “Type tip” (L=lance, P=flat, S=countersunk) must coincide with that set for the bore. A boring set with a given “Type tip” cannot be executed by a tip the “Type tip” field for which is not programmed.
- The “Height countersink” of the tool must be the same as that of the boring, if the hole has been programmed tapered and with taper other than zero.

4.1.4 Supplementary Tool Change (Tool Room)

If the machine features additional Tool Room (T) type tool change, the management of the tools and their magazine locations are completely automatic and therefore the fields “Number store” and “Positions store” of the external tool configuration should not be programmed.
The “Number total” field, if programmed with a value between 1 and 96, means that specific tool is considered (together with all the other tools having the same aggregate number) as belonging to a single head. This head is represented by the wording A*n*, with *n* that goes from 1 to 96 in the supplementary tool change management table, in the MDI environment (see: Machine Panel manual).
After programming the tooling file data, tools must be loaded in the machine magazines by means of the tool change management table, in the M.D.I. environment. Upon completion, the system will update the two parameters of tooling file mentioned above automatically and they will remain valid while the machine is being used. This way it is possible to program and manage more than one tooling file: when enabling a different tooling file used previously, the system will automatically configure all data saved in the file (and therefore the tool change management table now shows the old configuration) but the user must actually reset this condition by manually positioning the tools in the magazine of the machine as indicated in the configuration.

4.1.5 Reference to Tools in Programming

During programming, the field containing the tool number (field T if the instruction supports it) with a number *m* that identifies univocally the configured tool *n* according to the following rule:

a) If m is a number between 1 and 96, it indicated a fixed tool.

Example:

T = 3 Fixed tool 3 of the boring head.

b) In the case of boring operation with fixed tools, it is also possible to set a multiple operation. By programming more tools at the same time, (separated by a blank space or a comma, if the numbers are not contiguous, by a dash if they are in sequence) boring operation takes place with a unique downward movement of the boring head of the machine. In the case, the X, Y and Z positions set with boring instructions refer to the hole made by the first tool of the programmed list. The holes that come after the first one deviate in X, Y, Z compared to the first hole equal to the offset in X, Y, Z of the programmed tools compared to the first tool on the list.

Examples:

T = 5 2 1 Tools 5, 2 and 1 of the boring head, in one unique downward movement.
T = 1 – 4 Tools 1, 2, 3, 4 of the boring head, in one unique downward movement.

c) A group of three figures is used to indicate an external tool, the first figure of which indicates the electro-spindle or unit where the tool is mounted (for example, unit 100 will be indicated by the figure 1), and is followed by two figures that indicate the number of the tool (from 01 to 96).

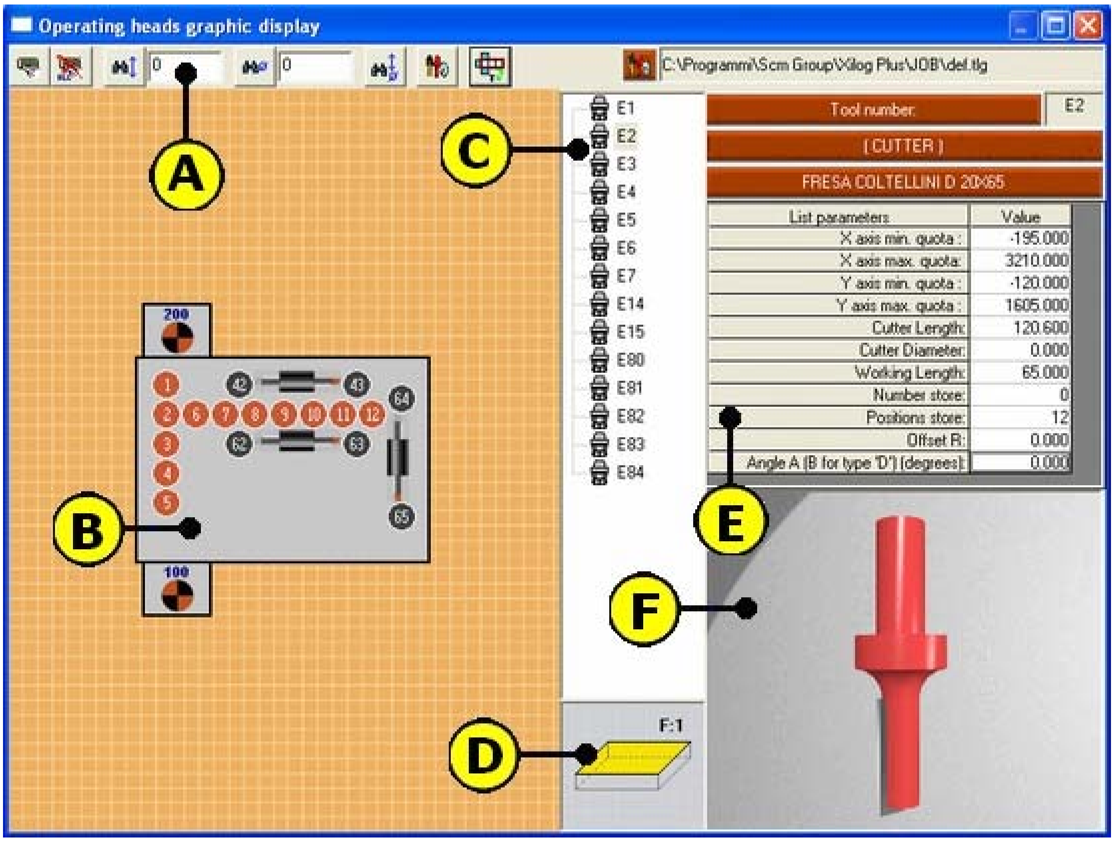
Examples:

T = 101 External tool 1 (E1), on main electro-spindle 1.
T = 280 External tool 80 (E80), on the Universal unit.

d) Entering "NULL" in field T allows you to indicate a "null" tool, i.e.: not configured in the tooling.
The null tool cannot be used for machining on the workpiece, but can be used for example in the edge banding instructions or in those used only for the graphical representation.
The null tool cannot be used with instruction XT.

4.1.6 Operating Heads Graphic Display

The Operating heads graphic display graphically displays the operative heads and information relative to the individual tools.

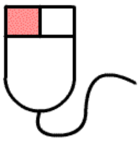


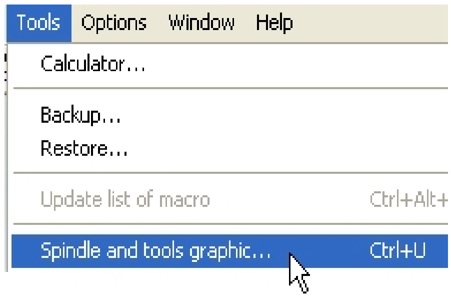
A) Toolbar (see: Appendix A). The text box to the right displays the selected tooling file.
B) Graphic representation of operative heads. The fixed tools can be selected with the mouse.

C) List of configured external tools. The tools on the list can be selected with the mouse.
D) Graphic representation of the machining surface involved with the selected tool.
E) Table of parameters for the tool selected. At the top are indicated: the tool number, the tool type and any comments are entered in the "Comment" box.
F) Picture of the type of tool selected.

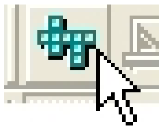
► Viewing the Operating heads graphic display.

Click on the **TOOLS/SPINDLE AND TOOLS** graphic menu.





… or on the button.



4.1.7 Heads Configuration

See: Configuration Manual for the Heads.

4.2 Program

4.2.1 Introduction to Machining Programs

4.2.1.1 Program Structure

The program is a sequence of instructions describing a machining cycle of the panel. It is saved in a binary file in the main PGM format and is translated into ISO language when it is executed by the machine.
Each program must the **Header** as the first instruction, followed by the other instructions (profiles, boring, movements).
The program can contain various profiles (or geometric routes that the tool must following on a continuous basis) and borings. Each file must have, as the first instruction, the **Automatic entry (GIN** or **XGIN)** followed by the **Start Milling (G0** or **XG0)**. These are followed by all other geometric commands and the final command **automatic exit** (**GOUT** or **XGOUT).**

Schematic example of a profile:

GIN	automatic entry
G0	start routing
…	geometry
…	geometry
GOUT	automatic exit

Schematic example of a program with boring and milling instructions:

H	header
GIN	automatic entry
G0	start routing
…	geometry
…	geometry
GOUT	automatic exit
B	boring
GIN	automatic entry
G0	start routing
…	geometry
…	geometry
GOUT	automatic exit

Instructions are classified in:

- **Operating instructions.** They are instructions defining the actual machining operations of the piece. When running the program, each of these instructions can correspond to either a single movement or a set of movements of the operating unit of the machine. For example, in the case of boring, the operating head of the machine carries out the operation in various pre-set phases.
- **Modal instructions.** Modal instructions can be written several times inside a program but the setting of a subsequent modal instruction cancels the previous one. These modify the

meaning of the X, Y and Z present in all operative instructions following the call of the modal instruction. For example, setting a different work face means that the X, Y, Z readings of all subsequent operative instructions refer to the origin of the new set face. In the operative instructions for extended language, that do not correspond to the intermediate machining of a profile, setting modal instructions is facilitated by the presence of a data entry screen with four standard fields for instructions F, C, K and P (in the guided text editor) or by buttons on the modal data bar (in the graphic editor). In the graphic editor however, only the face change buttons (equivalent to instruction F) and tool correction buttons (equivalent to instruction C) are active: in order to use them it is necessary to select face change or tool correction before entering a new instruction. NOTE. The PL/XPL instruction and the face F change instruction cancel each other.

- **Instructions for managing subprograms.** They allow to recall inside a program another program previously set. Here it is possible to pass new parameters using parametric programming.
- **Sending messages to the operator.** The messages to the operator contain information sent to the operator while the piece is being machined.
- **Programming the motorised table.** These instructions are to program the position of the supports of the machine with respect to the origin of the panel. In addition, they view or memorize diagnostic messages during graphic display of the program. These instructions are ignored in the program execution phase.

WARNING

The Header is a required instruction and must not be preceded by any other instruction. Incorrect programming of machining with slanted tools may cause damages to the structure of the machine. It is advisable that a SIMULATION of this type of operation be executed first.

4.2.1.2 Program Editor

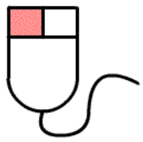
For writing and changing programs Xilog Plus provides two types of editor:

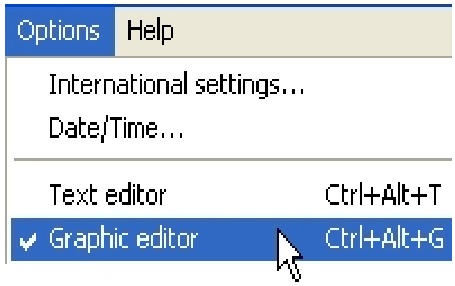
- **text** editor
- **graphic** editor

The type of editor must be selected **before** creating or opening a program.

► Selecting type of editor.

Click on the **OPTIONS** menu, bring the mouse pointer over the type of editor preferred and click to select it.

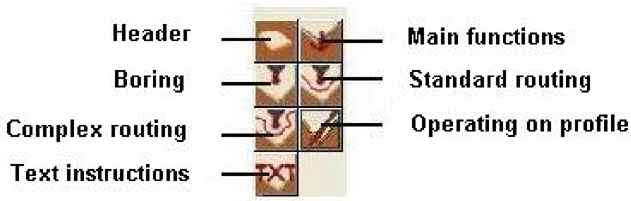




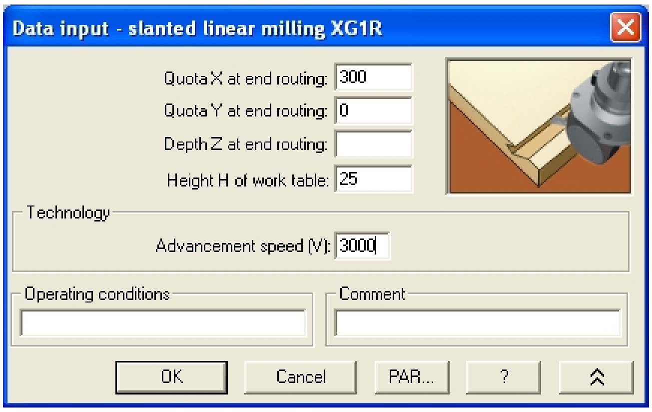
The instructions available for programming the machining belong to two classes:

- **basic** instructions (or text);
- **extended** instructions (or graphic), marked with an X in front to distinguish them from the corresponding basic instructions.

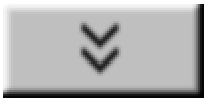
With the text editor it is possible to insert instructions that belong to both classes, while with the graphic editor it is possible to insert mainly extended instructions (even though it is capable of graphically interpreting basic instructions as well).
Inserting extended instructions (and some basic instructions in text editor) is facilitated in both the editors by using the icons on the **instructions bar**, that graphically represent the individual instructions collected into groups. Clicking on a group icon with the **right** mouse button will display a list of group instructions.



The text editor, in guided mode, also has a screen of its own with which to insert parameters, which can be used as an alternative to using the instructions bar.
If an instruction is inserted using the instructions bar, a graphic screen will appear in which to insert the data.



In most cases, when the screen opens it only shows the main or “basic” parameters; the other instructions parameters (“extended”) can be made to appear by clicking on the button

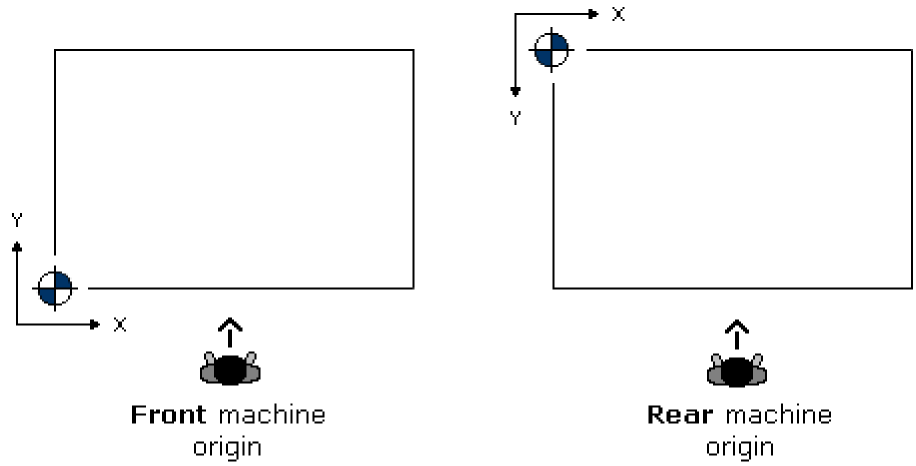


By clicking a second time on this button (if the screen is extended the arrows appear turned upwards), the extended parameters will be hidden again.
The following buttons are also available:

- **OK**, to confirm the instructions has been inserted;
- **DELETE**, to delete the selection of an instruction;
- **PAR…**, to open the parameters table;
- **?**, to access the Xilog Plus on-line Help

4.2.1.3 Machine Origin

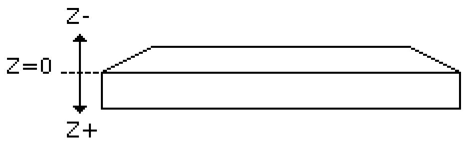
Depending on the type of machine, the point 0 from which the distances are measured (machine origin) may be **front** or **rear**.



For example:

- front origin, on the Record and Ergon machines;
- rear origin, on the author, Pratix and Tech machines.

The movement in depth (parameter **Z**) is indicated with **positive** values. For example, to program a 100mm deep drilled hole, starting from position Z=0, a value of 100 is inserted into the Z parameter. To the contrary, a value of -100 will cause the miller to be moved away from the work table.

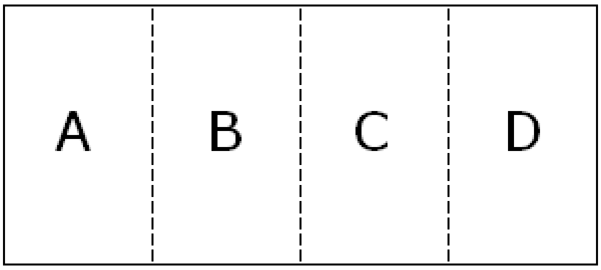


WARNING!

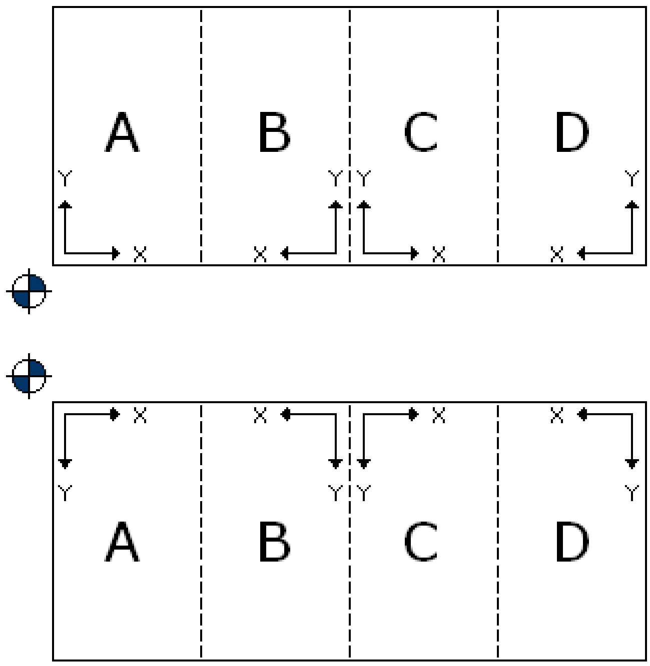
On the Record, Ergon and Pratix machines not equipped with CNC OSAI, the parameter Z is inverted (Positive Z=away from the work table, negative Z=towards the work table).

4.2.1.4 Work areas

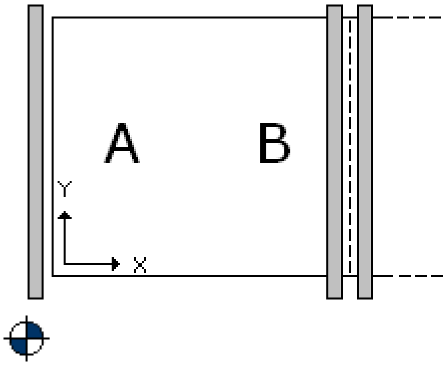
Every program is run in a **work area**. The work table is normally divided into 4 areas (A, B, C, D).



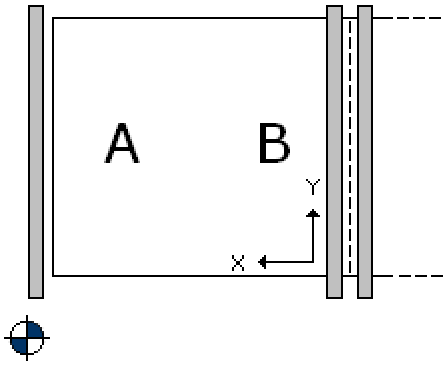
Each area has its own **axes origin**, oriented according to the machine origin.



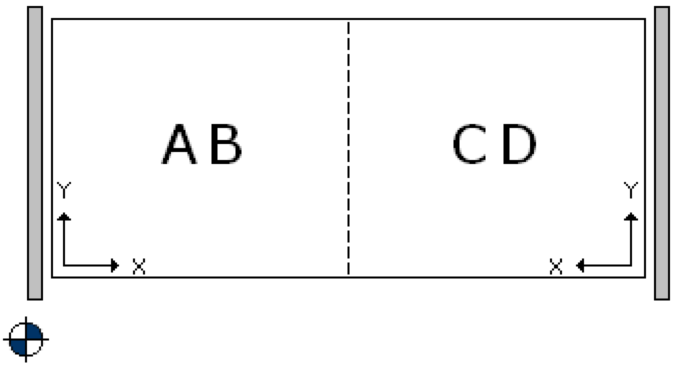
On **table with a centre zero and four mobile stops** (one to the left, one to the right, and two at centre), the presence of the centre mobile stops makes it possible to program single machining operations independently on the four areas. Additionally, the areas can be used in pairs: for example, if the panel to be machined has dimensions which occupy two areas, a double area can be used in the programming phase, i.e. area AB (with the axes origin in A)...



or BA (with the axes origin in B)

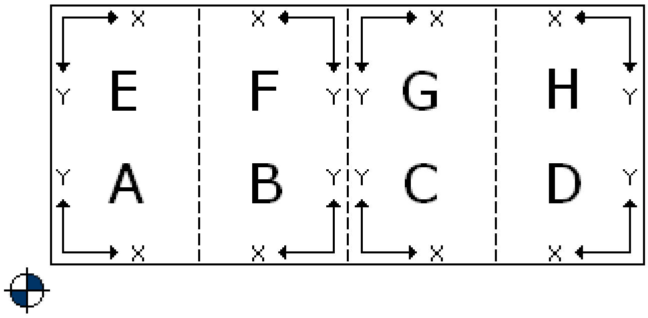


On **tables without a centre zero (without centre mobile stops)** the presence of the stops only at the left and right ends of the work table makes it possible to only use the areas AB and DC. In this case, it is not possible to set the machining program on the areas B, C, BA and CD, while the setting of areas A and D is automatically considered by Xilog Plus, respectively equivalent to areas AB and DC.



WARNING! For machines not equipped with CNC OSAI the following configuration pertains: left area=A; right area=B.

The table can also be configured with specular areas: for example, with respect to the above illustrated areas A, B, C, D the respective specular areas E, F, G, H are available on the work table.

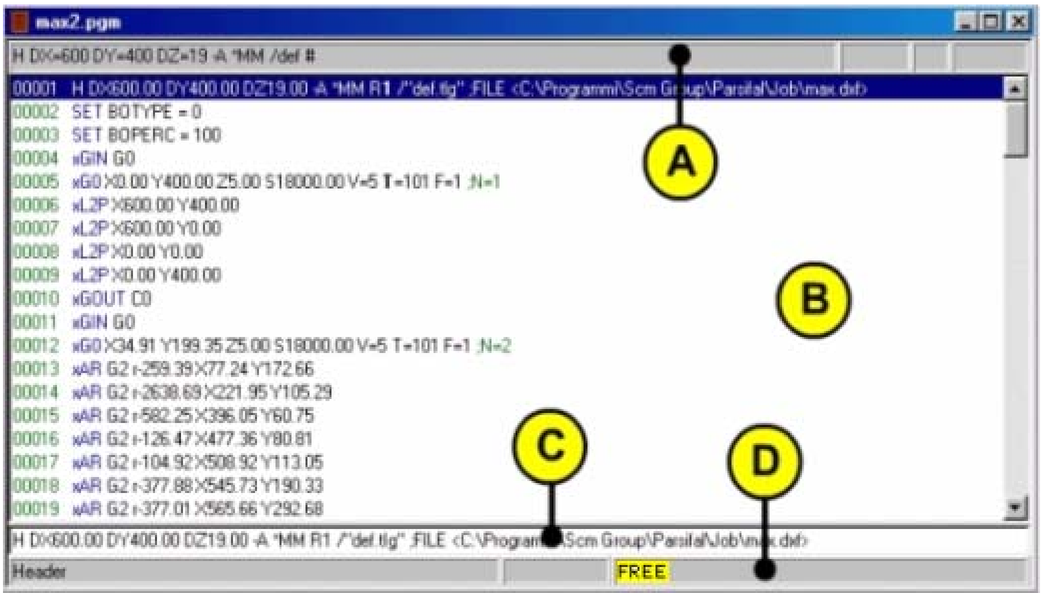


To set the work area of a program, see: instruction H.

4.2.2 Text Editor

4.2.2.1 Interface

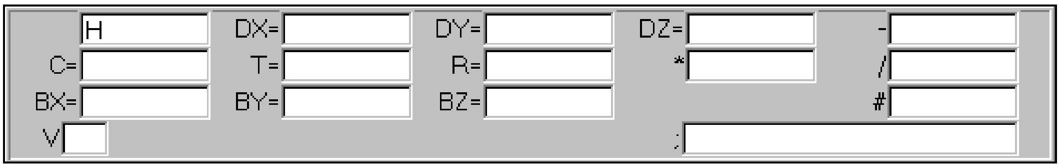
The editor presents itself as a window divided into four areas (A, B, C**,** D):



A) This always displays the program Header.
B) Scrolling window dedicated to displaying program steps.
C) Area reserved for entering or changing program instructions.
D) Area that displays help information.

The text editor can be used in two modes:

- **Free** (free editor). In this mode Area C presents itself as one sole box, inside of which it is possible to write and delete (the "FREE" sign will appear in the box to the right of area D).
- **Guided** (guided editor). In this mode, a screen will appear in place of the box to facilitate entering the parameters (the "GUIDED" sign will appear in the box to the right of area D).

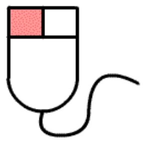


► Selecting free mode or guided mode from the text editor.

Click on the view menu, bring the mouse pointer over the preferred editor mode and click to select.

It is always possible to pass from free mode to guided mode, except if the editor is in error mode

(see further on).

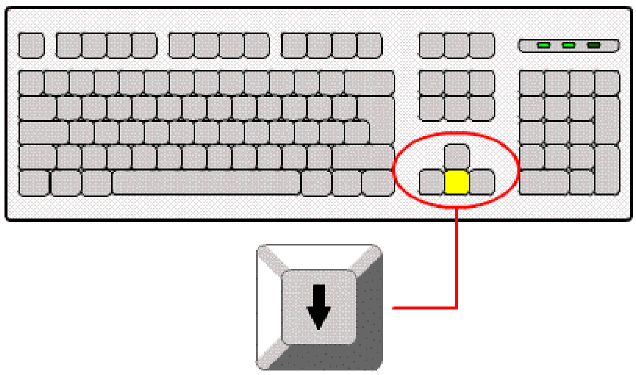


► Entering a new instruction in free mode.

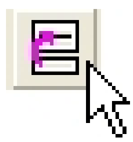
1

After having entered the Header, press the **[DOWN ARROW]** key.

The editor passes to insert mode: the number for a new instruction appears in area B and box C appears empty. The INS sign appears in area D.



To insert a new instruction in the main part of the program press the button or the **[INS]** key: the new instruction will be inserted after the one selected.

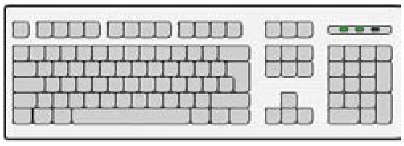


2

Write the new instruction in the box in area C.

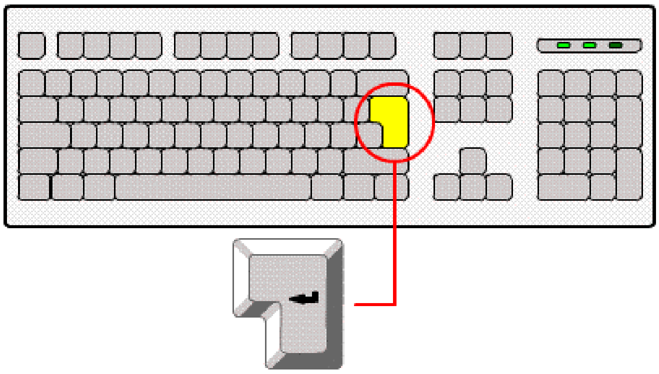
Suggestion

To prevent the numeric value from being converted into mm or inches, add a hatch sign (#) after it.



3

Press the **[ENTER]** key: the new instruction will be inserted into the program. After the instruction is confirmed Xilog Plus returns to insert mode.



Warning!

If an erroneous instruction is inserted, the editor will enter error mode (the sign BLK will appear in area D)

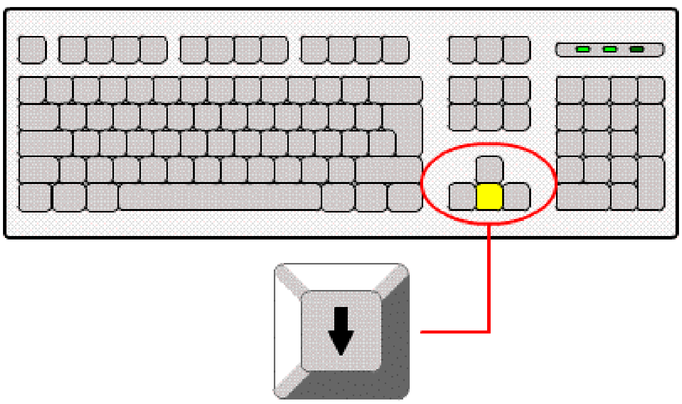
The error must be corrected so that an new instruction can be inserted.

► Inserting a new instruction in guided mode.

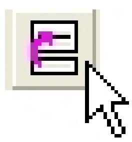
1

After having inserted the Header. press the key **[DOWN ARROW]**

The editor passes to insert mode: a number appears in area B for the new instruction and a new empty box appears on the insert screen. The sign INS appears in area D.



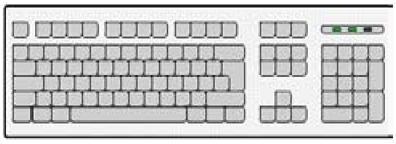
To insert a new instruction in the main part of the program press the button or the **[INS]** key: the new instruction will be inserted after the one



selected.

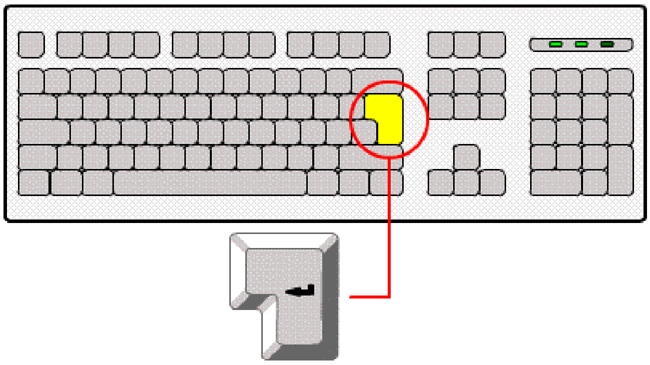
2

Write the abbreviation for the instruction in the box.



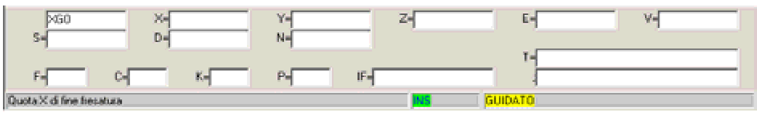
3

Press the **[ENTER]** key.



4

Boxes will appear in which to enter the parameters for the relative instruction. Write the parameters in the boxes. When finished press the **[ENTER]** key: the new instruction will be entered into the program. Once an instruction is confirmed, Xilog Plus returns to the insert mode.



Suggestion

To prevent the numeric value from being converted into mm or inches, add a hatch sign (#) after it.

Warning !

If an erroneous instruction is inserted, the editor will enter error mode (the sign BLK will appear in area D)

The error must be corrected so that a new instruction can be inserted.

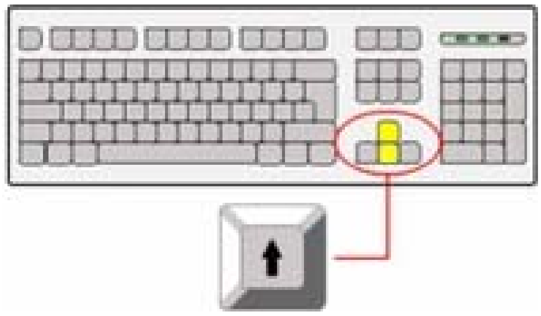
NOTE. An instruction can also be inserted using the instructions bar: see: chapter 4.2.3 – Graphic Editor.

► Changing an instruction (in free or guided mode).

1

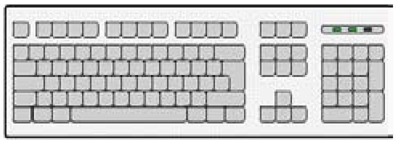
Press the **[UP ARROW]** or **[DOWN ARROW]** key to move between the program instructions.

The editor passes to edit mode.



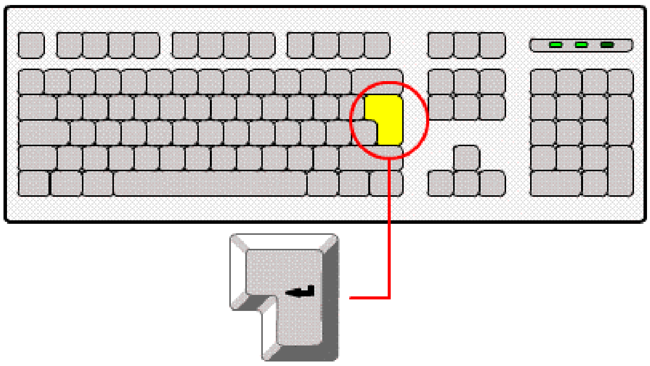
2

Edit parameters in the text box of the free editor or in the box on the screen of the guided editor.



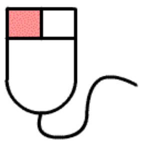
3

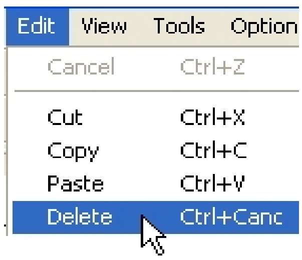
Press the **[ENTER]** key to confirm the edit.



4

To delete the selected instruction, click on the **EDIT/DELETE** menu.





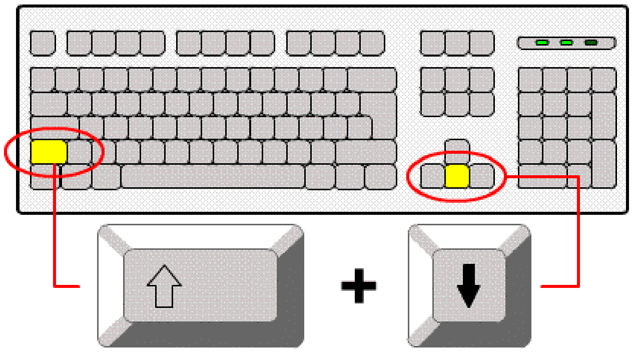
4.2.2.2 Block Mode

With the Xilog Plus text editor it is possible to cut or copy a block of lines from the program and paste it in a different position or inside another program.

► Cutting and copying a block of lines and pasting them into a different position.

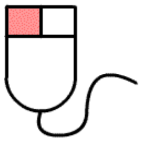
1

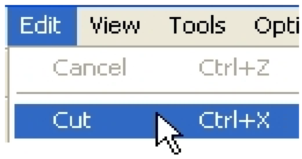
Select the first line to be cut by moving onto the program with the **[DOWN ARROW]** or **[UP ARROW]** keys. Press the **[SHIFT]** + **[DOWN ARROW]** key to select other lines below; **[SHIFT]** + **[UP ARROW]** to select other lines above.



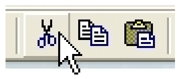
2

Click on the **EDIT/CUT** menu…





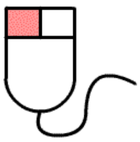
… or the **CUT** button.

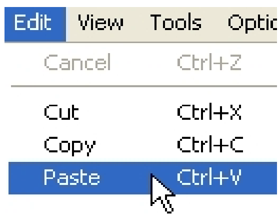


The cut lines will disappear from the screen.

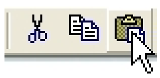
3

Select a line from the program. Click on the **EDIT/PASTE** menu …





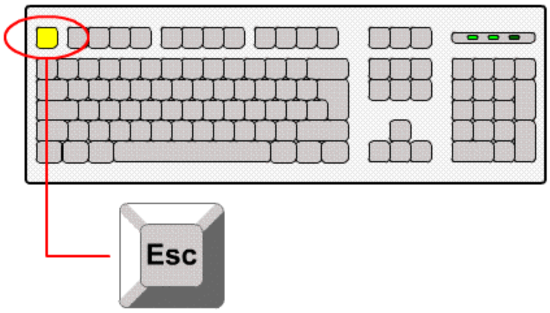
… or on the **PASTE** button



The cut lines will be pasted after the selected instruction.

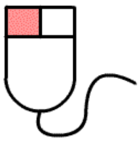
4

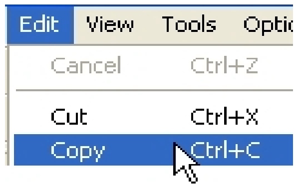
To exit the blocks mode press the **[ESC]** key.



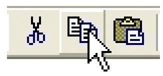
5

Repeat steps 1-3, but at step 2, instead of entering the **CUT** command, click on the **EDIT/COPY** menu.





… or on the **COPY** button.



The copied lines are not removed from their original position and can be pasted like the cut lines.

Suggestion

The cut or copied lines can be pasted more than once.

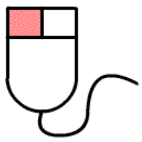
The Cut/Copy and Paste functions can be used in text editor, even outside the block mode, but only with one instruction line at a time.

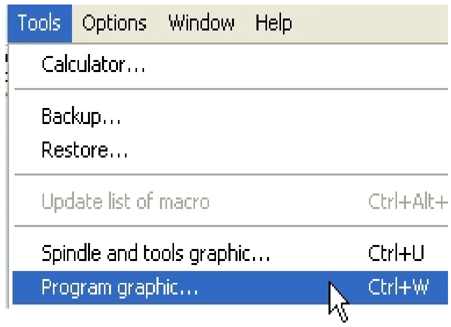
4.2.2.3 Displaying Graphics

The Displaying graphics graphically shows the machining that is being programmed with the text editor. For programs that refer to subprograms (with S/XS instruction), the main program must be closed and reopened again in order to update the graphics of the subprograms.

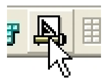
► Graphically viewing machining programmed with text editor.

Click on the tools/ program graphic **BUTTON**.





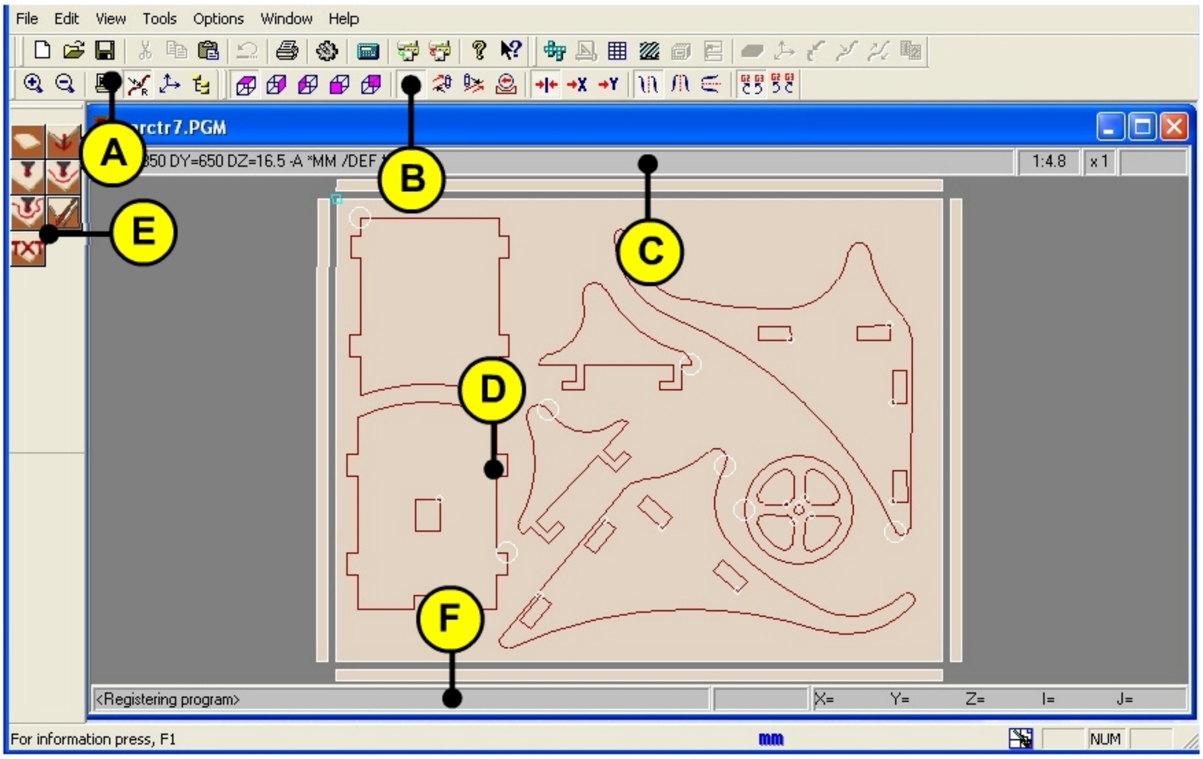
… or on the button.



4.2.3 Graphic Editor

The graphic editor graphically represents the machining instructions. The graphic image is updated as the program instructions are inserted; for programs that refer to subprograms (with S/XS instruction), the main program must be closed and reopened again in order to update the graphics of the subprograms.
In this environment it is possible to insert all the instructions listed in chapter 5.3 (extended instructions) – that represent a subgroup of the instructions available in text editor (basic instructions – see: chapter 5.2) – using the graphic mask on the instructions bar (see further on). The other instructions available in text editor are also represented, but they cannot be inserted or edited from the graphic editor, other than in free text mode.
N.B. It is suggested that the basic instructions be not inserted by the graphic editor, except for instructions for special use (like ISO and SET instructions).

The graphic editor appears as a window divided in three areas (C, D, F); in this editor mode an additional two new bars are active (A, B). The instructions bar (E) will be used to insert data.



A) Graphic bar (see: Appendix A).
B) Modal data bar (see: Appendix A).
C) This area always displays the program Header. In the box to the right they also appears information on the view scale and the value of the magnifying zoom
D) This area displays graphics on machining instructions. The upper part of the panel surface is represented with the side profiles all around. The instructions selected are graphically represented with various colors:
- **red**, for the individual sections;
- **green**, for the profiles;
- **fuchsia**, for the through sections (that is, those that exceed in depth the thickness of the panel);

- blue, for through profiles.

The selected section also displays a light blue square that marks the end of the section, and a circle positioned at the beginning of the section that graphically represents the correction of the tool radius.

E) Instruction bar.

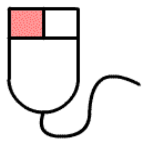
F) Area that displays help information.

► Inserting a new instruction.

1

The instructions are collected into groups in the instructions bar. We will attempt to insert a boring instruction.

Click on the boring button.

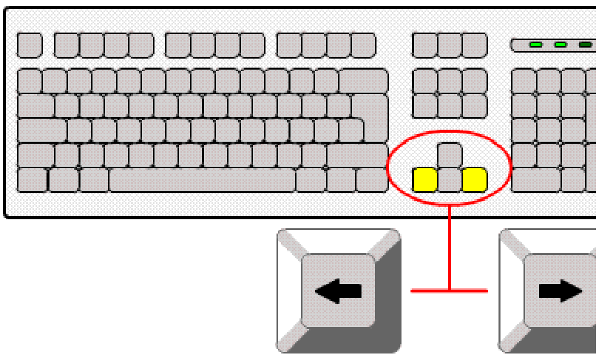


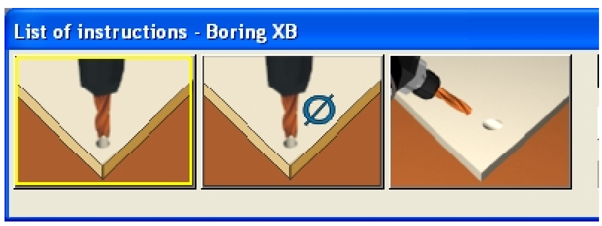
2

A window will appear showing the instructions available in the selected group.

Press the **[RIGHT ARROW]** and **[LEFT ARROW]** keys to scroll the list of instructions, represented by an image.

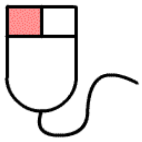
The image for the selected instruction is highlighted by a yellow outline.

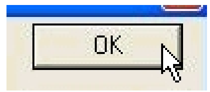




3

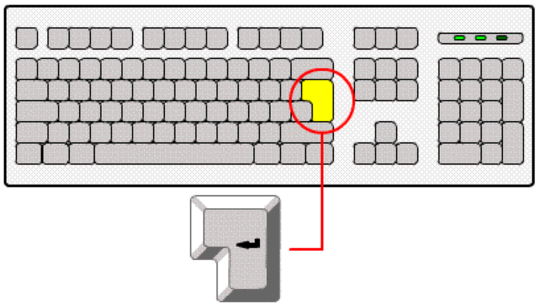
Click on the **OK** button to confirm the instruction selected.





Suggestion

To confirm the instruction selected, it is also possible to click directly on the image or press the **[ENTER]** key.



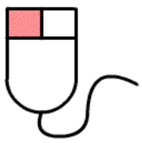
4

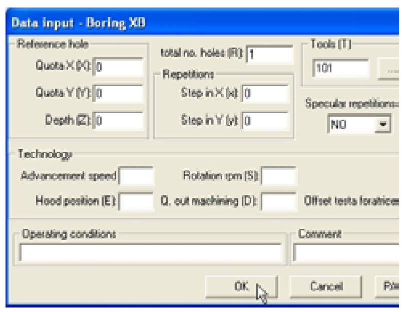
A screen will appear for inserting the parameters for the instructions.

Write the parameters inside the boxes. To enter the instruction into the program click on the **OK** button or press the **[ENTER]** key.

Suggestion

To prevent the numeric value from being converted into mm or inches, add a hatch sign (#) after it.





Warning!

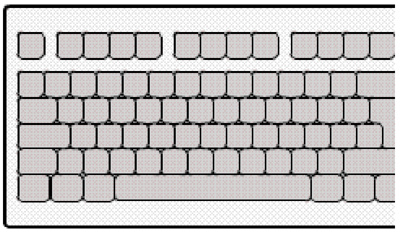
If an erroneous instruction is inserted, the editor will enter error mode (the sign BLK will appear in area D) and the graphic representation is not updated. Xilog Plus requires that the error be corrected in order to continue; if the error is not corrected, the last error free condition will be restored..

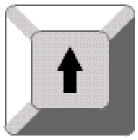
► Editing an instruction.

1

To select the instructions already inserted in the program, press the **[UP ARROW]** or **[DOWN ARROW]** key.

The name of the selected instruction appears in the box to the left of area C; the parameters X, Y, Z, I, J will appear in the box to the right.

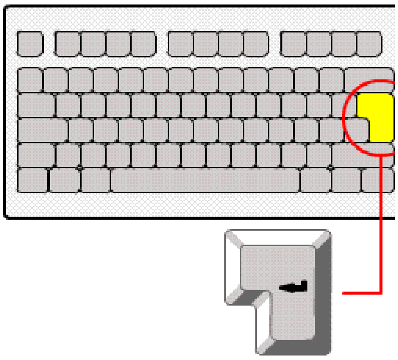




2

Press the **[ENTER]** key.

A window will appear for inserting the parameters relative to the selected instruction.

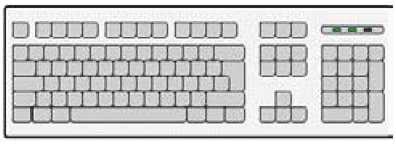


Suggestion

The instructions may also be chosen directly on the graphic display by clicking on the corresponding line or circle; a double click, or pressing the **[ENTER]** key opens the window for changing the parameters.
If no instruction has been selected, press the **[ENTER]** key to open the Heading parameters window (to deselect an instruction, click on the **EDIT/RE-INTERPRET** menu).

3

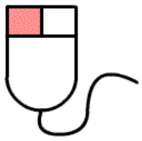
Edit parameters. Press the **[ENTER]** key to confirm the change.

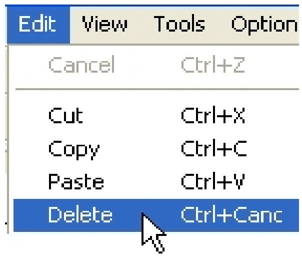


4

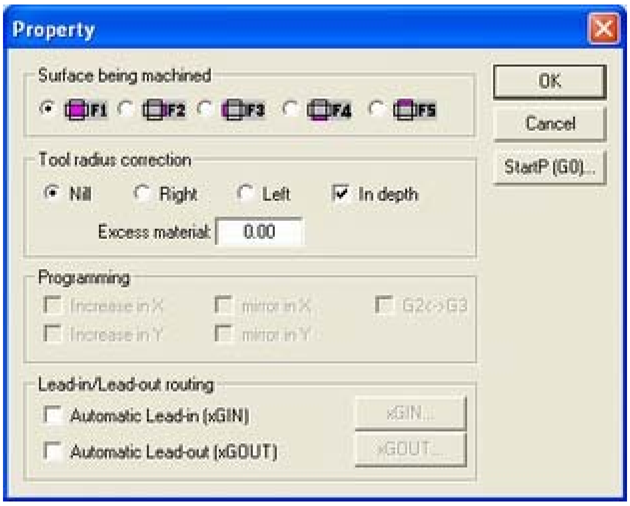
To delete the selected instruction, click on the **EDIT/DELETE** menu.

The single selected segment is eliminated by default. It is also possible to eliminate the entire profile that the selected segment is part of: select the proper option in the window which appears before confirming the deletion.





For the instructions complete with modal parameters, there is a window available for setting the Properties with different options. Select the instruction and click on the **EDIT/PROPERTIES** menu.



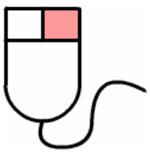
In the case of milling, there are also buttons for directly accessing the parameter windows for the XGO, XGIN and XGOUT instructions.

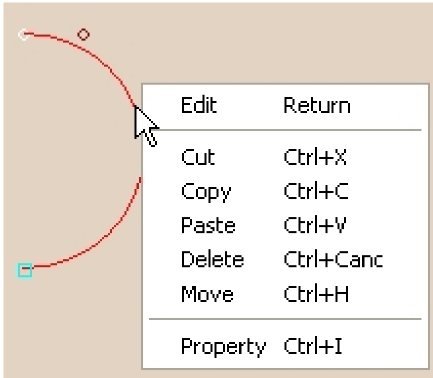
► Moving or duplicating an instruction.

1

The instructions represented graphically can easily be moved or duplicated with the quick pick menu.

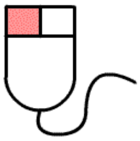
Select a section with the **left** mouse key, then click on the section with the **right** mouse key to view the quick pick menu.

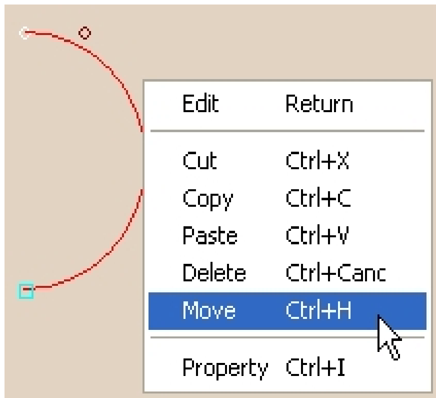




2

Click on **MOVE**.

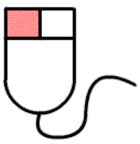


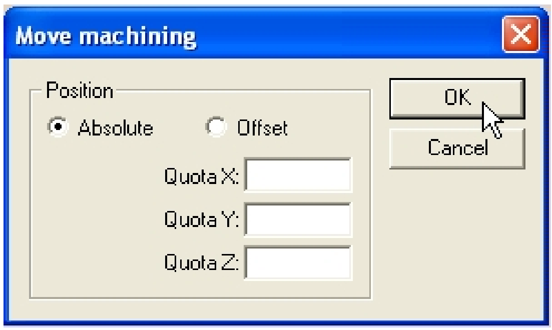


3

*The **MOVE MACHINING** screen will appear.*

Insert the quotas (absolute or offset) for the new position and click on the **OK** button.

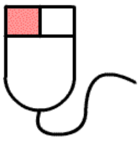


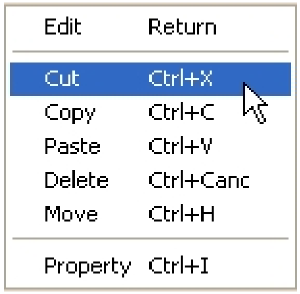


4

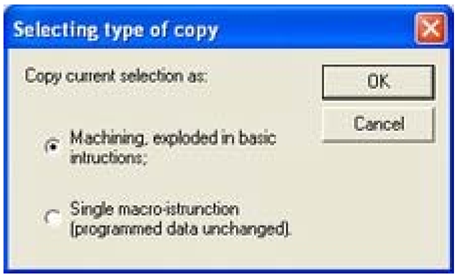
The **CUT** command (for moving) and the **COPY** command (for duplicating), which are also available in the quick pick menu, can be applied to the selected instructions as well.

Cut or copy the selected instruction.





The copy of a graphic boring instruction, an XEA instruction or a sub-program can be made in two ways: as one or more simple machining operations (default value) or as a single instruction.



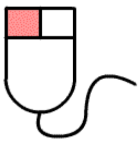
WARNING!
The first mode DOES NOT MAINTAIN THE PARAMETRIC SETTINGS since the machining is not copied as a set of instructions (each with its parameters) but as a machining operation represented by basic instructions programmed with the absolute coordinates (X, Y, Z, …). This mode is the only one that guarantees that the machining operation, regardless of where it is pasted into the program, maintains its characteristics.
To the contrary, the machining operation may modify its characteristics since the value of the characters may differ from one point in the program to another.

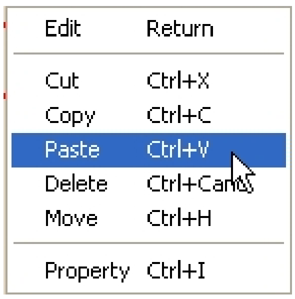
For example, an XB routing instruction which calls for 5 repetitions of the same hole, will be copied (and reproduced with the **PASTE** command, if necessary):

- in the first case (default value), as 5 distinct instructions;
- in the second case, as a single instruction.

5

Click on paste.



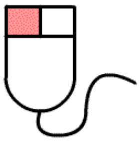


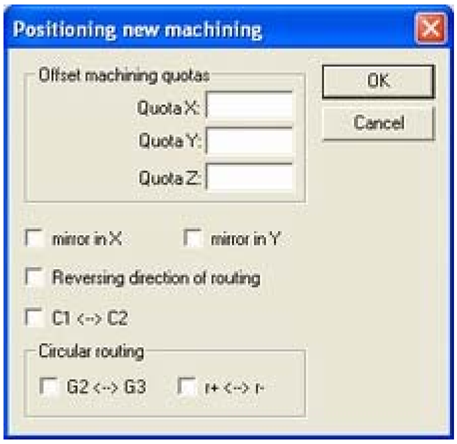
6

*The **POSITIONING NEW MACHINING** window will appear.*

Insert the offset quota for the new position and any specular positioning; confirm by clicking on the **OK** button.

If the Mirror X/Y or Reversing direction of routing options are chosen, Xilog Plus automatically selectes other suggested options for positioning the work.

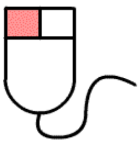




► Display the program structure.

For easier machining on a complex program, it is possible to display the program structure.

Click on the **VIEW/PROGRAM STRUCTURE** menu…



… or on the button.

The selected machining operation is highlighted in the graphic representation. The machining operations represented in the structure may be selected and moved by dragging them with the mouse; there are also commands in the rapid selection menu. The structure displays the comments inserted in the "Comment" field of the instructions (G0/XG0, in the case of profiles).

NOTE 1.

Old programs generated with the previous versions of Xilog3/Routolink: the program changes made with Xilog3/Routolink which contain errors such as "Unprogrammed speed – Tools not defined – Axis in end of run – etc." are guaranteed with 100% effectiveness only if re-edited with text editor.

NOTE 2.

In the graphic editor, comments like “AFX_BEGIN ... AFX_END” must absolutely not be edited.

4.3 Mix of Programs

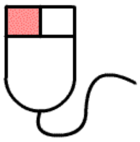
With the **mix** it is possible to create a file for executing, in sequence, a list of ready programs. The instructions for a mix begin with the letter P and are formatted by the name and the header for the programs to be executed. The mix editor is similar to the programs editor in text mode.

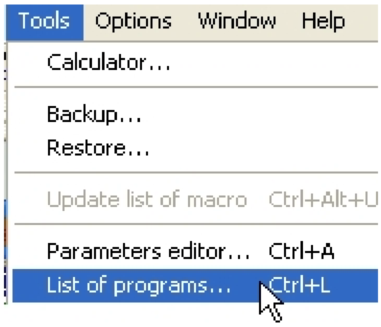
► Creating a mix.

1

To create a mix it is necessary to tell Xilog Plus which programs are to be executed.

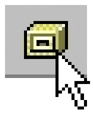
Click on the **TOOLS/LIST OF PROGRAMS** menu.





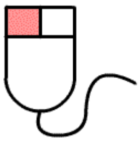
… or on the button.

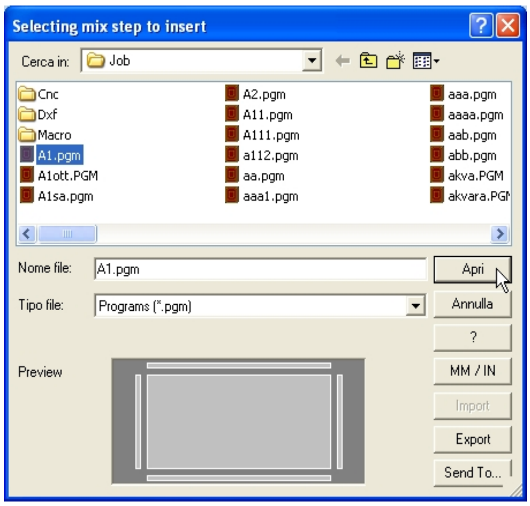
The window for opening programs will open.



2

Select the program to be inserted in the mix and click on the **OPEN** button.





The instructions for the mix can also be written directly. In this case, in free text mode the instructions must be written in the text insert box; in the guided mode it is necessary to write the letter P in the text box so that the screen for inserting parameters appears.

Parameters:

/	Program name.
DX	Dimension in X of the panel.
DY	Dimension in Y of the panel.
DZ	Dimension in Z of the panel.
-	Work area in which the program must be executed; the authorized values are A, B, C, D, AB, BA, CD, DC, AD, DA.
R	Number of identical panels to produce (max. 9999).
*	Unit of measure; the authorized values are MM (millimeters) and IN (inches); if the field is omitted, the unit of measure used is the one specified in the machine parameters.
/	Name of tooling-up configuration file.
#	Name of environment variables file.
BX	Distance in X between the panel zero and the field zero.
BY	Distance in Y between the panel zero and the field zero.
BZ	Dimension in Z of a shim positioned under the panel.
(empty)	Box for importing program parameters (PAR) .
C	Type of machining. the authorized values are: 0 for normal machining 1 for continued machining
T	Enables/disables the options. Equivalent to the T field of the H instruction (see paragraph 5.1)
V	Enables/disables piece blocking and the check of the position of the automatic suction cups (if present). Equivalent to V field of the H instruction (see paragraph 5.1)

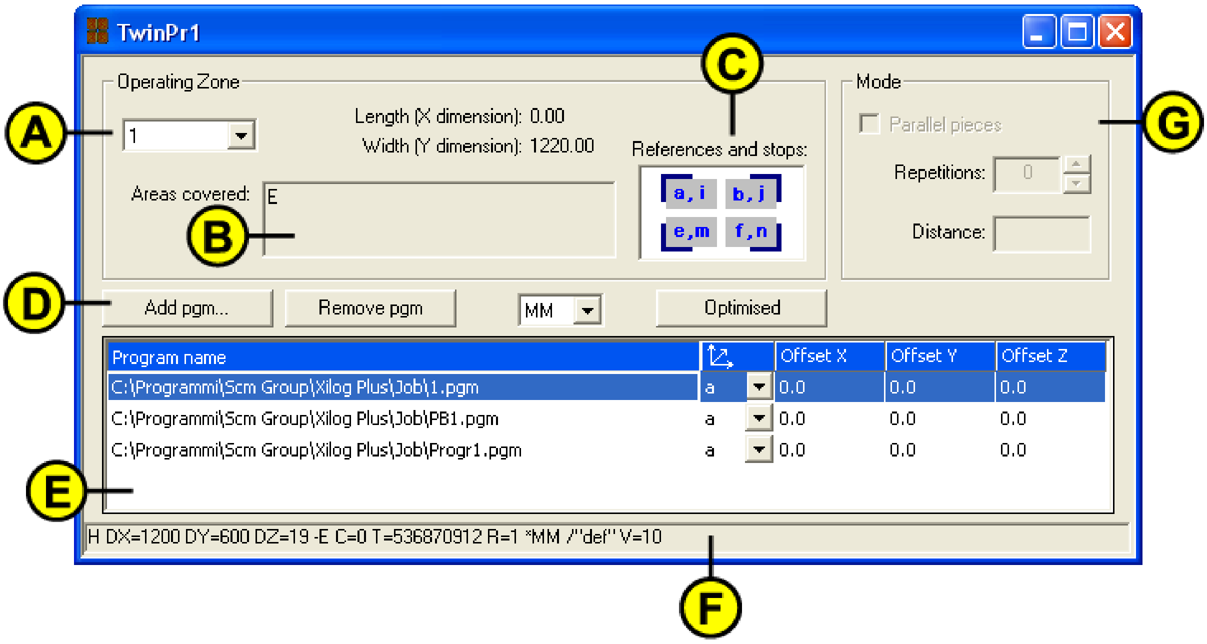
Example:

P /test DX1000 DY500 DZ20 -AB R10 *MM /DEF

4.4 Multiple Program

4.4.1 Multiple Programs Editor

Multiple programs allow you to associate different machining programs (from two to eight) in an area of the table consisting of one or more "zones". The division of the table into zones depends on the machine configuration; each zone may consist of several areas. Each program entered in the multiple program may be further positioned in one of the areas that are part of the selected zone, or with reference to the possible origins of the axes.
The multiple program is then saved in a normal machining program, optimised if necessary. The new program inherits the programming of the supports present in the various programs in the multiple program; it can be run and edited, including the programming of the supports. If the programs entered have been positioned relative to an origin of the axes, the resulting program may be run in all of the zones on the table.

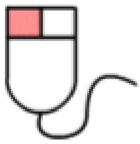


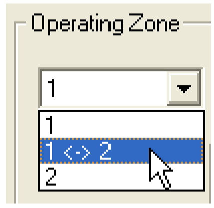
A) Setting the zone.
B) Areas available for the selected zone.
C) Diagram of possible origins of the axes (according to the table's machine origin)
D) Keys for multiple program composition and optimisation.
E) List of programs entered in the multiple program.
F) Heading of the program selected in the list of programs.
G) Programming mode:
When the programming **PIECES IN PARALLEL** mode is selected, if you set the **REPETITION** field with a number of repetitions which is higher than zero together with an Offset X in the **DISTANCE** field, you obtain that:
by selecting the first program, the list is automatically filled with a number of programs equal to the number of determined repetitions plus one.
The programs are all the same except for the position programmed in the **OffsetX** column which is zero for the first program and is increased of the value set in the **DISTANCE** field for each program following the first one (i.e. **Distance**, for the second one, **Distance***2, for the third one and so on).

► Let’s create a multiple program. (Standard Mode)

1

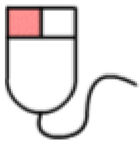
Select the zone.





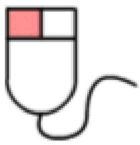
2

Click on the **ADD PGM** key.



3

Select a program and click on the **OPEN** key to enter it.



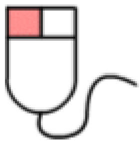
To eliminate a program from the list, select the program and click on the **REMOVE PGM** button.

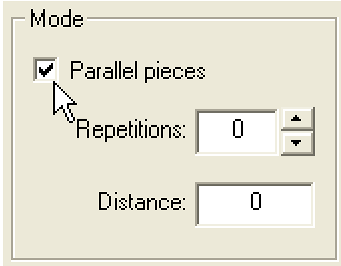
It is possible to directly create the programming file for the work table Editor supports; the file is automatically produced on the basis of the programming of the work tables for the individual PGM programs added in the multiple program.
It is also possible to modify the supports programming file several times before it is inserted, at the moment it is saved, in the PGM program.
To create or modify the programming file for the work table Editor supports, if it has been previously produced, click on the buttons on the function Bar.

► Let’s create a multiple program. (Piece in parallel mode)

1

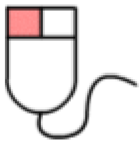
Select the Pieces in parallel mode.

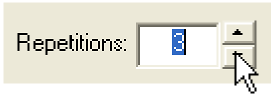




2

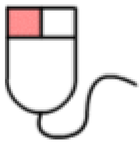
Set the number of repetitions.





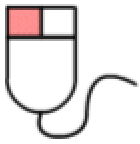
3

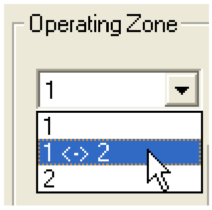
Set the X Offset in the Distance field.



4

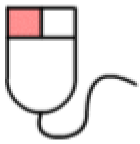
Select the zone.





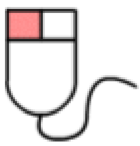
5

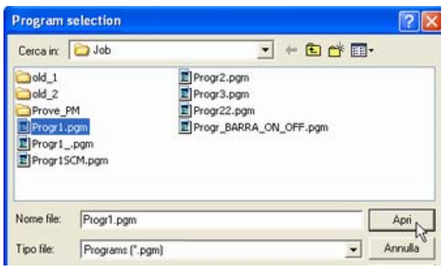
Click on the **ADD PGM** key.



6

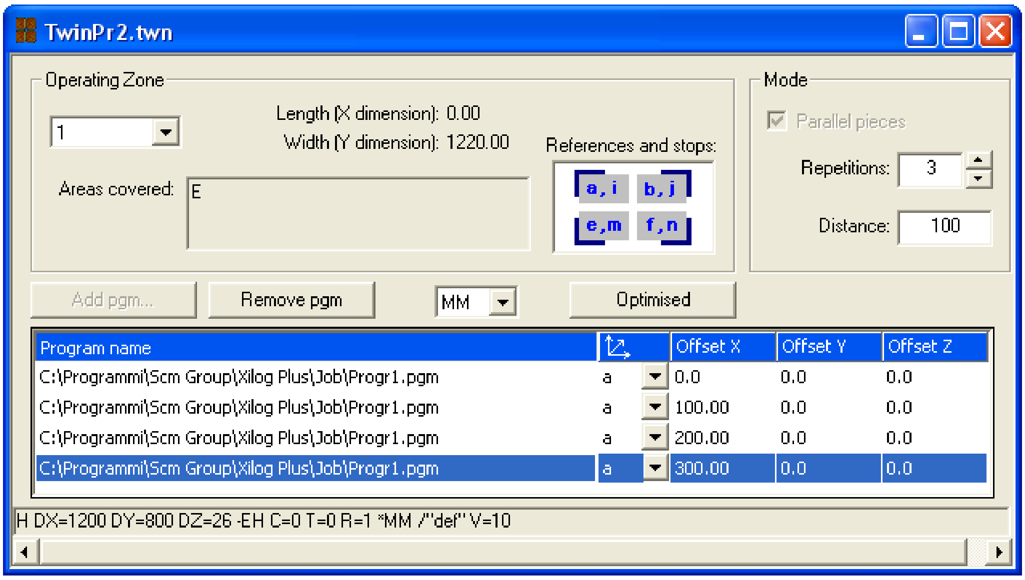
Select a program and click on the **OPEN** key to enter it.





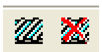
The created program will contain a compiled list with a number of programs equal to the number of preset repetitions plus one.

The additional programs are the same except for the position programmed in the **OFFSETX** field which is zero for the first program and is increased by the value set in the **DISTANCE** field for each program after the first.



To eliminate a program from the list, select the program and click on the **REMOVE PGM** button.

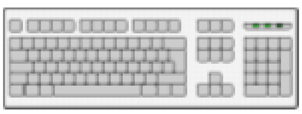
It is possible to directly create the programming file for the work table Editor supports; the file is automatically produced on the basis of the programming of the work tables for the individual PGM programs added in the multiple program.
It is also possible to modify the supports programming file several times before it is inserted, at the moment it is saved, in the PGM program.
To create or modify the programming file for the work table Editor supports, if it has been previously produced, click on the buttons on the function Bar.



► Changing the position of a program.

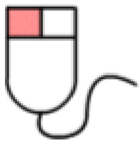
1

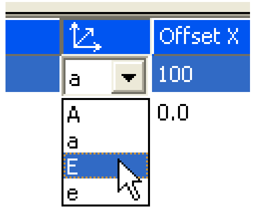
To change the position of a program, click on the **OFFSET X**, **OFFSET Y** and **OFFSET Z** boxes and enter the offset values (relative to the origin of the axes of the area in which the program is entered).



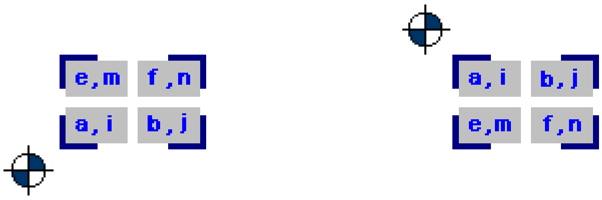
2

To position the program in an area in the selected zone other than the default area, click on the drop-down menu and select the area.





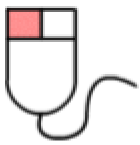
The lower case letters in the list of areas (a, b, c, d...) are generic references to the position of the stops against which the workpiece rests. These references are independent of the zones of the work table and indicate four different possible origins of the axes, according to the following patterns:

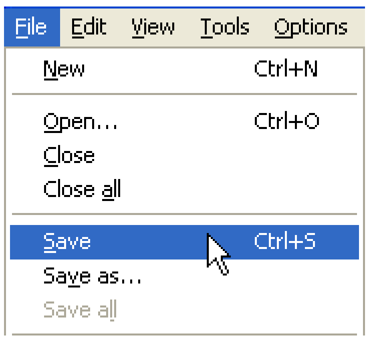


If the programs are positioned using such references, the program resulting from the multiple program may also be run in other zones (see: 4.4.3 - Examples).

► Saving the multiple program as a machining program.

Click on the menu **FILE /SAVE**.



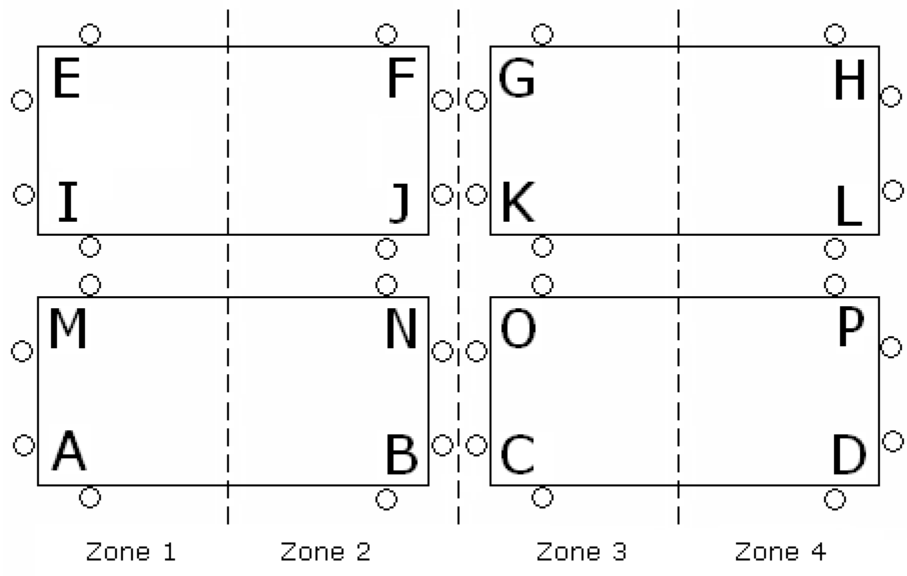


The multiple program is also automatically saved in the PGM format, by carrying out the relative conversion.
If you want to create an optimised PGM program, just click on the **OPTIMISATIONS** button before saving the program. At the right moment, you will be asked for the type of optimisation.

If there is another program open in the graphical editor with tooling different to that of the multiple program, the multiple program cannot be saved as a machining program, since only one set of tooling can be loaded in the memory for interpretation of the programs.

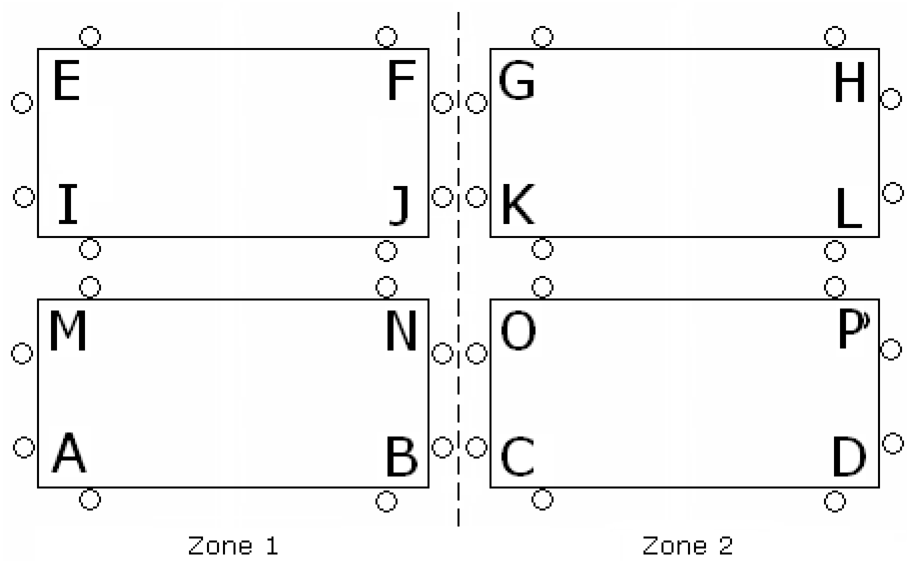
4.4.2 Dividing the Table into Operating Zones

Machines with central zero



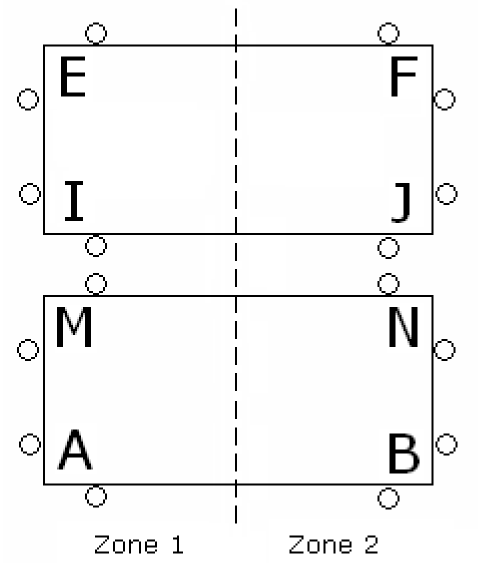
Zone 1	Areas A/E/I/M
Zone 2	Areas B/F/J/N
Zone 3	Areas C/G/K/O
Zone 4	Areas D/H/L/P
Zones 1 e 2	Areas AB/BA/EF/FE/IJ/JI/MN/NM
Zones 3 e 4	Areas CD/DC/GH/HG/KL/LK/OP/PO
Zones 1 - 4	Areas AD/DA/EH/HE/IL/LI/MP/PM

Machines with virtual central zero



Zone 1	Areas AB/BA/EF/FE/IJ/JI/MN/NM
Zone 2	Areas CD/DC/GH/HG/KL/LK/OP/PO
Zones 1 - 2	Areas AD/DA/EH/HE/IL/LI/MP/PM

Machines without central zero



Zone 1	Areas A/E/I/M
Zone 2	Areas B/F/J/N
Zones 1 - 2	Areas AB/BA/EF/FE/IJ/JI/MN/NM

Crossbeams and suction cups table

Programs entered in the same zone and sharing the same crosspieces must be the same.
For each zone the total number of crosspieces cannot be more than those physically available. The number of supports of a given type must not exceed the maximum number specified in the configuration file libsupp.cfg.

4.4.3 Examples

The examples refer to a machine with a virtual central zero.
Program Progr1.pgm is defined in area A with dimensions DX = 700 and DY = 500; program Progr2.pgm is defined in area B with dimensions DX = 500 and DY = 500. The multiple program is programmed in zone 1.

Example 1

The following programs are entered in the multiple program and are positioned with reference to the possible origins of the axes (a, e, b, f):

Progr1 -a Offset X = 0 Offset Y = 0 Offset Z = 0
Progr1 -e Offset X = 0 Offset Y = 1000 Offset Z = 0
Progr2 -b Offset X = 1500 Offset Y = 0 Offset Z = 0
Progr2 -f Offset X = 1500 Offset Y = 1000 Offset Z = 0

The result is the following program in .pgm format:

```
H DX=1500 DY=1000 -AB
SO /"PROGR1" BX=0 BY=0 BZ=0 FLD=a
```

SO /"PROGR1" BX=0 BY=1000 BZ=0 FLD=e
SO /"PROGR2" BX=1500 BY=0 BZ=0 FLD=b
SO /"PROGR2" BX=1500 BY=1000 BZ=0 FLD=f

If run in area AB or BA (or EF or FE), the program will run the sub-programs, according to the order of the SO instructions, in areas: A, E, B and F.
If run in area CD or DC (or GH or HG) the program will run the sub-programs in areas: C, G, D, H.

Example 2

The following programs are entered in the multiple program, with an indication of the areas in which they must be run (A, E, B, F):

Progr1 -A Offset X = 0 Offset Y = 0 Offset Z = 0
Progr1 -E Offset X = 0 Offset Y = 0 Offset Z = 0
Progr2 -B Offset X = 0 Offset Y = 0 Offset Z = 0
Progr2 -F Offset X = 0 Offset Y = 0 Offset Z = 0

The result is the following program in .pgm format:

H DX=1500 DY=1000 -AB
SO /"PROGR1" BX=0 BY=0 BZ=0 FLD=A
SO /"PROGR1" BX=0 BY=0 BZ=0 FLD=E
SO /"PROGR2" BX=0 BY=0 BZ=0 FLD=B
SO /"PROGR2" BX=0 BY=0 BZ=0 FLD=F

The programs entered in the multiple program list are the same as those in example 1 but, in this case, the resulting .pgm program cannot be run in areas CD and DC.

Example 3 (Pieces in parallel mode selected)

Program Progr1 is inserted into the multiple program with three repetitions and distance equal to 100:
Progr1 -AB Offset X = 0 Offset Y = 0 Offset Z = 0
Progr1 -AB Offset X = 100 Offset Y = 0 Offset Z = 0
Progr1 -AB Offset X = 200 Offset Y = 0 Offset Z = 0
Progr1 -AB Offset X = 300 Offset Y = 0 Offset Z = 0

The result is the following program, in .pgm format:

H DX=1500 DY=1000 -AB
SO /"PROGR1" BX=0 BY=0 BZ=0 FLD=AB
SET NOISO = 1
SO /"PROGR1" BX=0 BY=0 BZ=0 FLD=AB
SO /"PROGR1" BX=100 BY=0 BZ=0 FLD=AB
SO /"PROGR1" BX=200 BY=0 BZ=0 FLD=AB

5. Programming Instructions

5.1 Header

H (Header)

The header describes the panel. It is the first instruction (mandatory) of a program.

Basic parameters:

DX Panel X dimension (length).
DY Panel Y dimension (width).
DZ Panel Z dimension (thickness).
/ Name of tooling-up configuration file.
- Machining area on which the program must be executed. The maximum values are: A, B, C, D, AB, BA, CD, DC, AD, DA.

NOTE. If values 0, 0, 0 are entered respectively for parameters DX, DY, DZ, the program is automatically recognized as a macro.

Extended parameters:

***** Unit of measure. The allowed values are MM (millimeters) ad IN (inches); if the field is omitted, the unit of measures specified in the machine parameters will be used.
# Name of environment variables file.
C Machining type. The permitted values are 0 for normal machining and 1 for uninterrupted machining.
V Activates / deactivates the clamping of the piece and the position control of the automatic suction cups, if present, according to the table below:

Field V	*Blocking*	*Control of automatic suction cups*	*Blocking of power-driven clamps*
Empty	Yes, using the system configured in the configuration file xilog3.cfg; if both vacuum switches and pressure switches are included the first will be activated	Yes	Automatic
0	Mechanic	No	Not used
1	Mechanic	Yes	Not used
10	Yes, using the vacuum switches	No	Manual
20	Yes, using the pressure switches	No	Manual
11	Yes, using the vacuum switches	Yes	Automatic
21	Yes, using the pressure switches	Yes	Automatic
12	Yes, using the vacuum switches	Yes	Semiautomatic
22	Yes, using the pressure switches	Yes	Semiautomatic

30	Yes, using pressure and vacuum switches	No	Semiautomatic
31	Yes, using pressure and vacuum switches	Yes	Automatic
32	Yes, using pressure and vacuum switches	Yes	Semiautomatic
40	Equipment	No	Semiautomatic
50	Default	Yes	Automatic
51	Front Duomatic	No	Semiautomatic
52	Rear Duomatic	No	Semiautomatic
53	Duomatic front + rear	No	Semiautomatic
60	Yes, by using horizontal clamps	No	Manual
61	Yes, by using horizontal clamps	Yes	Automatic
62	Yes, by using horizontal clamps	Yes	Semiautomatic
100 – 153 (for Ergon)	As in cases 0 - 53 with combination 1 of vacuum subareas	As in cases 0 - 53 Subarea: 1	As in cases 0 - 53 Subarea: 1
200 – 253 (for Ergon)	As in cases 0 - 53 with combination 2 of vacuum subareas	As in cases 0 - 53 Subarea: 2	As in cases 0 - 53 Subarea: 2
300 – 353 (for Ergon)	As in cases 0 - 53 with combination 3 of vacuum subareas	As in cases 0 - 53 Subarea: 3	As in cases 0 - 53 Subarea: 3
400 – 453 (for Ergon)	As in cases 0 - 53 with combination 4 of vacuum subareas	As in cases 0 - 53 Subarea: 12	As in cases 0 - 53 Subarea: 12
500 – 553 (for Ergon)	As in cases 0 - 53 with combination 5 of vacuum subareas	As in cases 0 - 53 Subarea: 13	As in cases 0 - 53 Subarea: 13
600 – 653 (for Ergon)	As in cases 0 - 53 with combination 6 of vacuum subareas	As in cases 0 - 53 Subarea: 23	As in cases 0 - 53 Subarea: 23

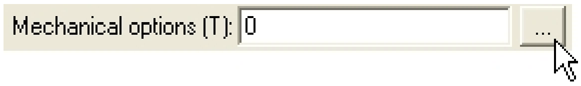
T Enables/disables lifters (if present) and laser lights for positions the piece (if present) according to the following table:

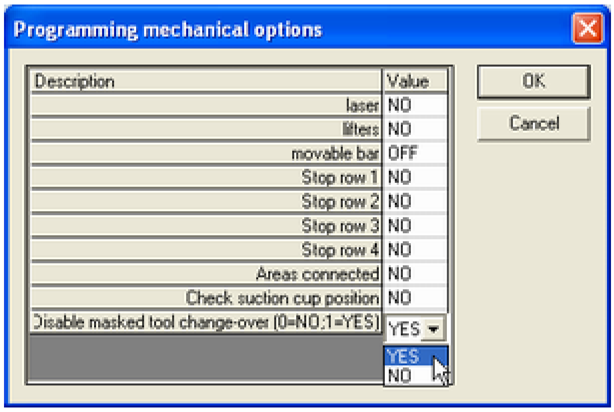
T field	*Laser lights*	*Lifters*	*TV Bar*
0	No	No	OFF
1	No	Yes	OFF
10	Yes	No	OFF
11	Yes	Yes	OFF
100	No	No	ON
101	No	Yes	ON
110	Yes	No	ON
111	Yes	Yes	ON

R Number of identical panels to produce (max. 9999).
BX Distance in X between the panel zero and the field zero.

BY Distance in Y between the panel zero and the field zero.
BZ Dimension in Z of a shim positioned under the panel.

Setting the parameters V and T is facilitated by special schedules.





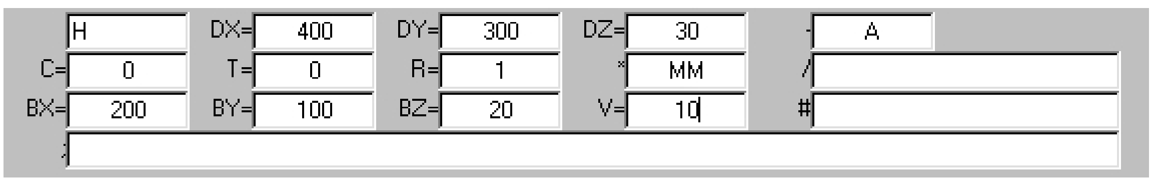
The table for the T parameter is also used to enable/disable:

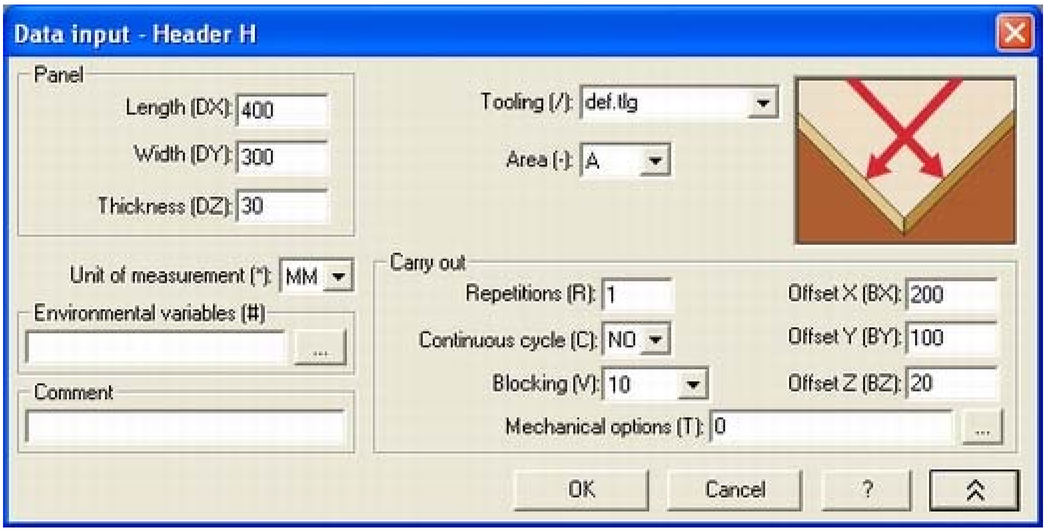
- all of the rows of stops (the main one plus any secondary stops) relative to the area specified in the program;
- the combination of the areas;
- the check of the suction cup position for the crosswise tables ad the motorised suction cups;
- the masked tool change.
- automatic track feeling for UNIX machine
- program optimisation (see paragraph 5.1)
- PAV optimisation (see paragraph 8.7.4)
- clamp optimisation for UNIX machine

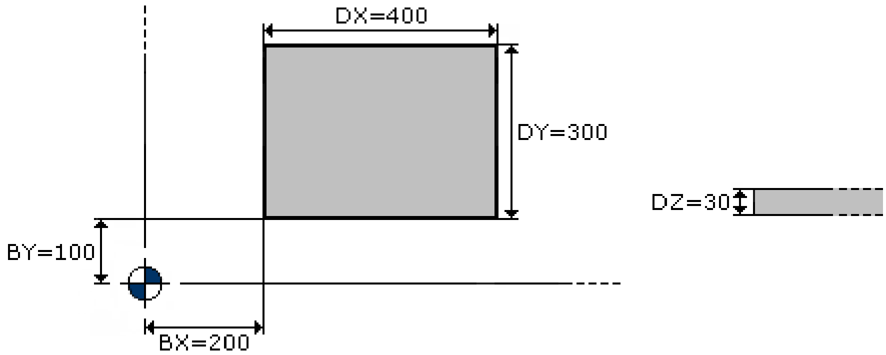
In the text editor, the schedules may be opened using two buttons on the function bar.

	Open the table to set parameter T.
	Open the table to set parameter V.

Example:







WARNING

The panel dimension in Z is used to optimize translation of head between machining on different surfaces. For this reason the DZ field must be compiled with the effective maximum overall dimensions of the piece in Z, otherwise the heads and the piece may collide.

5.2 Basic Instructions (Text)

5.2.1 Operating Instructions

5.2.1.1 General Functions

Automatic Entry in the Profile - GIN

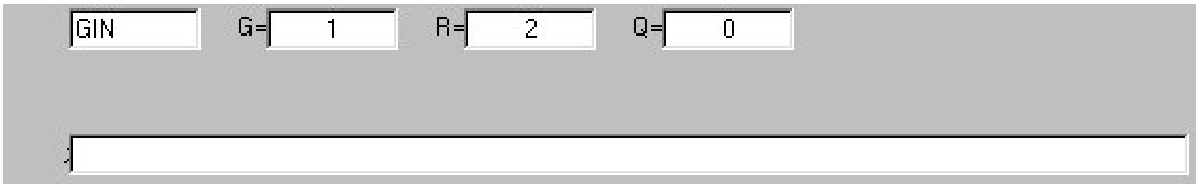
Defines a line or an arc of a circle tangent to the profile at the entry point. Works if programmed before an instruction for beginning profile (G0).

Parameters:

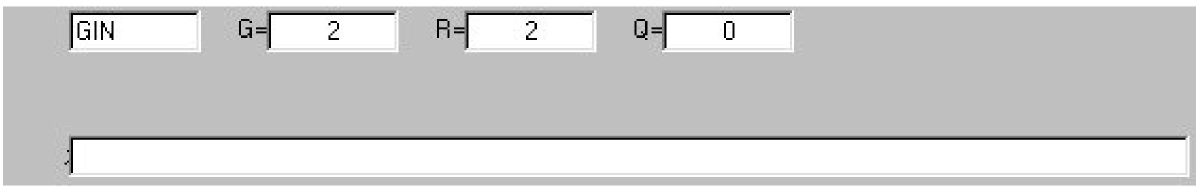
G Type of entry: 1=straight, 2=arc (the arc is valid only if the tool radius compensation is enabled). Entry with blade at a tangent line is enabled.
R Tool radius multiplication factor (default=2).
Q Type of onset: 0=in quota, 1=in descent.

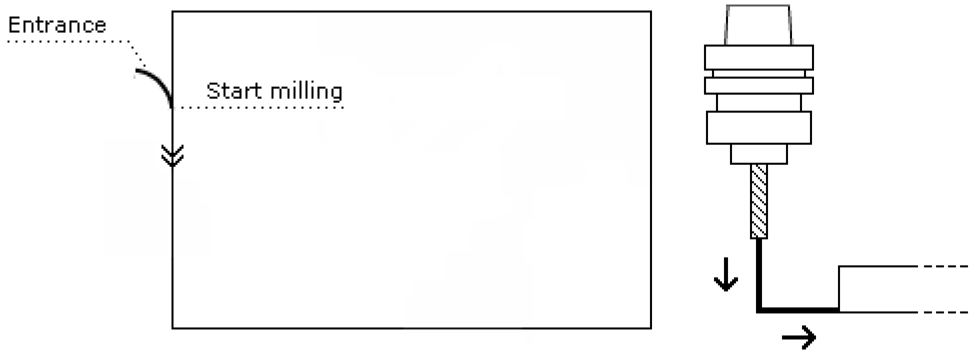
Examples: method for automatic entry.

1) Vertical linear entry.



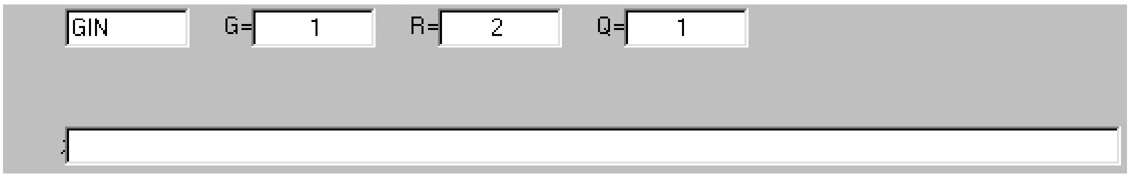
2) Vertical arc entry.

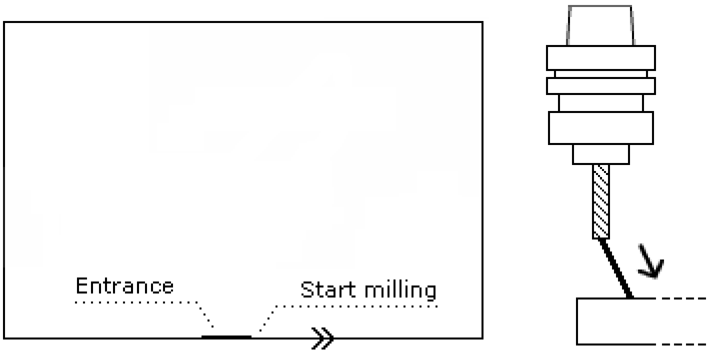




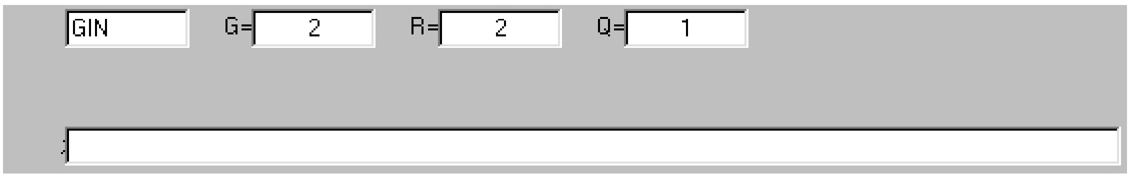
Ingresso = Entrance; Inizio Fresatura = Start Milling

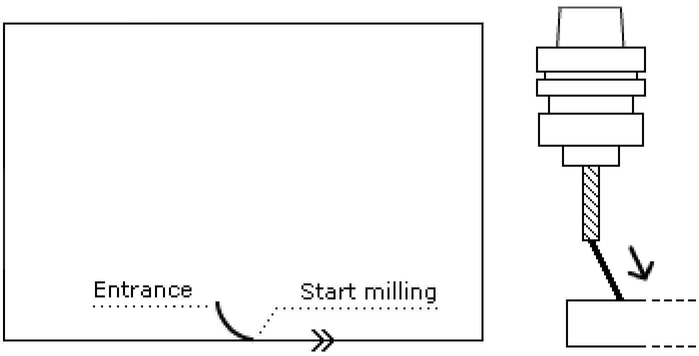
3) Slanted linear entry.





4) Slanted arc entry.





Ingresso = Entrance; Inizio Fresatura = Start Milling

WARNING

Automatic entrance is not enable if either instruction C=3, C=31 or C=32 has been set.
For C=0 and G=2 the direction for arc entry is established by the sign R
(if positive = arc clockwise; if negative = arc counter-clockwise.

Automatic Exit from the Profile - GOUT

Defines a line or an arc of a circle tangent to the profile at the exit point. It has effect if programmed after the last instruction of the profile.

Parameters:

G Type of exit: 1=straight, 2=arc (the arc is valid only if the tool radius compensation is enabled). Entry with blade in tangent line (G=1) is enabled.

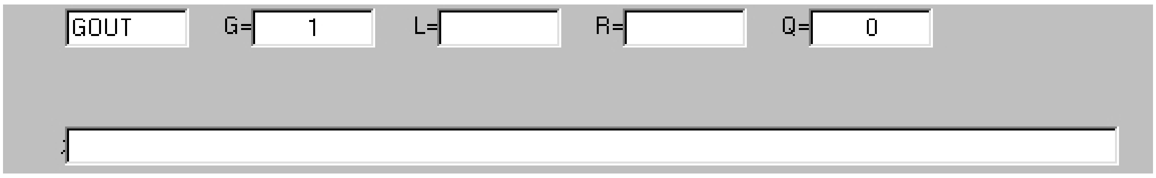
L Overlapping on profile. If the value of L is set to -1, overlapping operation occurs all along the last profile section; if L is set to -2, overlapping occurs al along the last profile section and also the panel exits perpendicularly (as per instruction N) ignoring the value of parameter Q.

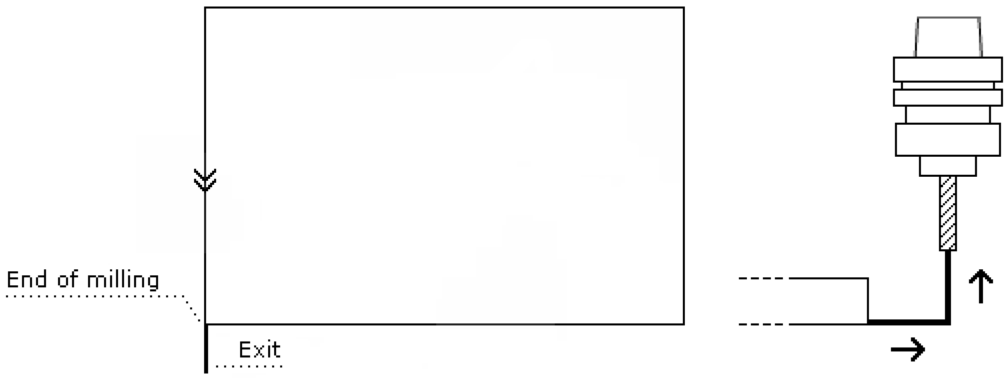
R Tool radius multiplication factor (default=2).

Q Type of expulsion: 0=in quota, 1= upwards.

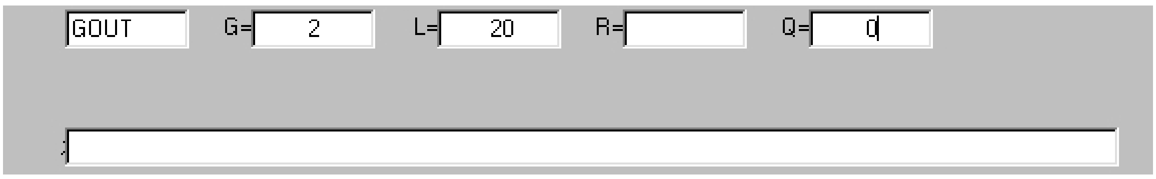
Examples: methods for performing automatic exit.

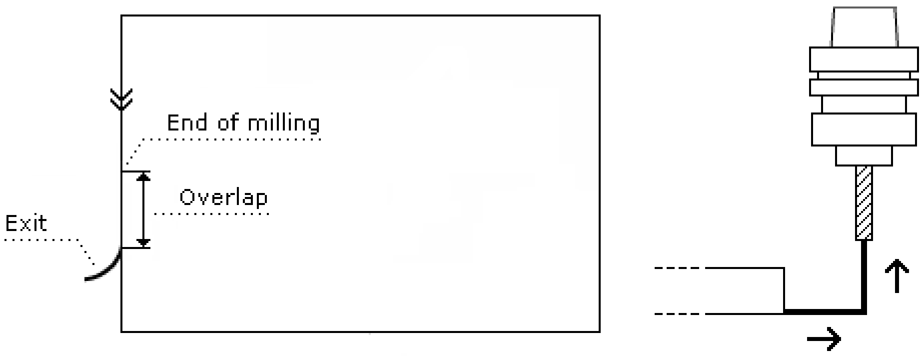
1) Vertical linear exit.



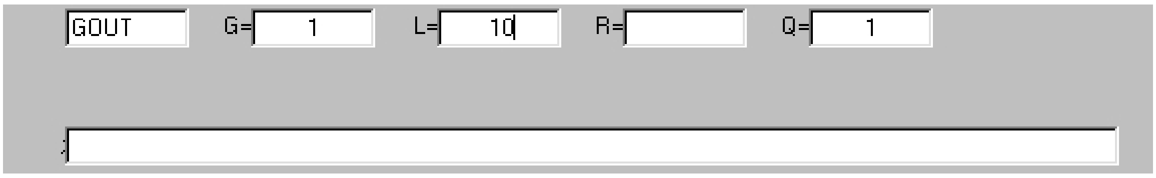


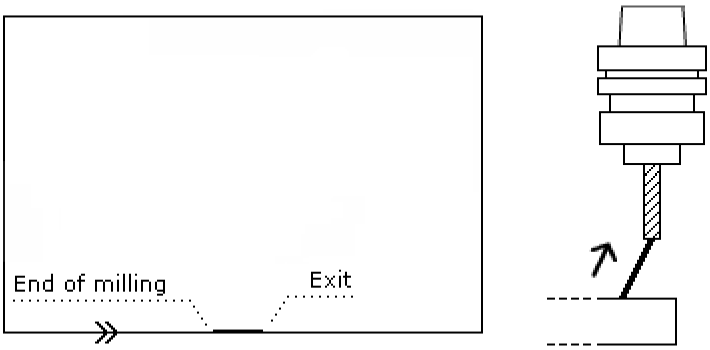
2) Vertical arc exit.



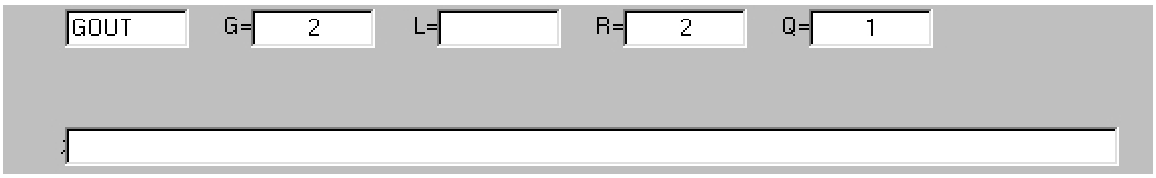


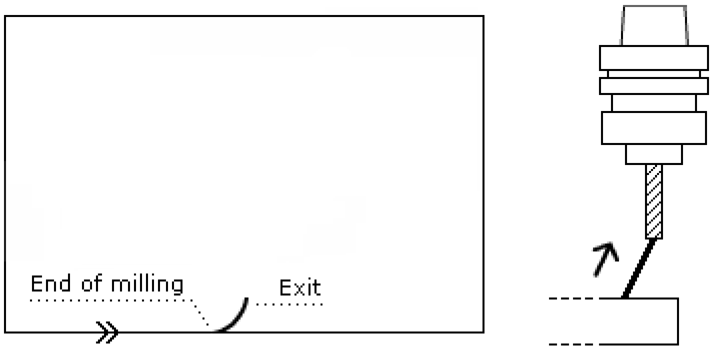
3) Slanted linear exit.





4) Slanted arc exit.





WARNING

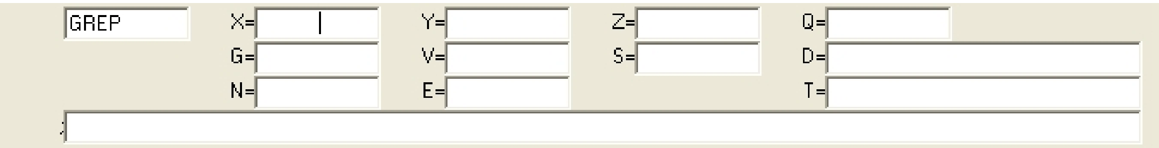
Automatic exit is not enable if either instruction C=3, C=31 or C=32 has been set.

Profile Repetition - GREP

Repeats a profile. ***The GREP instruction is not influenced by the SX and SY instructions.***

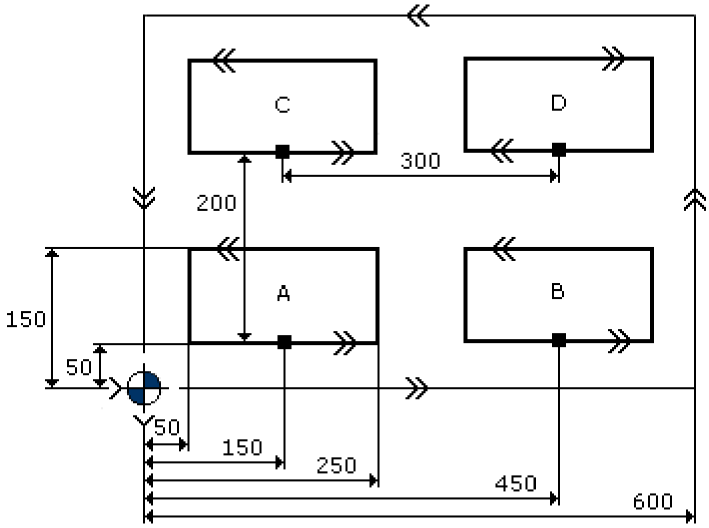
Parameters:

X Coordinate X of the beginning point of the profile, or offset X of the profile (see Q).
Y Coordinate Y of the beginning point of the profile, or offset X of the profile (see Q).
Z Offset of profile in depth Z.
Q 0 = X, Y absolute values; 1 = offset.
G Reverse direction (0=NO,1=YES).
V Milling speed.
S Rotation speed of tool.
D Out machining quota.
N Name of profile (see field N of instruction G0).
E Position of vacuum hood (see: Appendix D).
T Tool.

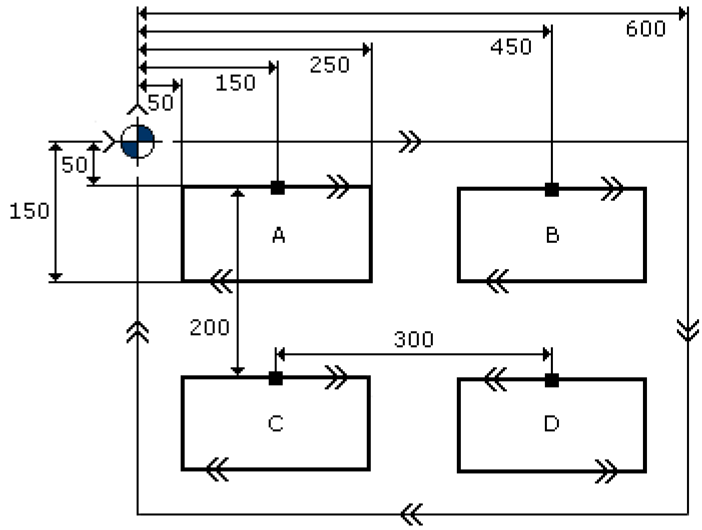


The following example shows the trimming of the panel in the figure that has a thickness of 30 mm with a tool called E1, the internal profiling of rectangle A with a tool called E2, the repeat of profile A in position B with absolute coordinates, the repeat of profile A in position C with offset, the repeat of profile A in position D with offset and a change in direction with a tool called E3.

► *Front machine origin*



►*Rear machine origin*



H DX=600 DY=400 DZ=30 -A C=0 T=0 R=1 *MM /"ANDREA" V10 ; header

;panel external trimming

GIN Q=0 R=2 G=1 C=... (front machine origin: C=1; rear machine origin: C=2); automatic entry into the profile with straight line in quota
G0 X=0 Y=0 Z=… T=101 V=4000 S=18000 E=1 D=20 ; begin trimming profile
G1 X=600 Y=0
G1 X=600 Y=400
G1 X=0 Y=400
G1 X=0 Y=0
GOUT Q=0 R=2 G=1 C=0 ; automatic exit from profile with straight line upwards

; start machining rectangle A

GIN Q=1 R=2 G=1 F=1 C=0; entry with straight line downwards no radius compensation G0 X=150 Y=50 Z=… E=1 V=5 S=18000 D=20 N=”TEST” T=101 start name profile “test”
G1 X=250 Y=50
G1 X=250 Y=150
G1 X=50 Y=150
G1 X=50 Y=50
G1 X=150 Y=50
GOUT Q=1 R=2 G=1 L=10; exit from profile with slanted line

; repeat of rectangle A called “test” in position B with absolute quotas

GREP X=450 Y=50 Q=0 G=0 N=”TEST”; same tool and same work quota

; repeat of rectangle A called “test” in position C with offset quotas

GREP Y=200 Q=1 G=0 N=”TEST”; quotes referred to starting point of first rectangle

; repeat of rectangle A called “test” in position D with offset quotas

GREP X=300 Y=200 Q=1 G=1 N=”TEST” T=103 ; tool E3 change of direction

N X=0 Y=0 T=101 F=1 ;stop spindle at machine zero and load tool E1

Speed Change - GSET

The instruction GSET describes some new characteristics of a profile retrieved next GREP.

Parameters:

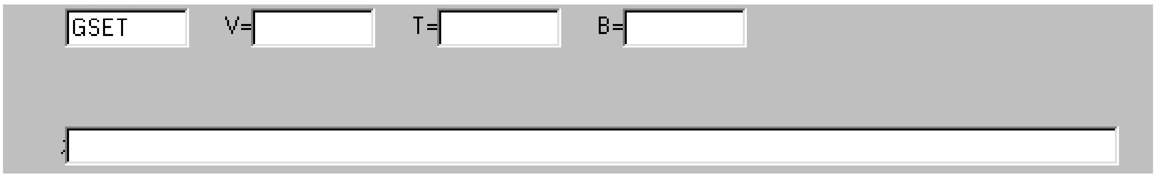
V Movement speed.

T Tools.

B Type of profile. The type of profile B associates the characteristics of GSET to the tool or process active at the moment of interpreting the geometry: 1=milling path, 2=planing path.

All instruction parameters are optional.

GSET instructions must be input before any G movement instructions. G movement instructions influenced by a GSET assume new characteristics during a repetition of the profile with GREP.
GSET instructions are only enabled if the specified tool is equal to that specified in GREP or if the identifier of the type of profile is represented by the active tool during the repetition of the profile.



Examples:

1. GSET has no effect when the specified tool in GREP is not 101

```
….
G0      X..Y.. Z.. V=1 S E T=101 N=”Prof”
G1      X.. Y.. Z.. V=5
GSET V=10 T=101
G1      X.. Y.. Z.. V=6
….
GREP V=2 T=102 N=”Prof”
….
```

2. GSET is significant when the specified tool in GREP coincides with that of GSET

```
….
G0      X.. Y.. Z.. V=1 S E T=101 N=”Prof”
G1      X.. Y.. Z.. V=5
GSET V=10 T=102
G1      X.. Y.. Z.. V=6
….
GREP V=2 T=102 N=”Prof”
….
```

3. GSET only has an effect if the tool E2 is a cutter

….

```
G0      X.. Y.. Z.. V=1 S E T=101 N=”Prof”
G1      X.. Y.. Z.. V=5
GSET V=10 B=1
G1      X.. Y.. Z.. V=6
….
GREP V=2 T=102 N=”Prof”
….
```

4. GSET only has an effect if the tool E2 is a cutter (due to presence of B=1)

```
….
G0      X.. Y.. Z.. V=1 S E T=101 N=”Prof”
G1      X.. Y.. Z.. V=5
GSET V=10 T=101 B=1
G1      X.. Y.. Z.. V=6
….
GREP V=2 T=102 N=”Prof”
….
```

As the profile can be repeated more than once, more than one GSET can be specified before a G movement instructions. The repetition underway determines the GSET associated with instruction G. In case of redundant GSETs the last one will be taken into consideration.

5.

```
….
G0      X.. Y.. Z.. V=1 S E T=101 D N=”Prof”
G1      X.. Y.. Z.. V=5
GSET V=10 T=102
GSET V=15 T=103
G1      X.. Y.. Z.. V=6
….
GREP V=2 T=102 N=”Prof”;has an effect only if GSET with T=102
GREP V=2 T=103 N=”Prof”;has an effect only if GSET with T=103
….
```

6.

….

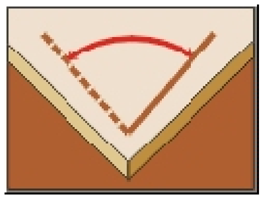
```
G0      X.. Y.. Z.. V=1 S E T=101 D N=”Prof”
G1      X.. Y.. Z.. V=5
GSET V=10 T=102
GSET V=15 T=103
GSET V=20 B=1
G1      X.. Y.. Z.. V=6
….
```

GREP V=2 T=102 N="Prof'; ;if E2 and E3 are cutters the instruction GSET
GREP V=2 T=103 N="Prof'; with B=1 invalidates the previous GSETs
….

Profile Rotation - ROT

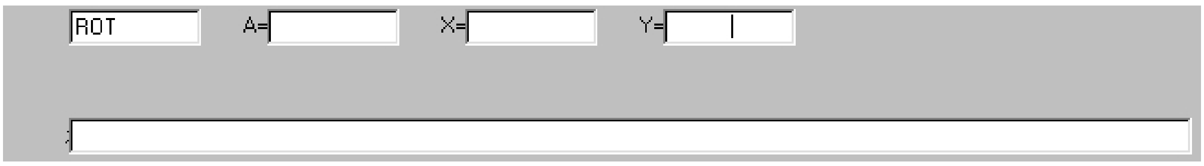
Rotated the machining of a macro. ***The function of the ROT instruction is only guaranteed for macros***.

Unit:
Main functions



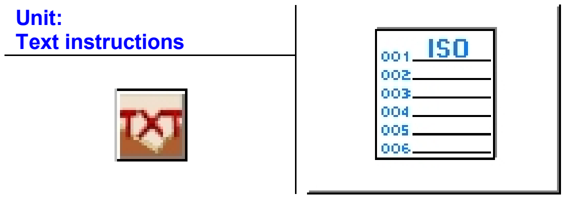
Parameters:

A Angle of rotation (front machine origin: positive in counter-clockwise direction; rear machine origin: positive in clockwise direction).
X Quota X from the center of rotation.
Y Quota Y from the center of rotation.



ISO Instruction - ISO

Used to program an instruction in the ISO language of the numerical control used. The instructions are placed between double quotation marks and may be followed by a list of parameters separated by at least one space (the structure and meaning of the parameters are described in the paragraph on the instruction PRINT). No syntax check is performed on the instruction and no graphic view is associated to it.

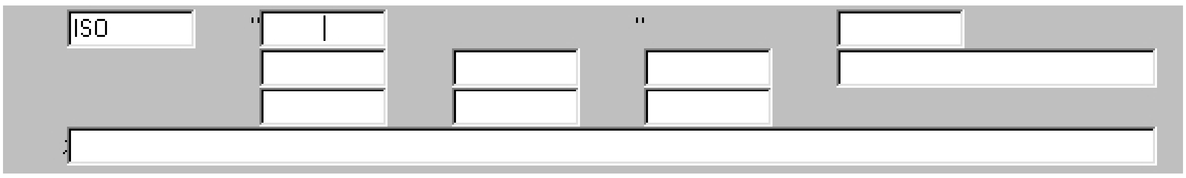


Parameters:

“<string>” ISO instructions.
(empty boxes) Optional parameters.

Note: if in a program there are MICRO POSITIONINGS, it is necessary to insert the following instruction at the tip of the program:

ISO “%B20”



Examples:

ISO “GX1000Y740F6000”
ISO “M71”
ISO “M?d” 110+ Hood

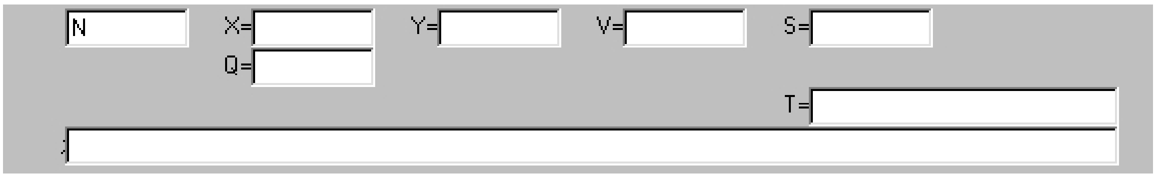
Null Operation - N

Turns the rotation off and moves the electro-spindles to their idle position.

Parameters:

X Position of head on axis X.
Y Position of head on axis Y.
V Movement speed.
S Rotation speed of tool Meaning of field S: if not programmed or the value is 0 the spindles switch off; if programmed on 1 the spindles remain on.
Q Meaning of field Q: if not programmed the value is 0 it is understood that quotas X and Y refer to machine zero; if programmed on 1 it is understood that quotas X and Y refer to panel zero.
T Number of external tools.

The tool specified in field T is removed from the tool store.



Panel Tracing - TA

Tracing is used to detect any irregularity or roughness in the panels and therefore enables them to be corrected

Parameters:

X Coordinate X of the point to feel referring to the current side (from 1 to 5).
Y Coordinate Y of the point to feel referring to the current side (from 1 to 5).
Q Defines the type of feeling.

(default)	The head moves up to the Z + limit switch.
0	The head moves up to the Z + limit switch.
1	The head remains at the feeling Z dimension.
2	The head moves up to the dimension for brushing over the workpiece.

G Controls the head return upstroke after feeling.

(default)	Feeling the panel faces.
0	Feeling the panel faces.
1	Feeling a point on the panel.

T Tool.

After detecting the difference (D) between the real panel dimensions and the theoretical dimensions, the feeler changes the numerical control in relation to the side on which it was carried out:

Face 1: D is algebraically added to dimension Z of panel (DZ)
Face 2: is algebraically added to dimension X of panel (DX)
Face 3: is algebraically added to offset X of panel (BX)
Face 4: if the face is opposite the stop, is algebraically added to dimension Y of panel (DY); otherwise, D is algebraically added to offset Y of panel (BY)
Face 5: same rule as Face 4

By touching the panel once or a number of times you can generally determine variations in its dimensions from those (theoretical) specified in instructions H. The new dimensions (real) are memorized in the variables @DX, @DY and @DZ which can be used in the parametric; the real dimensions are automatically used when the machining operations are programmed with specular instructions SX and SY; more than one face can be touched before machining. You can use the preset variables @BX, @BY, @BZ to read the workpiece movement relative to the stops.
For feeling a point on the panel (G = 1), the co-ordinates of the point felt are saved in the variables @X, @Y and @Z.

Examples:

H DX1000 DY800 DZ20 –A /DEF
F 2
TA X10 Y400 T91

```
F 3
TA X10 Y500 T91
F 1
B X100 Y400 Z8 T2
B X = @DX -50 Y500 Z8 T2
SX 1
B X200 T1
```

5.2.1.2 Boring

Boring - B

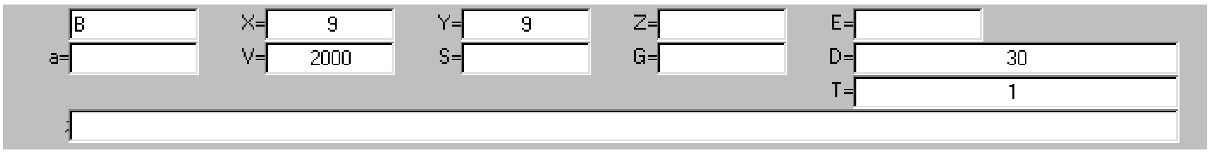
Executes one or more borings.

Parameters:

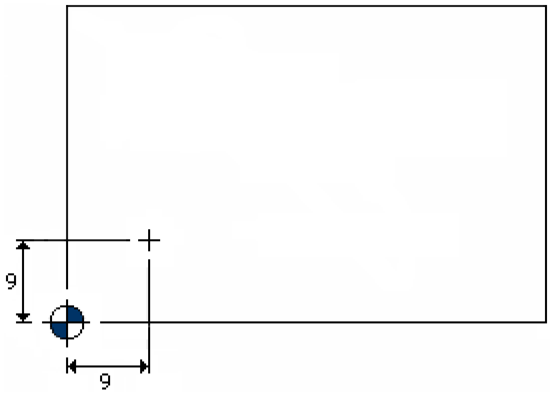
X Coordinate X of first hole.
Y Coordinate Y of first hole.
Z Hole depth.
E Position of vacuum hood (see: Appendix D).
a Offset boring head floating in Y (axis E, only if present)
V Boring.
S Rotation speed of tool
G Number of chip discharge steps
D Out machining quota.
T List of tools; in case of various tools the coordinates X, Y refer to the first tool indicated.

Examples:

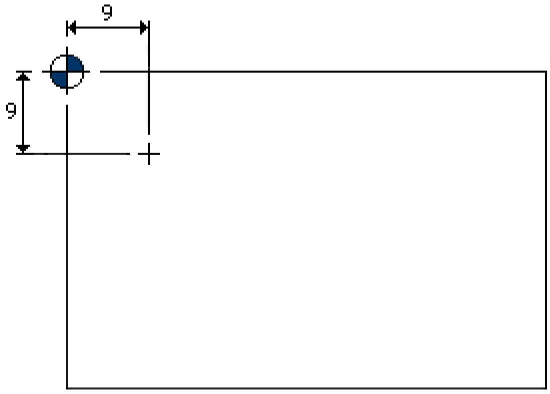
1) Programming a hole.
Parameters X, Y, Z, T are mandatory to produce a hole. The possibility of making a hole with shavings discharge is activated with parameter G = number of passages (divided in equal parts; for example G = 3).



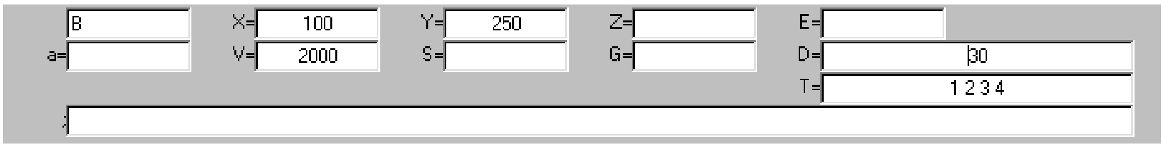
►*Front machine origin*



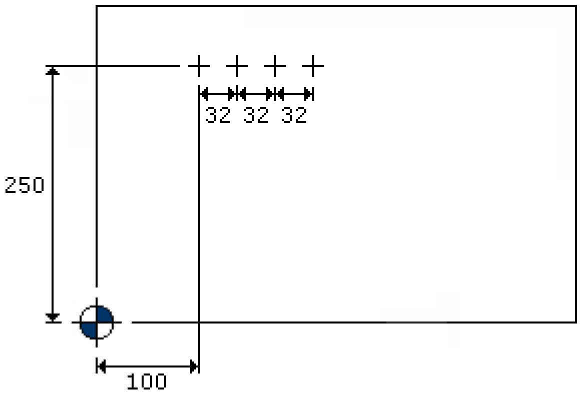
►*Rear machine origin*



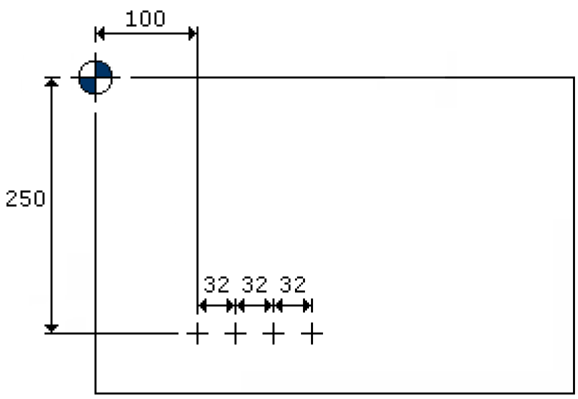
2) Programming a series of holes with the boring machine at step 32 using various spindles. Parameters X, Y, Z, T are mandatory. The first hole is programmed; the other holes, which are at a distance of 32 mm, are programmed by selecting various spindles with parameter T (there must be a space between one number and the other).



►*Front machine origin*



►*Rear machine origin*



Optimized Boring - BO

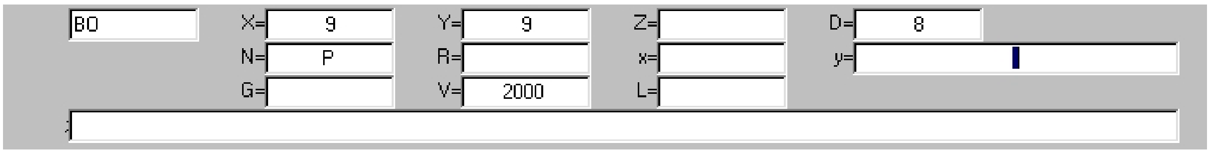
This executes one or more holes using the boring optimization algorithm. All BO instructions programmed in succession are placed in the same optimization session; a program may contain more than one block of BO instructions.

Parameters:

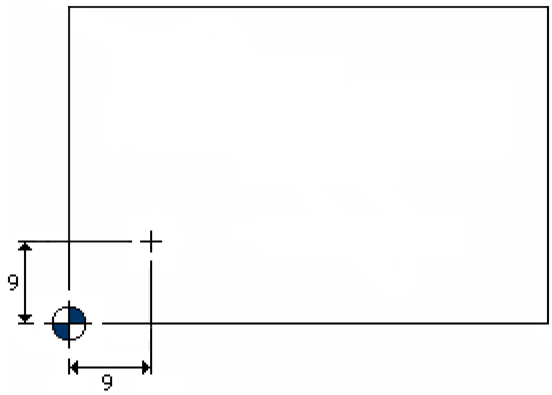
X Coordinate X for the first hole.
Y Coordinate Y for the first hole.
Z Depth of holes.
D Diameter of holes.
N Hole type (P=flat, L=lance, S=taper).
R Number of holes (including the original hole).
x X step for repetitions.
y Y step for repetitions.
G Number of chip discharge steps.
V Boring speed.
L Taper height (only if N=S).

Examples:

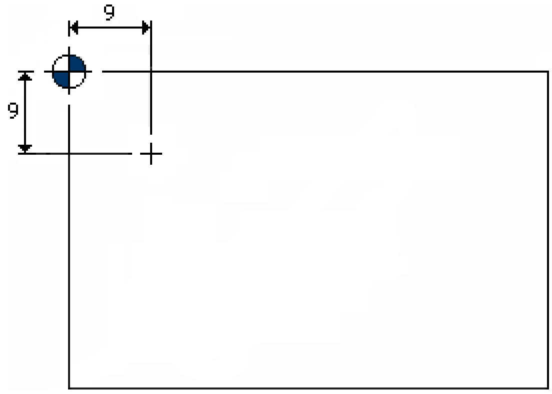
1) Programming a hole.
Parameters X, Y, Z, D, N are mandatory for making a hole.



►*Front machine origin*

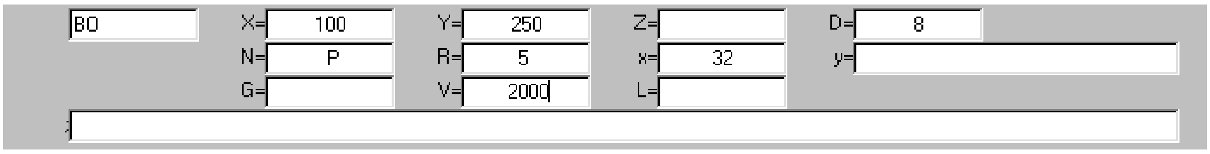


►*Rear machine origin*

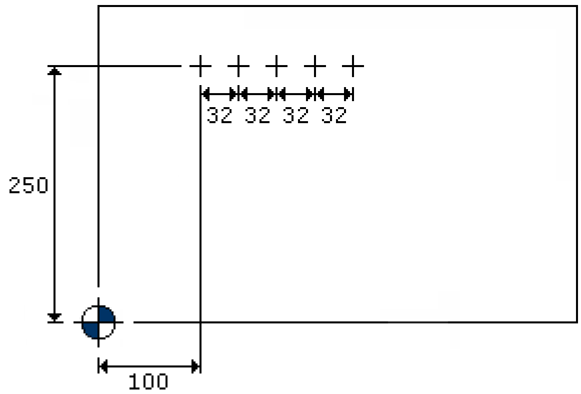


2) Programming a series of holes.

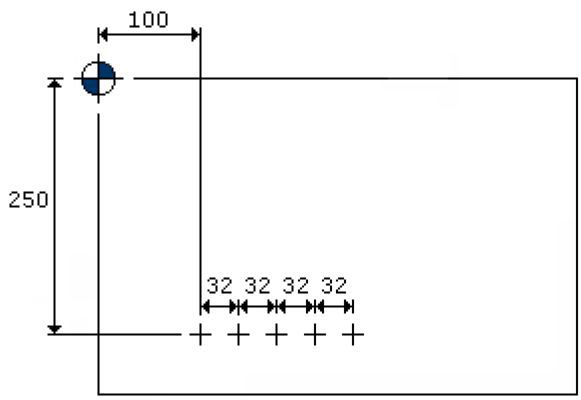
To make a series of holes, parameters X, Y, Z, R, x, y, D, N are mandatory. The first hole is to be programmed and subsequent holes are programmed with R = number of repeats (including the original hole), x = distance between the holes (can also be different from 32 mm).



► *Front machine origin*



►*Rear machine origin*



Slanted Boring - BR

Allows you to make one or more borings at an angle other than 90° with respect to the working table. BR is always referred to side 1. The current work side must always be side 1.

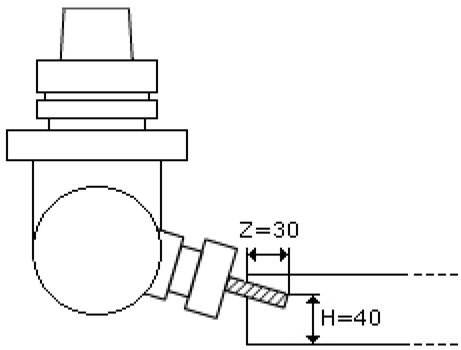
Parameters:

X Coordinate X of the first hole.
Y Coordinate Y of the first hole.
Z Depth of holes.
H Height of boring from the working table (position of the tool from the working table of the machine). Defaults to DZ.
A Rotation angle of bore (front machine origin: positive in counter-clockwise direction; rear machine origin: positive in clockwise direction).
Q If the value is 0, angle A is summed algebraically to tool's offset R. If the value is 1, angle A substitutes the tool's offset R.
E Position of vacuum hood (see: Appendix D).
V Boring speed.
S Rotation speed of tool.
G Number of chip discharge steps.
D Out machining quota.
B Tool angle relative to the vertical plane (only for the Prisma group).
T List of tools; in case of various tools the coordinates X, Y refer to the first tool .

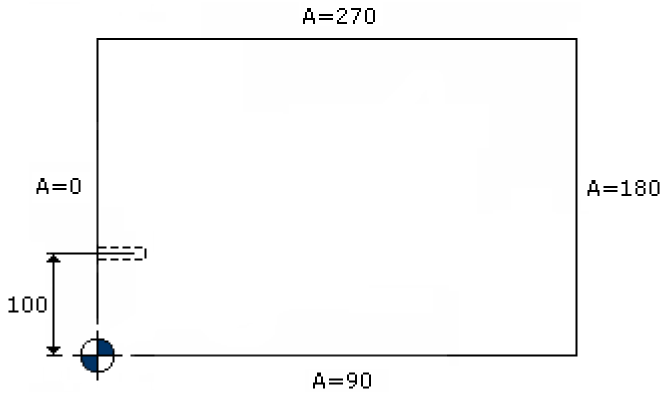
Example:

Programming a hole with parameters Q=1.

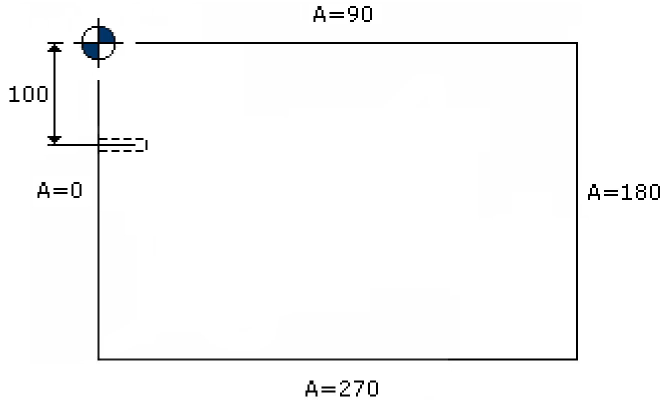
	BR	X=	0	Y=	100	Z=	30	H=	40
A=	0	Q=	1	E=	1	V=	2000	S=	6000
		G=		D=	50	B=		T=	105



►*Front machine origin*



►*Rear machine origin*



WARNING

Incorrect programming of machining with slanted tools may cause damages to the structure of the machine. It is advisable that a SIMULATION of this type of operation be executed first.

5.2.1.3 Routing

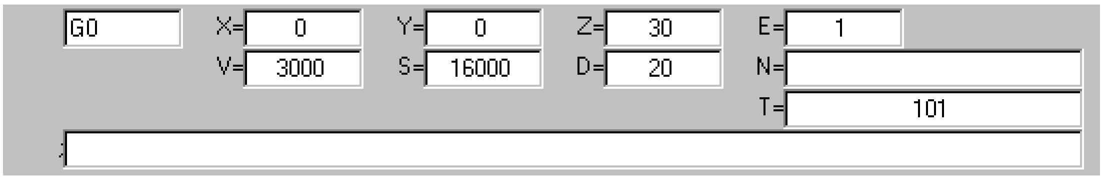
Cutting Start - G0

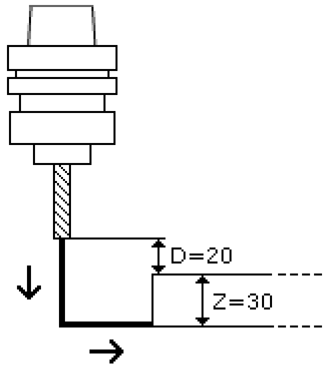
Defines the beginning point of a profile.

Parameters:

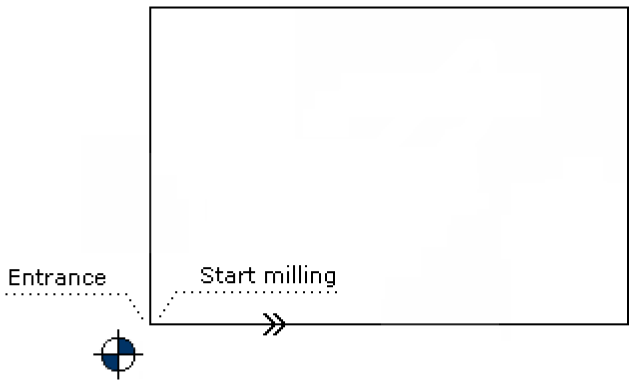
X Coordinate X for beginning of profile.
Y Coordinate Y for beginning of profile.
Z Depth of beginning of profile.
E Position of vacuum hood (see: Appendix D).
V Piece entry speed.
S Rotation speed of tool
D Outside work quota.
N Profile name (see GREP).
T Tool.

Example:

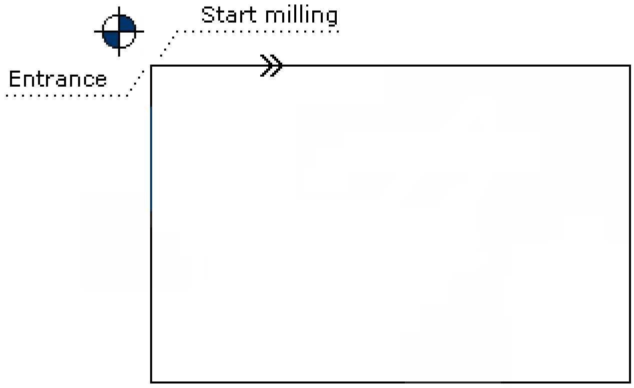




►*Front machine origin*



► *Rear machine origin*

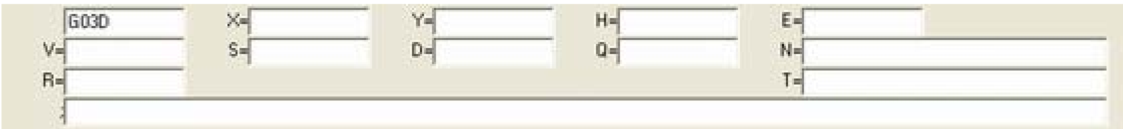


3D Begin Milling - G03D (Prisma Head)

Use these instructions to start linear routing on a plane set at an angle to the orthogonality of the panel surfaces; it controls the tool lead in/lead out into/from the workpiece with a null machining depth. It does not allow GIN/GOUT; with G03D you cannot switch on the hood. You can only use this instruction with the Prisma head, only with face 1 active and only with radius correction Off. See: Appendix H – Prisma Head.

Parameters:

X Start profile X co-ordinate.
Y Start profile Y co-ordinate.
H Height of machining above work table (dimension to which tool must descend relative to machine work table; if this is not set the value is DZ).
E Position of vacuum hood (see: Appendix D).
V Routing speed.
S Tool speed of rotation.
D Out of machining dimension (only with Q and R unprogrammed).
Q Angle of rotation about the Z-axis.
N Profile name (see GREP).
R Angle of rotation about the X-axis.
T Tool.



Linear Cutting - G1

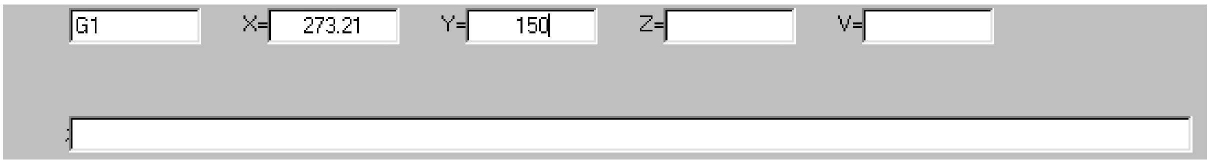
Defines a line segment.

Parameters:

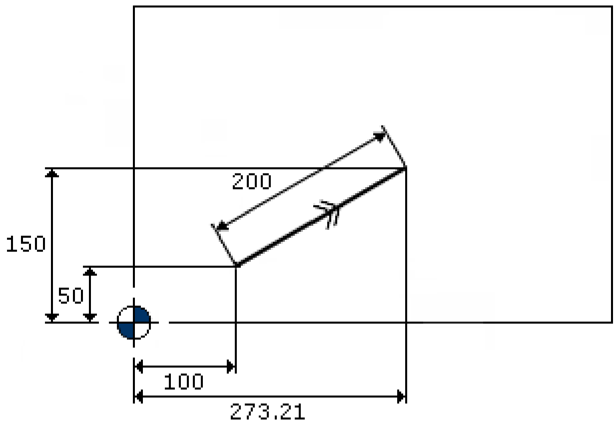
X Coordinate X for end of segment.
Y Coordinate Y for end of segment.
Z Depth of end of segment
V Milling speed.

Example:

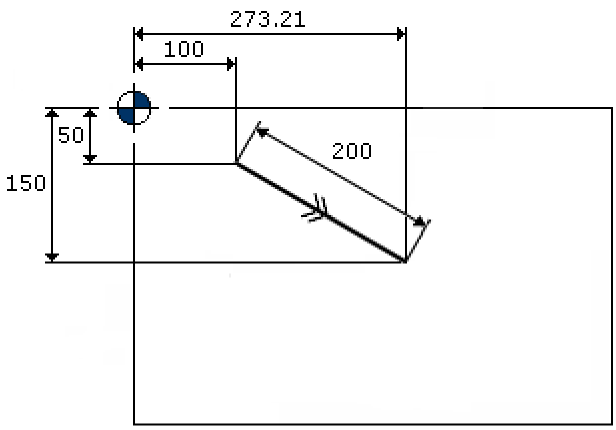
The starting point (X=100, Y=50) will be taken by the precious instruction.



► *Front machine origin*



► *Rear machine origin*

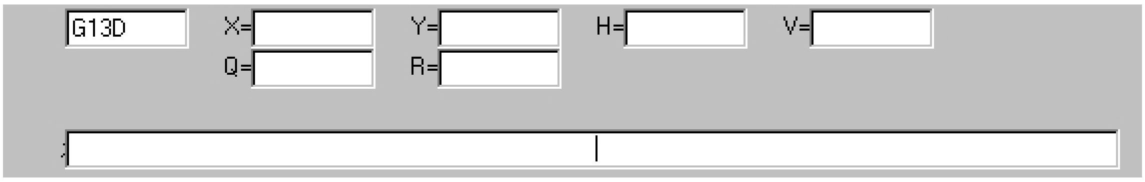


3D Linear Milling - G13D (Prisma Head)

Use these instruction to perform linear routing on a plane set at an angle to the orthogonality of the panel surfaces. You can only use this instruction with the Prisma head, only with face 1 active and only with radius correction Off. See: Appendix H – Prisma Head.

Parameters:

X End section X co-ordinate.
Y End section Y co-ordinate.
H Height of machining above work table (dimension to which tool must descend relative to machine work table; if this is not set the value is DZ).
V Routing speed.
Q Angle of rotation about the Z-axis.
R Angle of rotation about the X-axis.



Clockwise Circular Cutting - G2

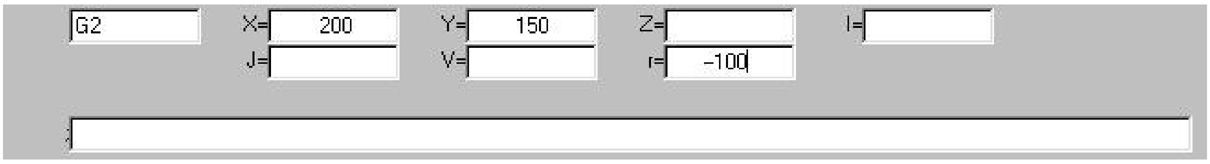
Defines the arc of circle in clockwise direction. In alternative to the definition of the center, it is possible to define the radius of the arc according to the following convention: if the radius is positive, the center falls to the left of the imaginary line going from the beginning to the end point of the arc, otherwise, the center falls to the right of this line.

Parameters:

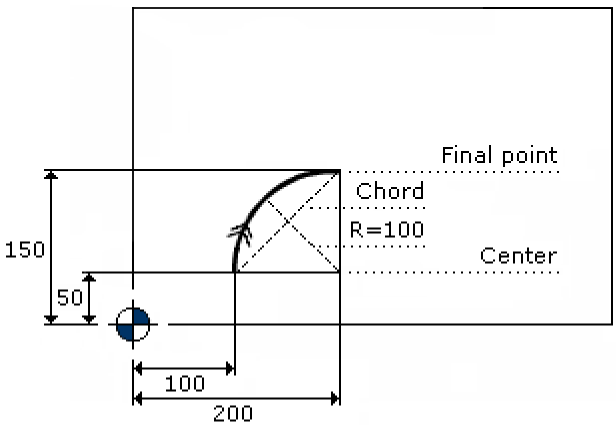
X Coordinate X for end of arc.
Y Coordinate Y for end of arc.
Z Depth of end of arc.
I Coordinate X of center of arc.
J Coordinate Y of center of arc.
V Milling speed.
r Arc radius.

Examples:

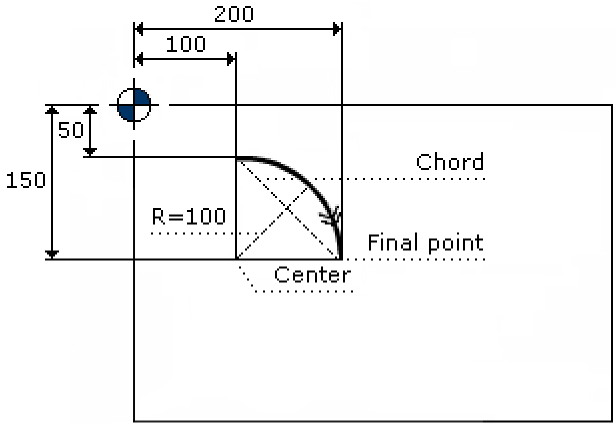
1) Arc given the radius. The starting point (x=100, Y=50) will be taken by the previous instruction.



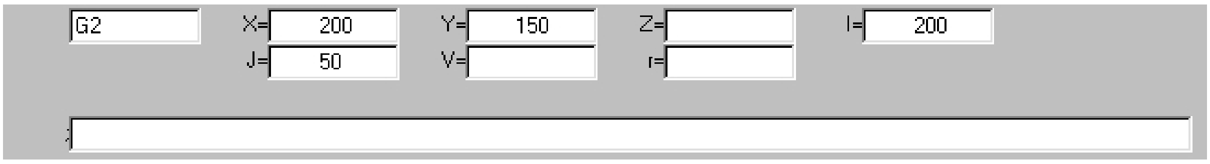
► *Front machine origin*



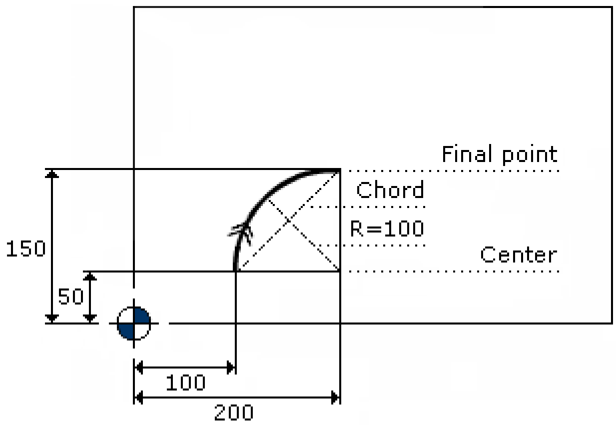
►*Rear machine origin*



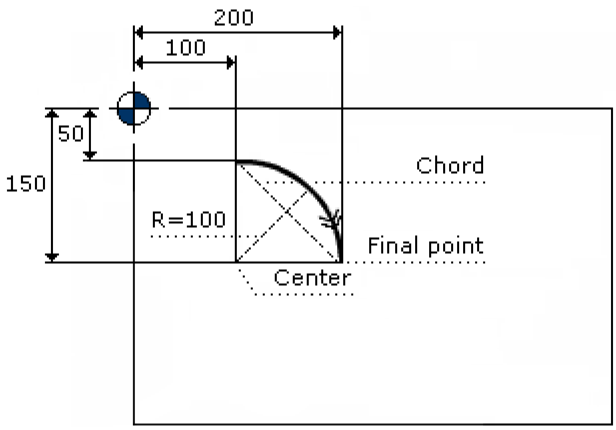
2) Arc given the center. The starting point (x=100, Y=50) will be taken by the previous instruction.



►*Front machine origin*



►*Rear machine origin*



Anticlockwise Circular Cutting - G3

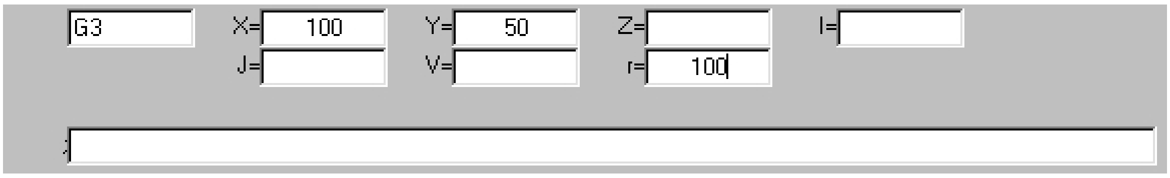
Defines an arc of a circle in counter-clockwise direction. In alternative to the definition of the center, it is possible to define the radius of the arc according to the following convention: if the radius is positive, the center falls to the left of the imaginary line going from the beginning to the end point of the arc, otherwise, the center falls to the right of this line.

Parameters:

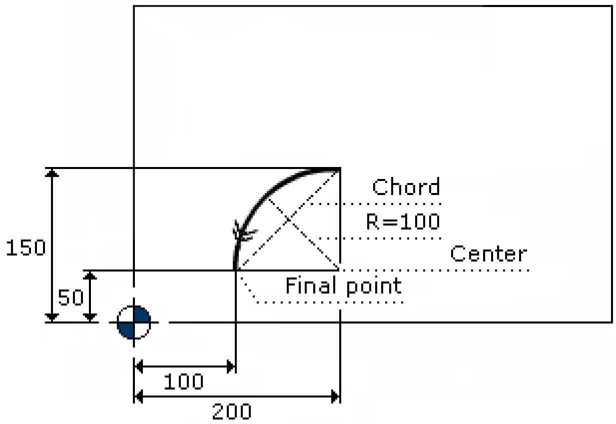
X Coordinate X of end of arc.
Y Coordinate Y of end of arc.
Z Depth of end of arc.
I Coordinate X of center of arc.
J Coordinate Y of center of arc.
V Milling speed.
r Radius of arc.

Examples:

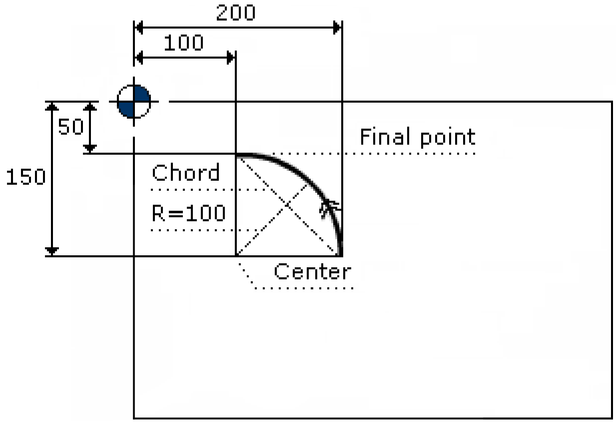
1) Arc given the radius. The starting point (x=200, Y=250) will be taken by the previous instruction.



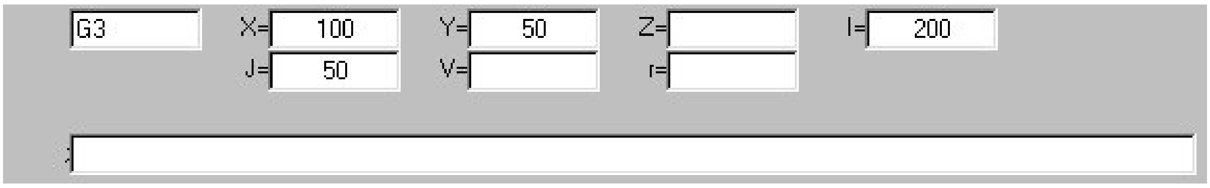
►*Front machine origin*



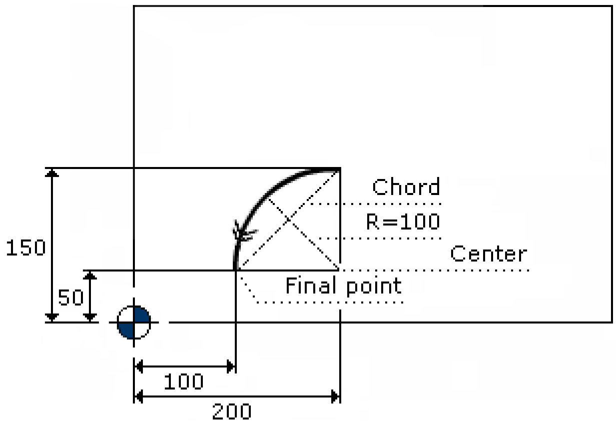
►*Rear machine origin*



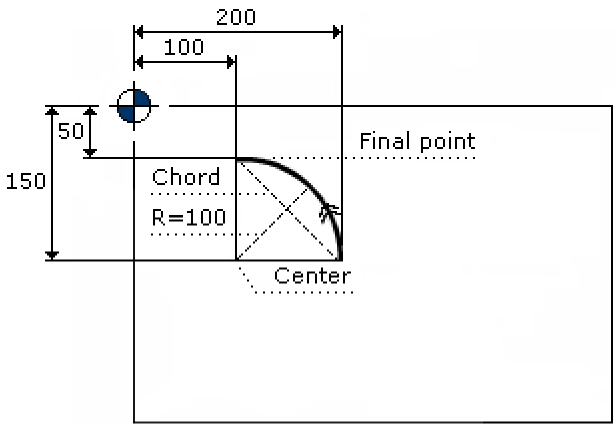
2) Arc given the center. The starting point (x= 200, Y=150) will be taken by the previous instruction.



►*Front machine origin*



►*Rear machine origin*



Section Tangent to Previous Section - G5

Defines a milling section tangent to the previous one.

Parameters:

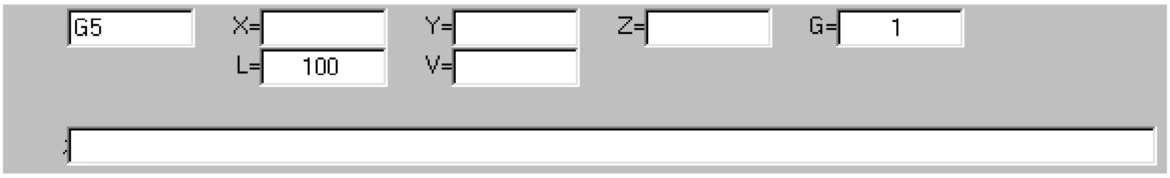
X Coordinate X of end of milling (only if G=2 or G=3).
Y Coordinate Y of end of milling (only if G=2 or G=3).
Z Depth of end of milling.
G Type of tangency:

- 1 Segment in the same running direction as the previous section.
- -1 Segment in the opposite running direction from the previous section.
- 2 Clockwise circular milling.
- 3 Counter-clockwise circular milling.

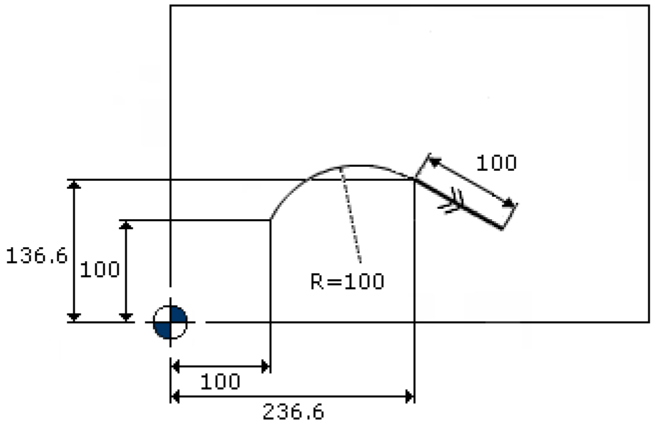
L Length of segment (only if G=1 or G=-1).
V Milling speed.

Examples:

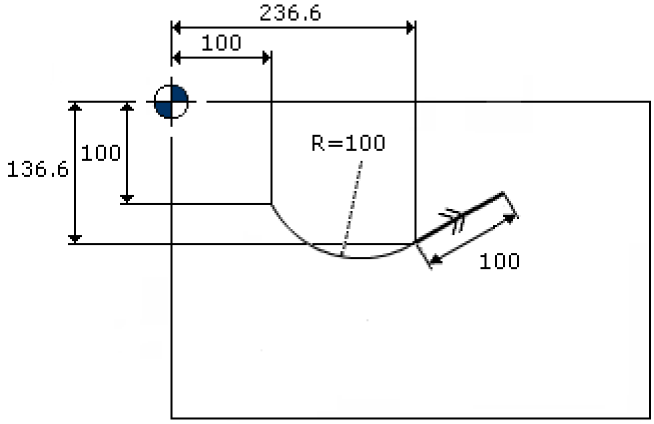
1) Programming a section of linear milling tangent to the arc in the same running direction. Parameters L, G are mandatory. The starting point for milling will be taken from the previous instruction.



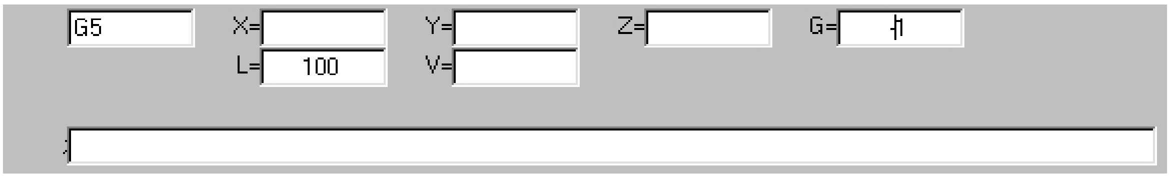
►*Front machine origin*



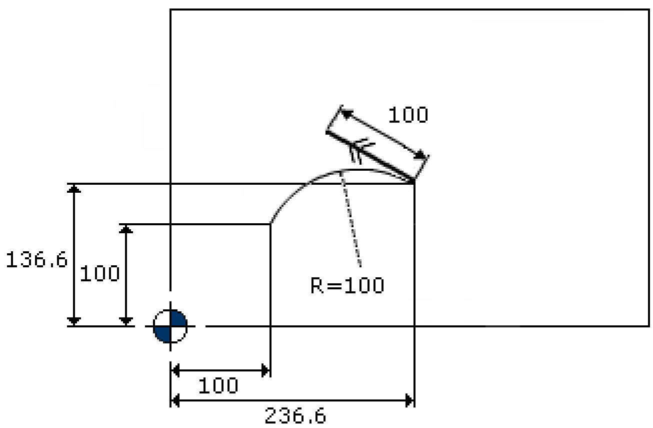
►*Rear machine origin*



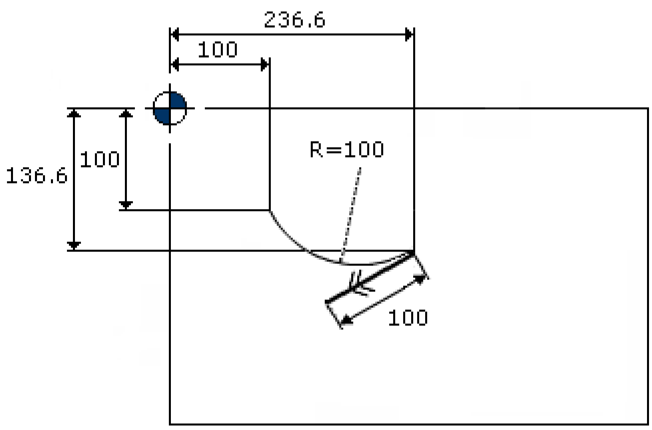
2) Programming a section of linear milling tangent to the arc in the opposite running direction. Parameters L, G are mandatory. The starting point for milling will be taken from the previous instruction.



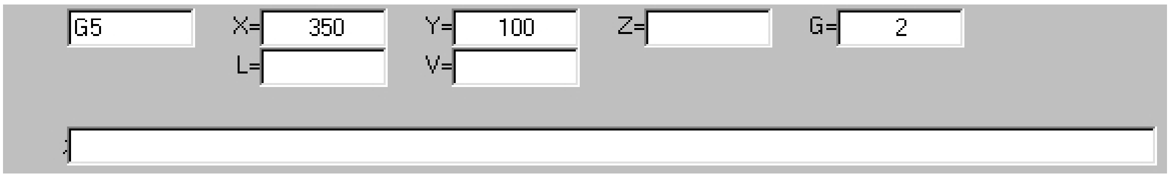
►*Front machine origin*



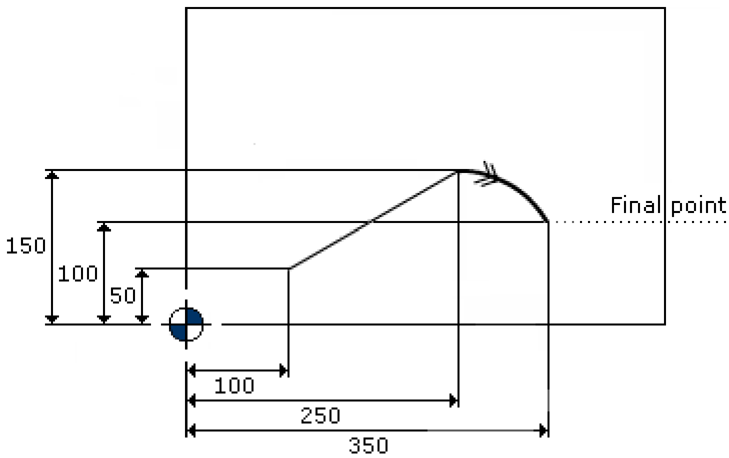
►*Rear machine origin*



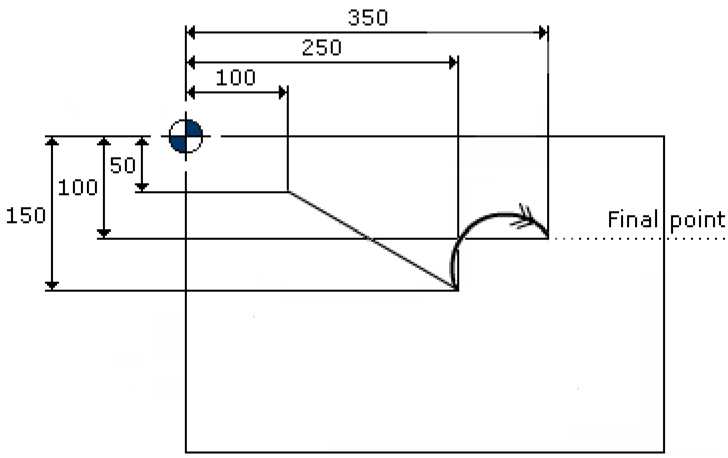
3) Programming a section of circular clockwise milling tangent to the previous section. Parameters X, Y, G are mandatory. The starting point for milling will be taken from the previous instruction.



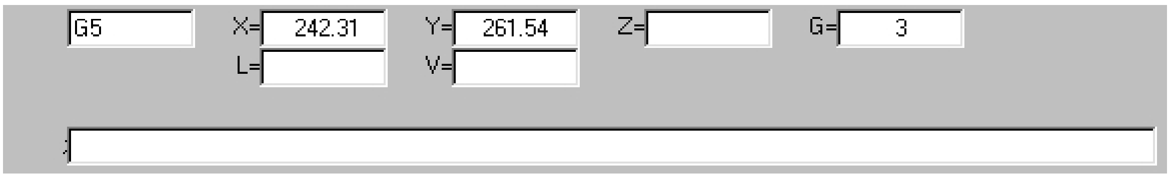
►*Front machine origin*



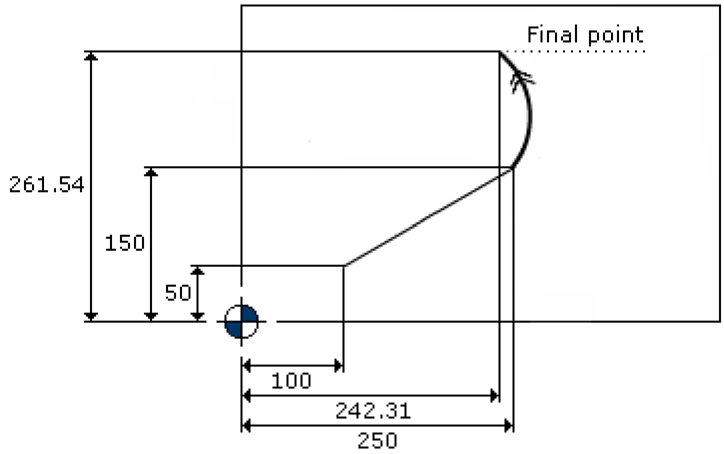
►*Rear machine origin*



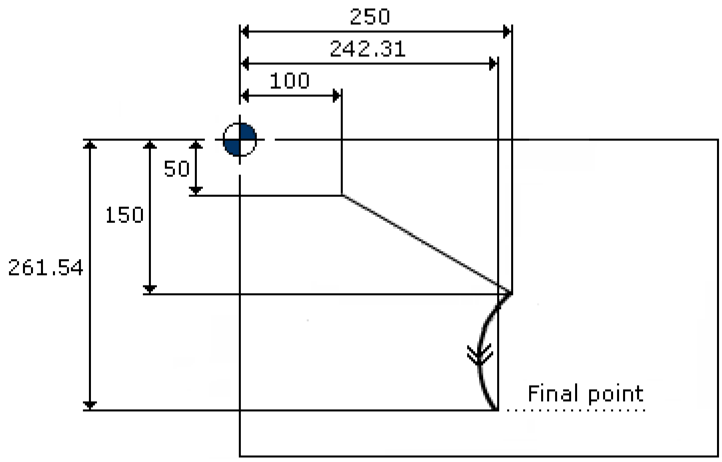
4) Programming a section of circular counter-clockwise milling tangent to the previous section. Parameters X, Y, G are mandatory. The starting point for milling will be taken from the previous instruction.



►*Front machine origin*



►*Rear machine origin*

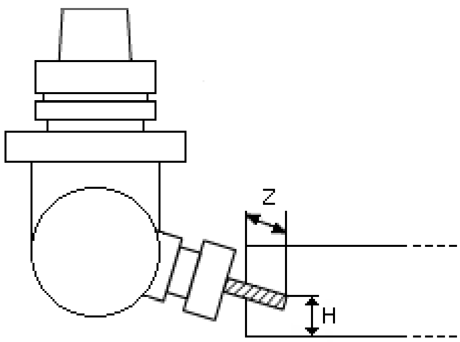


Slanted Begin Milling - G0R

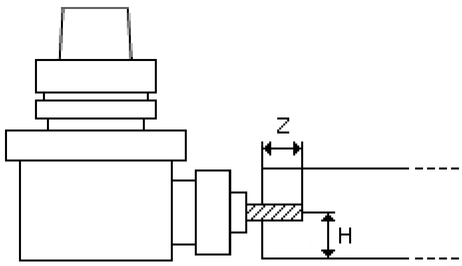
Begins a milling operation with a tool slanted on a plane which is not square with the surfaces of the panel. G0R is always referred to side 1. The current work side must always be side 1.

Parameters:

X Coordinate X for start of profile.
Y Coordinate Y for start of profile.
Z Depth of start of profile (measured according to the tool angle).
A Milling rotation angle.
H Height of machining from the working table (position of the tool from the working table of the machine). Defaults to DZ.
E Position of vacuum hood (see: Appendix D).
V Speed when entering the piece.
S Rotation speed of tool.
D Outside work quota.
Q Angular offset relative to parameter A (for special heads Benz configured in tooling with type M, N, O; see: Configuration Manuale for the Heads).
N Name of profile (see GREP).
IC Entrance side (0, 1, 2, 3, 4).
B Tool angle relative to the vertical plane (only for the Prisma group).
T Tool.

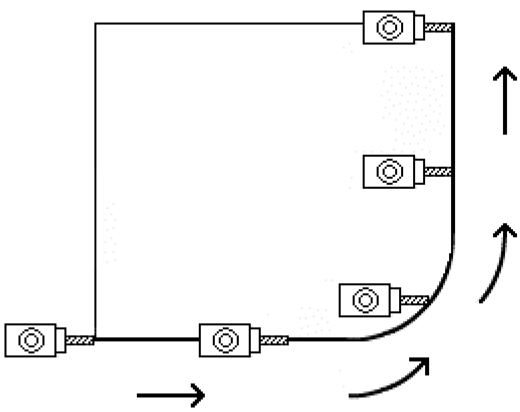


Inclined tool



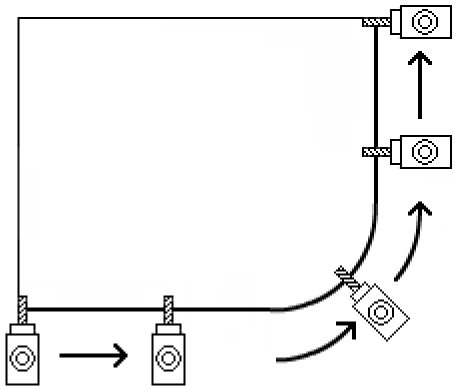
Tool at 90°

With parameter IC=0.



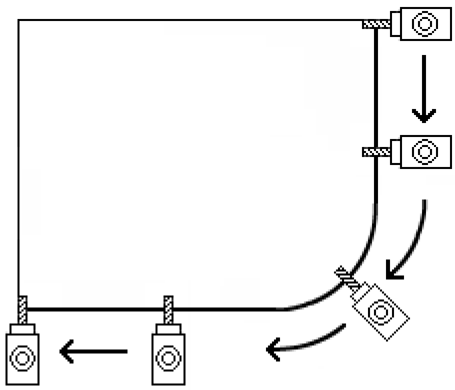
If parameter IC=0 and parameter A=0 the tool is parallel with the positive X-axis. Values other than 0 for parameter A cause an angular offset in tool orientation. The angle of the tool remains fixed during machining (fixed position).

With parameter IC=1.



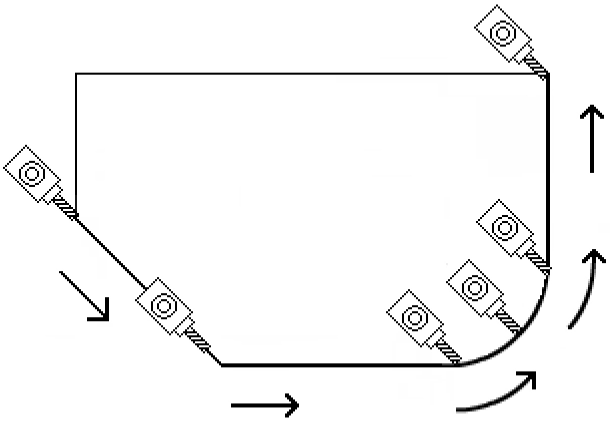
If parameter IC=1 and A=0 the tool is perpendicular to the programmed trajectory on the right-hand side of the feed path. Values other than 0 for parameter A cause an angular offset in tool orientation. The tool remains perpendicular to the path during machining (interpolated position).

With parameter IC=2.



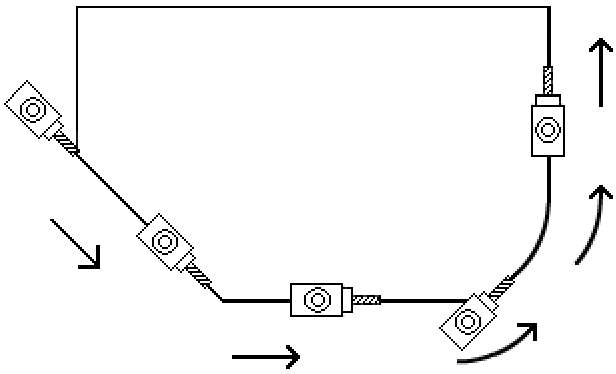
If parameter IC=2 and A=0 the tool is perpendicular to the programmed trajectory on the left-hand side of the feed path. Values other than 0 for parameter A cause an angular offset in tool orientation. The tool remains perpendicular to the path during machining (interpolated position).

With parameter IC=3.



If parameter IC=3 and A=0 the tool is parallel with the programmed trajectory. Values other than 0 for parameter A cause an angular offset in tool orientation. The angle of the tool remains fixed during machining (fixed position).

With parameter IC=4.



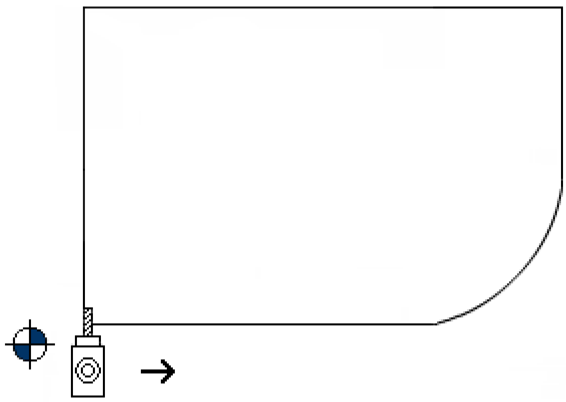
If parameter IC=4 and A=0 the tool is parallel with the programmed trajectory. Values other than 0 for parameter A cause an angular offset in tool orientation. The tool remains parallel with the path during machining (interpolated position).

Example:

►*Front machine origin*

Programming the start of inclined milling with perpendicular tool and on the right side compared to the direction to be followed:

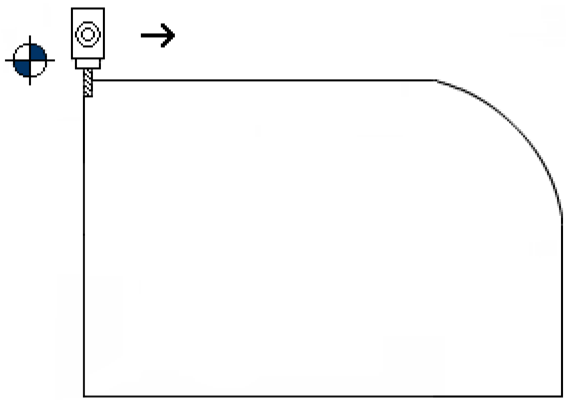
	G0R	X=	0	Y=	0	Z=	30	A=	0
H=	25	E=		V=		S=		D=	40
Q=		N=		IC=	1	B=		T=	101
;									



►*Rear machine origin*

Programming the start of inclined milling with a perpendicular tool and on the left side compared to the direction to be followed.

	G0R	X=	0	Y=	0	Z=	30	A=	0
H=	25	E=		V=		S=		D=	40
Q=		N=		IC=	2	B=		T=	101



N.B. Automatic entry and exit are not envisaged for inclined routing. The only instructions to follow the G0R could be G1R, G2R, G3R, G5R.

WARNING

Incorrect programming of machining with slanted tools may cause damage to the structure of the machine. It is advisable that a SIMULATION of this type of operation be executed first.

Slanted Linear Milling - G1R

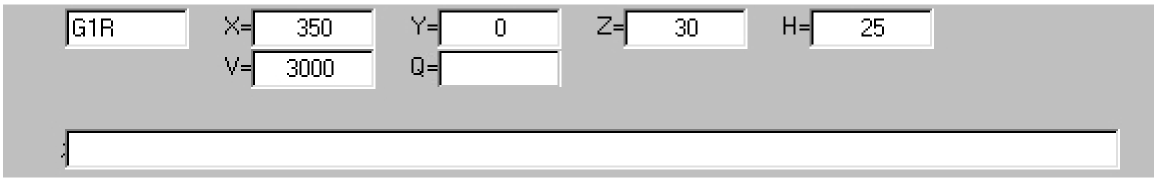
Makes it possible to perform linear milling on a surface that is inclined compared to the orthogonality of the panel surfaces; it must be used with slanted tools. The G1R instruction always refers to surface 1; for this reason the current work surface must always be surface 1.
This instruction must be preceded by a begin milling with slanted tool or by another milling instruction with slanted tool.

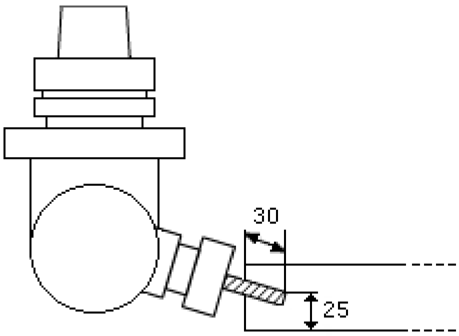
Parameters:

X Coordinate X for end of milling.
Y Coordinate Y for end of milling.
Z End of milling depth (measured according to the tool angle).
H Height of end point of machining from working table (position of the tool from the working table of the machine).
V Milling speed.
Q Angular offset relative to parameter A (for special heads Benz configured in tooling with type M, N, O; see: Configuration Manuale for the Heads).

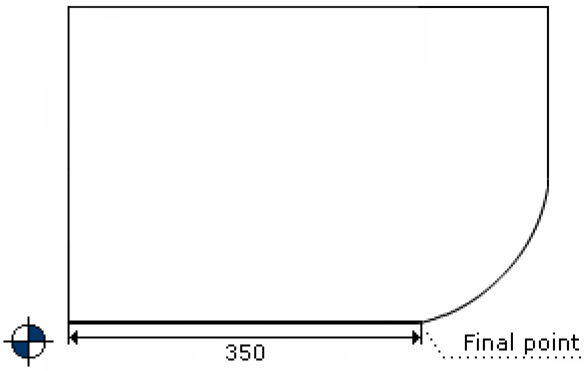
Example:

Programming inclined linear milling with starting point X=0, Y=0.

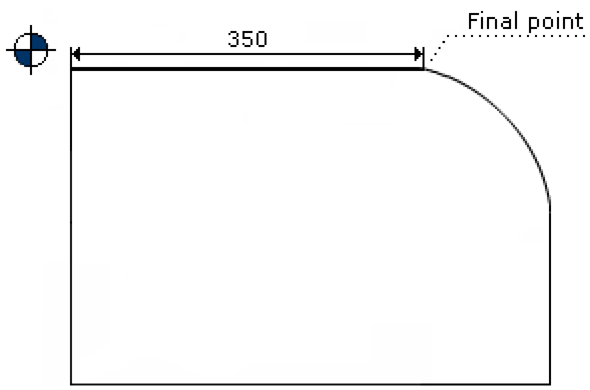




►*Front machine origin*



►*Rear machine origin*



WARNING

Incorrect programming of machining with slanted tools may cause damages to the structure of the machine. It is advisable that a SIMULATION of this type of operation be executed first.

Slanted Clockwise Circular Milling - G2R

Defines a section of circular (or circle arc) milling on a surface that is inclined compared to the orthogonality of the panel surfaces. In alternative to the definition of the center, it is possible to define the radius of the arc according to the following convention: if the radius is positive, the center falls to the left of the imaginary line going from the beginning to the end point of the arc, otherwise, the center falls to the right of this line. The G2R instruction always refers to surface 1; for this reason the current work surface must always be surface 1.
This instruction must be preceded by a begin milling with slanted tool or by another milling instruction with slanted tool.

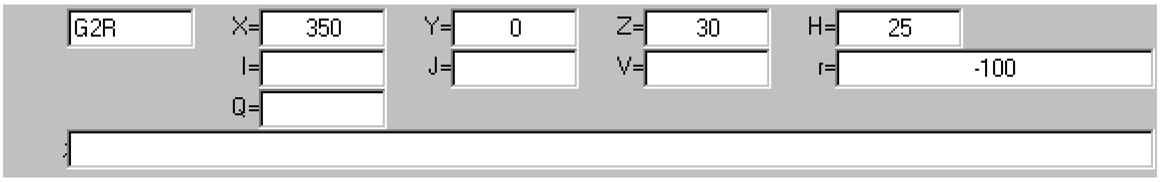
Parameters:

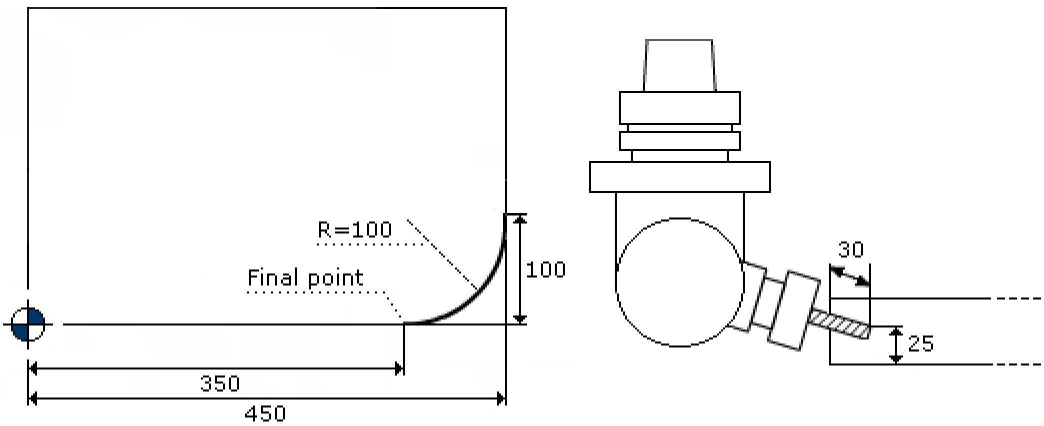
X Coordinate X for end of milling.
Y Coordinate Y for end of milling.
Z End of milling depth (measured according to the tool angle).
H Height of end point of machining from working table (position of the tool from the working table of the machine).
I Coordinate X of center of arc.
J Coordinate Y of center of arc.
V Milling speed.
r Arc radius.
Q Angular offset relative to parameter A (for special heads Benz configured in tooling with type M, N, O; see: Configuration Manuale for the Heads).

Example:

►*Front machine origin*

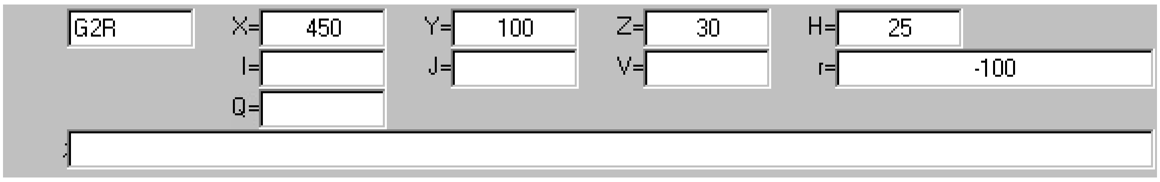
Programming inclined clockwise circular milling with starting point X=450, Y=100.

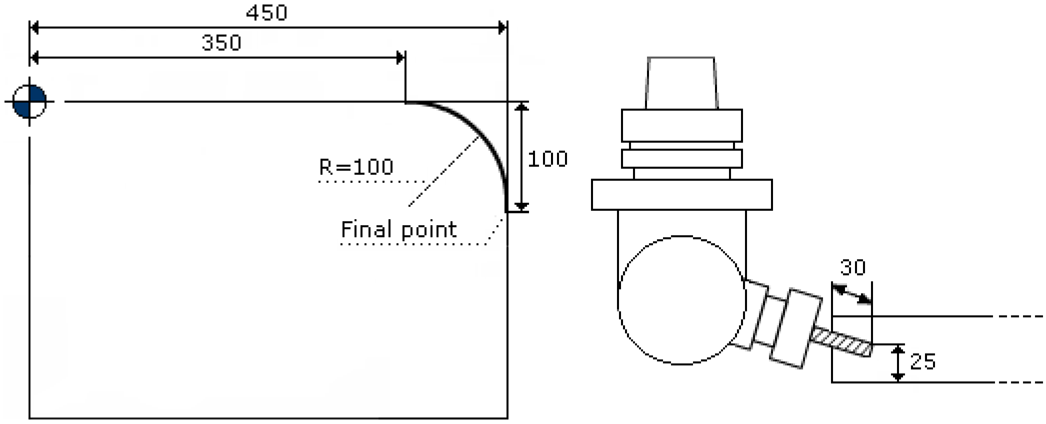




►*Rear machine origin*

Programming inclined clockwise circular milling with starting point X=350, Y= 0.





WARNING

Incorrect programming of machining with slanted tools may cause damages to the structure of the machine. It is advisable that a SIMULATION of this type of operation be executed first.

Slanted Counter-Clockwise Circular Milling – G3R

Defines a section of circular (or circle arc) milling on a surface that is inclined compared to the orthogonality of the panel surfaces, with a counter-clockwise sense of direction. In alternative to the definition of the center, it is possible to define the radius of the arc according to the following convention: if the radius is positive, the center falls to the left of the imaginary line going from the beginning to the end point of the arc, otherwise, the center falls to the right of this line. The G3R instruction always refers to surface 1; for this reason the current work surface must always be surface 1.
This instruction must be preceded by a begin milling with slanted tool or by another milling instruction with slanted tool.

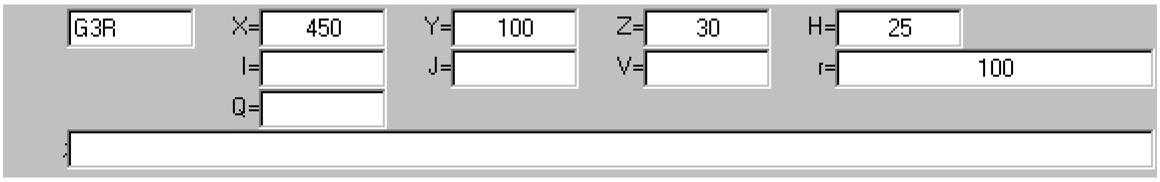
Parameters:

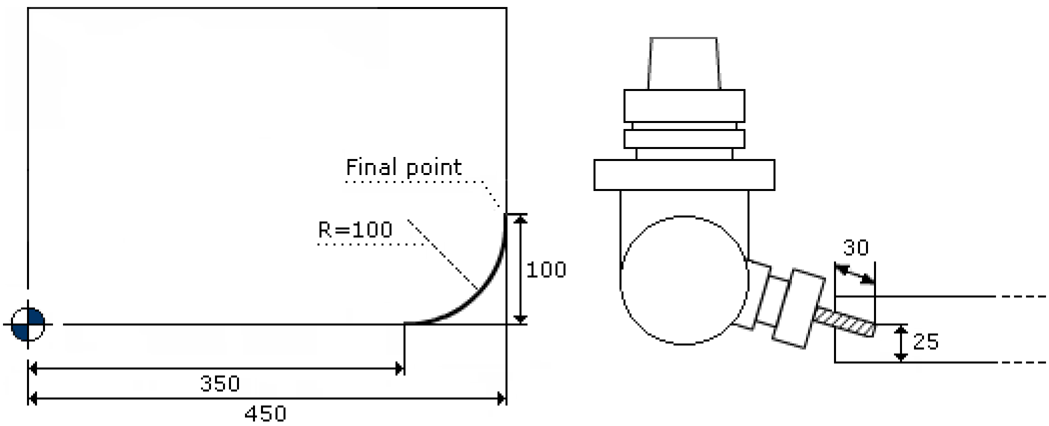
X	Coordinate X for end of milling.
Y	Coordinate Y for end of milling.
Z	End of milling depth (measured according to the tool angle).
H	Height of end point of machining from working table (position of the tool from the working table of the machine).
I	Coordinate X of center of arc.
J	Coordinate Y of center of arc.
V	Milling speed.
r	Arc radius.
Q	Angular offset relative to parameter A (for special heads Benz configured in tooling with type M, N, O; see: Configuration Manuale for the Heads).

Example:

►*Front machine origin*

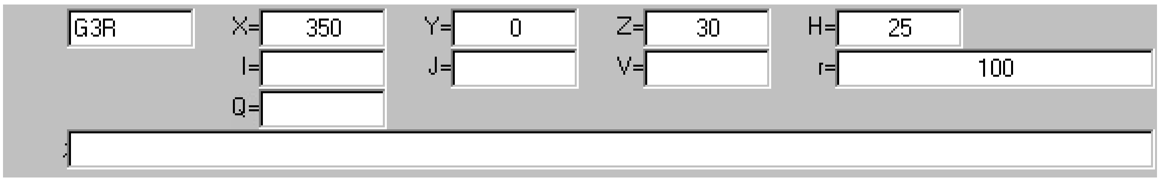
Programming inclined Counter-clockwise circular milling with starting point X=350, Y=0.

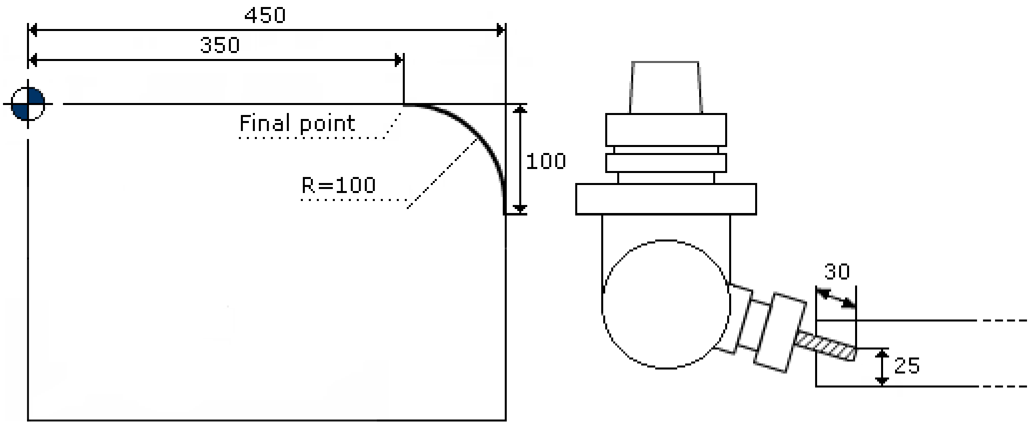




►*Rear machine origin*

Programming inclined Counter-clockwise circular milling with starting point X=450, Y=100.





WARNING

Incorrect programming of machining with slanted tools may cause damages to the structure of the machine. It is advisable that a SIMULATION of this type of operation be executed first.

Section Tangent to Previous Inclined Section - G5R

Defines a section of milling tangent to previous section with slanted tool. Instruction G5R always refers to surface 1; for this reason the current work surface must always be surface 1.
This instruction must be preceded by a begin milling with slanted tool or by another milling instruction with slanted tool.

Parameters:

X Coordinate X of end of milling (only if G=2 or G=3).
Y Coordinate Y of end of milling (only if G=2 or G=3).
Z End of milling depth (measured according to the tool angle).
H Height of end point of machining from working table. (position of the tool from the working table of the machine).
G Type of tangency:

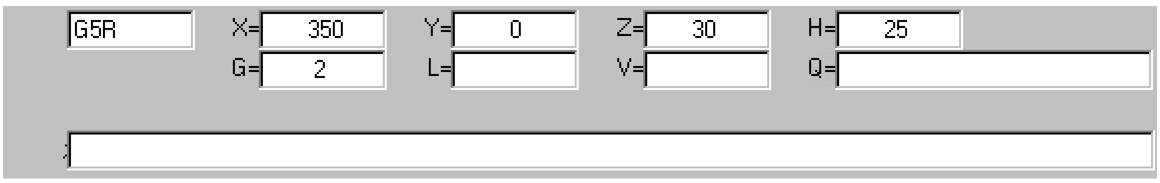
- 1 Segment in the same running direction as the previous section.
- -1 Segment in the opposite running direction from the previous section.
- 2 Clockwise circular milling.
- 3 Counter-clockwise circular milling.

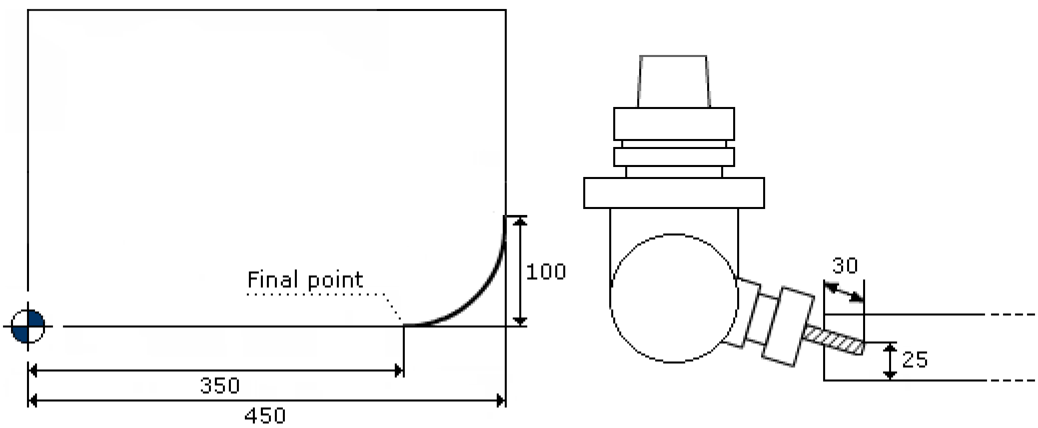
L Length of segment (only if G=1 or G=-1).
V Milling speed.
Q Angular offset relative to parameter A (for special heads Benz configured in tooling with type M, N, O; see: Configuration Manuale for the Heads).

Examples:

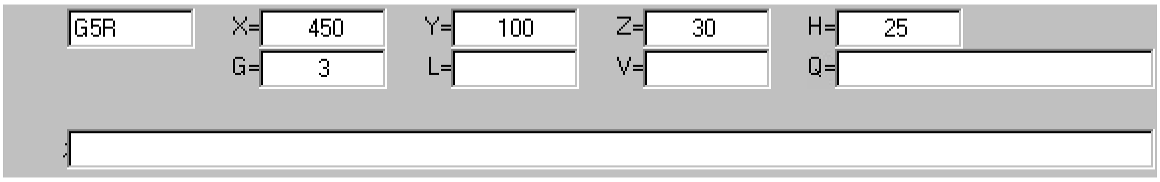
►*Front machine origin*

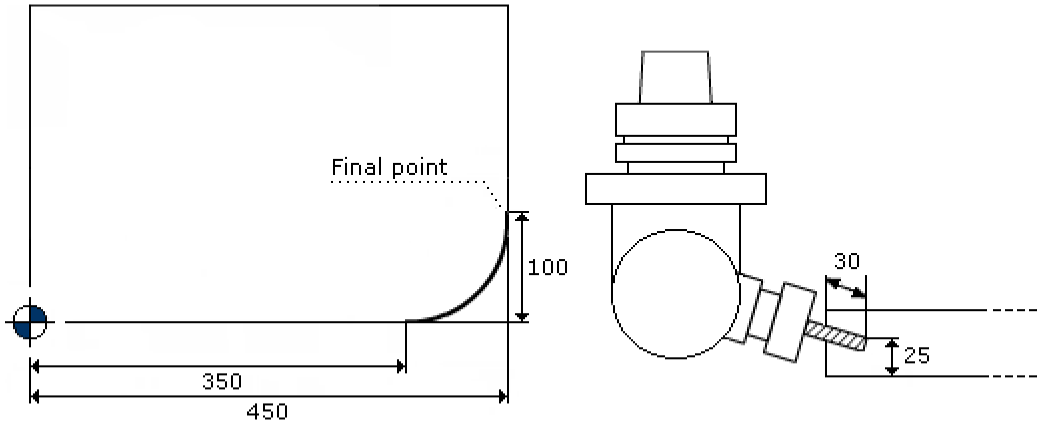
1) Programming a section of inclined milling tangent to a segment with starting point X=450, Y=100.



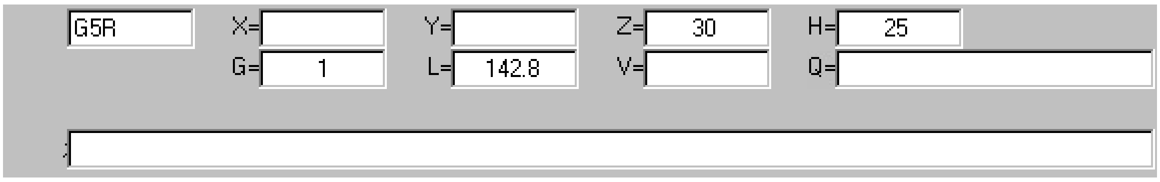


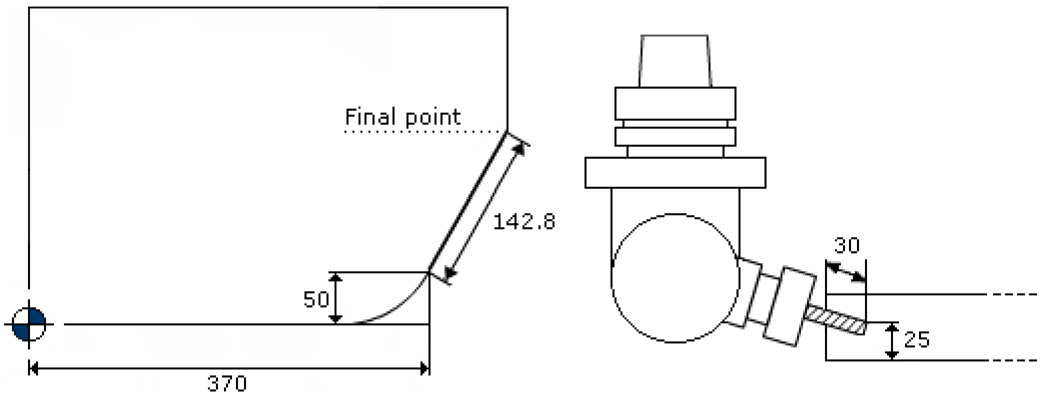
2) Programming a section of inclined milling tangent to a segment with starting point X= 350, Y=0.





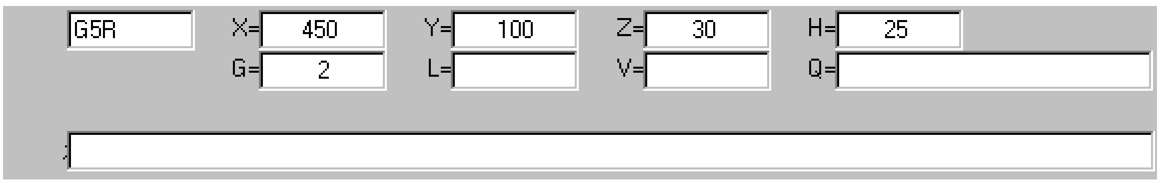
3) Programming a section of inclined milling tangent to an arc with starting point X=370, Y=50.

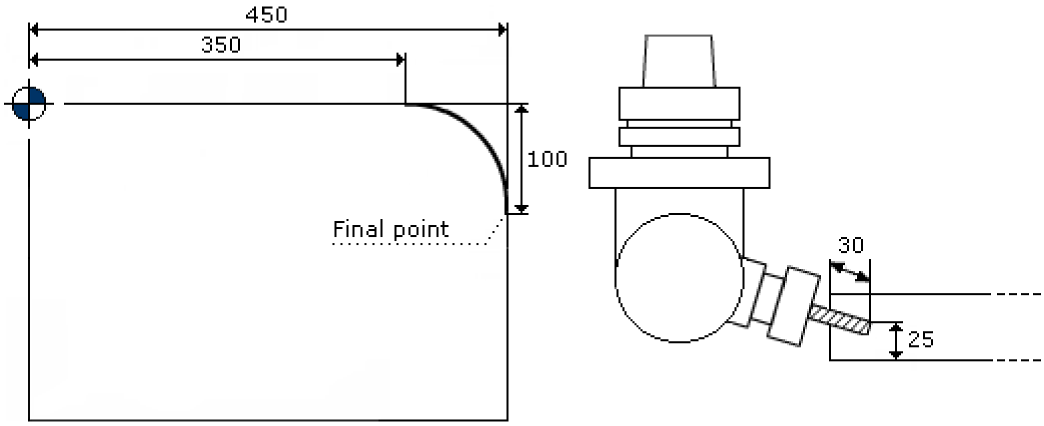




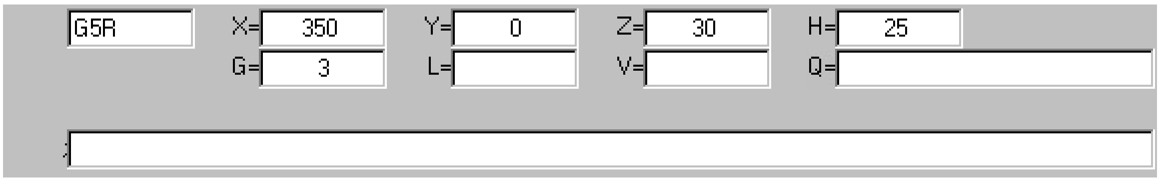
►*Rear machine origin*

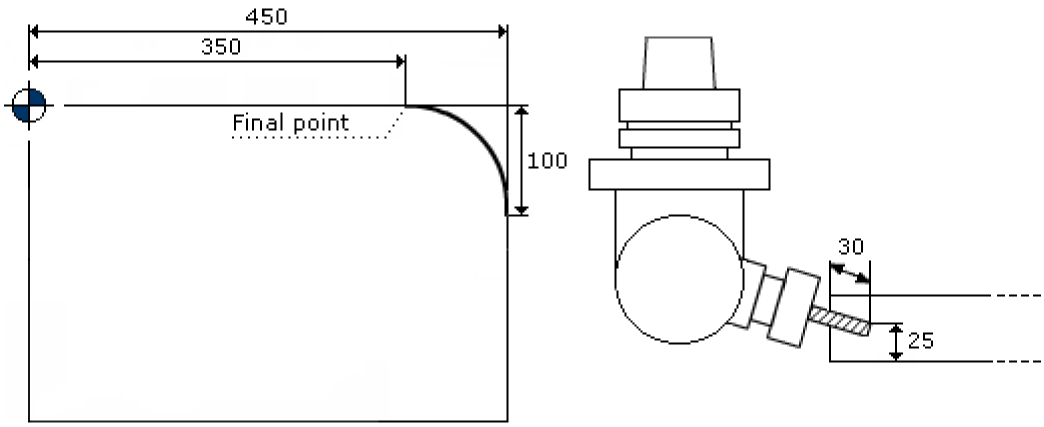
1) Programming a section of inclined milling tangent to a segment with starting point X=350, Y=0.



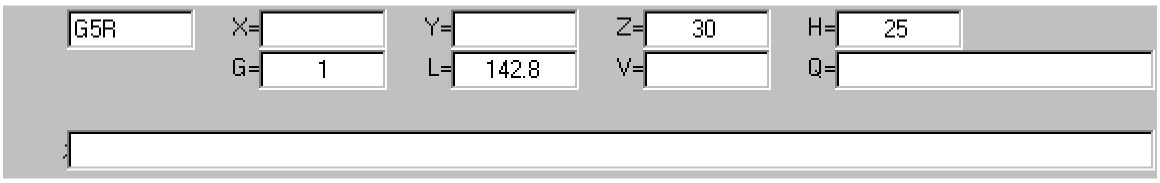


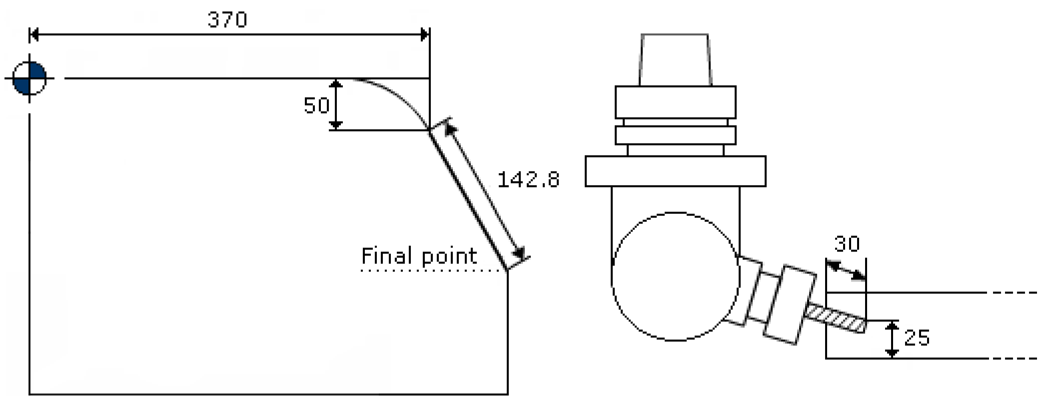
2) Programming a section of inclined milling tangent to a segment with starting point X=450, Y=100.





3) Programming a section of inclined milling tangent to an arc with starting point X=370, Y=50.





WARNING

Incorrect programming of machining with slanted tools may cause damages to the structure of the machine. It is advisable that a SIMULATION of this type of operation be executed first.

Link Between Millings - GFIL

Makes a circular connecting milling between the milling programmed before this instruction and the one programmed after this instruction. This instruction connects any linear or circular milling to any other linear or circular milling.

Parameters:

r Radius of connecting arc.
V Milling speed.

Use the following programming sequence:

1) digit the first section;
2) digit the second section;
3) insert a line with GFIL between the two previous.

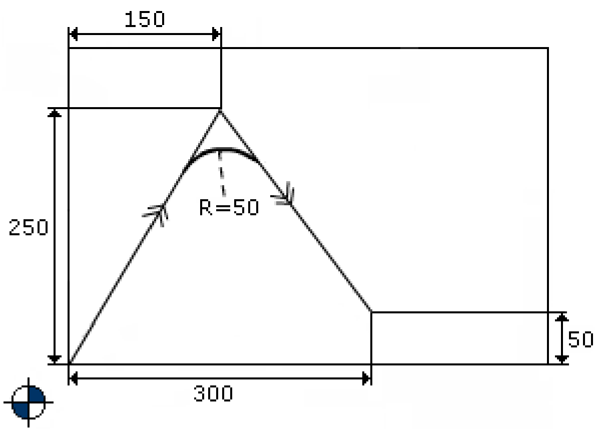
The system automatically calculates the starting and end points of the fillet.

Examples:

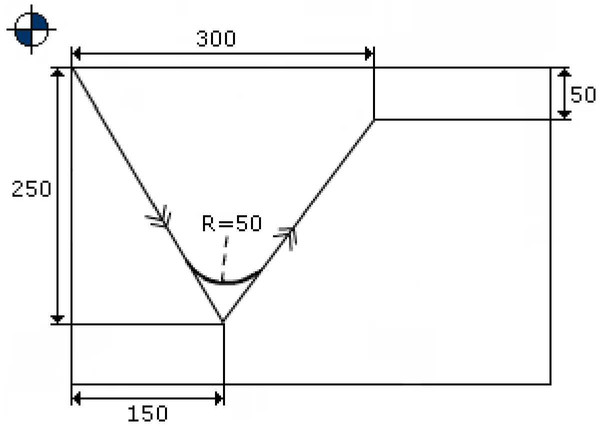
1) Programming a connection with a 50 mm radius between the two linear millings.

Header	H DX=450 DY=300 DZ=20 -A C=0 T=0 R=1 *MM /"test"
Start milling	G0 X=0 Y=0 Z=10 T=101 V=5000 S=16000 E=1 D=30
First segment	G1 X=150 Y=250
Fillet	**GFIL r=50**
Second segment	G1 X=300 Y=50

►*Front machine origin*



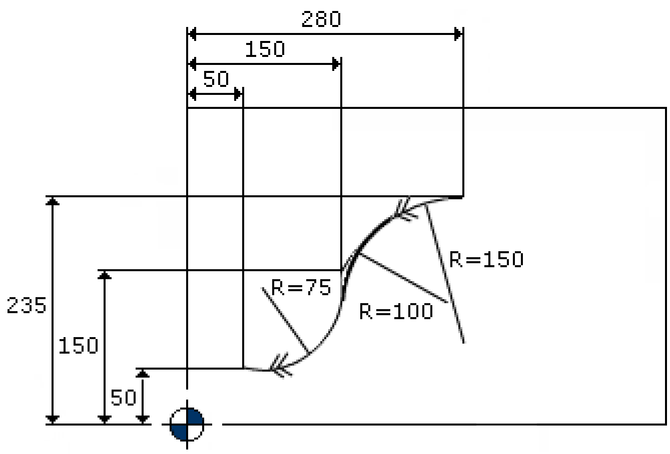
►*Rear machine origin*



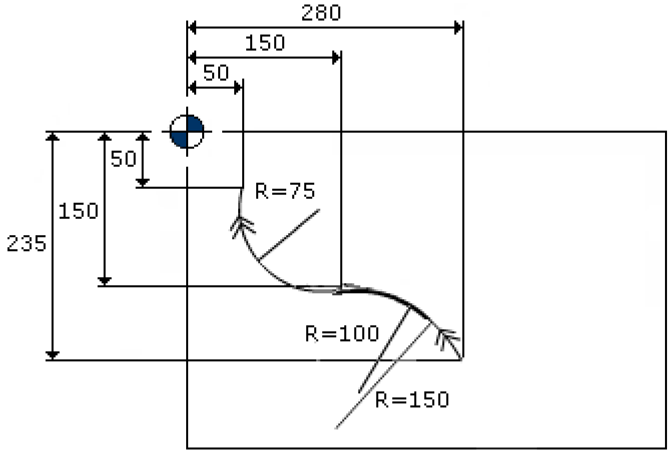
2) Programming a connection with a 100 mm radius between the two circular millings

Header	H DX=450 DY=300 DZ=20 -A C=0 T=0 R=1 *MM /"test"
Start routing	G0 X=280 Y=235 Z=10 T=101 V=5000 S=16000 E=1 D=30
First arc	G3 X=150 Y=150 r=150
Fillet	**GFIL r=100**
Second arc	G2 X=50 Y=50 r=-75

►*Front machine origin*



►*Rear machine origin*



Bevel Between Millings - GCHA

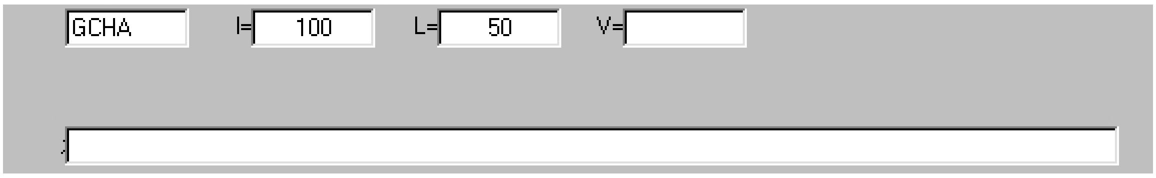
Makes a circular rounding milling between the milling programmed before this instruction and the one programmed after it. The previous and following rounding instruction can be any linear milling.

Parameters:

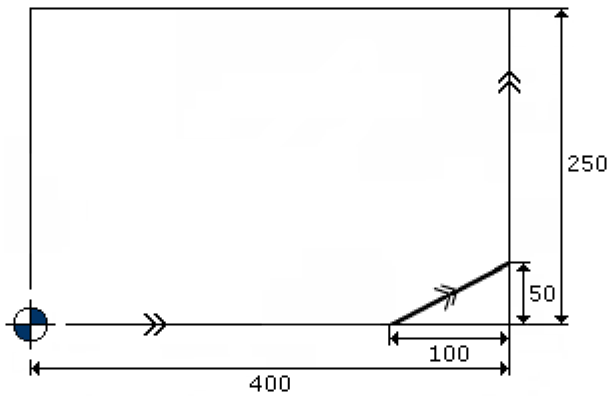
I Length of the first section to be rounded.
L Length of the second section to be rounded.
V Milling speed.

Example:

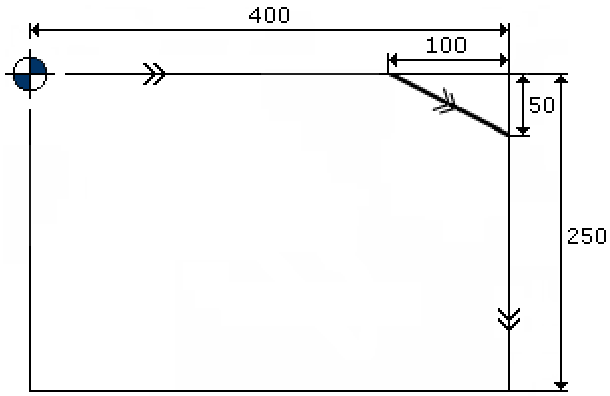
Programming a chamfer between two linear millings.



► *Front machine origin*



►*Rear machine origin*



Use the following programming sequence:

1) digit the first section;
2) digit the second section;

3) insert a line with GCHA between the two previous.

The system automatically calculates the starting and end points of the fillet.

Header	H DX=400 DY=250 DZ=20 -A C=0 T=0 R=1 *MM /"ROUTE"
Start routing	G0 X=0 Y=0 Z=10 T=101
Segment	G1 X=400 Y=0
Chamfer	**GCHA l=100 L=50**
Segment	G1 X=400 Y=250

5.2.2 Modal Instructions

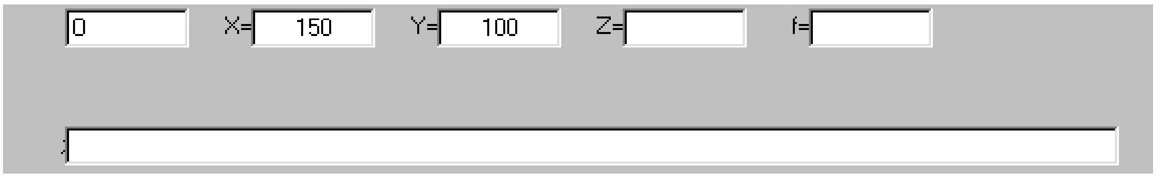
Origin Displacement of Panel in Stop - O

Moves the origin of the panel to the programmed position. All the following instructions will refer to the new origin.

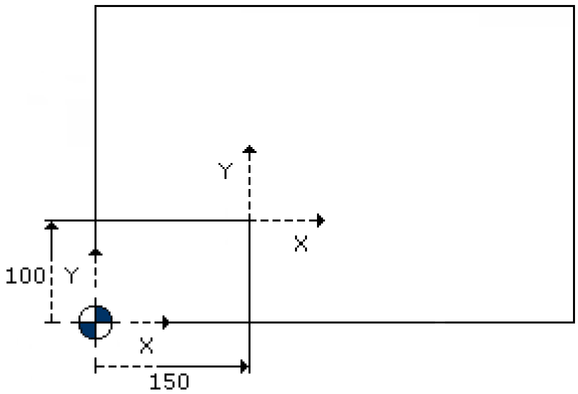
Parameters:

X Origin in X.
Y Origin in Y.
Z Origin in Z.
f If programmed with the number of one face (1-5), it enables the instruction only for the origin of the set face.

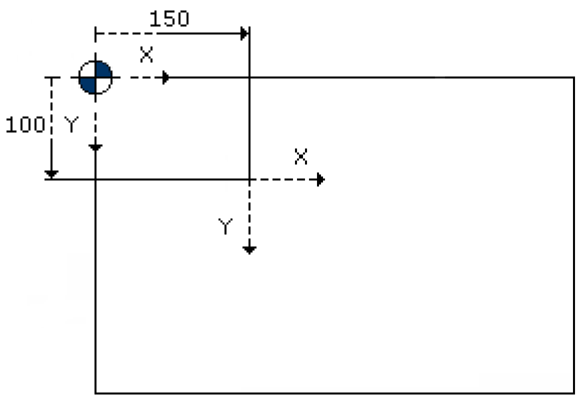
Example:



►*Front machine origin*



►*Rear machine origin*



Machining Side - F

Defines the current machining face.

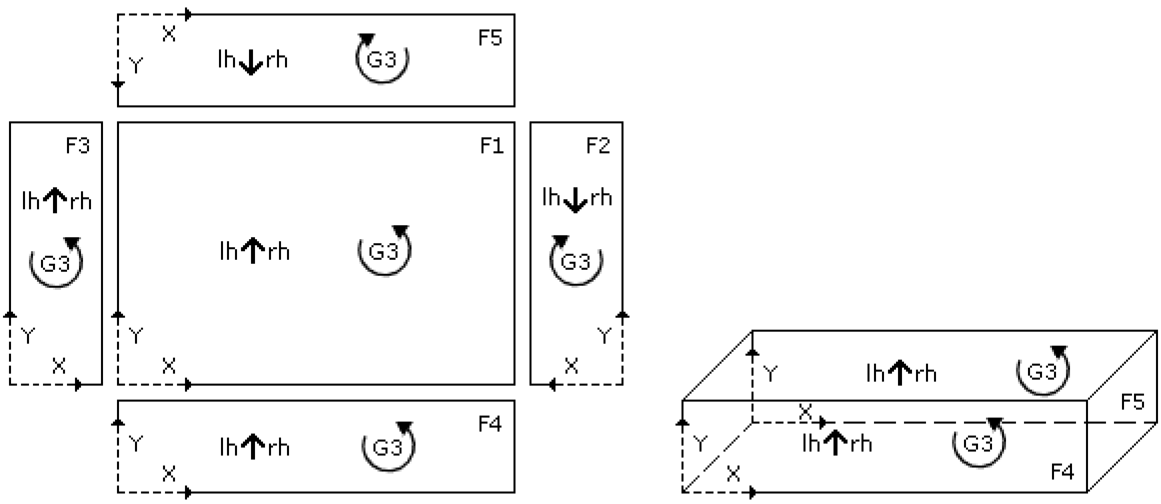
Unit:
Text instructions

= Accepted values:

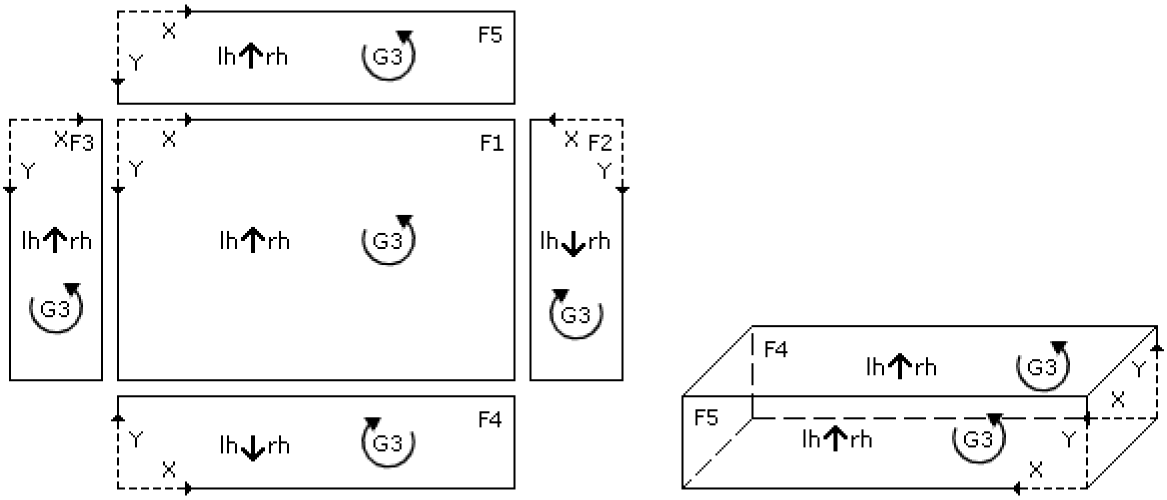
1 Upper face.
2 Right face.
3 Left face.
4 Front face
5 Rear face.

Example:

References for front machine origin:

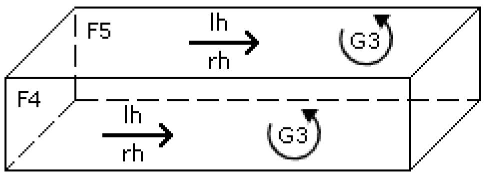


References for rear machine origin:



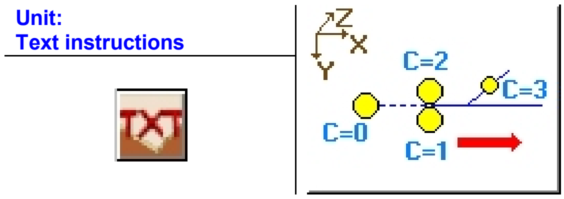
N.B. For interpolations on the side faces (F2 F3 F4 F5) Xilog Plus sees the panel as if it were transparent from the front face (F4) to the rear face (F5) and from the left-hand face (F3) to the right-hand face (F2), thus inverting the circular arcs (if on F4 it is clockwise, on F5 it becomes anti-clockwise, if on F3 it is clockwise, on F2 it becomes anti-clockwise) and the radius correction (if on F4 it is LH, on F5 it becomes RH, if on F3 it is LH, on F3 it becomes RH).

Example:



Tool Adjustment - C

Makes it possible to correct the path of the spindle based on the characteristics of the boring machine that is mounted. If the boring machine is fluted (type F) the correction is equal to the radius declared in the tooling file at the voice "actual diameter"; if the milling machine is disk (type D) the correction is equal to half the blade thickness declared in the tooling file.

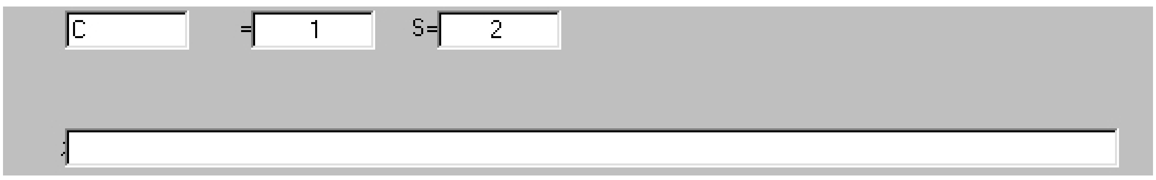


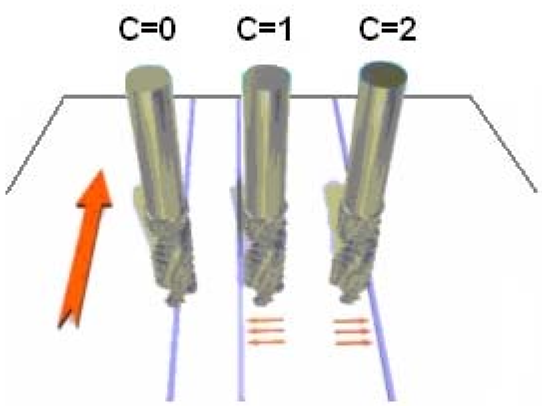
= Accepted values:

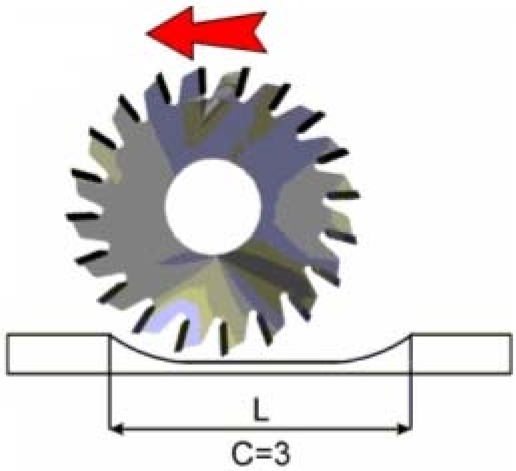
0 Void correction.
1 Right correction.
2 Left correction.
3 In depth correction (only for disk millers).
13 Correction 1 + correction 3 (only for disk millers).
23 Correction 2 + correction 3 (only for disk millers).

S Quota of over-metal.

Example:

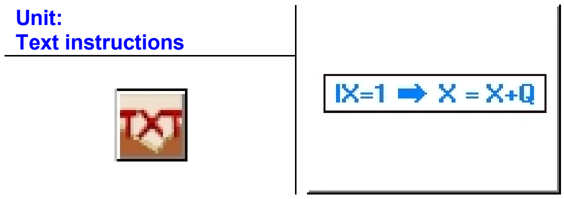






Incremental in X - IX

Enables incremental programming in the X direction. The quota that is activated as incremental, when programmed in an operating instruction assumes significance not for its position compared to the panel origin but for the deviation from the value that the quota itself had assumed after executing the previous operating instruction.



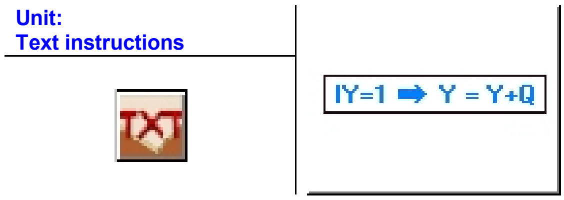
= Accepted values:

0 Disables incremental programming in X.
1 Enables incremental programming in X.

Example:

Incremental in Y - IY

Enables incremental programming in the Y direction. The quota that is activated as incremental, in an operating instruction assumes significance not for its position compared to the panel origin but for the deviation from the value that the quota itself had assumed after executing the previous operating instruction.



= Accepted values:

0 Disables incremental programming in Y.
1 Enables incremental programming in Y.

Example:

Specular in X - SX

Changes the origin in X by moving it into the opposite corner. This instruction is activated by automatically entering the profile or by starting milling.

Unit:
Text instructions

SX=1 ➡ X = DX-Q

Parameters:

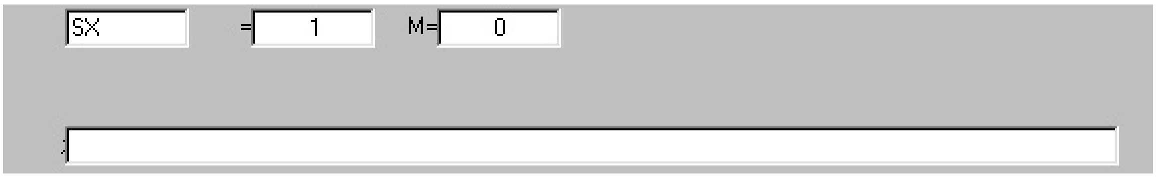
= Changes the origin in X.

0 Restores the normal reference in X.
1 Moves reference in X.

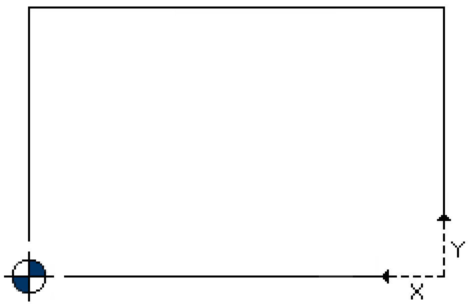
M Reverses the direction of circular milling.

0 Not to reverse.
1 To reverse.

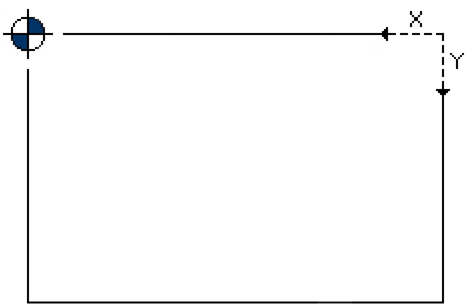
Example:



► *Front machine origin*

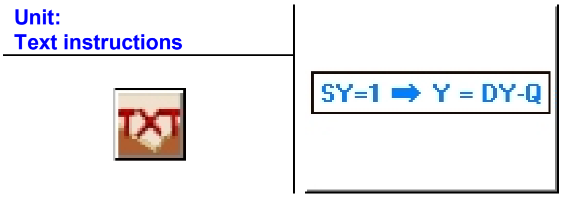


► *Rear machine origin*



Specular in Y - SY

Changes the origin in Y by moving is into the opposite corner. This instruction is activated by automatically entering the profile or by starting milling.



Parameters:

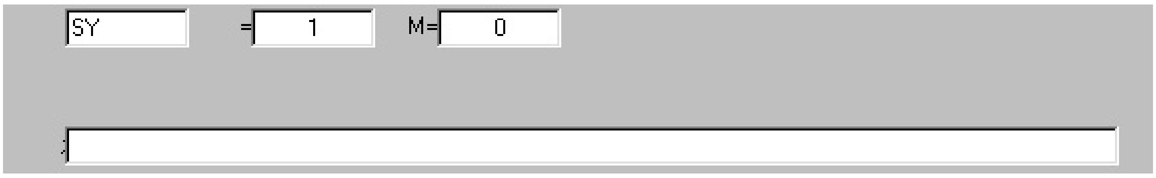
= Changes the origin in Y.

0 Restores the normal reference in Y.
1 Moves reference in Y.

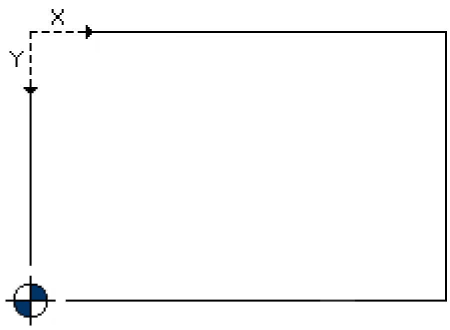
M Reverses the direction of circular milling.

0 Not to reverse.
1 To reverse.

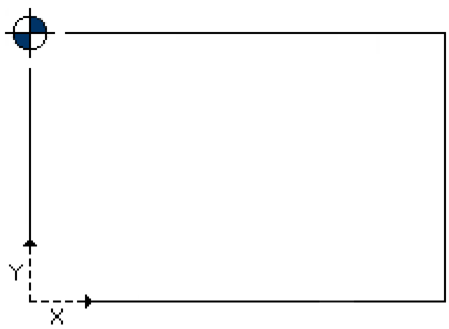
Example:



►*Front machine origin*



►*Rear machine origin*



Inclined Plane - PL

Defines a surface that differs from the 5 faces that Xilog Plus creates in automatic. It can be rotated either around axis Z or around axis X. An inclined surface server to create geometries that are perpendicular to the surface itself.

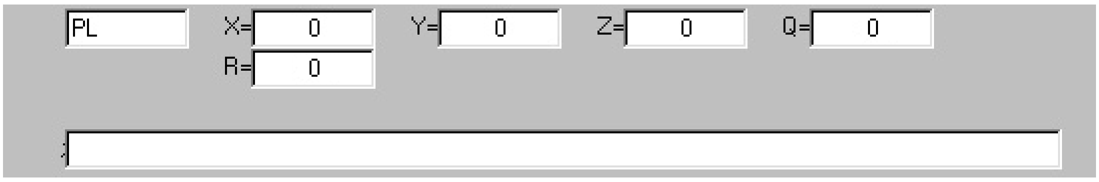
Parameters:

X Coordinate X of the origin of the plane (referred to the origin of the panel).
Y Coordinate Y of the origin of the plane (referred to the origin of the panel).
Z Coordinate Z of the origin of the plane (in relation to panel base).
Q Angle of rotation around axis Z (front machine origin: positive in counter-clockwise direction; rear machine origin: positive in clockwise direction).
R Angle of rotation around axis X (front machine origin: positive in counter-clockwise direction; rear machine origin: positive in clockwise direction).

The F parameter must always be set on 1 with the modal instruction F. Parameters X, Y, Q, R are mandatory.
A PL instruction is cancelled by:

- another instruction PL;
- the retrieval of a standard face.

To reset and inclined panel insert another PL instruction with all parameters on 0, as shown in the following example.

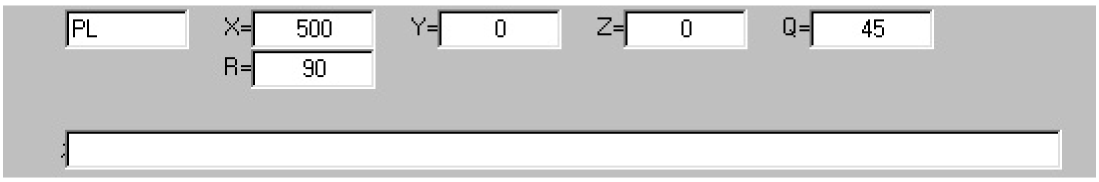


If all parameters have a value of zero face 1 is reset.

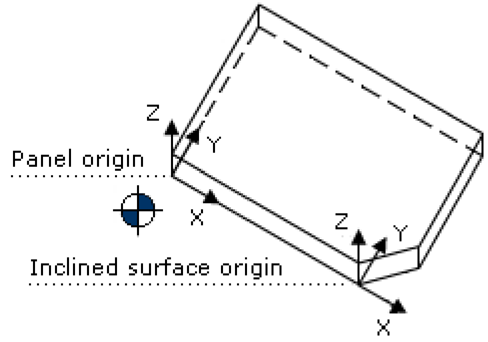
Example:

Programming an inclined surface with origins X=500, Y=0, Z=0, rotated 45° about the Z-axis and 90° about the X-axis.

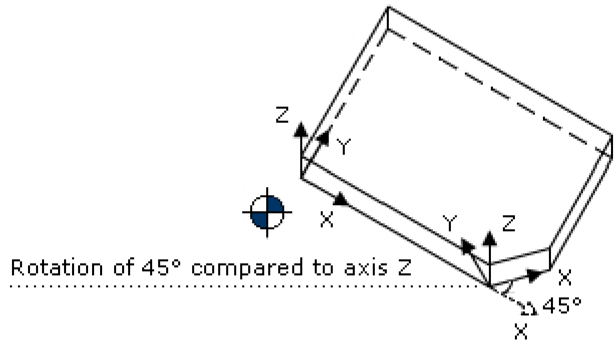
►*Front machine origin*



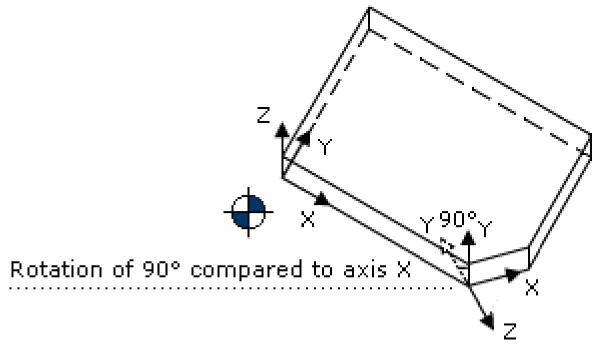
X=500, Y=0 and Z=0 are origin coordinates of the inclined surface.



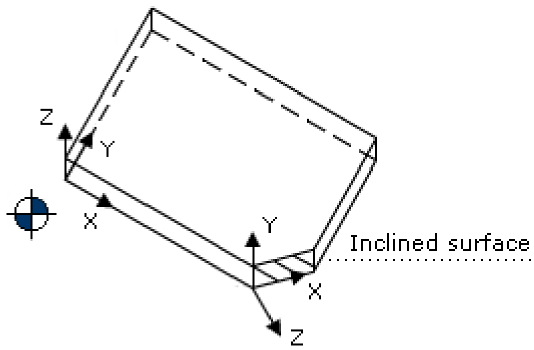
After having changed origins, the axis system must rotate around axis Z, by entering the parameter Q=desired angle (Q=45).



At this point, the axis system must rotate around axis X by entering parameter R=desired angle (R=90).

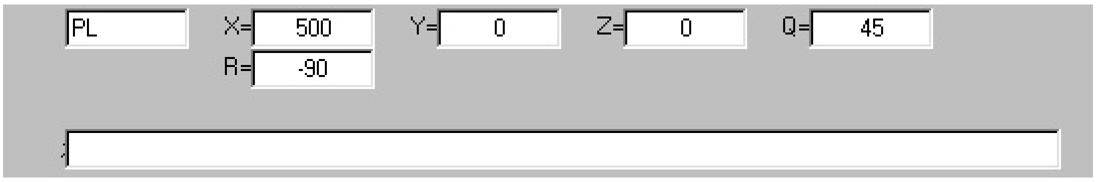


The final result will be an inclined surface like the one shown in the following figure.

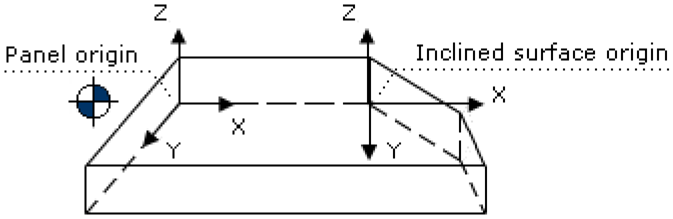


N.B. The direction of the Z-axis must be always **negative towards the outside** of the panel.

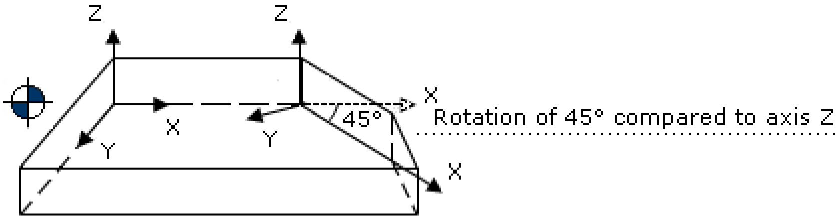
►*Rear machine origin*



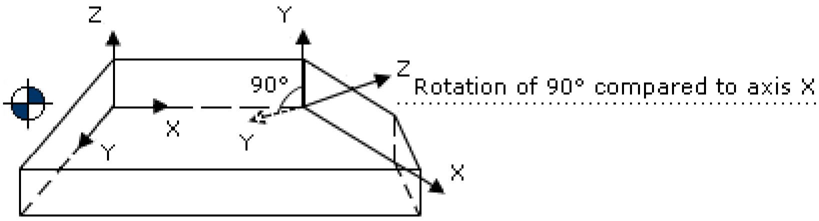
X=500, Y=0 and Z=0 are origin coordinates of the inclined surface.



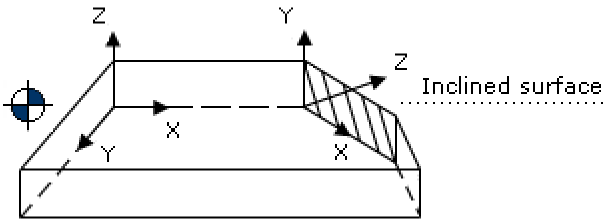
After having changed origin the axis system must be rotated around the Z axis by entering the parameter Q=desired angle (Q=45).



At this point, the axis system must rotate around axis X by entering parameter R=desired angle (R=-90).



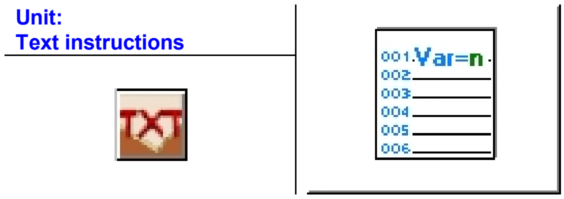
The final result will be an inclined surface like the one shown in the following figure.



N.B. The direction of the Z-axis must be always **negative towards the outside** of the panel.

Assigning a Value to Variable - SET

With this instructions it is possible to temporarily change (only for the program in which they are inserted) some system parameters established by the manufacturer of the work center. In addition to the SET command, you must enter a preset name indicating the parameter to be edited.



The names predefined by the SET instruction are:

AUTOTOOL Enables/disables change of overlapping tool (change of toll on main spindle, while the boring machine is working) when the machine model and configuration allows.

0 = disables overlapping tool change.
1 = enables overlapping tool change with movement in both X and Y
2 = enables overlapping tool change with movement only in X.

DONTCARE The SET DONTCARE=1 instruction forces Xilog Plus into not releasing the “There is no safety quota above the piece” message or the following work.
The instruction is only effective in recoverable cases (see: Appendix N – Notes: Transfer above Panel) and only on the following machining. There must not be any other instructions between SET DONTCARE=1 and the machining. You can use the SET DONTCARE=1 instruction to take responsibility for direct control of the movements (by means of N or XN), so as to prevent collisions with the panels and guarantee that any Vector rotations occur outside the work piece.

JERK In order to optimize machine use in the various machining stages, a special software function has been enabled (called Jerk), which can vary the machine response (and, therefore, its dynamic behavior) according to the work piece being machined. This function is capable of progressively accelerating the machine and adjusting the law of evolution. In this manner, it is possible to obtain a considerable reduction in stop and acceleration times, with notable gains on the total work time for each piece and reduced machine solicitation. The results which can be obtained are linked to the type of work piece being machined and are more obvious the more significant are the rapid movement phases (G0) and acceleration and deceleration. For example, the maximum gain is in boring cycles, whilst the minimum gain is in routing only cycles with slow consecutive movements (G1).
Expert users are given the possibility of optimizing the machine’s response as they wish, by modifying the dynamics based on the function of the speed-precision compromise The operator can set the Jerk function

adjustment "level", to favor one above the other, or select a neutral situation which takes into account both requirements.
We are speaking of **expert user**, since the use of this function required in depth knowledge of various work requirements, as well a various programming tools (manual program editing, capability of reading the ISO or XISO list in the presence of Xilog Plus, knowledge of editing tools in the list itself are required).
Those who do not want to, or do not feel able to, make these changes can simply forget this function, leaving the default values set by SCM Group S.p.A., which still guarantee a great compromise between finishing and roughing requirements, without having to make adjustments every time.
This instruction must be **rigorously** entered before the G0 for piece onset to start working.
Suggested values:

60 = boring.
90 = trimming.
160 or > 160 = finishing.

From one machine to another, the typical values detected by SCM Group S.p.a. are (neutral machine – default) are:

RECORD series, range 100-121-130	>= 180 (finishing)
RECORD series, range 240	>= 180 – 200 (finishing)
RECORD series, range 260	>= 260 (finishing)
Ergon series	>= 90 (finishing)

0 = restore default values, inside a program where they were purposely changed previously (a Reset from CNC also restores default conditions).

Activating Macros for Jerk levels, by the operator

The above adjustment is carried out by recalling the following ISO instruction line in the part-program, which, by specifying the value desired for the "XXX" parameters, will provided the desired adjustment.

G120EJxxx (with this instruction line, I recall the ISO system program %10120 and request activation of Jerk)

where "XXX" = Jerk adjustment value (minimum value to be set =60)

Note. Only tests and experience will tell which is the correct value to use for each program. The program line in question may be repeated several times in the same program, each time you feel that this is necessary. At present, this optimization is not controlled by the post-processors distributed by SCM Group S.p.A., therefore, selection of the position in which to enter the programming line in the program is entirely the operator's responsibility. Use manual program editing. If unsure, leave the default value!

WARNING!

Do not change or delete:

- **program %11000 (SCM Group S.p.a. file reserved for machine configuration)**
- **program %10120 (SCM Group S.p.a. file reserved for Jerk enabling)**

JERK3D (gruppo Prisma) Set the edge adjustment level in a macro.

0 = disabled.
1 = trimming.
2 = intermediate.
3 = finishing.

See: Appendix H – Prisma Head.

NOPF45 Performs a return upstroke to a skimming dimension between successive machining (bringing the Z axis to the skimming dimension).Useful on work centers with bars and suction cups to avoid collision between tools and bars for side movements. See also: XNOP instruction.

-1 = always performs a return upstroke to a skimming dimension between successive machining.
23 = performs a return upstroke to a skimming dimension between successive machining on face 2 or face 3.
1 = performs a return upstroke to a skimming dimension between successive machining on face 4 or face 5.
0 = cancels the last condition set (with values greater than or less than 0) with no return upstroke between successive machining.

PRU Makes it possible to use the mechanical presser, if present.

0 = press up.
1 = presser low (active).

SCALE Makes it possible to automatically scale a program by inserting a multiplication factor, that is, a number by which the values of coordinates X and Y are multiplied. It must be programmed before the operating instructions and cannot be repeated.

SIZEFLD Modify the volume of the piece in Z.
Example: if dz=20 is specified in the program Header and SIZEFLD=30 is set, the successive instructions are as if the dz of the panel were 30.

STANDBY Determines a temporary work halt (that can be restores from the interruption point by pressing the **Start** key).

0 = no stop.
1 = stop without possibility of blocking the piece (useful for removing scraps).
2 = stop with the possibility of releasing the piece.

TAU Copier control which accepts the following values:

0 = disables the device.
> 0 = device gain.

Negative values are not accepted.

For the use of the SET TAU instruction with the Ergon machines, see: Appendix F - Ergon Machine.

TRACEMODE Controls generation of the trace file (see: TRACE instruction).

(default)/0 = trace file is only generated in program editing environment.
1 = trace file is only generated in execution environment.
2 = trace file is generated in both environments.

The name of the report file generated in the execution environment is taken from the program name, to which the date and time are added.
Example:

CheckPositions 2004-11-23 14-24-44.log

The report file is saved in the folder indicated in the [TRACE] section of file Xilog3.ini.
Example:

```
...
[TRACE]
reportDir=c:\TraceDir\
…
```

If the folder is not indicated in the file Xilog3.ini, the report file is saved in the same folder as the program.

TWIN (macchine Ergon) Use of the TWIN does not require you to specify in field T the list of tools for all of the heads which must work in a synchronised fashion. You can simply indicate the tool (electrospindle) or spindles (boring head) for the "master" head, that is to say, the head relative to which the program dimensions are processed.

► The value assigned to the TWIN parameter must consist of increasing figures, since the PLC, as already indicated, requires that the "master" head, with which the other heads to be synchronised are aligned, always belongs to the lowest Z unit:

Examples:

SET TWIN=1234; indicates synchronisation of units 1, 2, 3 and 4.
SET TWIN=34; indicates synchronisation of units 3 and 4.
SET TWIN=23; indicates synchronisation of units 2 and 3.

► The tool indicated in field 'T' in the instructions must refer to a head belonging to a Z unit whose number is equal to the first or last figure in the TWIN value:

Examples:

SET TWIN=1234; T=1nn or T=11nn or T=4nn or T=14nn.
SET TWIN=34; T=3nn or T=13nn or T=4nn or T=14nn.
SET TWIN=23; T=2nn or T=12nn or T=3nn or T=13nn.

The only change relates to the head to which the calculations for the X/Y/Z dimensions refer.

► By nature, this instruction must necessarily be inserted at the beginning of the profile to which it is to be applied. Therefore, for example, it is inserted before an XGIN, if present:

H DX=… DY=… DZ=… BX=… BY=… BZ=… /"Def"
SET TWIN=13
XGIN
XG0 X=… Y=… Z=… **T=305**; 300 and 100 work (tool 5 – ref. T3)
XG1 X=… Y=… Z=…
XGOUT
SET TWIN=24
XGIN
XG0 X=… Y=… Z=… T=205; 200 and 400 work (tool 5 – ref. T2)
XG1 X=… Y=… Z=…
XGOUT

► SET TWIN is modal, that is to say, its value is considered valid from the moment it is programmed until reprogramming is performed.
For example:

H DX=… DY=… DZ=… BX=… BY=… BZ=… /"Def"
SET TWIN=13
XG0 X=… Y=… Z=… **T=305**; 300 e 100 (ut. 5) operate
XG1 X=… Y=… Z=…
SET TWIN=24
XB X=… Y=… Z=… **T=1407 1408**; 1400 e 1200 (spindles 7,8) operate
XB X=… Y=… Z=… **T=1401 1402 1403**; 1400 e 1200 (spindles 1, 2, 3) operate
SET TWIN=0; Cancel TWIN

XG0 X=… Y=… Z=… **T=305**; 300 operates alone
XB X=… Y=… Z=… **T=1407 1408**; 1400 operates alone
XG0 X=… Y=… Z=… **T=306 115**; 300 (tool 6) and 100 (tool 15) operate

See: Appendix F – Ergon Machines.

USAW Modal, makes it possible to define a movement s of the blade with value -0.5mm<=s<=0.5mm, orthogonally to the programmed linear trajectory. This is only applied to machining with a blade for linear milling with the same X,Y co-ordinates as the previous linear milling.

WARNING

If the instruction SET USAW is programmed, always check that all the linear millings with blade, carried out on another linear milling with the same X and Y co-ordinates as the previous, (corresponding to all the linear millings "in place"), are programmed OUT OF PANEL DIMENSIONS !

ZFAST Add another intermediate positioning to the programmed Z position, for all of the machinings programmed after this instruction.

Example:

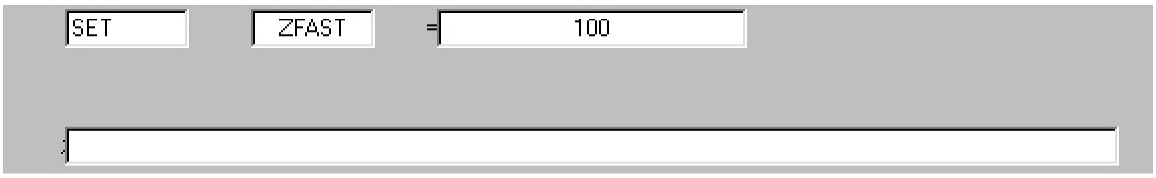
if **rapid/slow axis Z = 20** (in the **GENDATA** parameters) the machine goes in RAPID up to z=20 and then goes in SLOW to the machining position.

If I set **ZFAST** = 100 it stops at 100 in RAPID then still goes in RAPID to 20 and then goes in SLOW to the machining position.

If I set D = 100 it stops at 100 in RAPID then still goes in SLOW to the machining position.

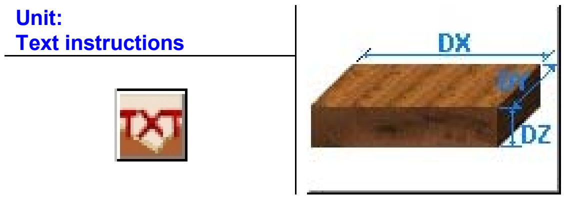
If I set **ZFAST**=100 and D = 80 it stops at 100 in RAPID then still goes in RAPID to 80 and then in SLOW to the machining position.

Example:



Change Reference - REF

REF instructions change the characteristics of the panel defined in the program header, therefore subsequent machining refers to the new characteristics.



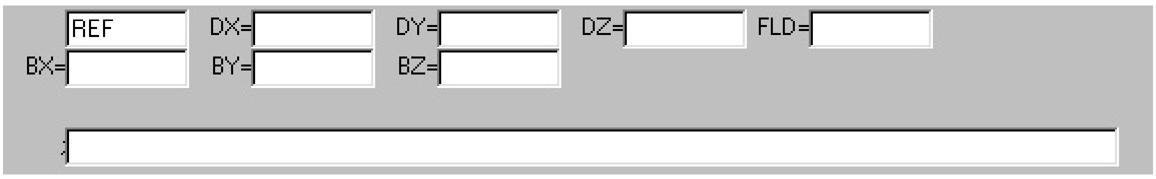
Parameters:

DX Dimension in X of the panel.
DY Dimension in Y of the panel.
DZ Dimension in Z of the panel.
FLD Work area to which to be associated (A, B, C, D, AB, BA, CD, DC, AD, DA).
BX Distance in X between the panel zero and the field zero.
BY Distance in Y between the panel zero and the field zero.
BZ Dimension in Z of a shim positioned under the panel.

All parameters are optional, if present, these replace the corresponding ones programmed in the program header.
An REF without parameters resets all the characteristics specified in the program header.

WARNING
The panel dimension in Z is used to optimize translation of head between machining on different faces. For this reason the DZ field must be compiled with the effective maximum overall dimensions of the piece in Z, otherwise the heads and the piece may collide.

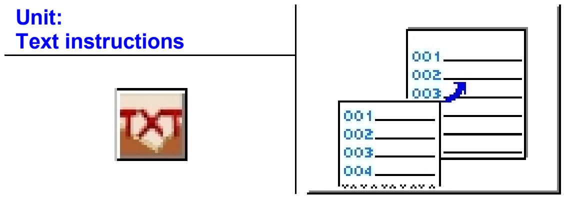
Machining carried out following an REF instruction start from the stop in the work area specified in the REF.



5.2.3 Instructions for Managing Subprograms

Call Sub-Program - S

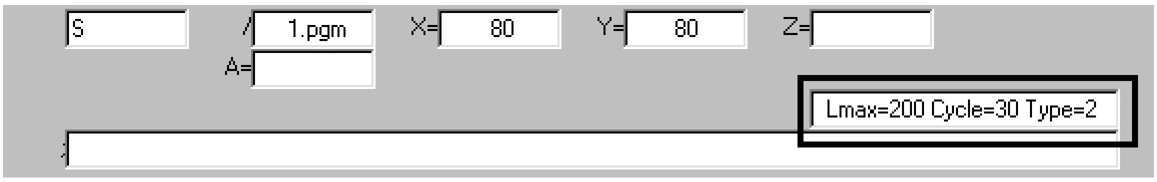
Recalls a normal program like a subprogram.



Parameters:

/	Name of subprogram.
X	Origin in X of subprogram.
Y	Origin in Y of subprogram.
Z	Origin in Z of subprogram.
A	Angle of rotation of subprogram (front machine origin: positive in counter-clockwise direction; rear machine origin: positive in clockwise direction).

In the empty box to the right, a list of parameters can be entered; the value of the latter must be modified within the subroutine/macro called (see: chapter 6.4 - Passing Parameters to Subprograms and Macro).



For unprogrammed X, Y, Z the current values pertain (for Z the last work quota pertains)

An open program stores subprograms in its memory: in order for changes to graphics in the subprograms to become effective the main program must be closed down and reopened.

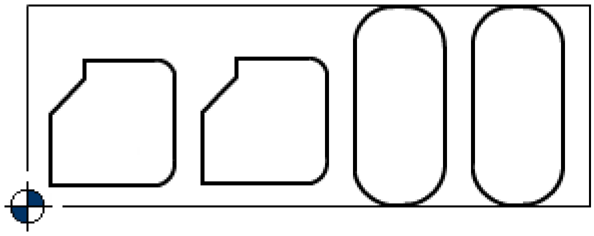
Caution! The S instruction is not suitable for retrieving programs containing angled surfaces (PL instruction), since the translation is only applied to machining and not to the origin of the angled surface. Use the SO instruction for this.

Warning! If the subprogram is dependent on a file of ambient variables, in order for the main program to function properly (the program which calls up the file), the same file of ambient variables in the subprogram must also be programmed in the heading of the main program.

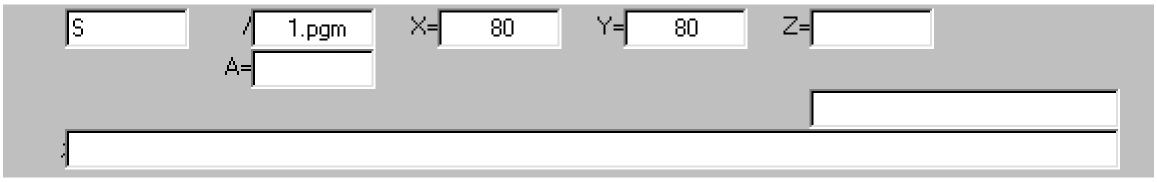
Example:

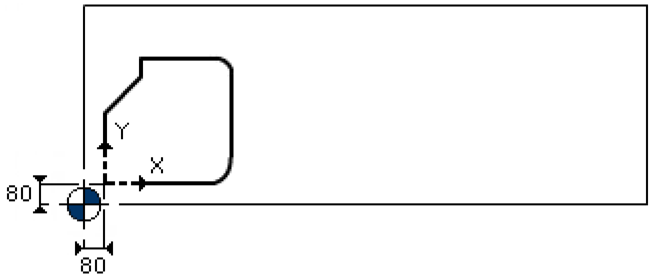
►*Front machine origin*

Obtaining four work pieces (sub-programs) from a large panel, as illustrated:

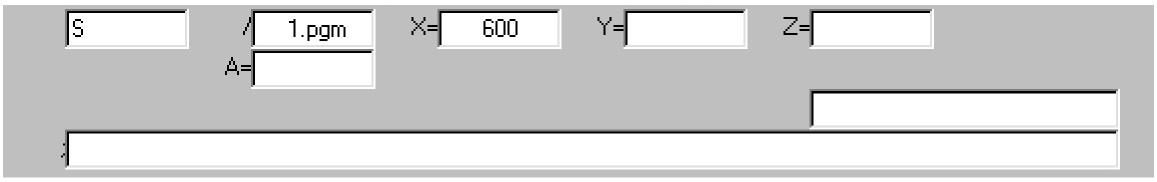


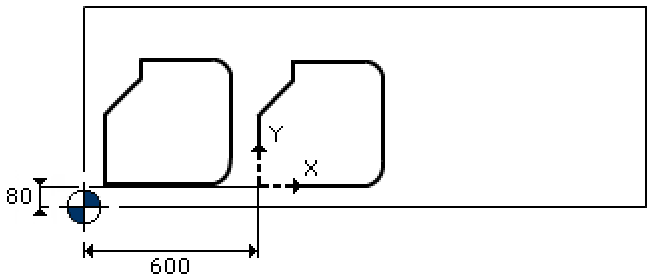
Proceed as follows:
Create a new program in the programs editors, fill in the Header with the panel data.
At this point we will recall the first subprogram with instruction S.



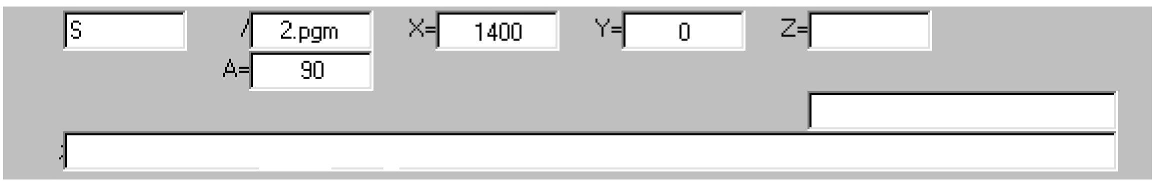


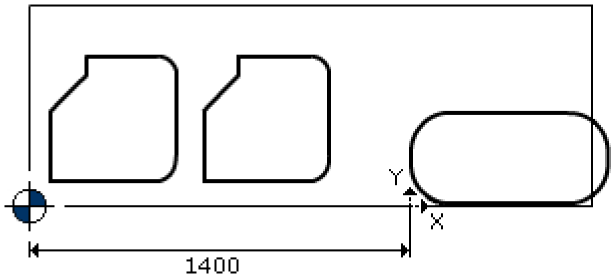
We will then recall the first subprogram a second time, inserting it into the second position.

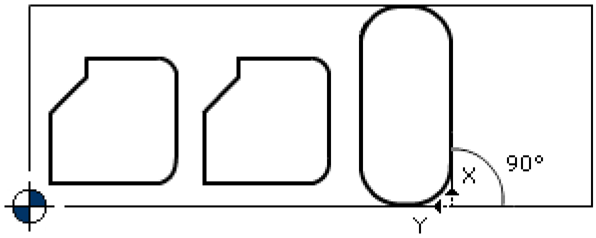




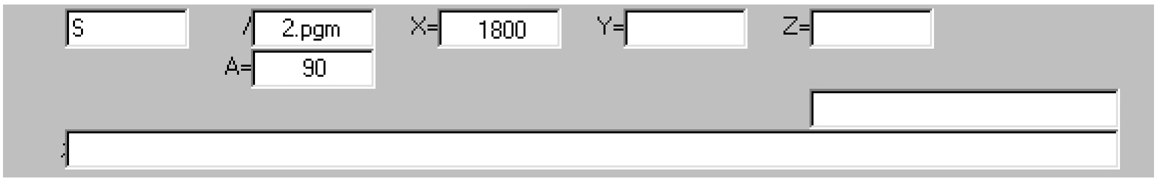
We will recall the second subprogram and insert it with origin like in the figure, rotating it then by 90°.

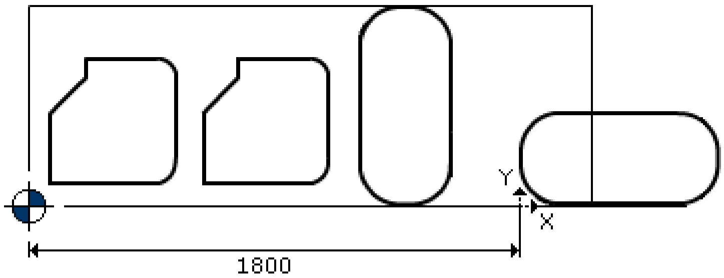


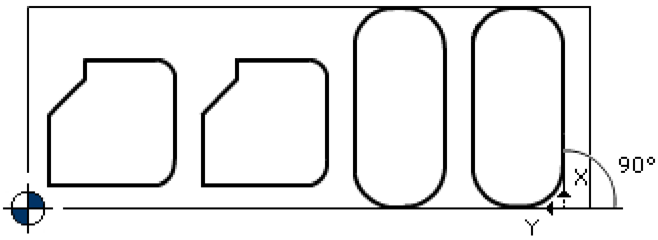




Repeat the operation for inserting the last subprogram.

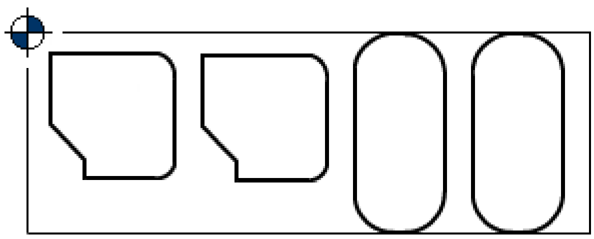




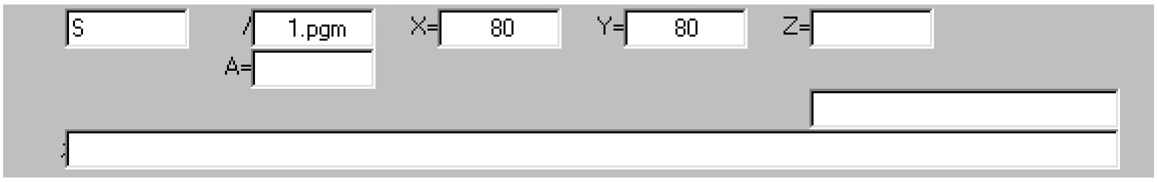


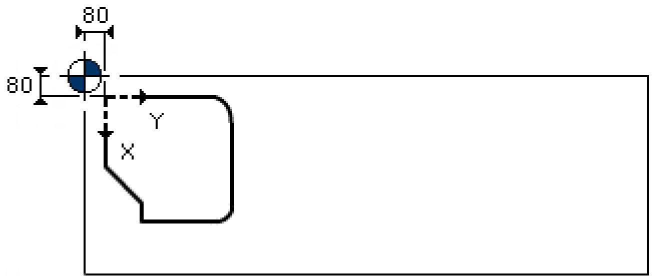
►*Rear machine origin*

To obtain four work pieces (sub-programs) from a large panel, as illustrated:

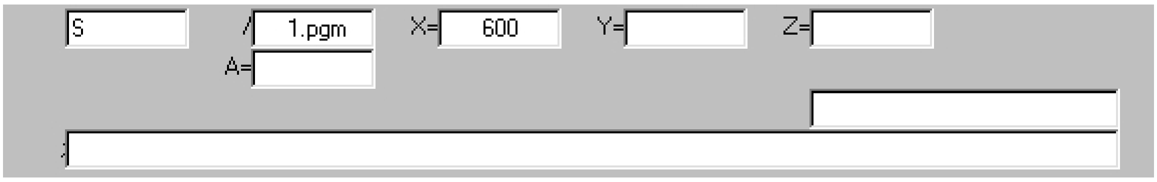


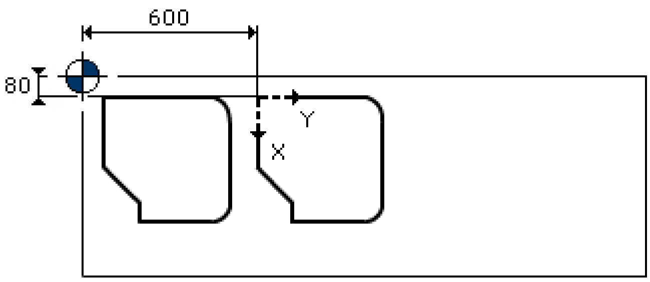
Proceed as follows:
Create a new program in the programs editor, by filling in the Header with the panel data.
At this point recall the first subprogram with instruction S.



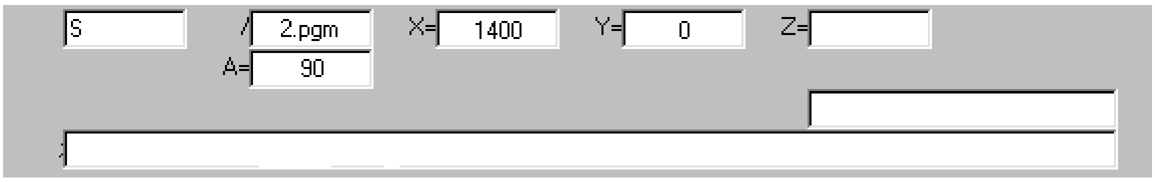


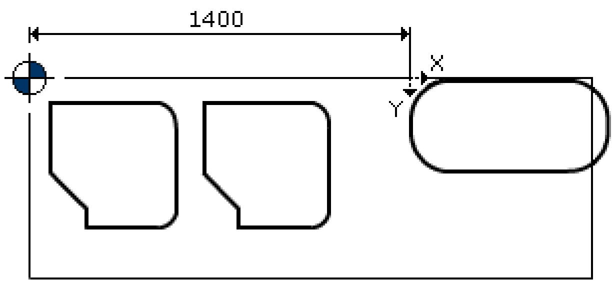
Then recall the first subprogram a second time, inserting it into the second position.

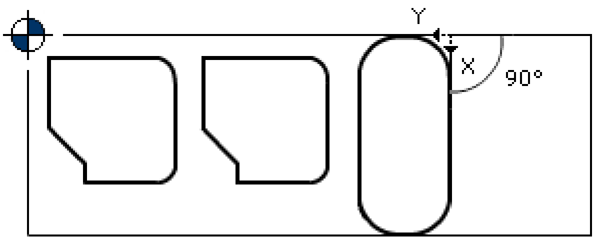




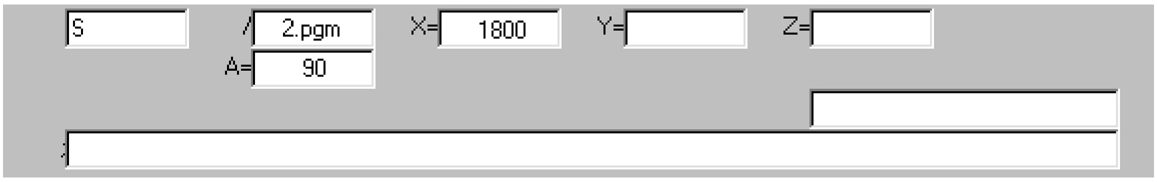
Recall the second subprogram and insert it with the origin, like in the figure, rotating it for 90°.

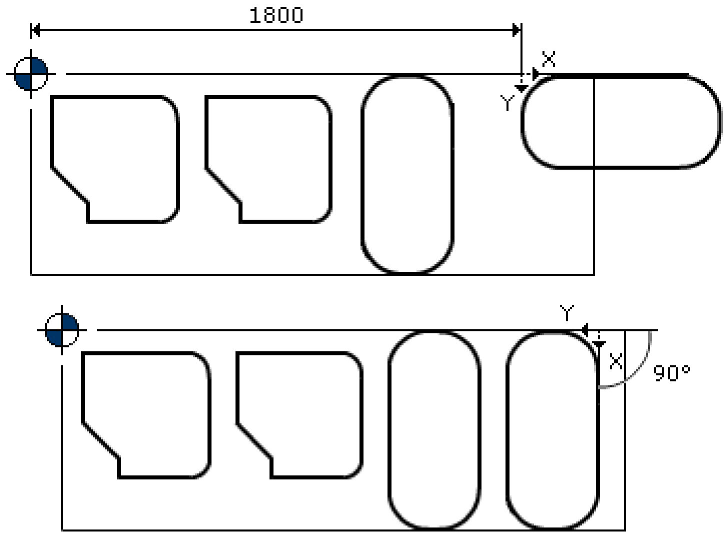






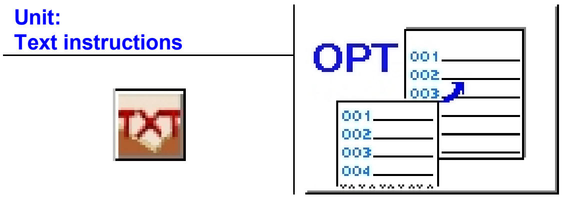
Repeat the operation for inserting the last subprogram.





Optimized Sub-Program - SO

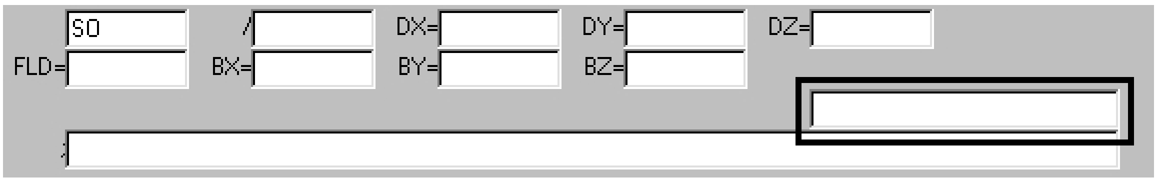
The instructions SO enable the retrieval of a subprogram associating it to a work zone, while the work is being carried out the real dimensions of the piece are maintained and therefore the position of the face.



Parameters:

/	Name of subprogram.
DX	Dimension in X of the panel.
DY	Dimension in Y of the panel.
DZ	Dimension in Z of the panel.
FLD	Work area to which it must be associated (A, B, C, D, AB, BA, CD, DC, AD, DA).
BX	Distance in X between the panel zero and the field zero.
BY	Distance in Y between the panel zero and the field zero.
BZ	Dimension in Z of a shim positioned under the panel.

In the empty box to the right, a list of parameters can be entered; the value of the latter must be modified within the subroutine/macro called (see: chapter 6.4 - Passing Parameters to Subprograms and Macro).



At the exclusion of the subprogram name, all the other parameters are optional, if present the optional parameters replace those programs corresponding to the subprogram header.

- SO instructions can only be recalled from within the main program. SO instructions define the characteristics of a panel which becomes a sub-panel of the main panel defined in the main program header. The latter defines the total machining characteristics (including locking pieces), and dimensions DX, DY and DZ, these specifications must include the sub-panels.
- The panel dimension in Z is used to optimize translation of head between machining on different faces. For this reason the DZ field must be compiled with the effective maximum overall dimensions of the piece in Z, otherwise the heads and the piece may collide.
- To retrieve a program containing an angled surface (PL/XPL instructions), assign the X, Y, Z values of the angled surface origin to the parameters BX, BY, BZ.
- SO machining starts from the stop in the SO specified work area.
- For programs containing SO instructions, it may be requested that the tool change be optimized (see: chapter 7 – Programs Optimizer); the optimizations consists in reordering the machining in order to minimize the number of tool changes between one machining

an another. The optimization of the tool change only has an effect on the SO instructions in the main program.

Warning! If the subprogram is dependent on a file of ambient variables, in order for the main program to function properly (the program which calls up the file), the same file of ambient variables in the subprogram must also be programmed in the heading of the main program.

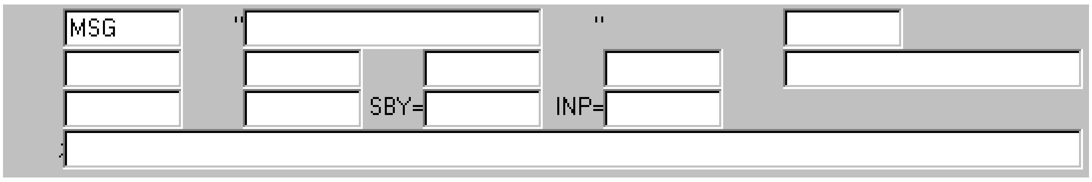
5.2.4 Sending Messages to the Operator

Print Message - MSG

The messages to the operator contain information sent to the operator while the piece is being machined.

Parameters:

“<string>”	Message.
(empty boxes)	Inserting parameters.
SBY	Type of standby to be carried out.
INP	Name of variable for input value.



An MSG without parameters cancels the previous message.
A request to input data can be associated to every message (parameter INP).
After a message is sent the piece program may carry on or stop waiting for the operator to input required data and/or press start cycle pushbutton.
The standby indication (parameter SBY at 1 or 2) stops the piece program waiting for the cycle start pushbutton to be pressed. When data is requested the piece program stops simulating the indication SBY=1.
The instructions MSG are treated individually and displayed with logic according to the type:

MSG without INPUT
The message is displayed in the message window; when in standby mode the message is cancelled by pressing the **Start** button, otherwise the message is cancelled when a new MSG arrives, even without messages; in both cases the cancellation also occurs when a Reset is accepted.

MSG with INPUT
The message is displayed in the messages window and an icon is displayed to signal the need to insert data (**CONTROLS/INPUT OPERATOR MESSAGE** menu). The message is cancelled when the **Start** button is pressed and when a Reset is accepted.

Example:

…
MSG “Wait completion of unload cycle”
…
MSG
…
MSG “Remove rejects and press start-cycle!” SBY=1
…
L ADX = 0
L ASX = 90

```
MSG “Input right cutting angle” INP=ADX
MSG “ Input left cutting angle” ” INP=ASX
IF ADX > 90 THEN
…
FI
…
L Min = 0
L Max = 4
L Hood = 0
MSG “Position of vacuum hood (from ?d to ?d)” Min Max INP=Hood
IF Hood >=Min AND Hood <= Max THEN
ISO “M?d” 110+ Hood
ELSE
PRINT “Position Hood=?d error!” Hood
FI
```

5.2.5 Programming the Motorised Table

PB

The PB instruction makes it possible to program the motorised table.

Parameters:

B Bar number. This can be left unprogrammed: if it is programmed, it indicates the bar to be moved; if it is not programmed, Xilog Plus will choose which bar to move to the value indicated in field X.

X Position in X at which the bar is to be positioned.

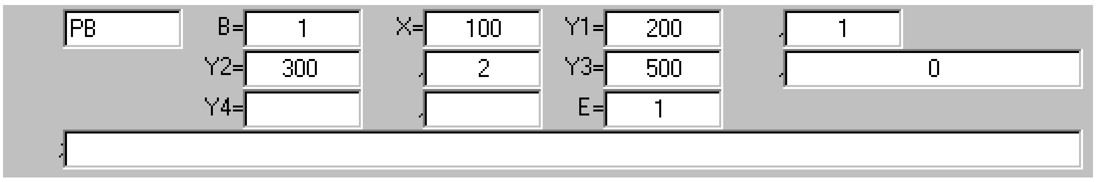
Y1, Y2… The values Y1, Y2 etc. indicate the position in Y of the first, second, etc. clamp or suction cup. For each value, the condition of the clamp or suction cup must be indicated in the next empty box.

0 = clamp closed.
1 = clamp open.
2 = clamp closet in blocking position (with piece blocked).

E Indicates the type of operation being executed on the work table:

0 = ignored.
1 = table fitting operations (only one table in a piece program).
2 = piece blocking operations (only one in a piece program).
3 = PB block which must be executed (optional value if the PB instruction is followed by a movement instruction).

All of the PB E=3 blocks contiguous to the PB E=2 block (blocking) and contiguous to each other are associated to the piece blocking phase. the contiguity cannot be interrupted by any instructions, not even non-operative ones. If a PB E=3 block can be assigned to both a blocking phase as well as to a releasing phase, it is assigned to the blocking phase.



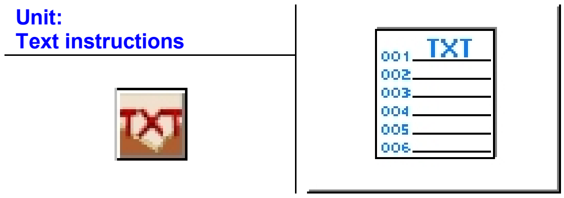
Programming the bars does not require a specific order. If the bar number is specified, this number may never be modified. If, instead, the clamp number is specified and there is an error in programming the Y co-ordinates, the clamps are automatically inverted (with three clamps, the only inversions possible are: clamp 1 – clamp 2 or clamp 2 – clamp 3).

The information relative to the table fitting operations is important because fitting out a table on a physical area may occur simultaneously with the execution of other operations, for example, the machining of a piece, on another physical area. Field E therefore makes it possible to anticipate table fitting for the next machining operation.

5.2.6 Programming Help Instructions

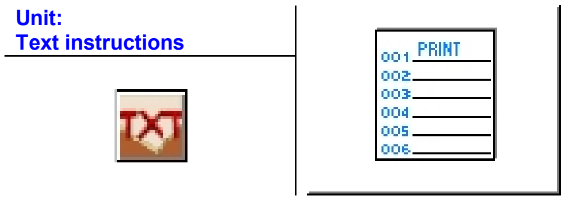
Comment - ;

*A comment is a line of text that begins with the character **;** or with the character *****. Comments beginning with **;** can also be placed at the end of instruction lines.*



Display Message - PRINT

Allows a diagnostic message to be displayed; after display any further interpretation of the program is blocked.



The instruction consists of a string between double apexes, sometimes followed by a list of parameters separated by at least one space.
The parameters are associated in succession to print formats inserted in the message; therefore, the programmer must ensure that a format for each parameter is inserted in the program
A print format starts with a question mark (?); when «PRINT» meets the first format (from left to right) it converts the value of the first parameter after the message and inserts it into the message in place of the format; the second format determines the conversion of the second parameter and so on.
Any parameters in excess of the format are ignored; any formats in excess of the parameters are ignored too.
A print format must have one of the following forms, each of which is assigned a type of display of the corresponding parameter:

?x a hexadecimal number between 0000 and FFFF;
?d a whole decimal number between -32768 and 32767;
?u a whole decimal number between 0 and 65535;
?D a whole decimal number between -2147483648 and 2147483647;
?U a whole decimal number between 0 and 4294967295;
?f a number between 1.7E-308 and 1.7E+308 displayed in the form [-] dddd.dddd;
?e a number between 1.7E-308 and 1.7E+308 displayed in the form [-] d.ddd and [+/-]

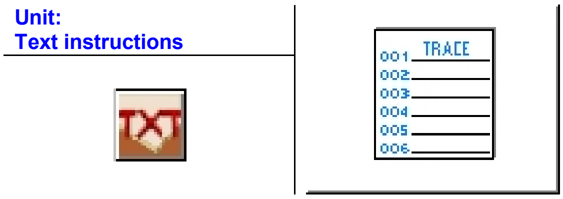
	ddd;
?g	a number between 1.7E-308 and 1.7E+308 displayed in the format ?f or ?e which gives the most compact display;
?c	character;
?s	string of characters.

If a question mark is followed by a character that does not identify any print format, that character is displayed; and so to display a question mark the form ?? must be used.

Example:

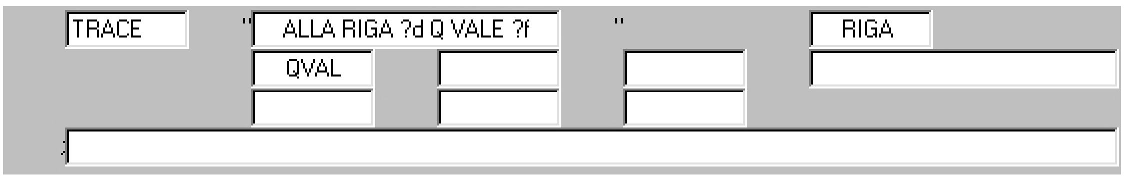
Store Message - TRACE

Allows a message to be written in a dedicated file; after writing it, the program interpretation continues as normal.



The instruction consists of a string between double apexes, sometimes followed by a list of parameters separated by at least one space.
The parameters are associated in the position mode to print formats in the message. The print formats are the same as those for the PRINT instruction.
The TRACE instruction is only effective in the editor when the piece design has been completed; therefore, it has no effect during program memorization or during the various automatic mode phases.

Example:



5.3 Extended Instructions (Graphic)

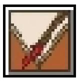
5.3.1 Operating Instructions

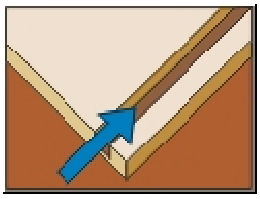
5.3.1.1 General Functions

Automatic Profile Entry - XGIN

Defines a line or an arc of a circle tangent to the profile at the entry point. Works if programmed before a begin profile instruction (XG0).

Unit:
Operating on profile





Basic parameters:

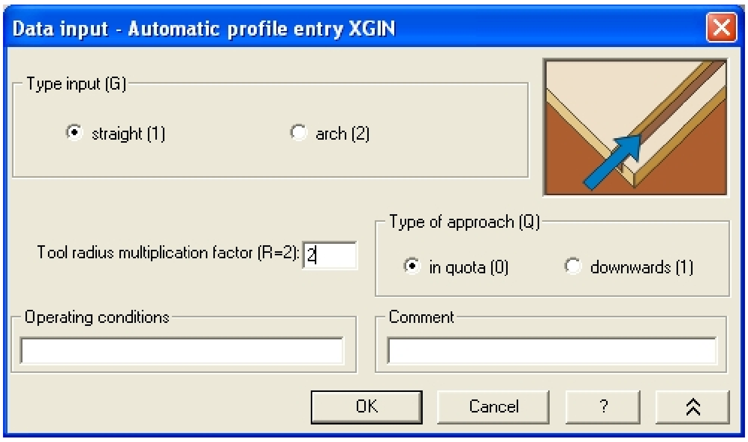
G Type of entry: 1=straight, 2=arc (the arc is only valid when the tool radius compensation is enabled). Entry is enabled with blade in straight tangent.

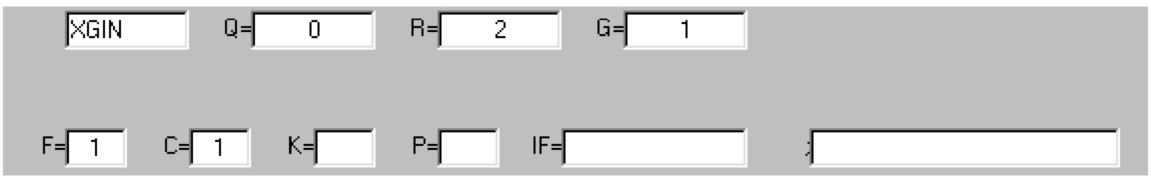
Extended parameters:

R Tool radius multiplication factor (default=2).
Q Type of approach: 0=in quota, 1=in descent.

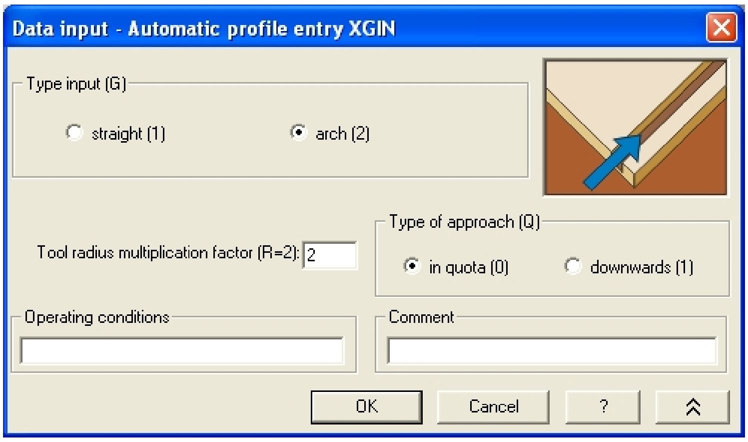
Examples: methods for performing an automatic entry.

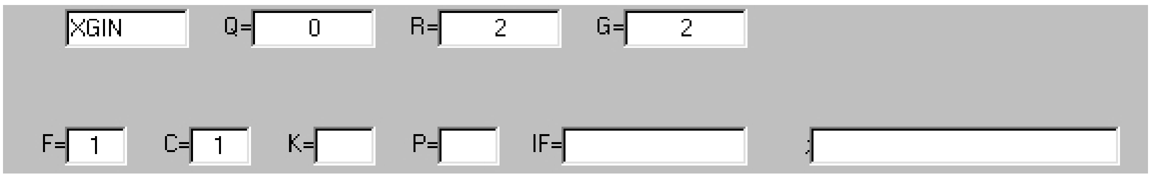
1) Vertical straight entry.

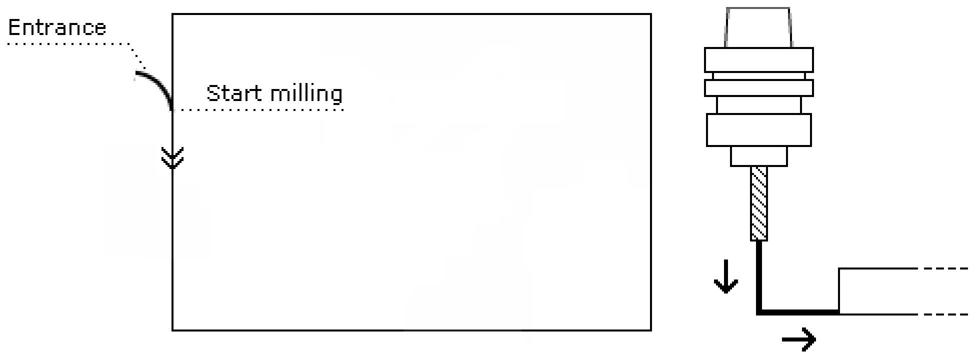




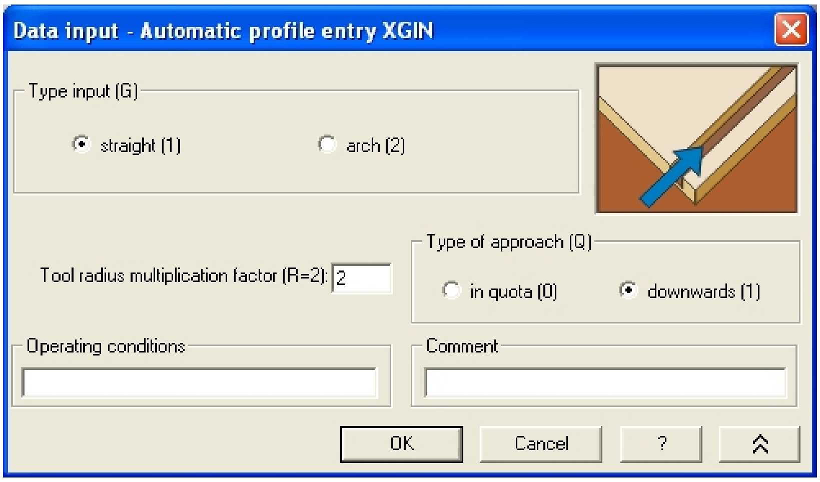
2) Vertical arc entry.

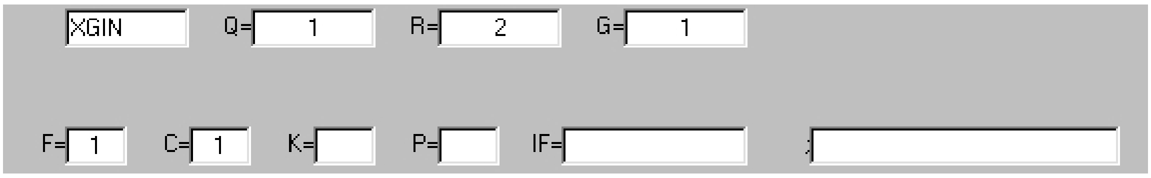


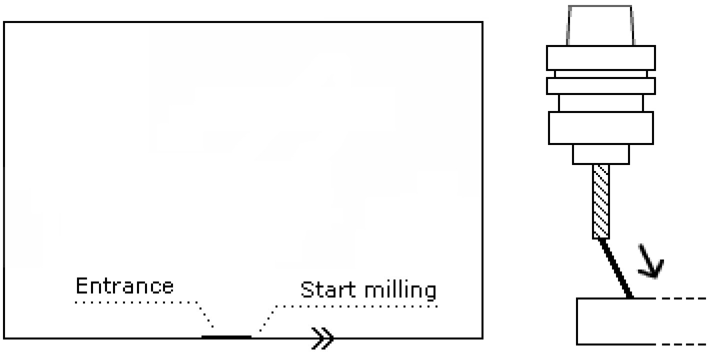




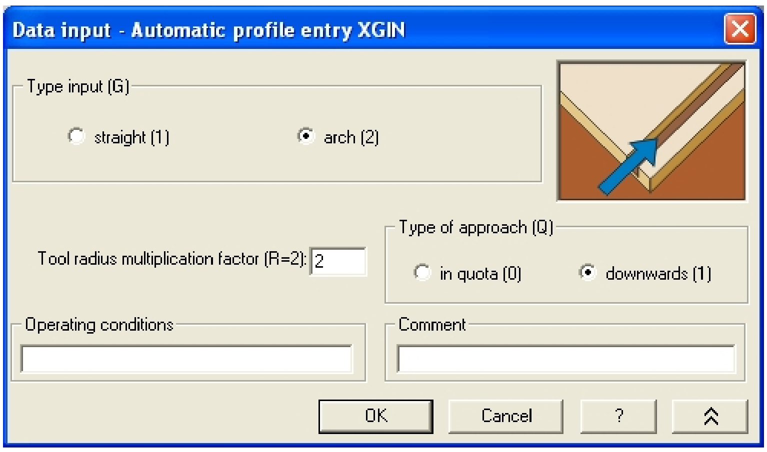
3) Slanting straight entry.

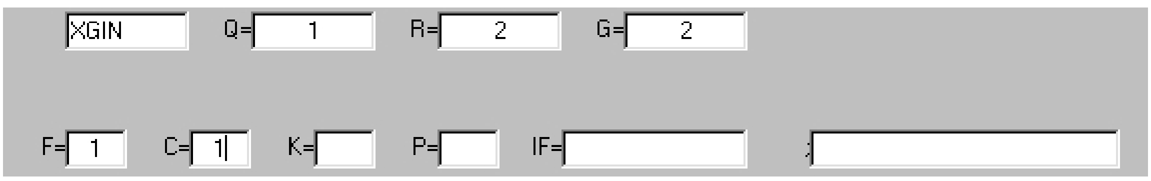


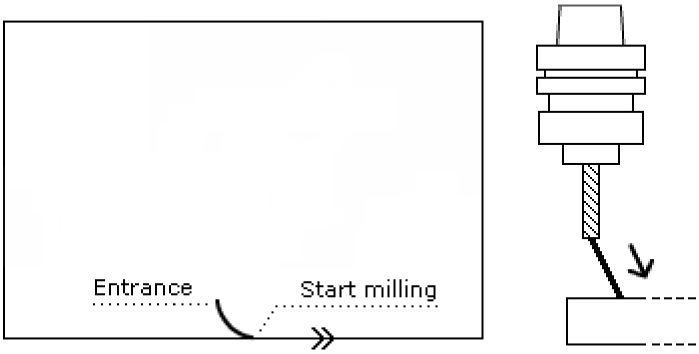




4) Slanting arc entry.







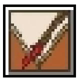
WARNING

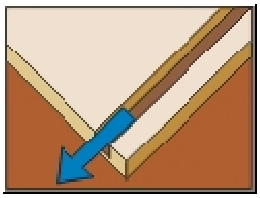
The automatic entry is not enabled if: C=3 or C=31 or C=32. If C=0 and G=2, the entry direction of the arc is determined by the sign R (if positive – arc clockwise; if negative = arc counter-clockwise

Automatic Profile Exit - XGOUT

Defines a line or an arc of a circle tangent to the profile at the exit point. It has effect if programmed after the last instruction of the profile.

Unit:
Operating on profile





Basic parameters:

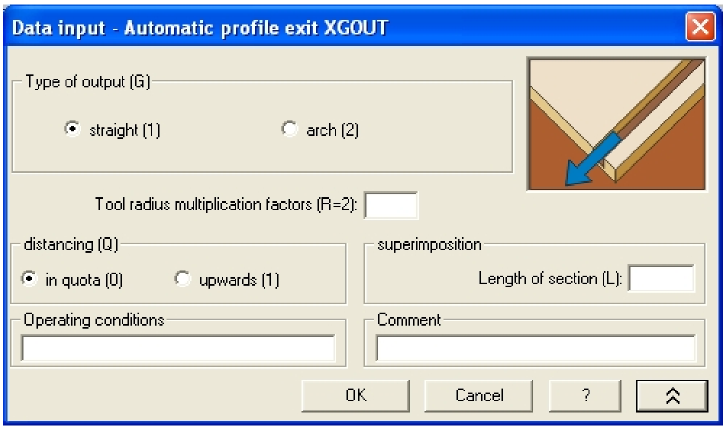
G Type of exit: 1=straight, 2=arc (the arc is only valid if the tool radius compensation is enabled). Entry with blade in tangent line (G=1) is enabled.

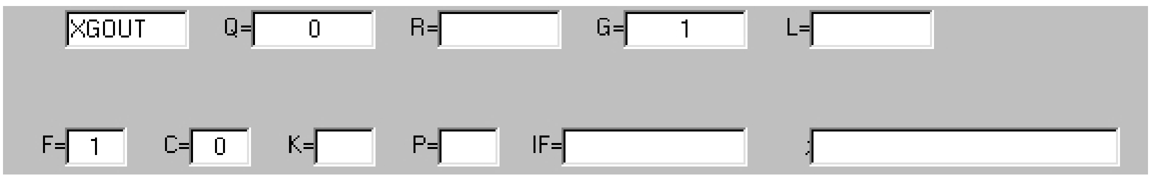
Extended parameters:

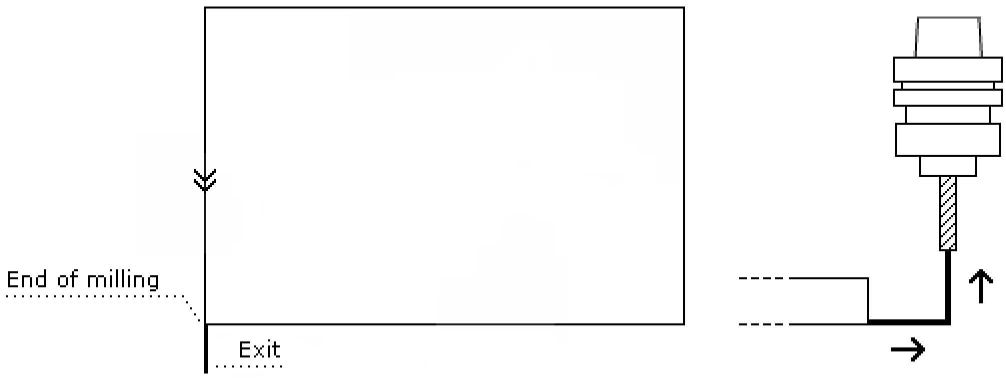
R Tool radius multiplication factor (default=2).
Q Type of distancing: 0=in quota, 1=uphill.
L Overlapping on profile. If the value of L is set to -1, overlapping operation occurs all along the last profile section; if L is set to -2, overlapping occurs al along the last profile section and also the panel exits perpendicularly (as per instruction N) ignoring the value of parameter Q.

Examples: methods for performing an automatic exit.

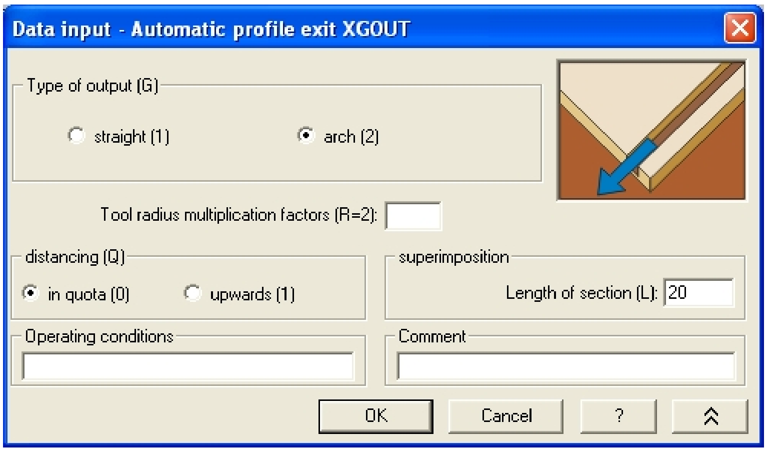
1) Vertical straight exit.

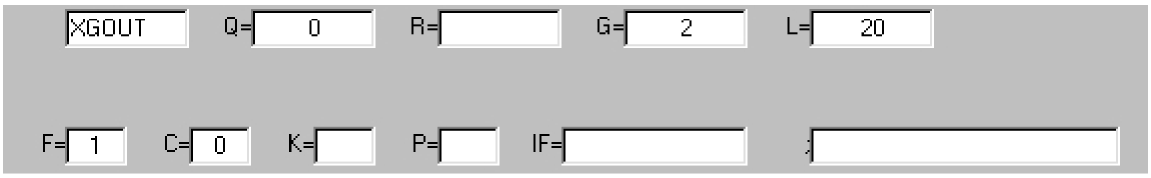


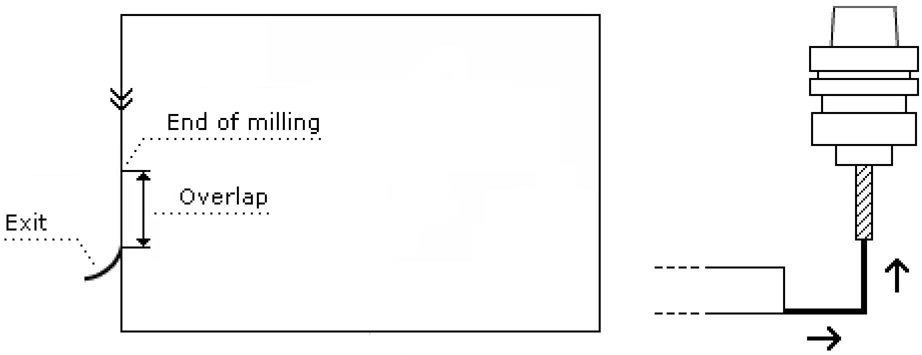




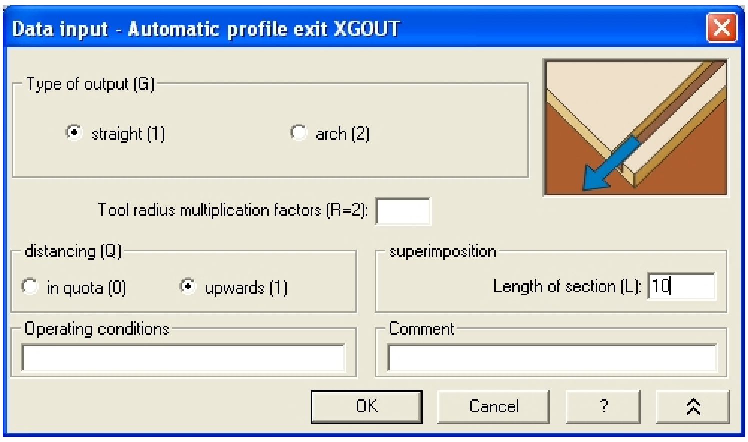
2) Vertical arc exit.

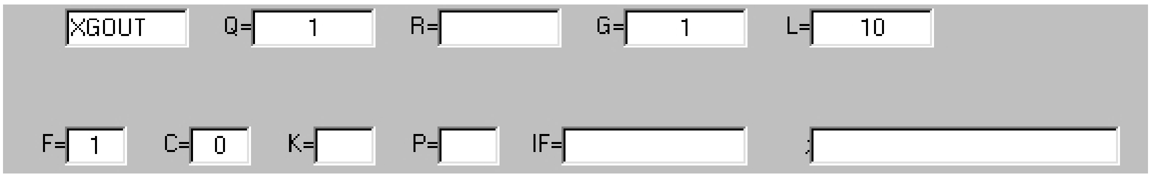


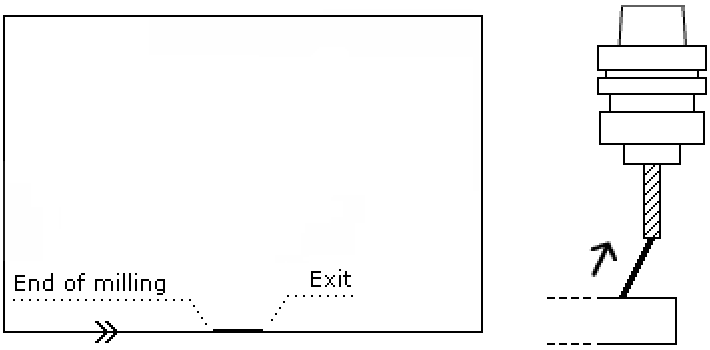




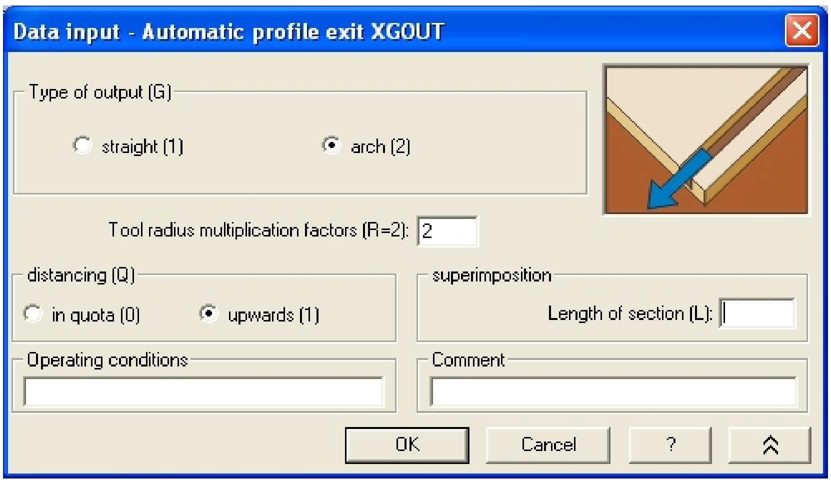
3) Slanted straight exit.

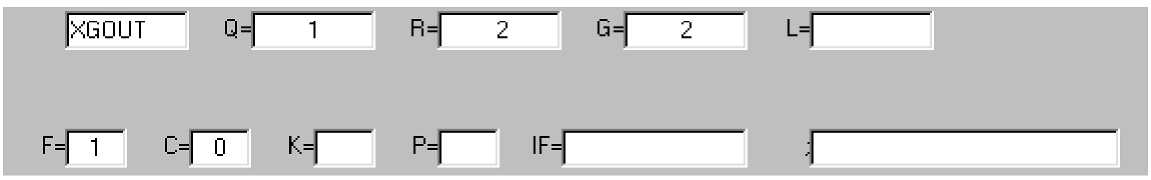


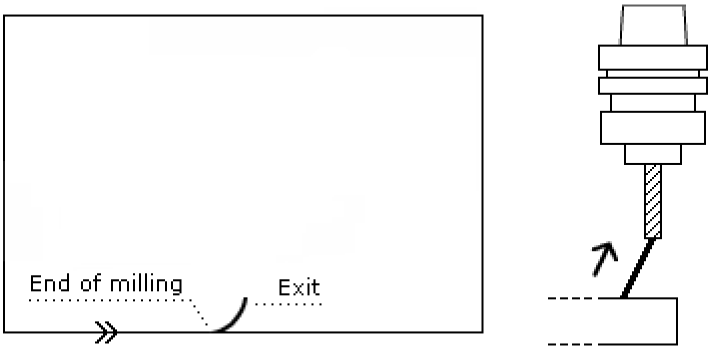




4) Slanted arc exit.







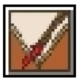
WARNING

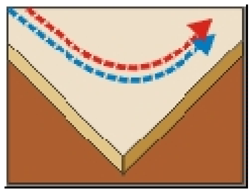
Automatic exit is not enabled if: C=3 or C=31 or C=32.

Profile Repetition - XGREP

Repeats a profile. The XGREP instruction is not influenced by the SX and SY instructions.

Unit:
Operating on profile



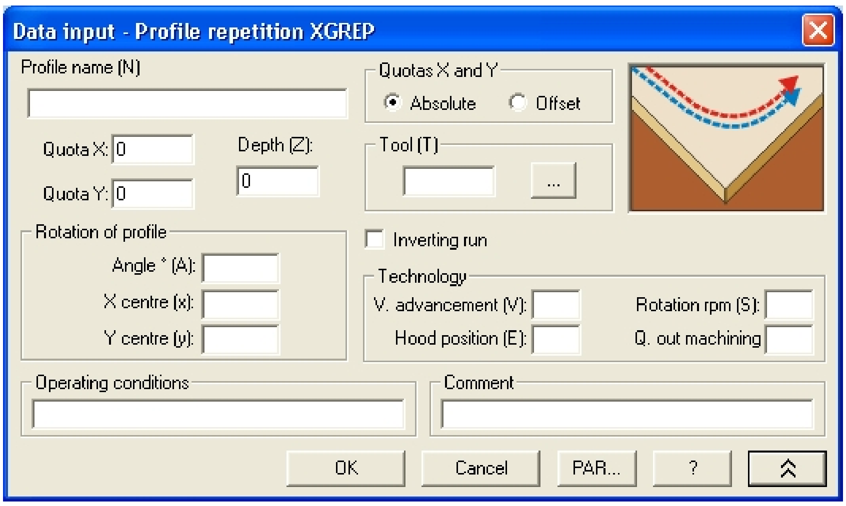


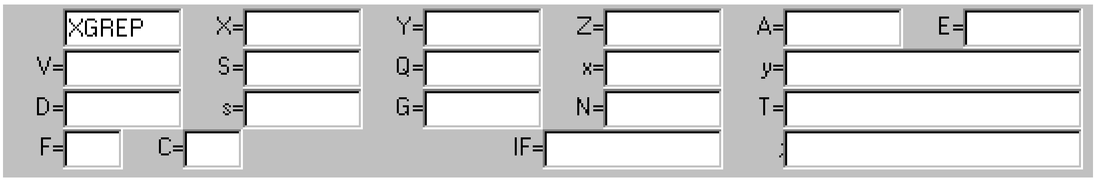
Basic parameters:

N Name of profile (see field N of instruction XG0).
X Coordinate X of the beginning point of the profile, or offset X of the profile (see Q).
Y Coordinate Y of the beginning point of the profile, or offset Y of the profile (see Q).
Z Offset of profile in depth Z.
Q 0 = X,Y absolute values; 1 = offset.
T Tool.

Extended parameters:

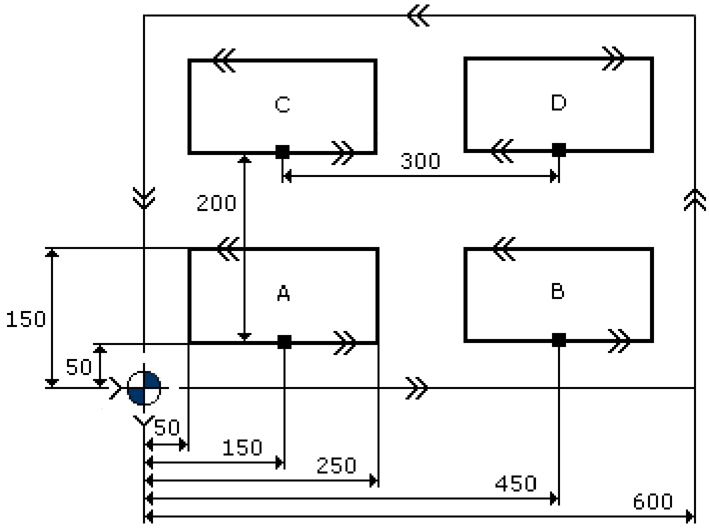
A Angle of rotation (front machine origin: positive in counter-clockwise direction; rear machine origin: positive in clockwise direction).
x Quota x from the center of rotation.
y Quota y from the center of rotation.
G Reverse direction (0=NO,1=YES).
V Milling speed.
S Rotation speed of tool.
E Position of vacuum hood (see: Appendix D).
D Quota out machining.
s Quota of over-metal (only text instruction).



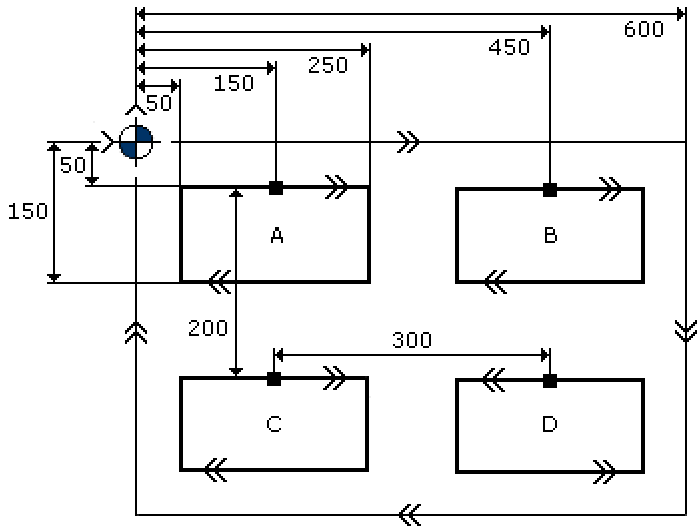


The following example contains the panel trimming in the figure with a thickness of 30mm with tool called E1, the internal trimming of rectangle A with the tool called E2, the repeat of profile A in position B with absolute coordinates, the repeat of profile A in position C with offset, the repeat of profile A in position D with offset and an inverted distance with a tool called E3.

►*Front machine origin*



►*Rear machine origin*



H DX=600 DY=400 DZ=30 -A C=0 T=0 R=1 *MM /"ANDREA" V10 ; header

;panel external trimming

XGIN Q=0 R=2 G=1 C=… ((front machine origin: C=1; rear machine origin: C=2); automatic entry in the profile with straight line in quota.
XG0 X=0 Y=0 Z=-31 T=101 V=4000 S=18000 E=1 D=20 ; start of trimming profile
XL2P X=600
XL2P Y=400
XL2P X=0
XL2P Y=0
XGOUT Q=0 R=2 G=1 C=0 ;automatic exit from profile with straight line upwards

; start machining rectangle A

XGIN Q=1 R=2 G=1 F=1 C=0; entry with downward straight line no radius compensation
XG0 X=150 Y=50 Z=… E=1 V=5 S=18000 D=20 N=”TEST” T=101 start profile name “test”
XL2P X=250
XL2P Y=150
XL2P X=50
XL2P Y=50
XL2P X=150
XGOUT Q=1 R=2 G=1 L=10; exit from profile with slanted line

; repetition of rectangle A called “test” in position B with absolute quota
XGREP X=450 Y=50 Q=0 G=0 N=”TEST”; same tool and same machining quota

; repetition of rectangle A called “test” in position C with offset quotas

XGREP Y=200 Q=1 G=0 N=”TEST”; quotas referred to the starting point of the first rectangle

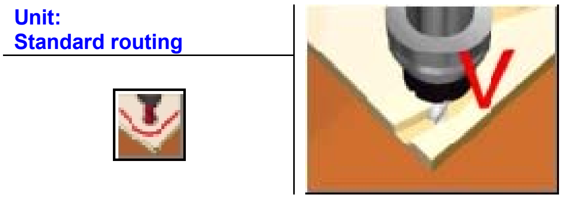
; petition of rectangle A called “test” in position D with offset quotas

XGREP X=300 Y=200 Q=1 G=1 N=”TEST” T=103; tool E3 path inversion

XN X=0 Y=0 T=101 F=1; stop spindle at machine zero and load tool E1

Speed Change - XGSET

The instruction XGSET describes some new characteristics of a profile retrieved with next XGREP.



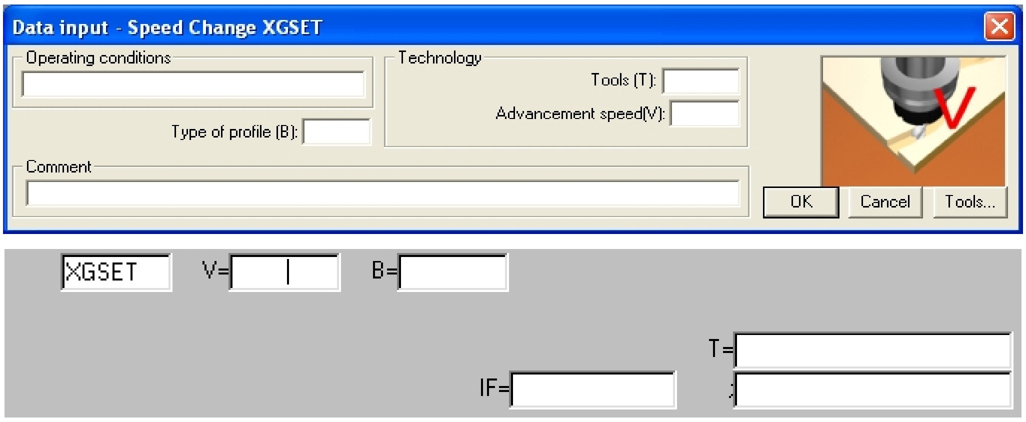
Parameters:

V Approach speed.
T Tool.
B Type of profile. The type of profile B associates the characteristics of GSET to the tool or process active at the moment of interpreting the geometry.

B=1 Milling (with milling unit “F” type tool)
B=2 Cut with blade (with disk “D” type tool)

All instruction parameters are optional.

The XGSET instruction must be inserted before any G movement instruction; the G movement instruction is influenced by an XGSET and takes on the new characteristics during a repetition of the profile with XGREP.
XGSET instructions are only enabled if the specified tool is equal to that specified in XGREP or if the identifier of the type of profile is represented by the active tool during the repetition of the profile.



Examples:

1. XGSET has no effect when the specified tool in XGREP is not 101

….
XG0 X..Y.. Z.. V=1 S E T=101 N=”Prof”
XG1 X.. Y.. Z.. V=5
XGSET V=10 T=101

```
XG1 X.. Y.. Z.. V=6
….
XGREP V=2 T=102 N=”Prof”
….
```

2. XGSET is significant when the specified tool in XGREP coincides with that of XGSET

```
….
XG0 X.. Y.. Z.. V=1 S E T=101 N=”Prof”
XG1 X.. Y.. Z.. V=5
XGSET V=10 T=102
XG1 X.. Y.. Z.. V=6
….
XGREP V=2 T=102 N=”Prof”
….
```

3. XGSET only has an effect if the tool E2 is a cutter

```
….

XG0 X.. Y.. Z.. V=1 S E T=101 N=”Prof”
XG1 X.. Y.. Z.. V=5
XGSET V=10 B=1
XG1 X.. Y.. Z.. V=6
….
XGREP V=2 T=102 N=”Prof”
….
```

4. XGSET only has an effect if the tool E2 is a cutter (due to presence of B=1)

```
….
XG0 X.. Y.. Z.. V=1 S E T=101 N=”Prof”
XG1 X.. Y.. Z.. V=5
XGSET V=10 T=101 B=1
XG1 X.. Y.. Z.. V=6
….
XGREP V=2 T=102 N=”Prof”
….
```

Since the profile can be repeated more than once, more than one XGSET can be specified before the XG movement instruction. The repetition underway determines the XGSET associated to instruction XG. If XGSET is redundant, the last one is taken into consideration.

5.

```
….
XG0 X.. Y.. Z.. V=1 S E T=101 D N=”Prof”
XG1 X.. Y.. Z.. V=5
XGSET V=10 T=102
```

```
XGSET V=15 T=103
XG1 X.. Y.. Z.. V=6
….
XGREP V=2 T=102 N=”Prof”        ;has an effect only if GSET with T=102
XGREP V=2 T=103 N=”Prof”        ;has an effect only if GSET with T=103
….
```

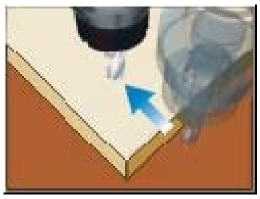
6.

```
….

XG0 X.. Y.. Z.. V=1 S E T=101 D N=”Prof”
XG1 X.. Y.. Z.. V=5
XGSET V=10 T=102
XGSET V=15 T=103
XGSET V=20 B=1
XG1 X.. Y.. Z.. V=6
….
XGREP V=2 T=102 N=”Prof”        ; ;if E2 and E3 are cutters the instruction XGSET
XGREP V=2 T=103 N=”Prof”        ; with B=1 invalidates the previous XGSETs
….
```

Null Operation - XN

Turns the rotation off and moves the electro-spindles to their idle position.

Unit:
Standard routing



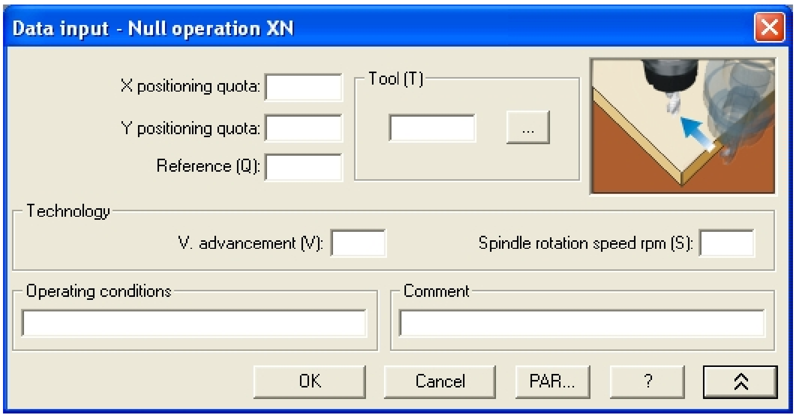
Basic parameters:

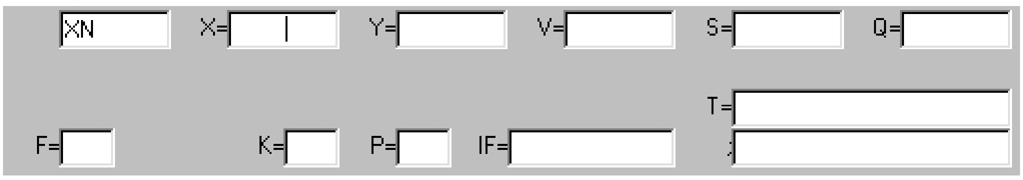
X Position of head on axis X.
Y Position of head on axis Y.
Q Meaning of field Q: if not programmed or if the value is 0, quotas X and Y are considered referring to machine zero; if programmed on 1, quotas X and Y are considered referring to panel zero.
T Number of an external tool.

Extended parameters:

V Movement speed.
S Rotation speed of tool Meaning of field S: if not programmed or the value is 0 the spindles will switch of; if 1 is programmed the spindles will remain on.

Values X and Y are referred to machine zero. The tool indicated in field T is collected from the tool magazine.

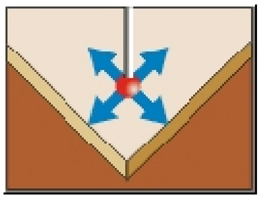




Panel Tracing - XTA

Tracing is used to detect any irregularity or roughness in the panels and therefore enables them to be corrected

Unit:
Main functions



Parameters:

X Coordinate X of the point to feel referring to the current side (from 1 to 5).
Y Coordinate Y of the point to feel referring to the current side (from 1 to 5).
Q Defines the type of feeling.

(default)	The head moves up to the Z + limit switch.
0	The head moves up to the Z + limit switch.
1	The head remains at the feeling Z dimension.
2	The head moves up to the dimension for brushing over the workpiece.

G Controls the head return upstroke after feeling.

(default)	Feeling the panel faces.
0	Feeling the panel faces.
1	Feeling a point on the panel.

T Tool.

After detecting the difference (D) between the real panel dimensions and the theoretical dimensions, the feeler changes the numerical control in relation to the side on which it was carried out:

Face 1: D is algebraically added to dimension Z of panel (DZ)
Face 2: is algebraically added to dimension X of panel (DX)
Face 3: is algebraically added to offset X of panel (BX)
Face 4: if the face is opposite the stop, is algebraically added to dimension Y of panel (DY); otherwise, D is algebraically added to offset Y of panel (BY)
Face 5: same rule as Face 4

By touching the panel once or a number of times you can generally determine variations in its dimensions from those (theoretical) specified in instructions H. The new dimensions (real) are memorized in the variables @DX, @DY and @DZ which can be used in the parametric; the real dimensions are automatically used when the machining operations are programmed with specular instructions SX and SY; more than one face can be touched before machining. You can use the preset variables @BX, @BY, @BZ to read the workpiece movement relative to the stops.
For feeling a point on the panel (G = 1), the co-ordinates of the point felt are saved in the variables @X, @Y and @Z.

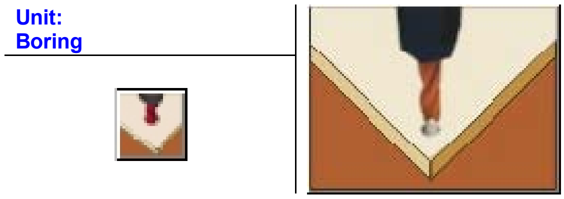
Examples:

```
H DX1000 DY800 DZ20 –A /DEF
F 2
XTA X10 Y400 T91
F 3
XTA X10 Y500 T91
F 1
B X100 Y400 Z8 T2
B X = @DX -50 Y500 Z8 T2
SX 1
B X200 T1
```

5.3.1.2 Boring

Boring - XB

Executes one or more borings.



Basic parameters:

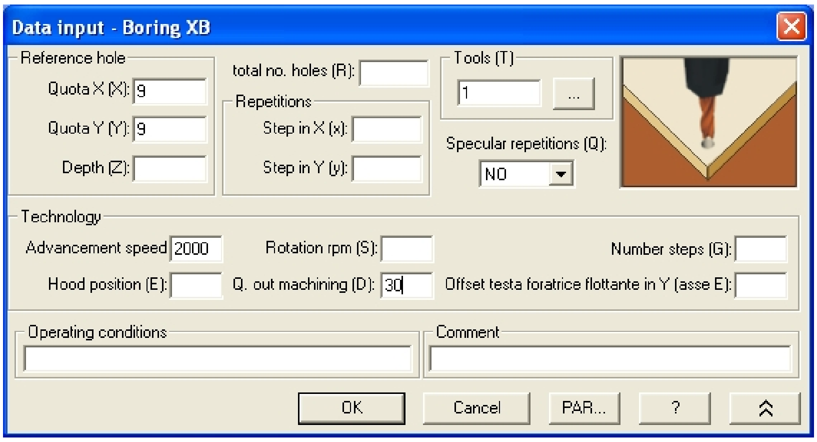
X Coordinate X for first hole.
Y Coordinate Y for first hole.
Z Hole depth.
R Number of holes (including the original hole).
x X step for repetitions.
y Y step for repetitions.
T List of tools; if there is more than one tool, coordinates X and Y refer to the first tool indicted.
Q Holes repetition according to specular in X/Y (0=No, 1=SX, 2=SY, 3=SXSY).

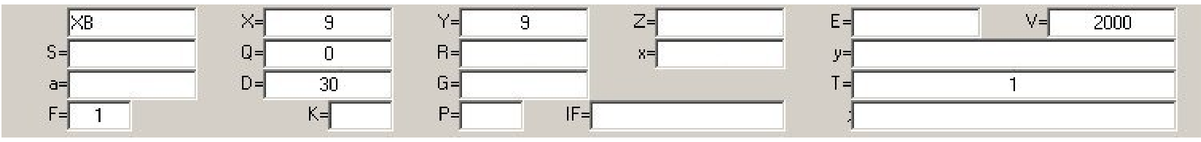
Extended parameters:

V Boring speed.
S Rotation speed of tool.
G Number of chip discharge steps.
E Position of vacuum hood (see: Appendix D).
D Quota out machining.
a Offset of boring head floating in Y (axis E, only if present).

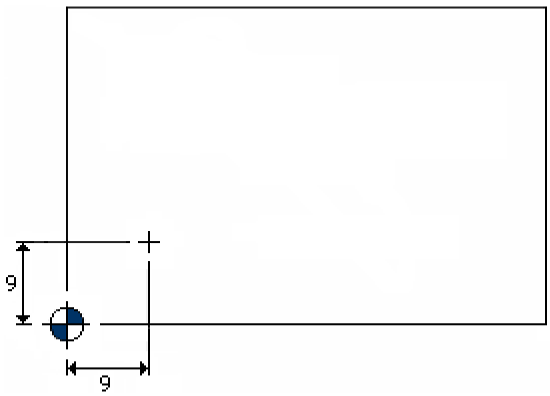
Examples:

1) Hole programming.
To produce a hole, parameters X, Y, Z, T, F are mandatory. The possibility of making a hole with chip discharge is activated with parameters G = number of steps (divided into equal parts; for example, G = 3).

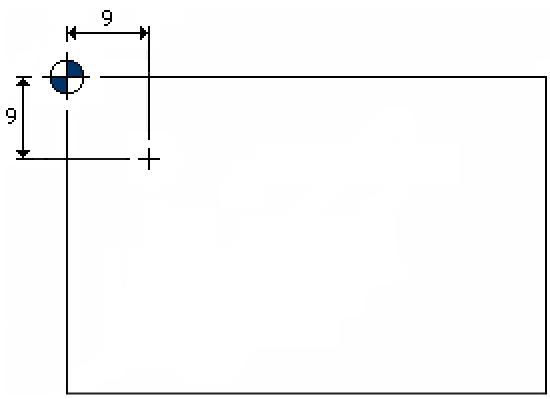




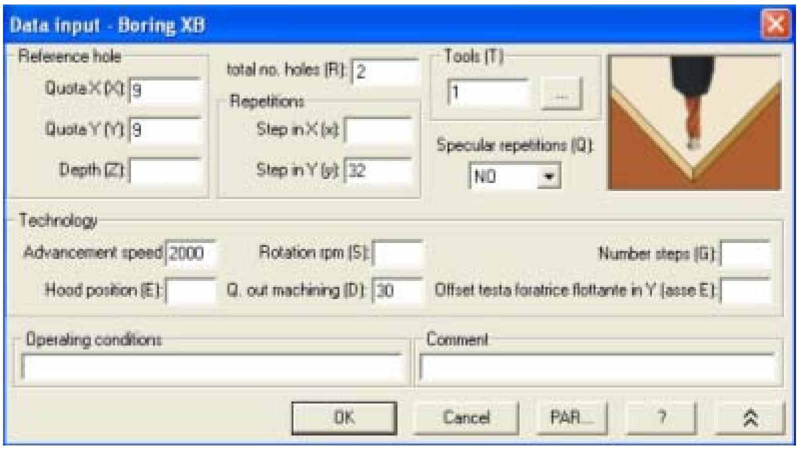
►*Front machine origin*

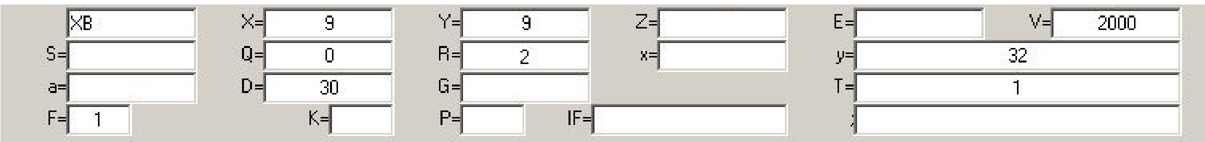


►*Rear machine origin*

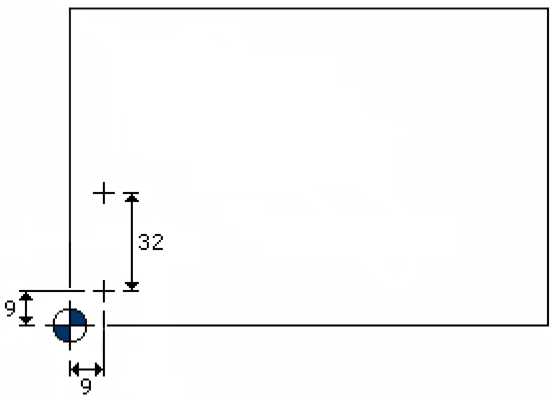


2) Programming repeated holes.
To produce repeated holes the parameters X, Y, Z, R, x, y, T, F are mandatory. The first hole is programmed with coordinates X=9, Y=9, Z=.. and the second one that has coordinates X=9, Y=41, Z=..is programmed with R = number of holes, y=32 (distance between second hole and first).

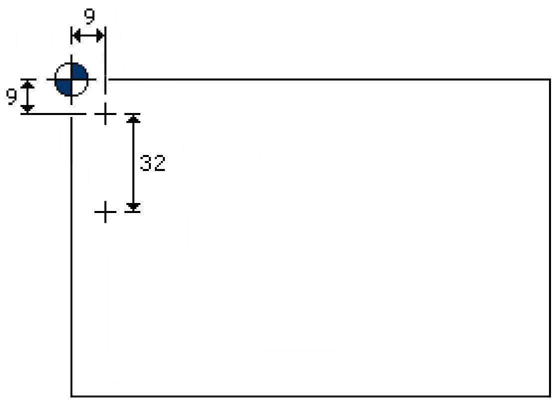




►*Front machine origin*

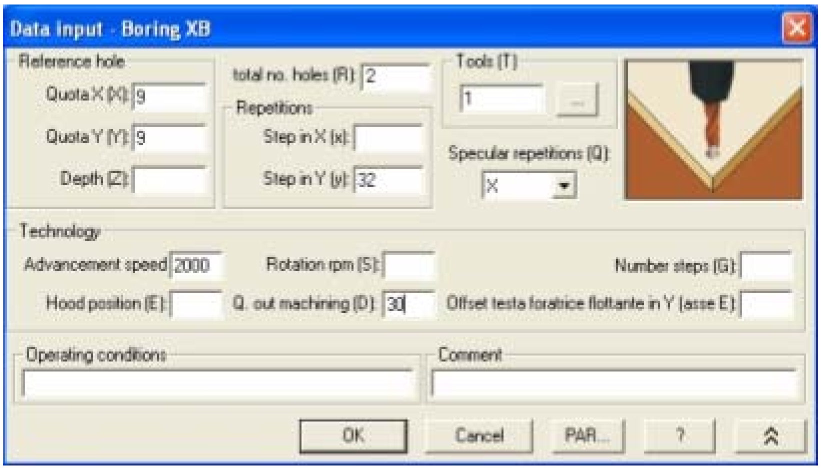


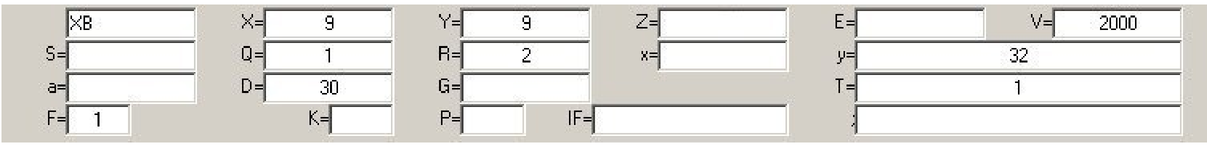
►*Rear machine origin*



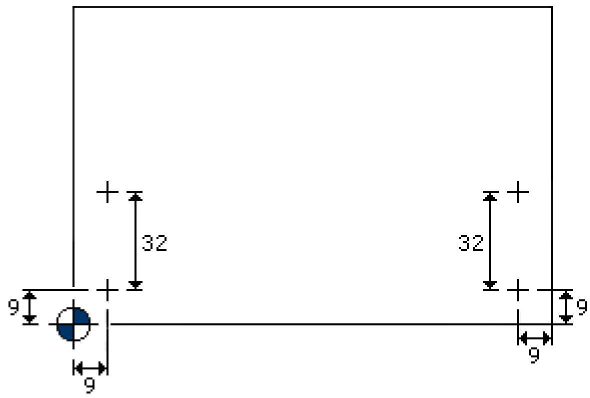
3) Programming two holes and the specular in X.

To produce two holes and their specula in X the parameters X, Y, Z, R, x, y, T, Q, F are mandatory. The first hole is programmed with coordinates X=9, Y=9, Z=.. and the second one that has coordinates X=9, Y=41, Z=..is programmed with R = number of holes, y=32 (distance between second hole and first) and their specular with Q=1.

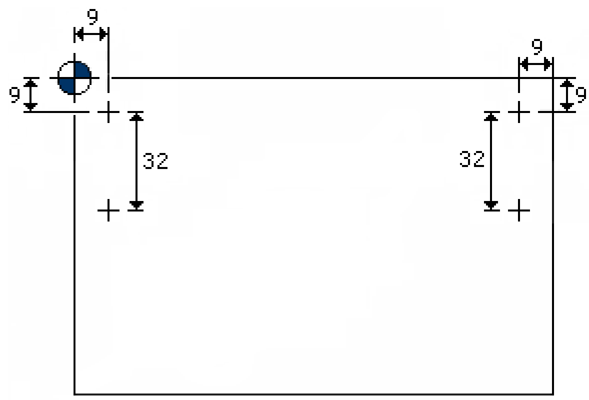




►*Front machine origin*

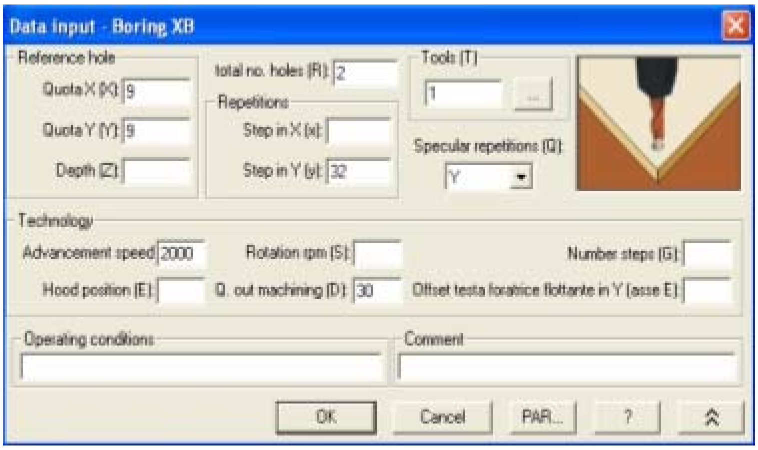


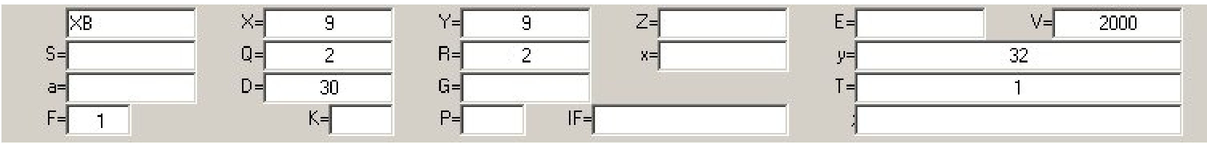
►*Rear machine origin*



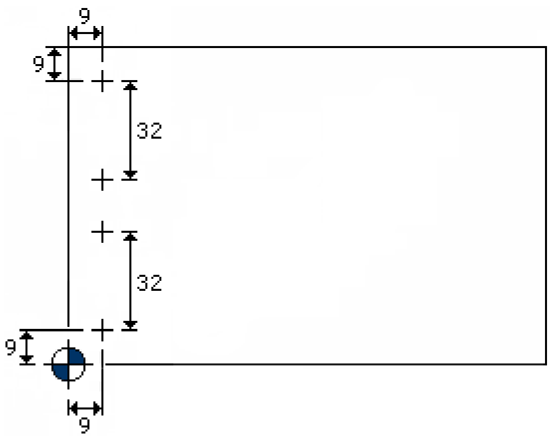
4) Programming two holes and the specular in Y.

To produce two holes and their specula in Y the parameters X, Y, Z, R, x, y, T, Q, F are mandatory. The first hole is programmed with coordinates X=9, Y=9, Z=.. and the second one that has coordinates X=9, Y=41, Z=..is programmed with R = number of holes, y=32 (distance between second hole and first) and their specular with Q=2.

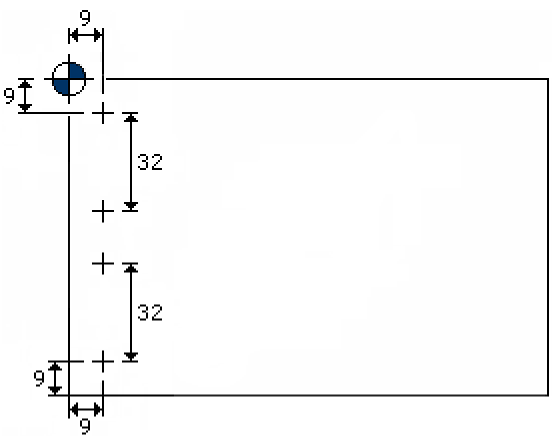




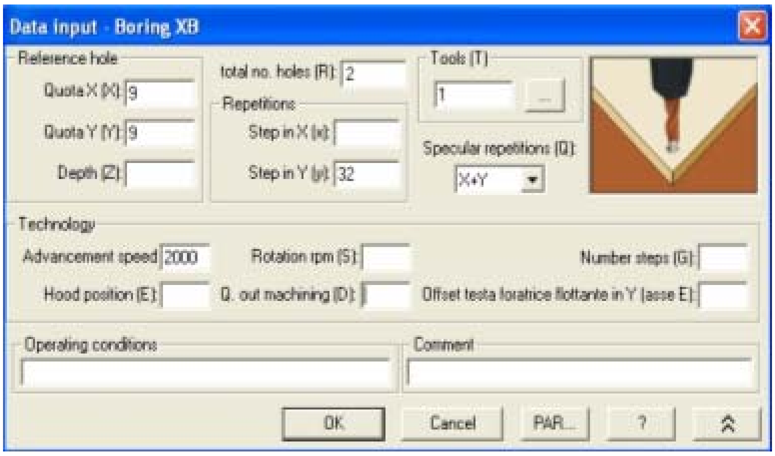
►*Front machine origin*

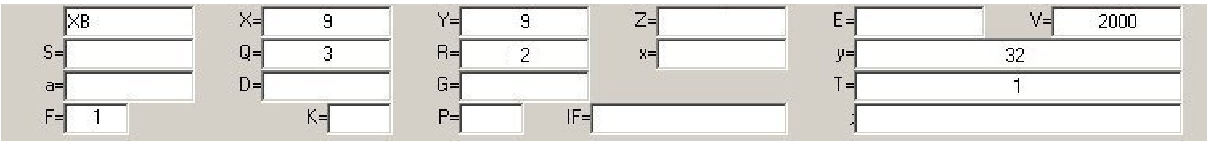


►*Rear machine origin*

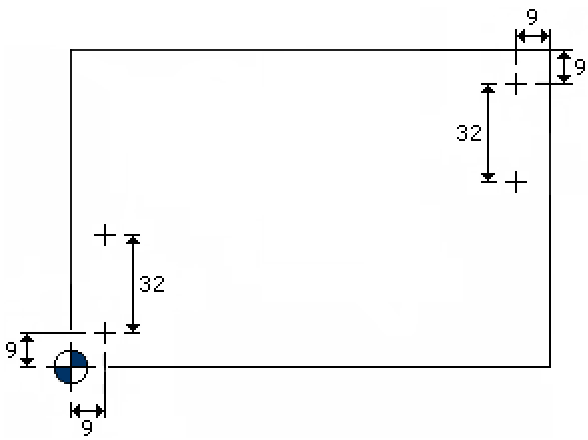


5) Programming two holes and the specular in X, Y.
To produce two holes and their specula in X, Y the parameters X, Y, Z, R, x, y, T, Q, F are mandatory. The first hole is programmed with coordinates X=9, Y=9, Z=.. and the second one that has coordinates X=9, Y=41, Z=..is programmed with R = number of holes, y=32 (distance between second hole and first) and their specular with Q=3.

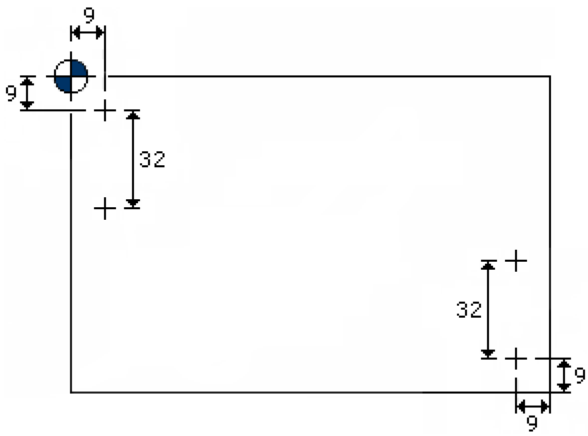




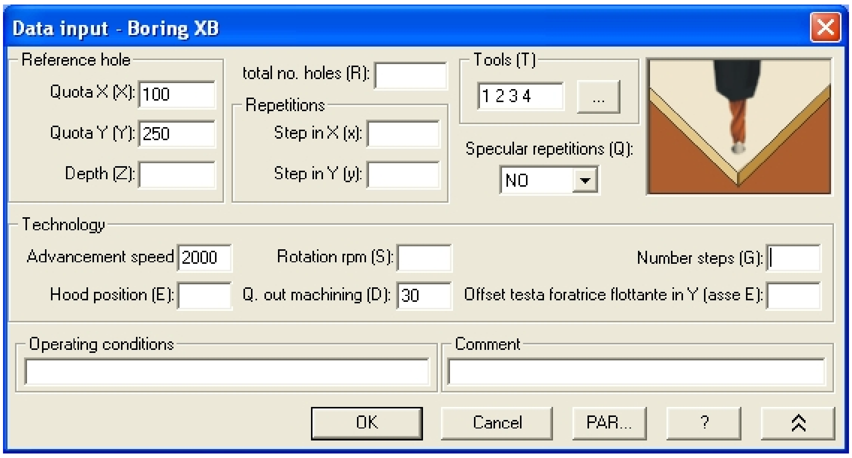
► *Front machine origin*

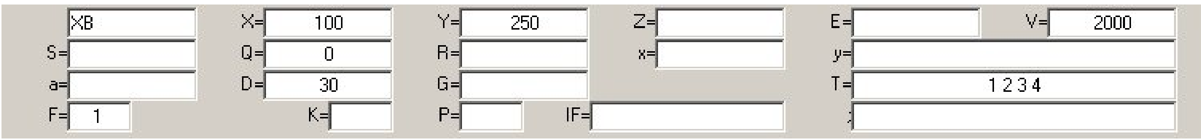


► *Rear machine origin*

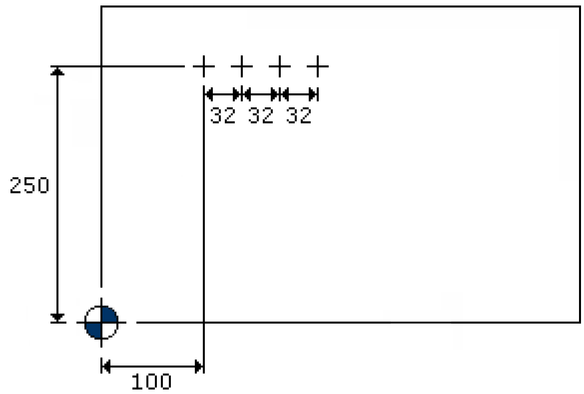


6) Programming a series of holes with step 32 boring machine using more than one spindle. Parameters X, Y, Z, T, F are mandatory. The first hole is programmed; the other holes, that are distanced at 32 mm, are programmed by selecting more than one spindle with parameter T (there must be space between one number and another).

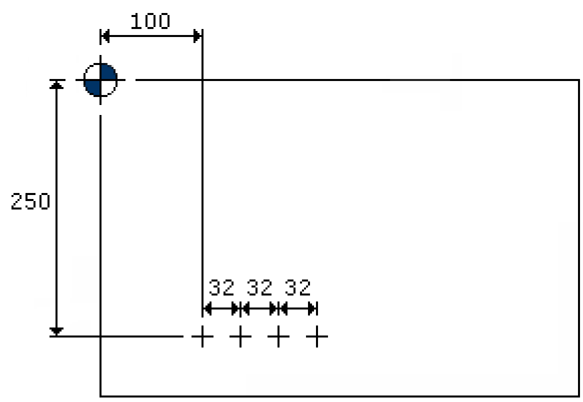




► *Front machine origin*

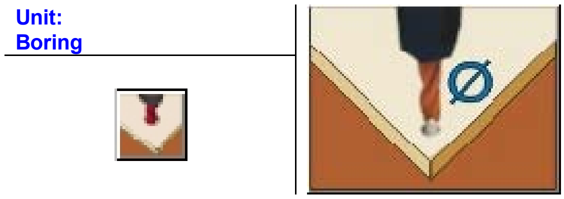


► *Rear machine origin*



Optimized Boring - XBO

This executes one or more holes using the boring optimization algorithm. All XBO instructions programmed in succession are placed in the same optimization session; a program may contain more than one block of XBO instructions.



Basic parameters:

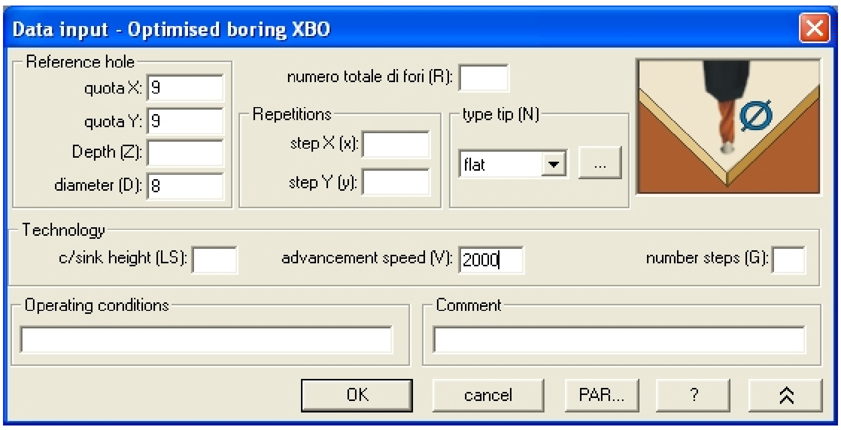
X Coordinate X for first hole.
Y Coordinate Y for first hole.
Z Hole depth.
D Hole diameter.
R Number of holes (including the original hole).
x X step for repetitions.
y Y step for repetitions.
N Hole type (P=flat, L=lance, S=taper).

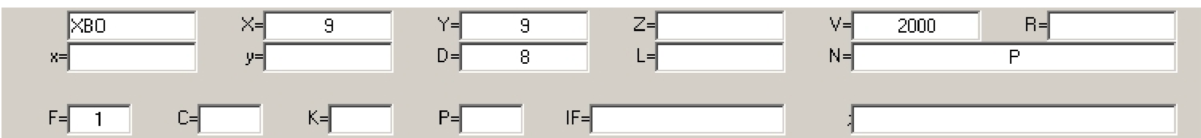
Extended parameters:

L Taper height (only if N=S).
V Boring speed.
G Number of chip discharge steps.

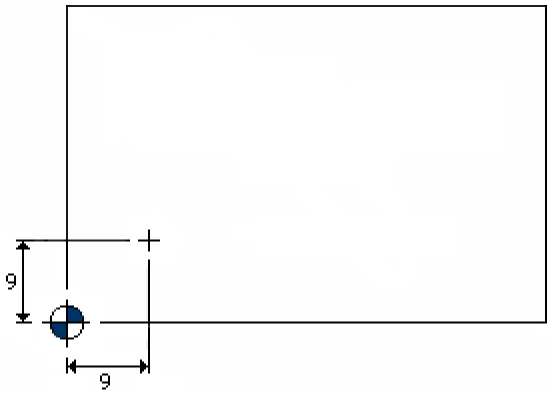
Examples:

1) Programming a hole.
To make a hole, parameters X, Y, Z, D, N, F are mandatory.

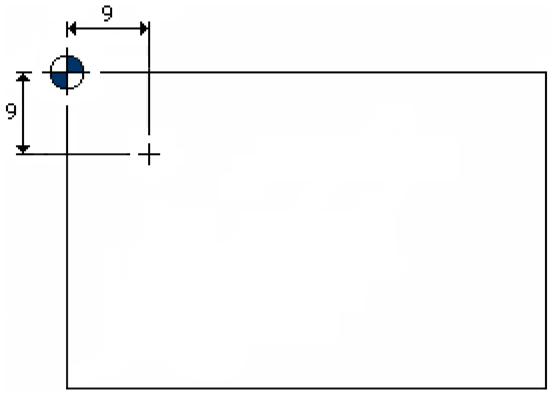




►*Front machine origin*

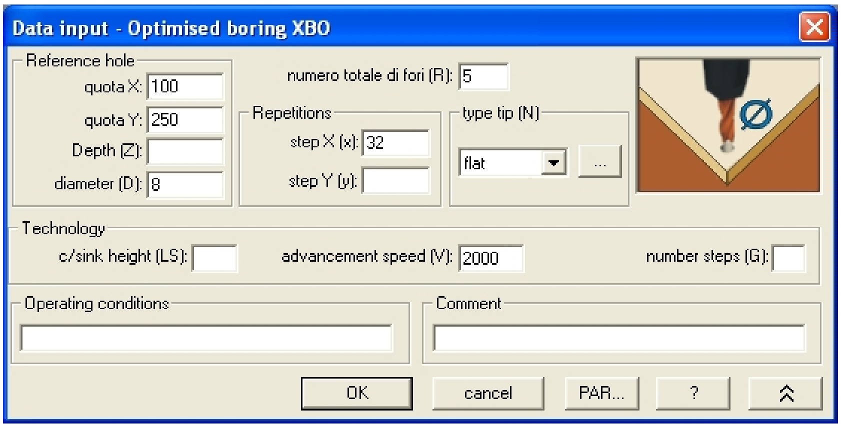


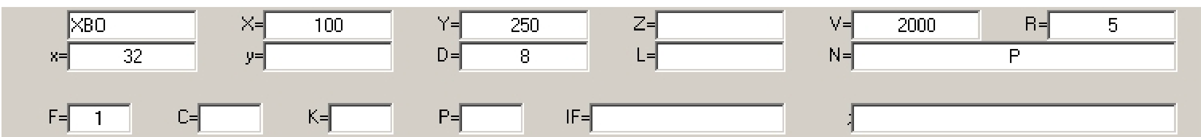
►*Rear machine origin*



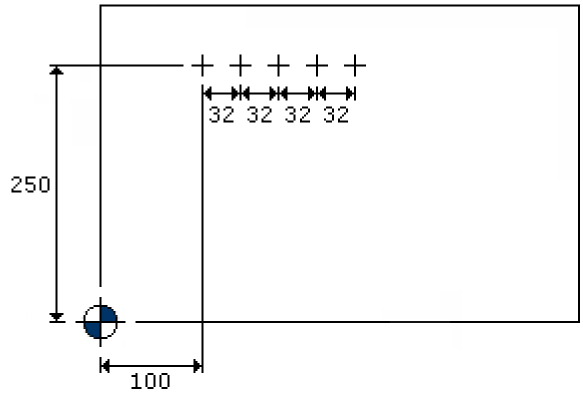
2) Programming a repetition of holes.

To produce a repetition of holes, the parameters X, Y, Z, R, x, y, D, N, F are mandatory. The first hole is programmed and the subsequent holes are programmed with R= umber of repetitions (including the original hole), x= distance between the holes (can also be other than 32 mm).

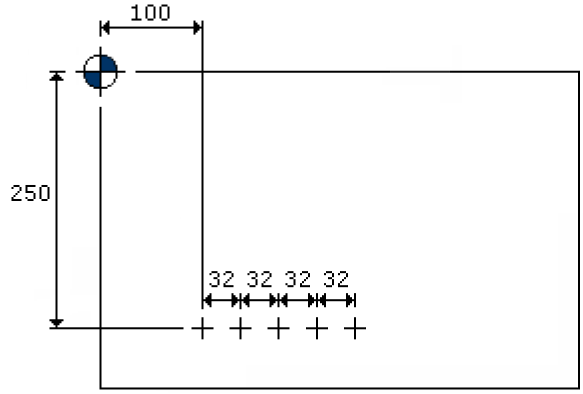




►*Front machine origin*



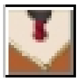
►*Rear machine origin*

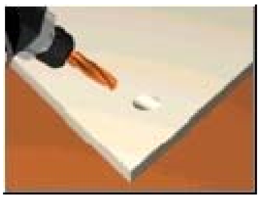


Slanted Boring - XBR

Allows you to make one or more borings at an angle other than 90° with respect to the working table. XBR is always referred to side 1. The current work side must always be side 1.

Unit: Boring





Basic parameters:

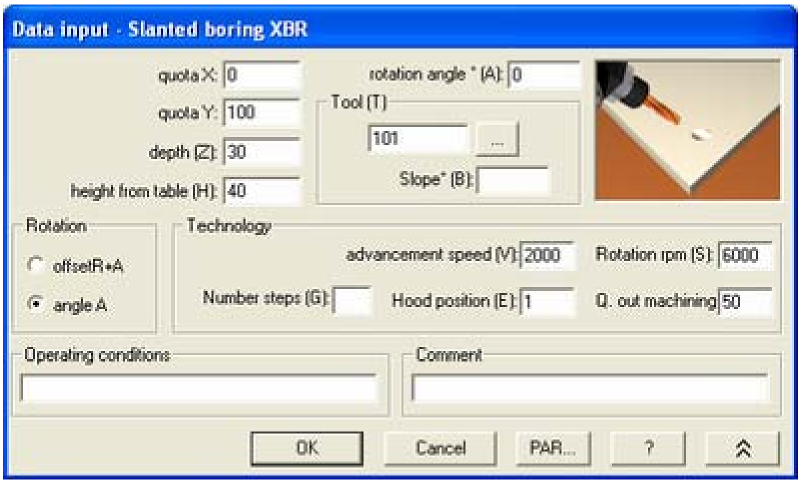
X Coordinate X for the first hole.
Y Coordinate Y fort he first hole.
Z Hole depth.
H Height of boring from the working table (position of the tool from the working table of the machine). Defaults to DZ.
A Boring rotation angle (front machine origin: positive in counter-clockwise direction; rear machine origin: positive in clockwise direction).
T List of tools; if there is more than one tool the coordinates X, Y refer to the first tool indicated.
B Tool angle relative to the vertical plane (only for the Prisma group).

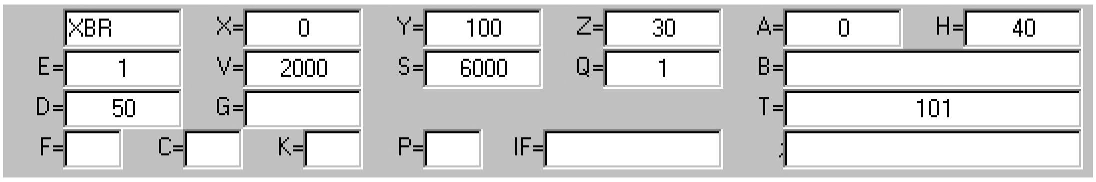
Extended parameters:

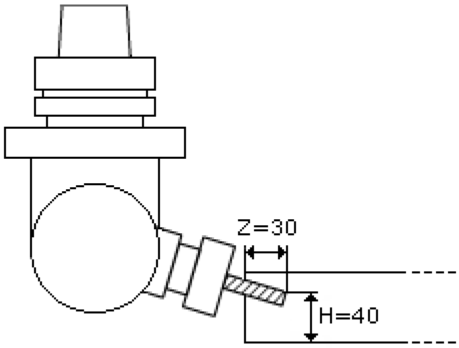
Q If the value is 0, angle A is summed to the tool's offset R, If the value is 1, angle A substitutes the tools offset R.
V Boring speed.
S Rotation speed of tool.
G Number of chip discharge steps.
E Position of vacuum hood (see: Appendix D).
D Out machining quota.

Example:

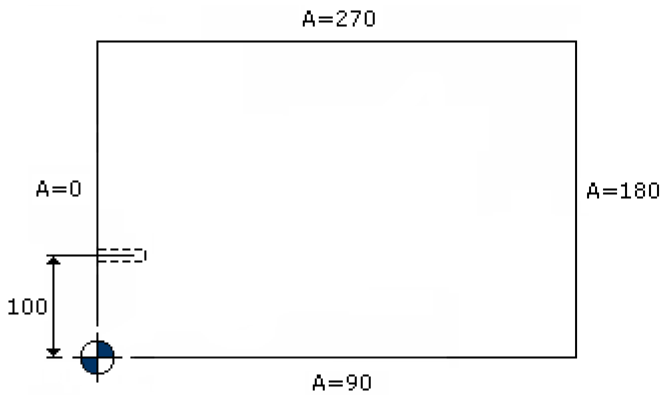
Programming a holes with parameter Q=1.



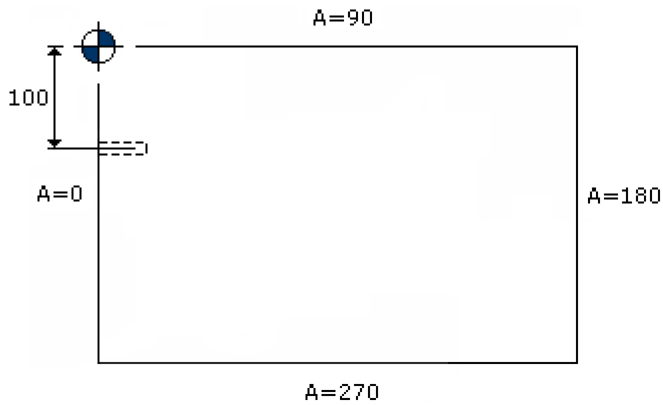




►*Front machine origin*



►*Rear machine origin*



WARNING

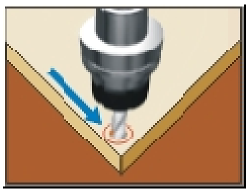
Incorrect programming of machining with slanted tools may cause damages to the structure of the machine. It is advisable that a SIMULATION of this type of operation be executed first.

5.3.1.3 Routing

Cutting Start - XG0

Defines the beginning point of a profile.

Unit:
Standard routing



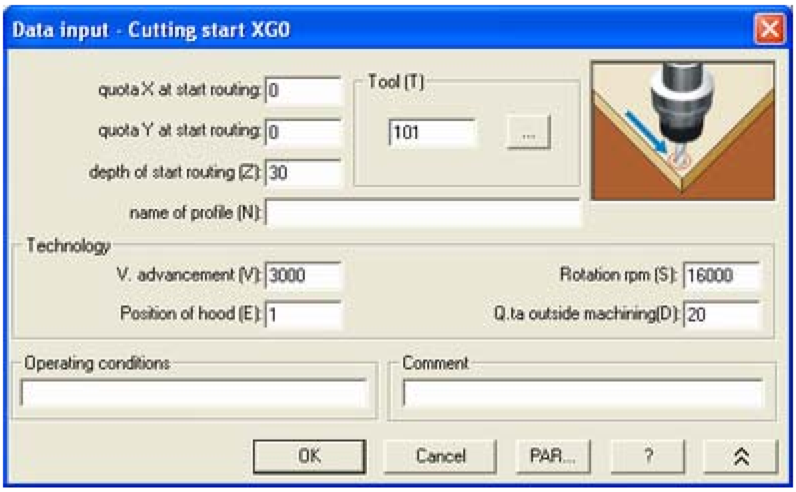
Basic parameters:

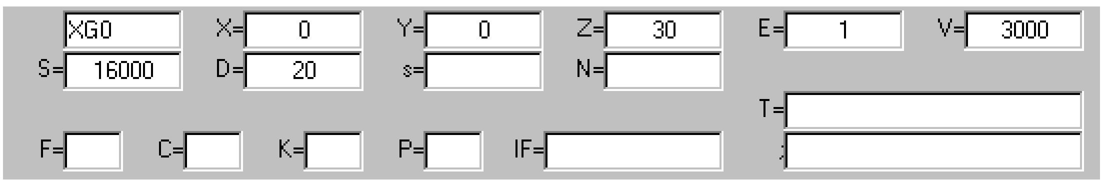
X Coordinate X for beginning of profile.
Y Coordinate Y for beginning of profile.
Z Depth of beginning of profile.
T Tool.

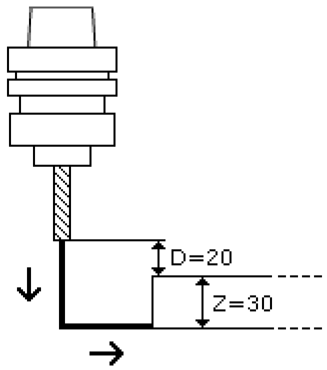
Extended parameters:

N Name of profile (v. XGREP).
V Entry speed into piece.
S Rotation speed of tool.
E Position of vacuum hood (see: Appendix D).
D Out machining quota.
s Quota of over-metal (only text instruction).

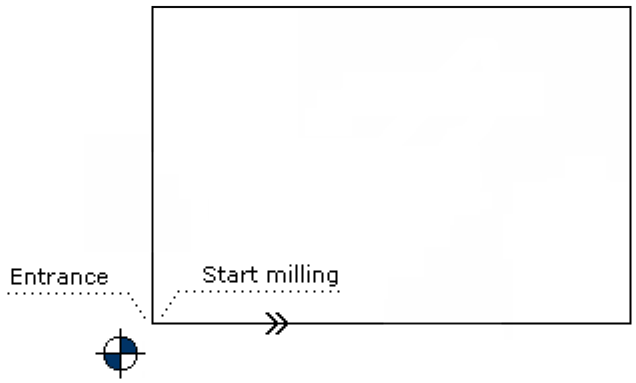
Example:



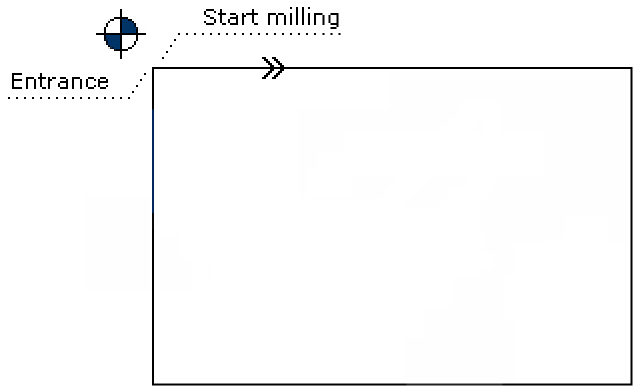




►*Front machine origin*



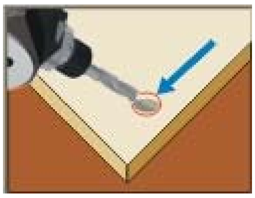
►*Rear machine origin*



3D Begin Milling - XG03D (Prisma Head)

Use these instructions to start linear routing on a plane set at an angle to the orthogonality of the panel surfaces; it controls the tool lead in/lead out into/from the workpiece with a null machining depth. It does not allow XGIN/XGOUT; with XG03D you cannot switch on the hood. You can only use this instruction with the Prisma head, only with face 1 active and only with radius correction Off. See: Appendix H – Prisma Head.

Unit:
Standard routing

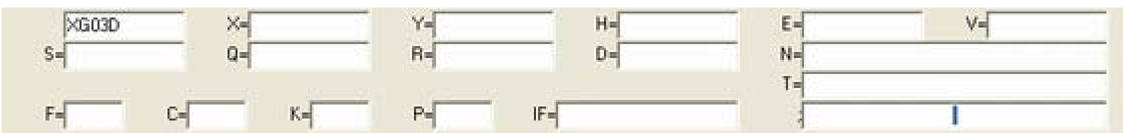


Basic parameters:

X Start profile X co-ordinate.
Y Start profile Y co-ordinate.
Q Angle of rotation about the Z-axis.
R Angle of rotation about the X-axis.
H Height of machining above work table (dimension to which tool must descend relative to machine work table; if this is not set the value is DZ).
T Tool.

Extended parameters:

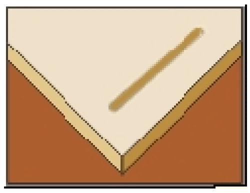
N Profile name (see XGREP).
V Routing speed.
E Position of vacuum hood (see: Appendix D).
S Tool speed of rotation.
D Out of machining dimension (only with Q and R unprogrammed).



Segment Through 2 Points - XL2P

Defines a line segment.

Unit:
Standard routing



Basic parameters:

X Coordinate X for end of segment.
Y Coordinate Y for end of segment
Z End of segment depth.

Extended parameters:

L Segment length
B Segment angle compared to axis X (front machine origin; rear machine origin: positive in clockwise direction).
A Angle of rotation (front machine origin: positive in counter-clockwise direction; rear machine origin: positive in clockwise direction).
V Milling speed.

The following combinations are valid:

1. quota X (Y=last set value);
2. quota Y (X=last set value);
3. quota X e quota Y;
4. angle B is one of the two quotas X and Y;
5. angle B is length L.

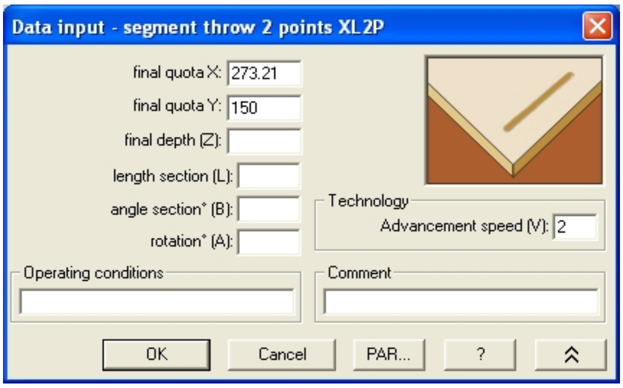
WARNING

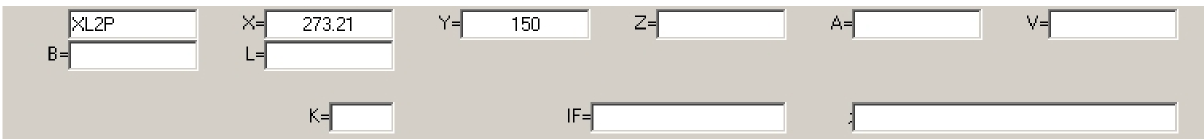
If the length L is set but the angle B is not, it is assumed that angle B is equal to 0 degrees.

Example:

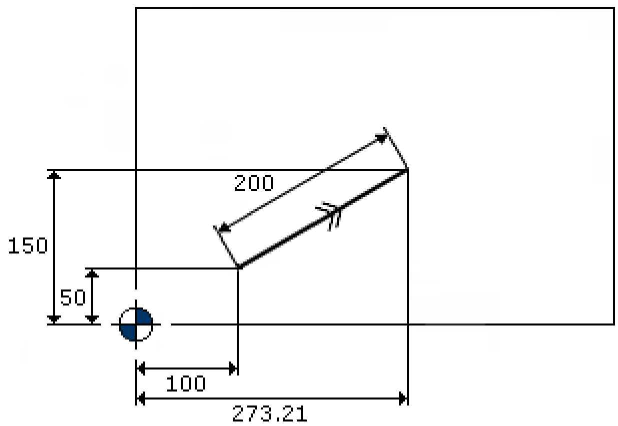
Programming a segment with starting point X = 100, Y = 50. The starting point for milling will be obtained from the previous instruction.

With Cartesian coordinates X, Y, Z.

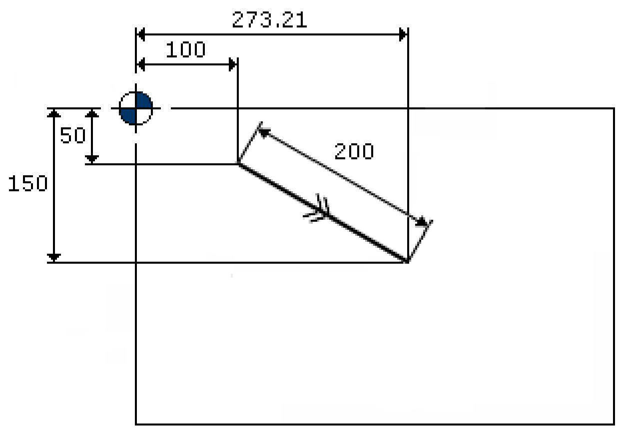




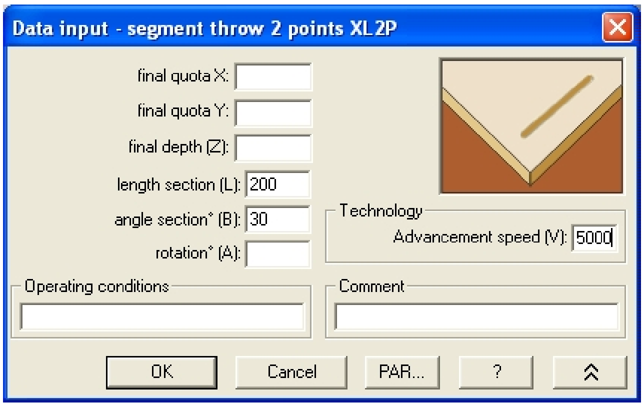
► *Front machine origin*

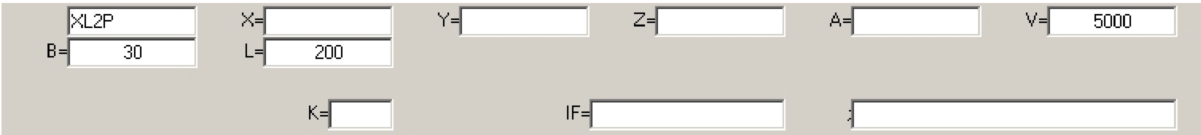


► *Rear machine origin*

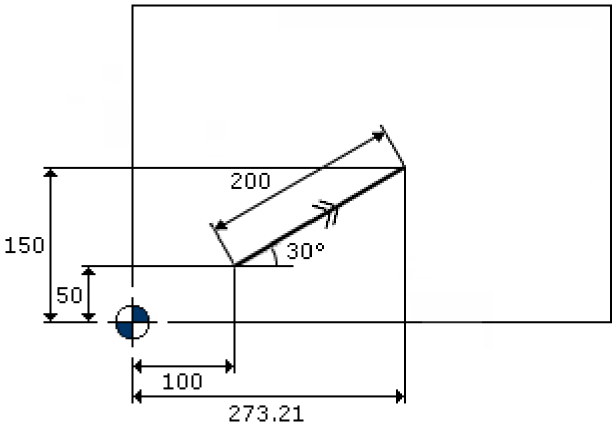


With polar coordinates B, L, Z.

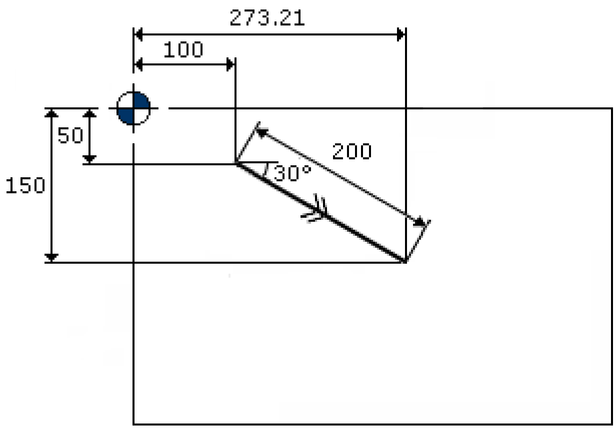




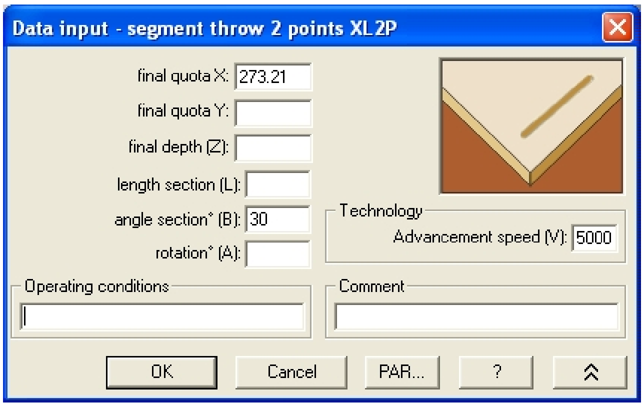
►*Front machine origin*

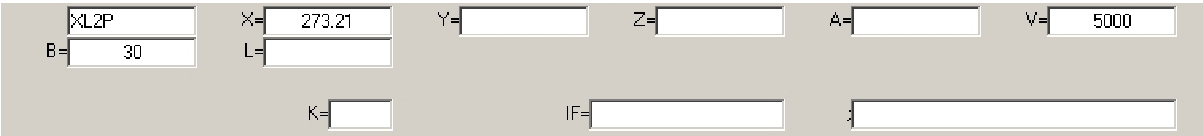


►*Rear machine origin*

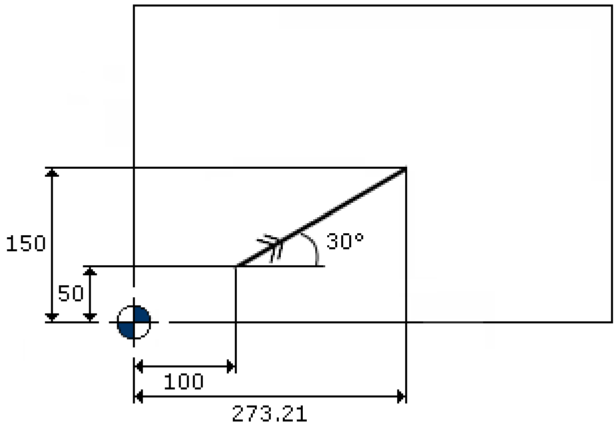


With mixed coordinates B and a Cartesian coordinate X or Y.

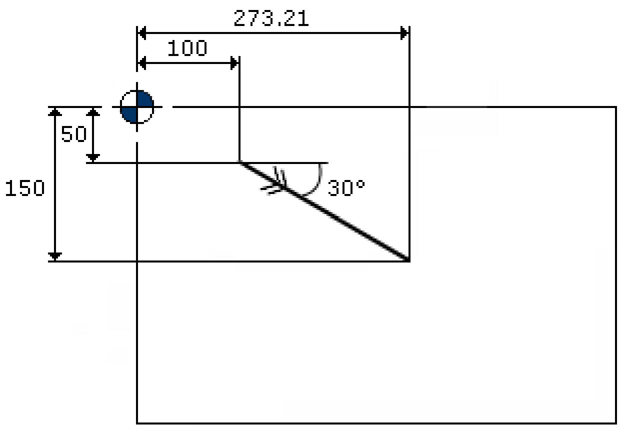




►*Front machine origin*



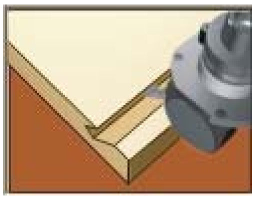
►*Rear machine origin*



3D Linear Milling - XG13D (Prisma Head)

Use these instruction to perform linear routing on a plane set at an angle to the orthogonality of the panel surfaces. You can only use this instruction with the Prisma head, only with face 1 active and only with radius correction Off. See: Appendix H – Prisma Head.

Unit:
Standard routing

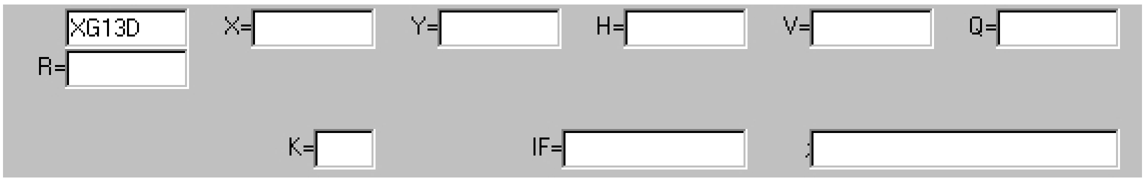


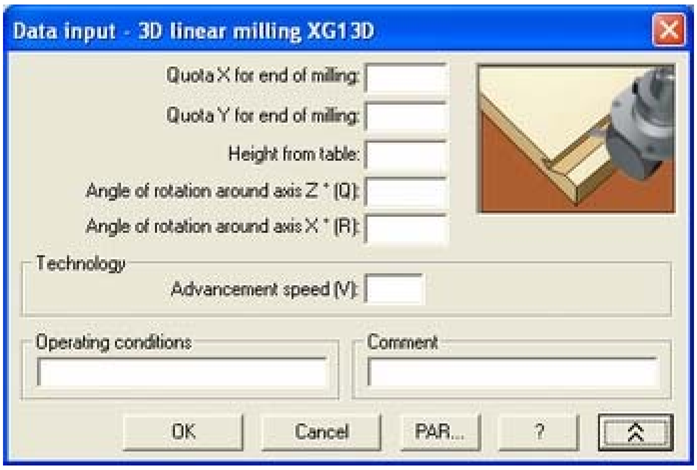
Basic parameters:

X End section X co-ordinate.
Y End section Y co-ordinate.
H Height of machining above work table (dimension to which tool must descend relative to machine work table; if this is not set the value is DZ).
Q Angle of rotation about the Z-axis.
R Angle of rotation about the X-axis.

Extended parameters:

V Routing speed.

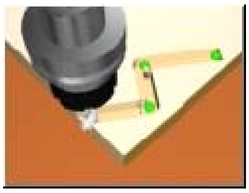




Broken Line - XSP

Defines two line segments.

Unit:
Complex routing



Parameters:

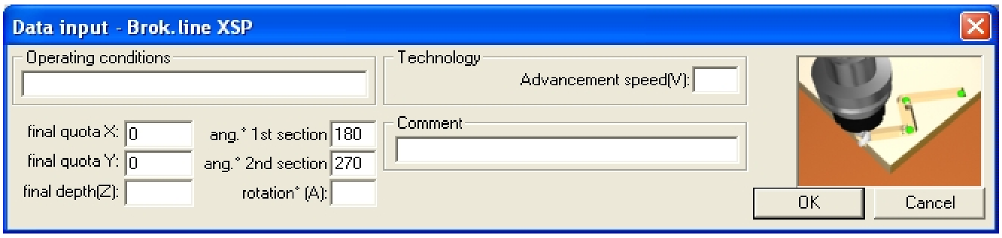
X Coordinate X of end of second segment.
Y Coordinate Y of end of second segment.
Z Depth of end of second segment.
A Angle of rotation (front machine origin: positive in counter-clockwise direction; rear machine origin: positive in clockwise direction).
V Milling speed.
B Angle of first segment compared to axis X (front machine origin: positive in counter-clockwise direction, rear machine origin: positive in clockwise direction).
D Angle of second segment compared to axis X (front machine origin: positive in counter-clockwise direction, rear machine origin: positive in clockwise direction).

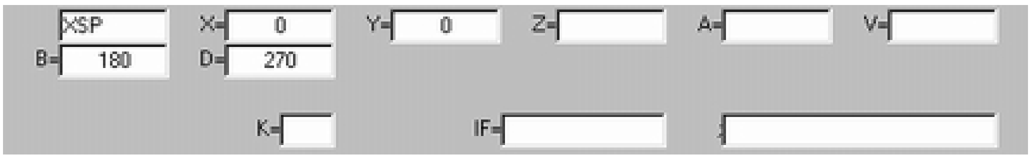
Parameters X, Y, B, D are required.

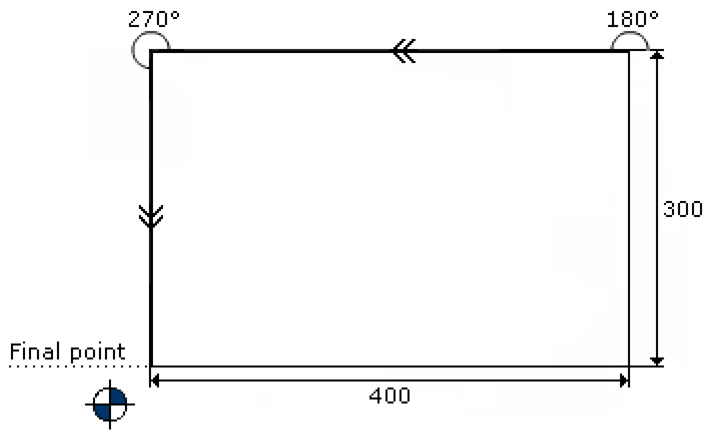
Example:

►*Front machine origin*

Programming two segments with starting points X=400, Y=300. The starting point for milling will be obtained from the previous instruction.

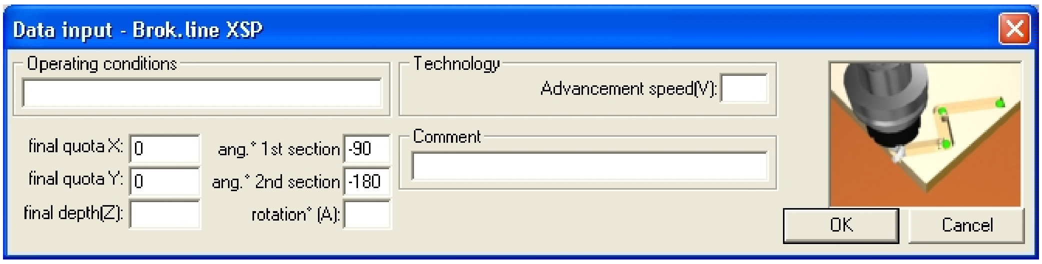


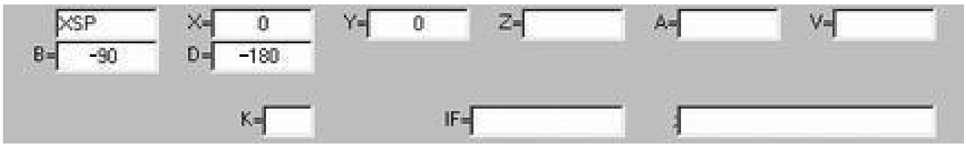


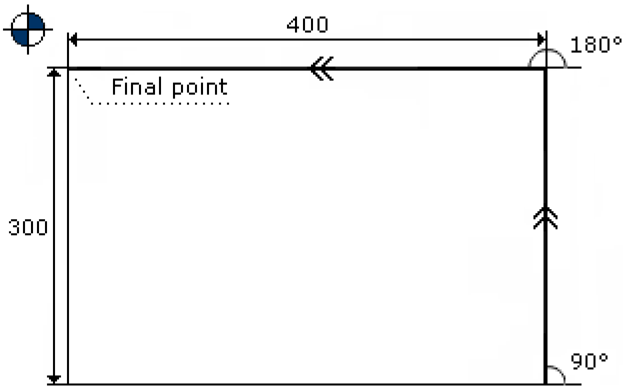


►*Rear machine origin*

Programming two segments with starting points X=400, Y=300. The starting point for milling will be obtained from the previous instruction.



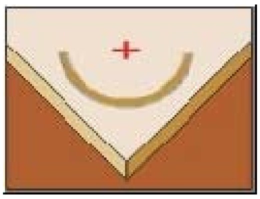




Arc Given 2 Points - XA2P

Defines an arc of a circle with two points known.

Unit:
Standard routing



Basic parameters:

X Coordinate X of end of arc.
Y Coordinate Y of end of arc.
Z Depth of end of arc.
I Coordinate X of center of circle.
J Coordinate Y of center of circle.
G Direction: 2=clockwise, 3=counter-clockwise.

Extended parameters:

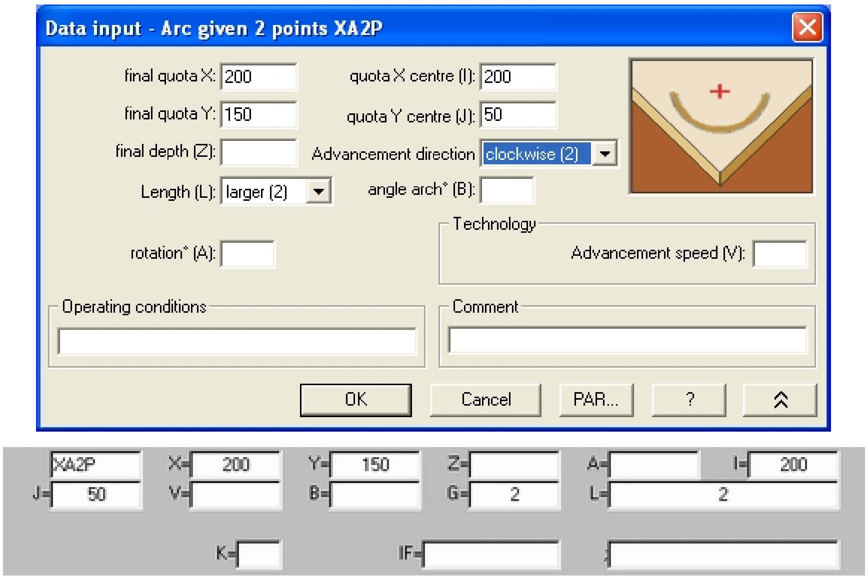
L Length of arc: 1=smaller, 2=greater.
B Angle of arc.
A Angle of rotation (front machine origin: positive in counter-clockwise direction; rear machine origin: positive in clockwise direction).
V Milling speed.
.
The parameters I, J, G are obligatory. One of the following combinations of parameters must also be programmed:

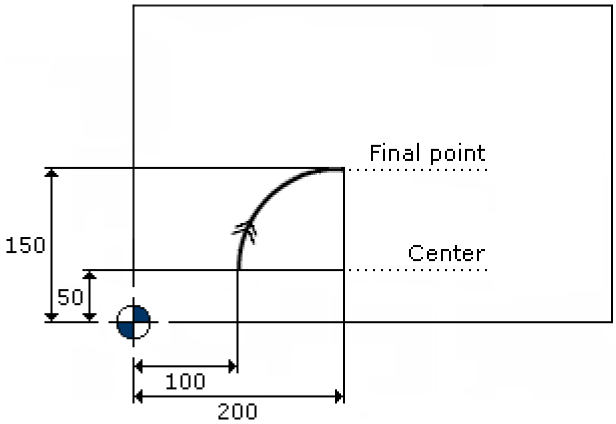
1) angle B;
2) final coordinates X and Y;
3) final coordinate X (the current coordinate is considered the final Y coordinate);
4) final coordinate Y (the current coordinate is considered the final X coordinate);
5) coordinate X and length L;
6) coordinate Y and length L.

Examples:

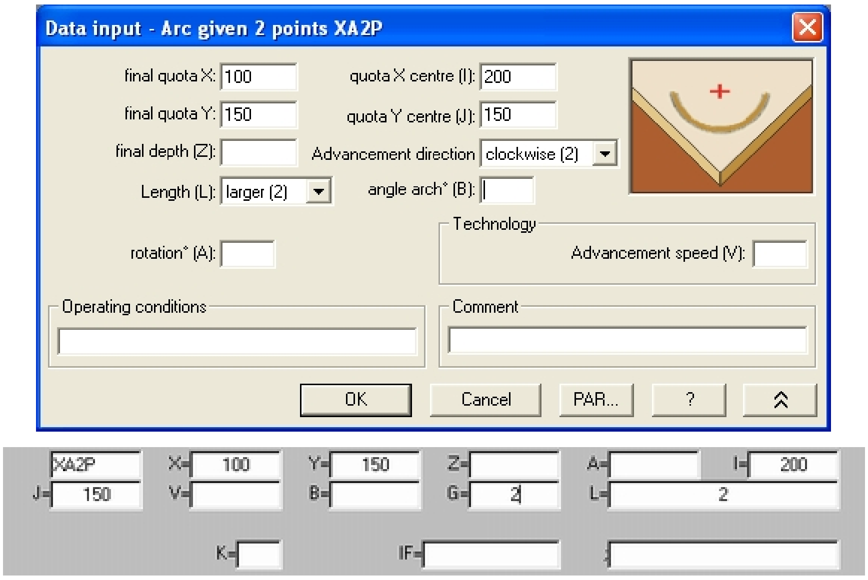
►*Front machine origin*

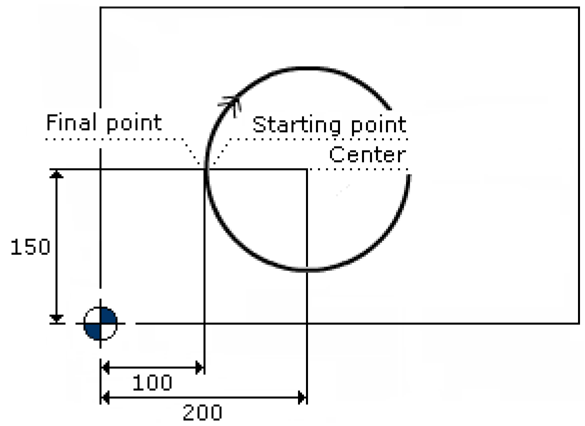
1) Programming clockwise circular milling with a starting point of X=100, Y=50. The starting point for the milling will be obtained form the previous instruction.





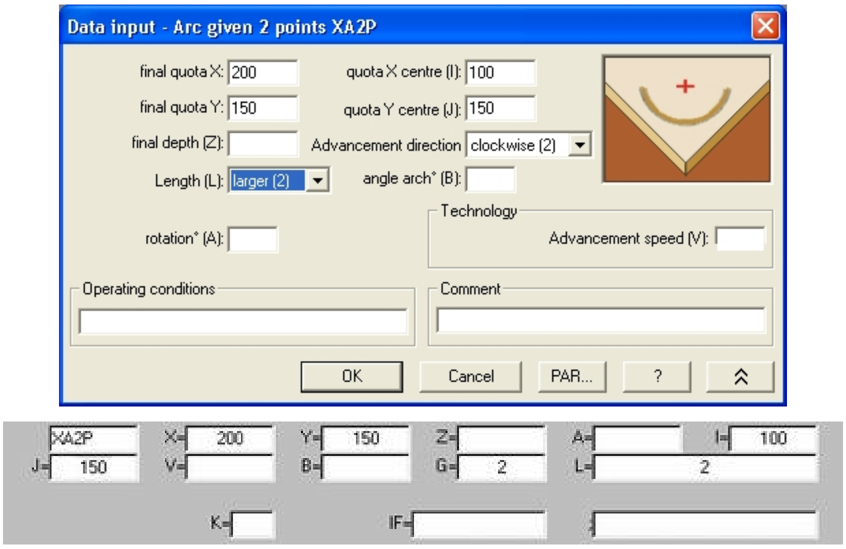
2) Programming clockwise circular milling (circumference) with starting point X=100 Y=150. The starting point for the milling will be obtained from the previous instruction.

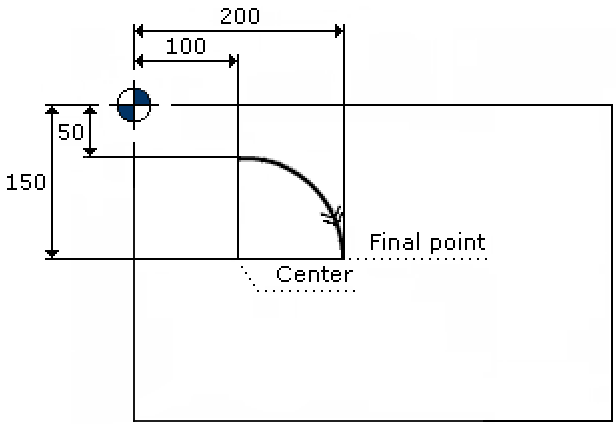



Final point
Starting point
Center
150
100
200

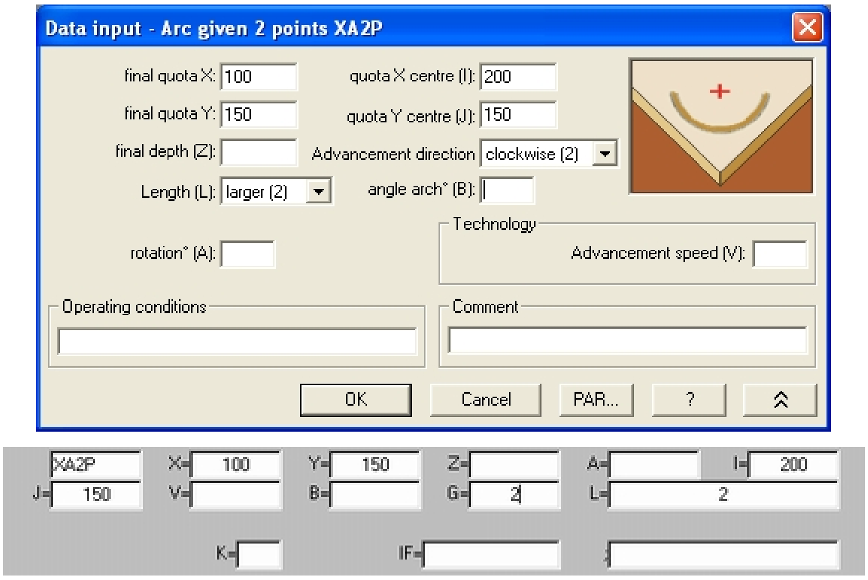
►*Rear machine origin*

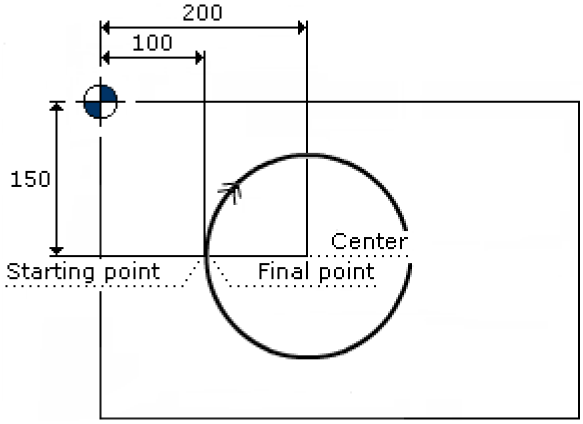
1) Programming clockwise circular milling with starting point X=100, Y=50. The stating point for milling will be obtained for the previous instruction.





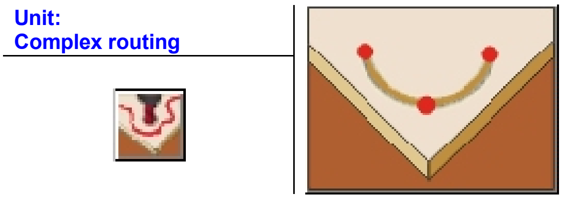
2) Programming clockwise circular milling (circumference) with starting point X=100 Y=150. The starting point for milling will be obtained from the previous instruction.




200
100
150
Center
Starting point
Final point

3 Point Data Arc - XA3P

Defines an arc of circle giving three points. The depth of intermediate point can be different from end point.



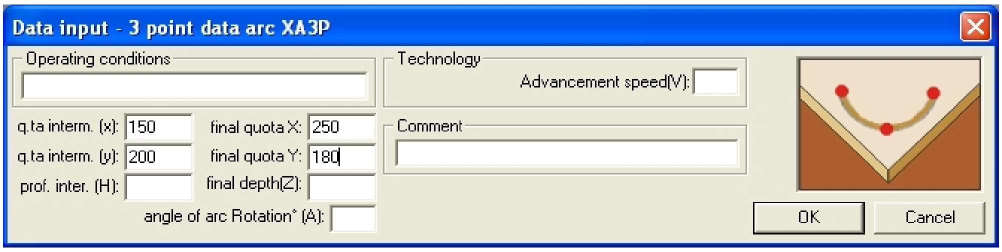
Starting from the current point, circular milling is performed for the coordinates point x, y, H) up to the end of coordinates (X, Y, Z). The following data must be programmed: x, quota X of the intermediate point, y, quota of the intermediate point, X, quota X of the final point; Y, quota Y of the final point. Quota H can be omitted, in this case all milling is performed at the same machining depth Z. The parameters X, Y, x, y are mandatory.

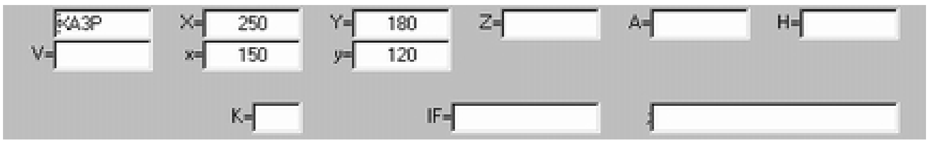
Parameters:

X	Coordinate X for end of arc.
Y	Coordinate Y for end of arc.
Z	Depth of end of arc.
A	Angle of rotation (front machine origin: positive in counter-clockwise direction; rear machine origin: positive in clockwise direction).
H	Intermediate depth.
V	Milling speed.
x	Coordinate X intermediate.
y	Coordinate Y intermediate.

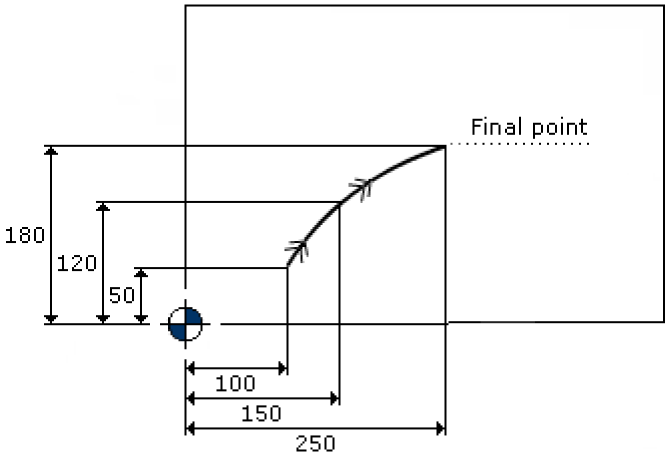
Example:

Programming circular milling that passes three point with starting point X=100, Y=50. The starting point for the milling will be obtained from the previous instruction.

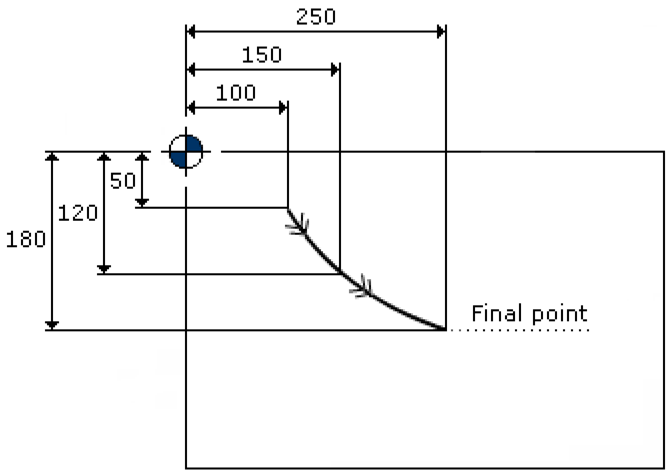




►*Front machine origin*



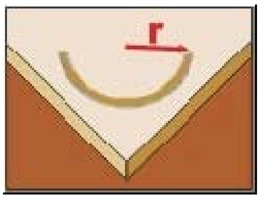
►*Rear machine origin*



Arc Given Radius - XAR

Defines an arc of a circle from its radius. The radius r can assume positive and negative values: If positive, the center is on the left of the imaginary line joining the beginning point with the end point of the arc. If negative, the center is on the right.

Unit:
Standard routing



Basic Parameters:

X Coordinate X for end of arc.
Y Coordinate Y for end of arc.
Z End of arc depth.
r Radius.
G Direction: 2=clockwise, 3=counter-clockwise.

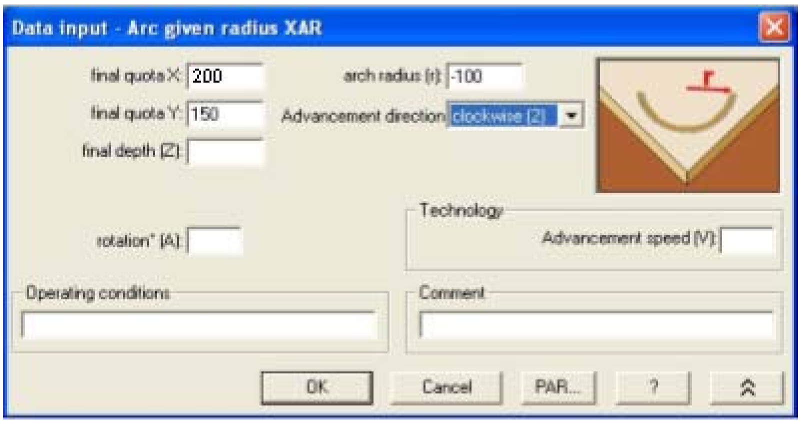
Extended parameters:

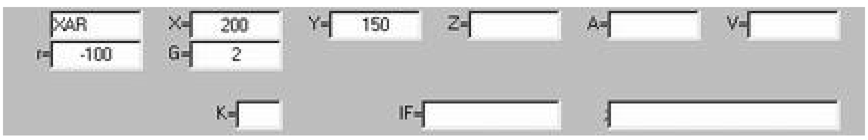
A Rotation angle (front machine origin: positive in counter-clockwise direction; rear machine origin: positive in clockwise direction).
V Milling speed.

The parameters X, Y, r, G are mandatory.

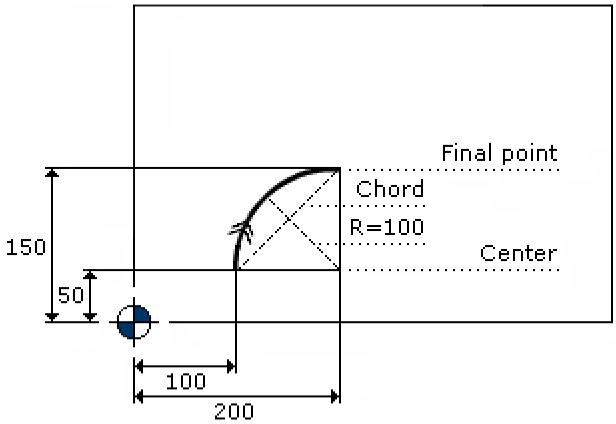
Examples:

1) Programming clockwise circular milling with starting point X=100, Y=50 (circle arc with center to the right of the chord). The starting point for the routing will be taken from the previous command.

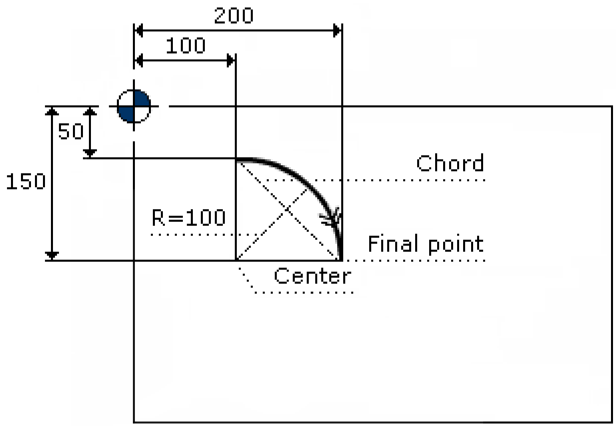




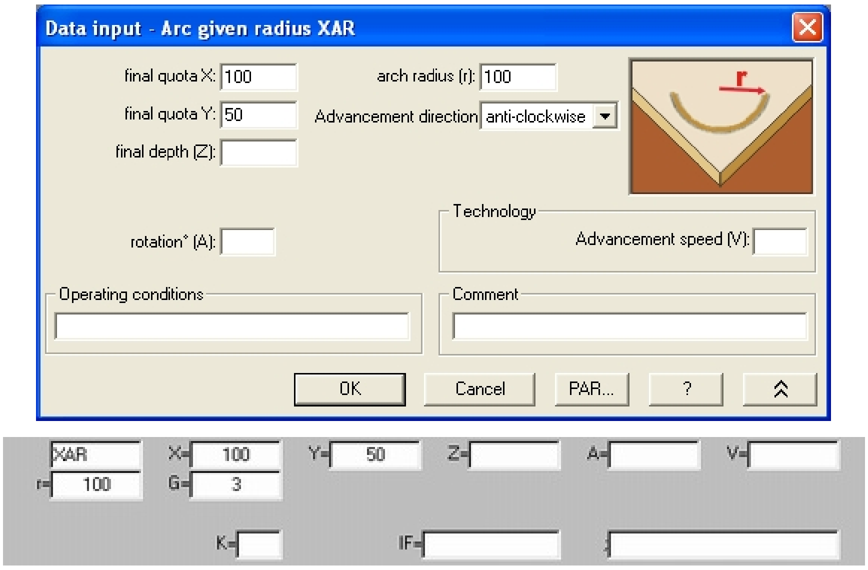
►*Front machine origin*



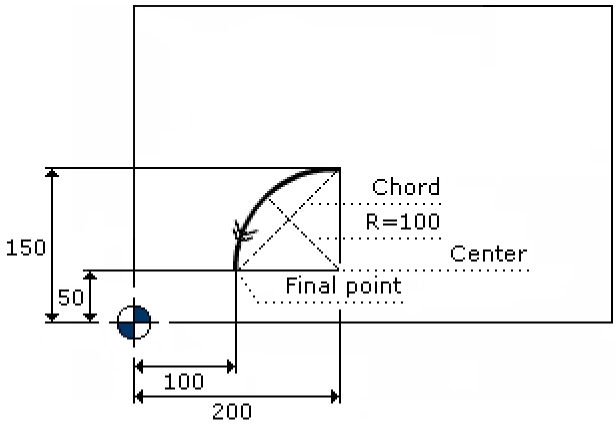
►*Rear machine origin*



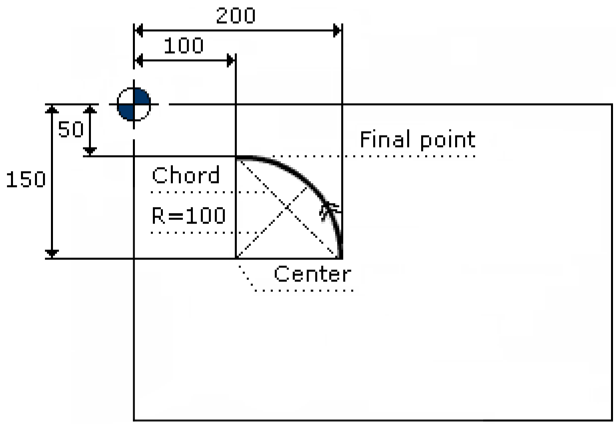
2) Programming counter-clockwise circular milling with starting point X=200, Y=150 (circle arc with center to the left of the chord). The starting point for the routing will be taken from the previous command.



►*Front machine origin*

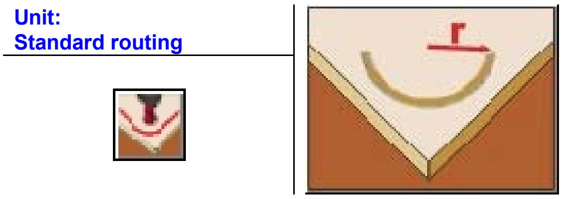


►*Rear machine origin*



Arc Given Radius 2 - XAR2

Defines an arc of a circle from its radius. Radius r can assumed positive and negative values. If it is positive, it is always considered as the shortest radius, if it is negative, it is always considered as the longest radius.



Basic Parameters:

X Coordinate X of end of arc.
Y Coordinate Y of end of arc.
Z Depth of arc end.
r Radius.
G Direction: 2=clockwise, 3=counter-clockwise.

Extended Parameters:

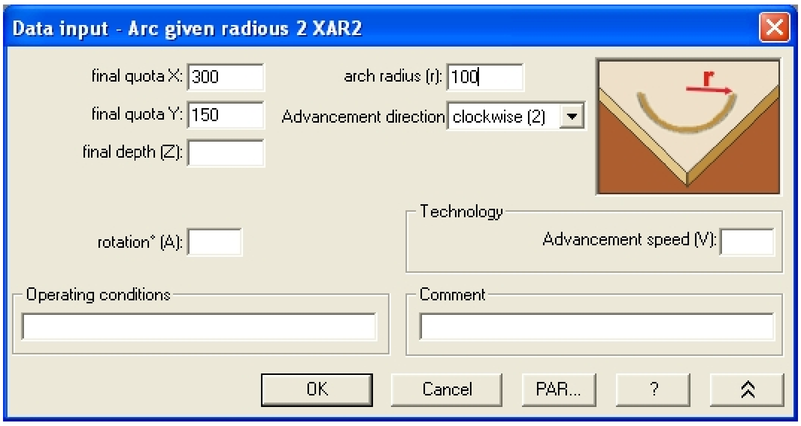
A Angle of rotation (front machine origin: positive in counter-clockwise direction; rear machine origin: positive in clockwise direction).
V Milling speed.

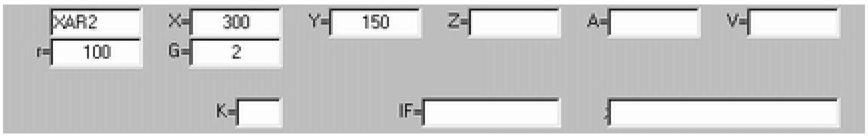
Parameters X, Y, r, G are mandatory.

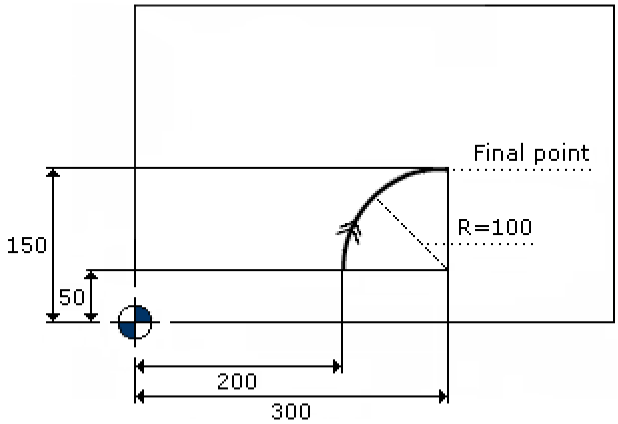
Examples:

► *Front machine origin*

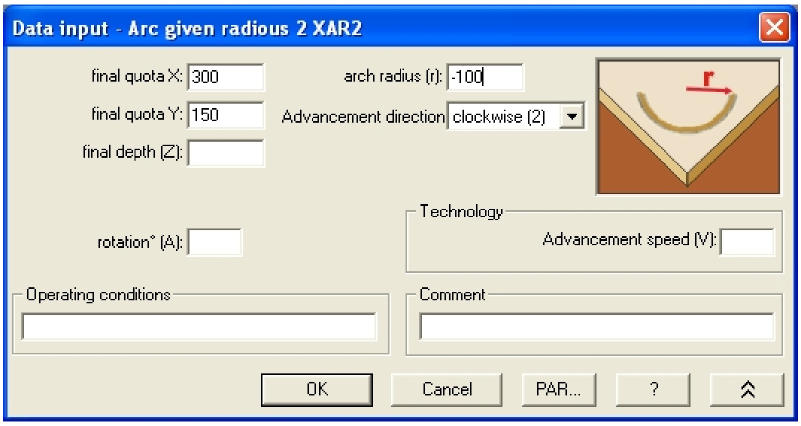
1) Programming clockwise circular milling with a starting point of X=200, Y=50 (the circle arc is less than 180°). The starting point for the routing will be taken from the previous command.

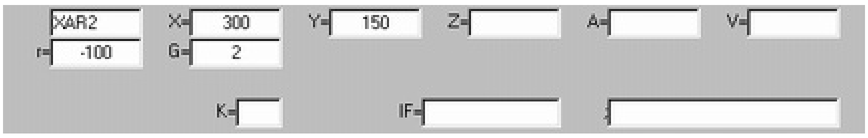


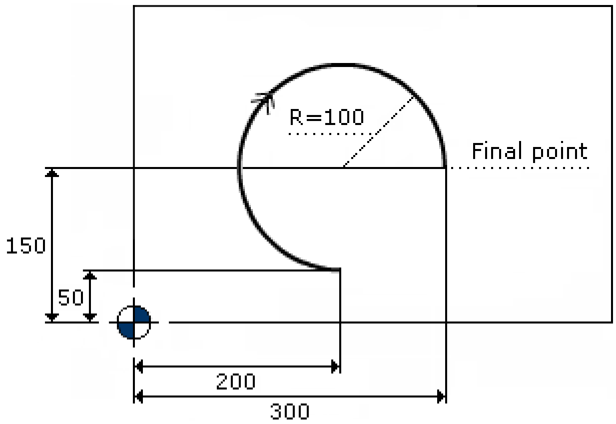




2) Programming clockwise circular milling with a starting point of X=200, Y=50 (the circle arc is greater than 180°). The starting point for the routing will be taken from the previous command.

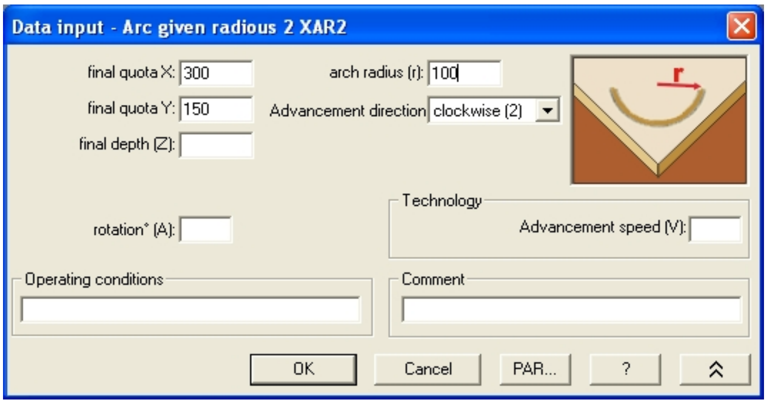


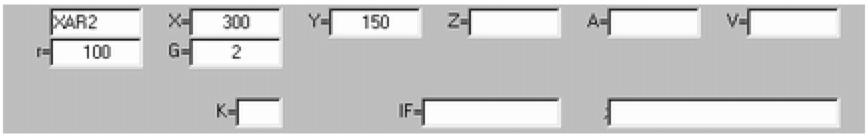


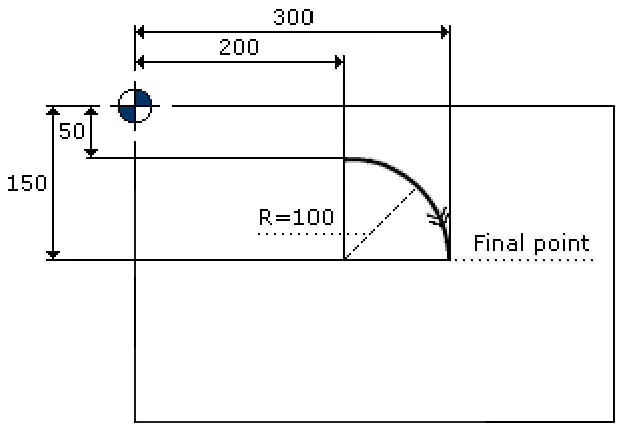


►*Rear machine origin*

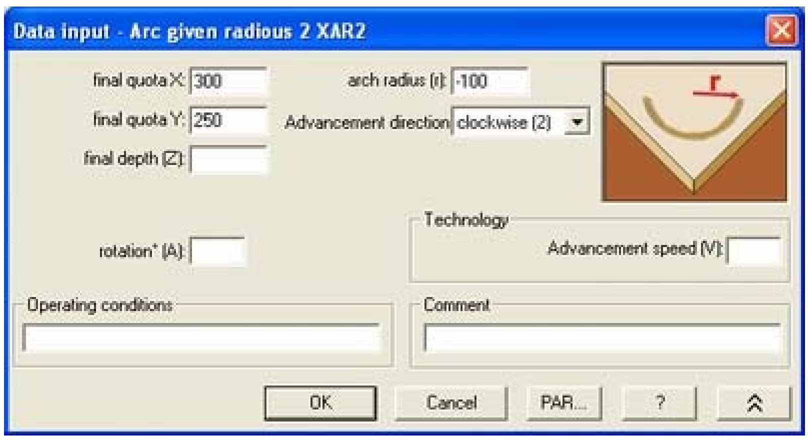
1) Programming clockwise circular milling with a starting point of X=200, Y=50 (the circle arc is less than 180°). The starting point for the routing will be taken from the previous command.

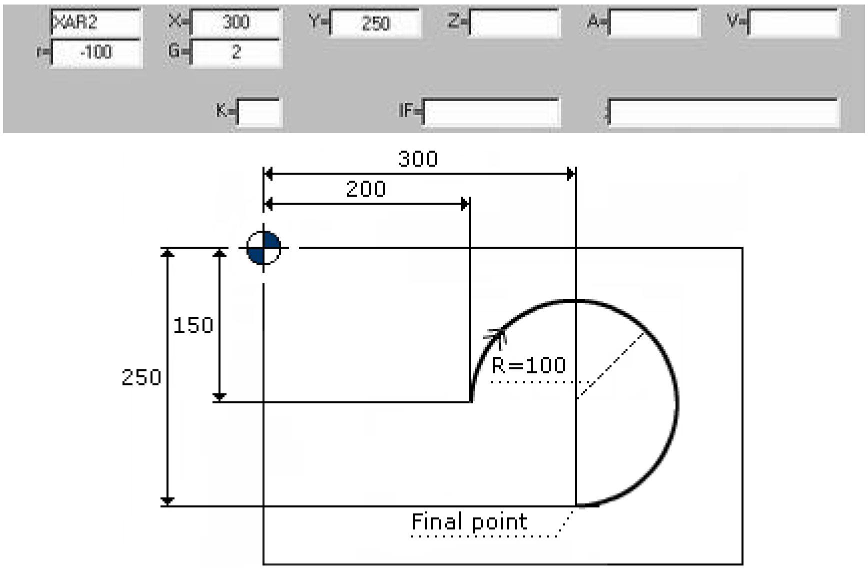






2) Programming clockwise circular milling with a starting point of X=200, Y=150 (the circle arc is greater than 180°). The starting point for the routing will be taken from the previous command.

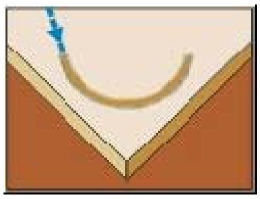



XAR2
X= 300
Y= 250
Z=
A=
V=
r= -100
G= 2
K=
IF=
300
200
150
250
R=100
Final point

Section Tangent to Previous Section - XG5

Defines a milling section tangent to the previous one.

Unit:
Standard routing



Basic parameters:

X Coordinate X of end of milling (only if G=2 or G=3).
Y Coordinate Y of end of milling (only if G=2 or G=3).
Z End of milling depth.

Extended parameters:

G Type of tangency:

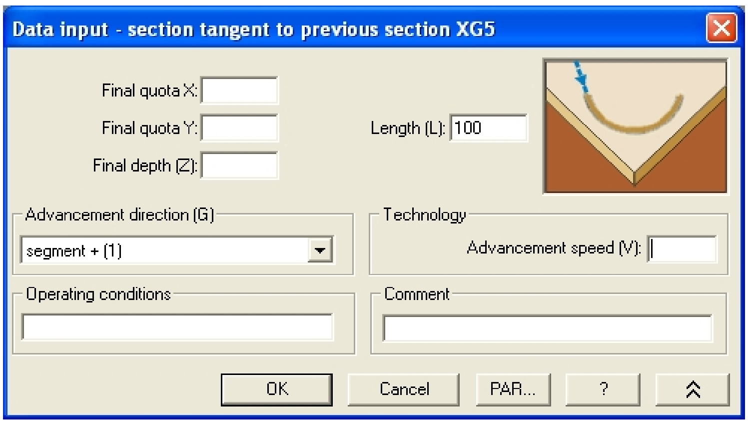
- 1 Segment in the same running direction as the previous section.
- -1 Segment in the opposite running direction from the previous section.
- 2 Clockwise circular milling.
- 3 Counter-clockwise circular milling.

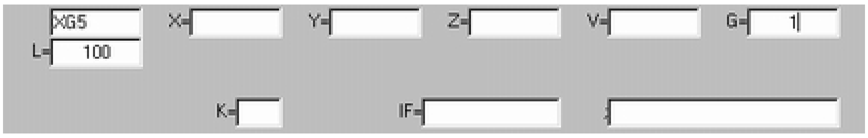
V Milling speed.
L Length of segment (only if G=1 or G=-1).

Parameters X, Y, G, L are mandatory.

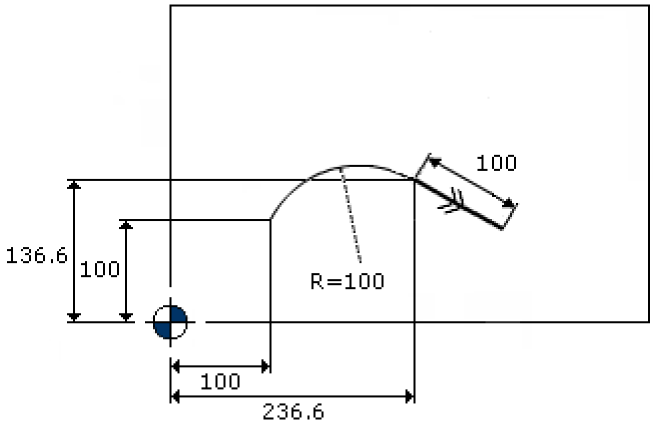
Examples:

1) Programming a section of linear milling tangent to an arc in the same routing direction. The starting point for the routing will be taken from the previous command. You must enter parameters L G.

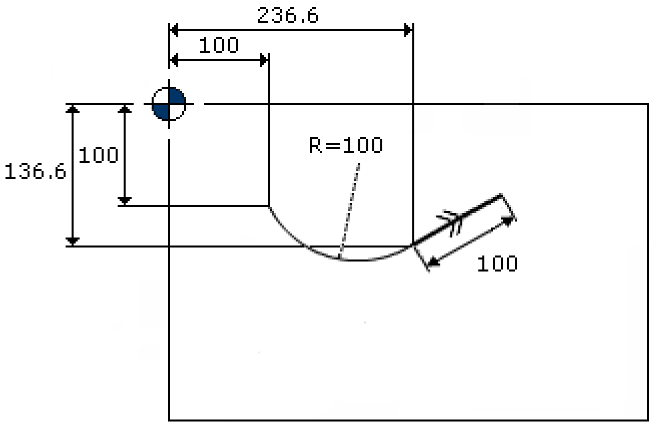




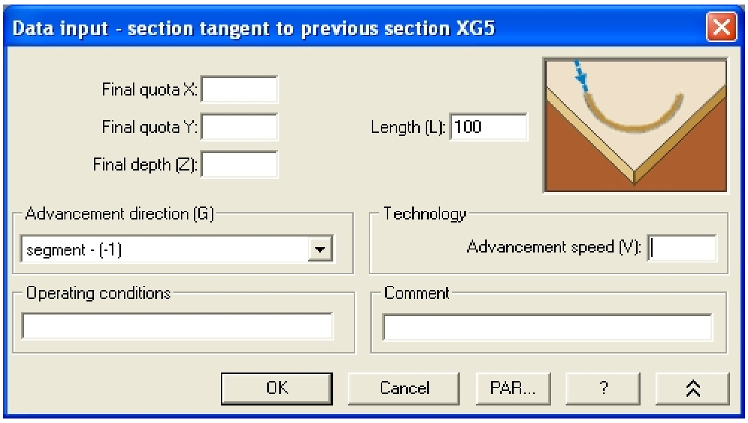
► *Front machine origin*

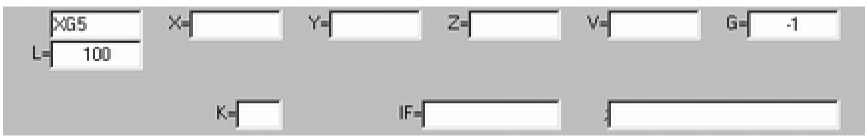


► *Rear machine origin*

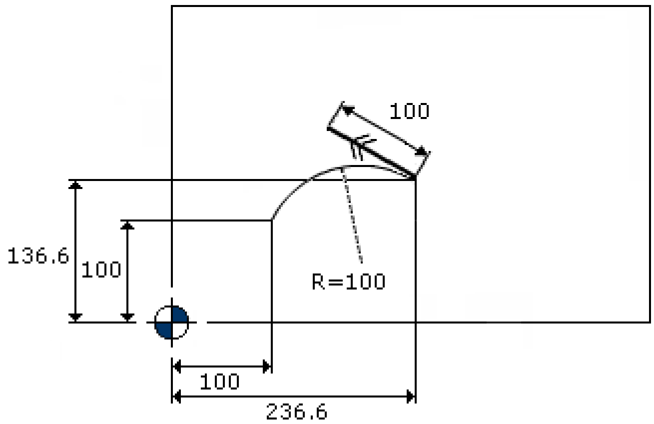


2) Programming a section of linear milling tangent to an arc in the opposite routing direction. The starting point for the routing will be taken from the previous command. You must enter parameters L G.

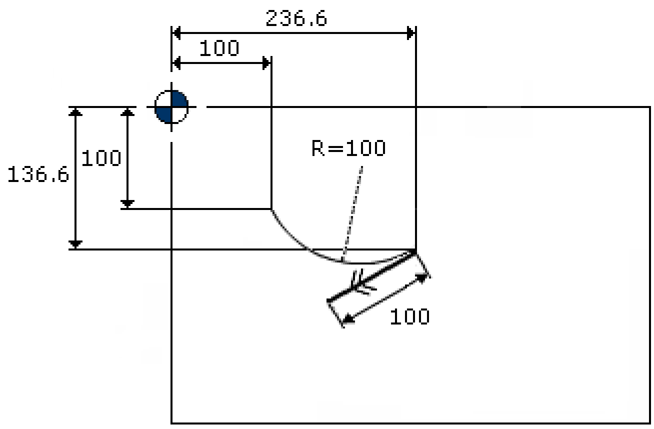




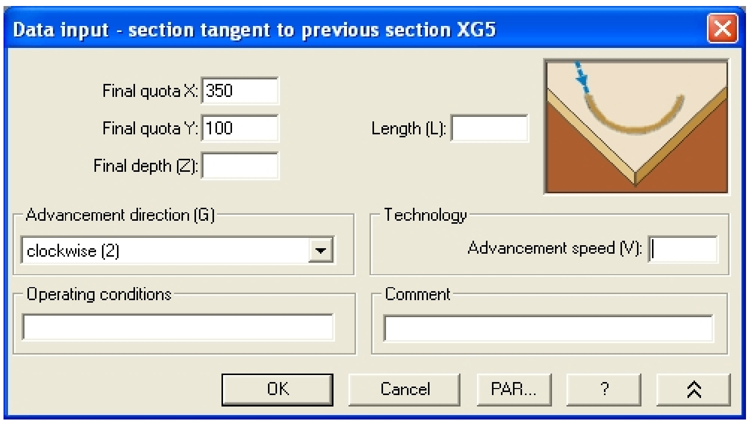
►*Front machine origin*

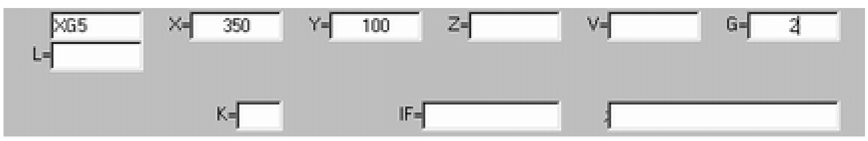


►*Rear machine origin*

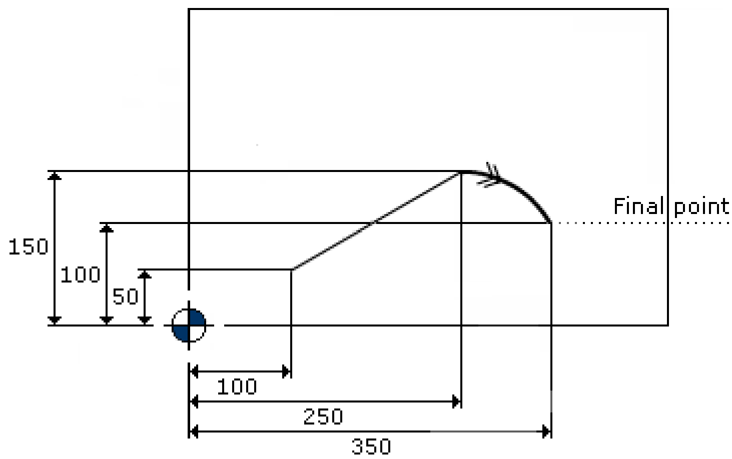


3) Programming a section of clockwise circular milling tangent to previous section. The starting point for the routing will be taken from the previous command. You must enter parameters X Y G.

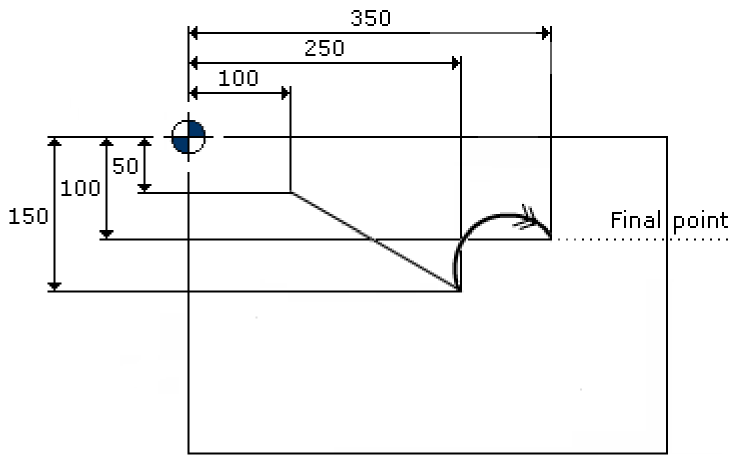




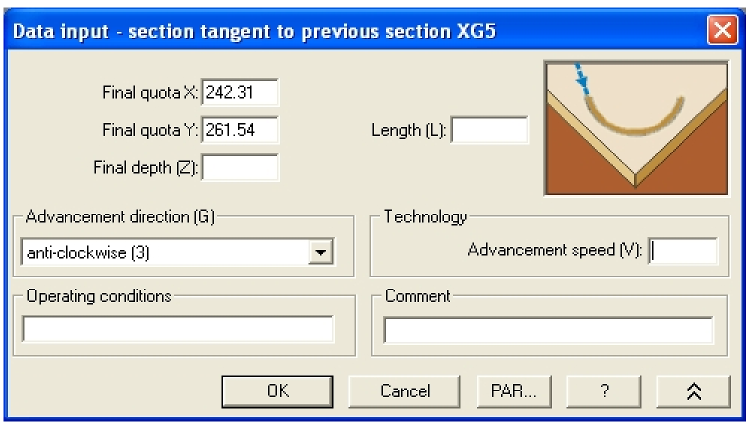
►*Front machine origin*

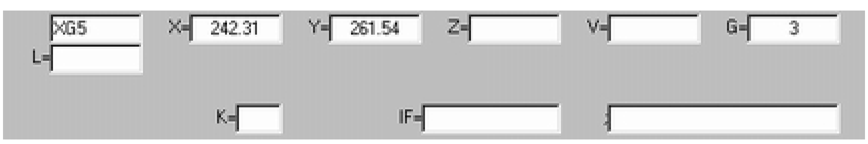


►*Rear machine origin*

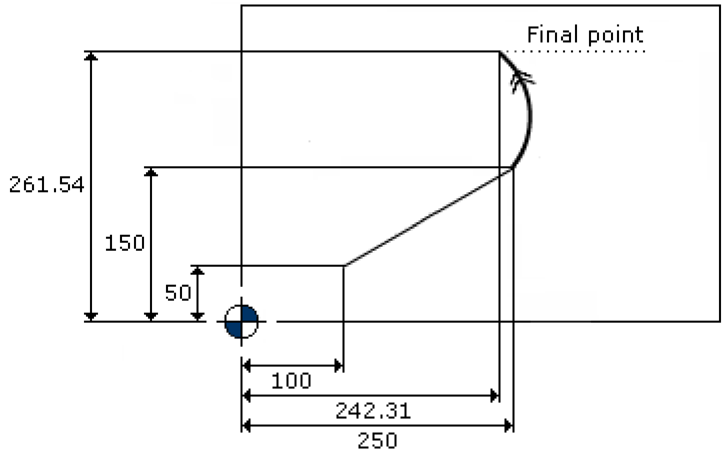


4) Programming a section of counter-clockwise circular milling tangent to previous section. The starting point for the routing will be taken from the previous command. You must enter parameters X Y G.

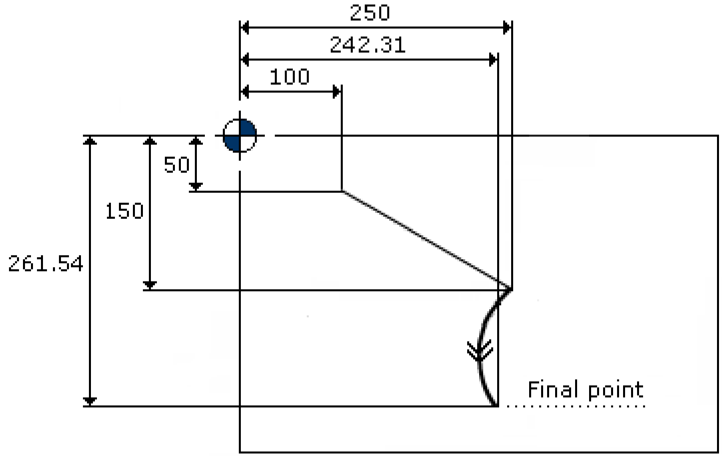




►*Front machine origin*



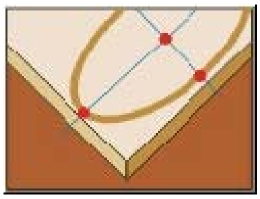
►*Rear machine origin*



Ellipse Arc - XEA

Defines an ellipse arc generating also start milling instruction.

Unit:
Complex routing



Parameters:

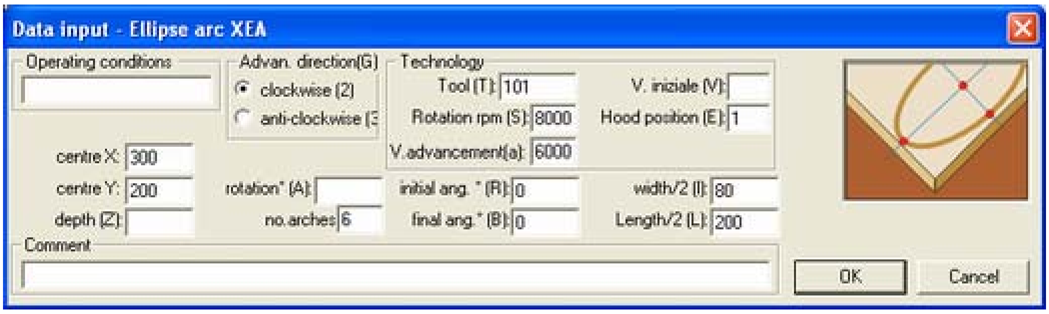
X Coordinate X of ellipse center.
Y Coordinate Y of ellipse center.
Z Milling depth.
A Angle of rotation compared to the center (front machine origin: positive in counter-clockwise direction; rear machine origin: positive in clockwise direction).
E Position of vacuum hood (see: Appendix D).
V Entry speed into piece.
S Rotation speed of tool.
Q Number of circular arches with which to approximate the ellipse per quadrant.
R Angle with axis X of initial point (front machine origin: positive in counter-clockwise direction; rear machine origin: positive in clockwise direction).
a Approach speed.
B Angle with axis X of final point (front machine origin: positive in counter-clockwise direction; rear machine origin: positive in clockwise direction).
I Length of minor semi-axis.
G Direction of path along arc: 2=clockwise, 3=counter- clockwise.
L Length of major semi-axis.
T Tool.

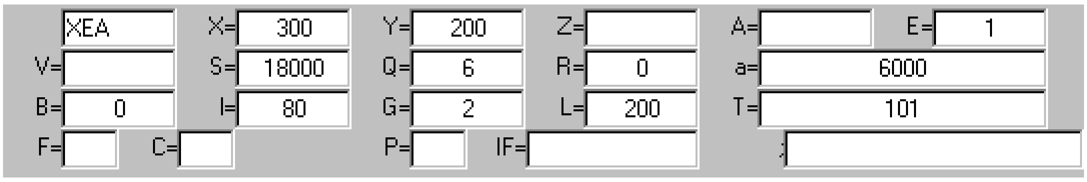
Parameters X, Y, R, B, l, L, G are required. Parameter I can never be greater than parameter L.

WARNING.
The XEA macro is a geometric approximation for arcs of an elliptical curve. The initial and final angles are referred to the circumference that circumscribes the arc of the ellipse, therefore they correspond perfectly to those represented by the perpendiculars to the tangents of the ellipse only in the points of intersection with the Cartesian axes: 0, 90, 180, 270. For all other cases in which it is necessary to make the initial and/or final points of the arc of the ellipse correspond to the initial or final point of another machining operation, the angles must be manually corrected in order to obtain the desired result.

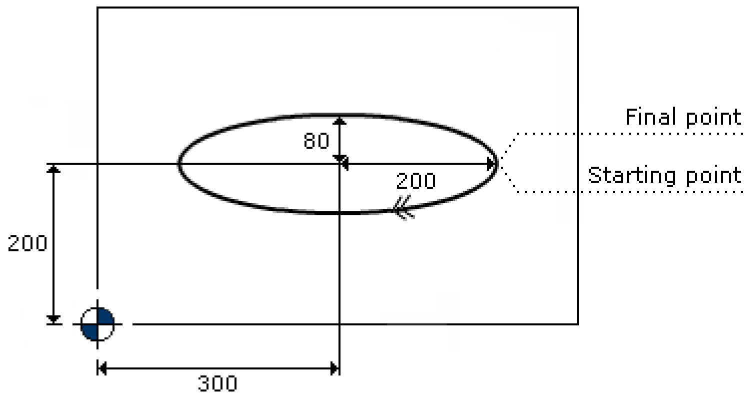
Examples:

1) Ellipse programming complete with clockwise direction with starting point towards positive X.

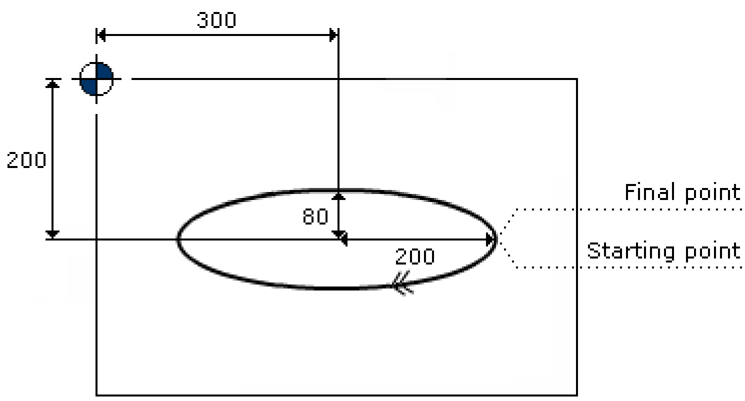




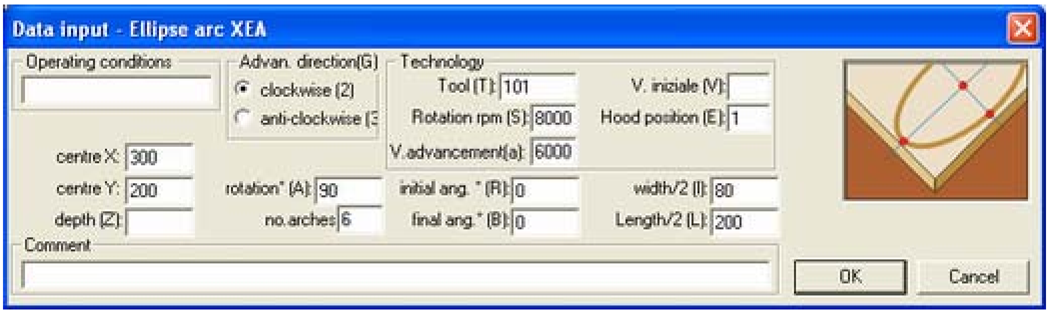
►*Front machine origin*

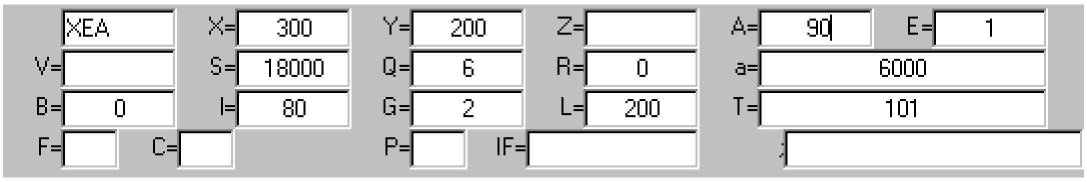


►*Rear machine origin*

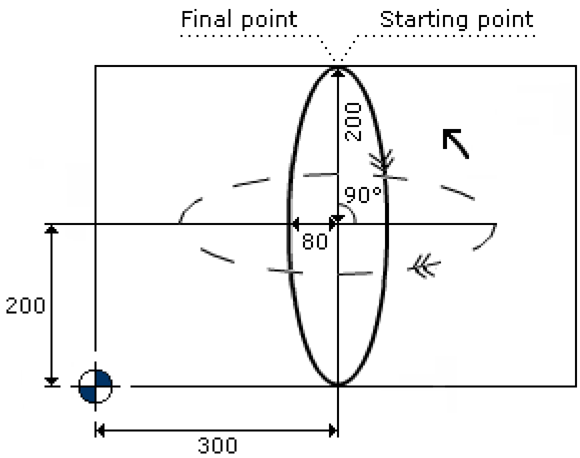


2) Ellipse programming complete with clockwise direction with starting point towards positive X, but rotated 90 degrees. Program horizontally, then rotate with the “rotation (A)” parameter.

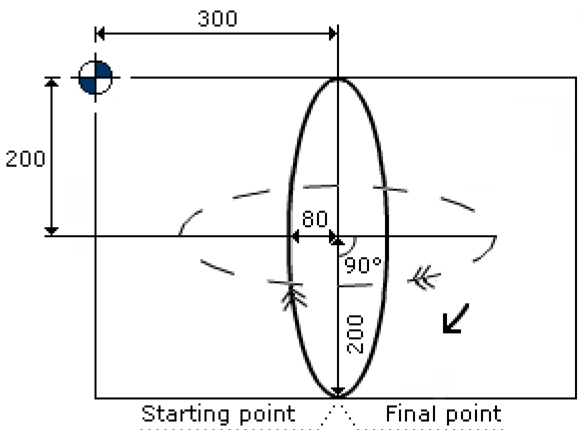




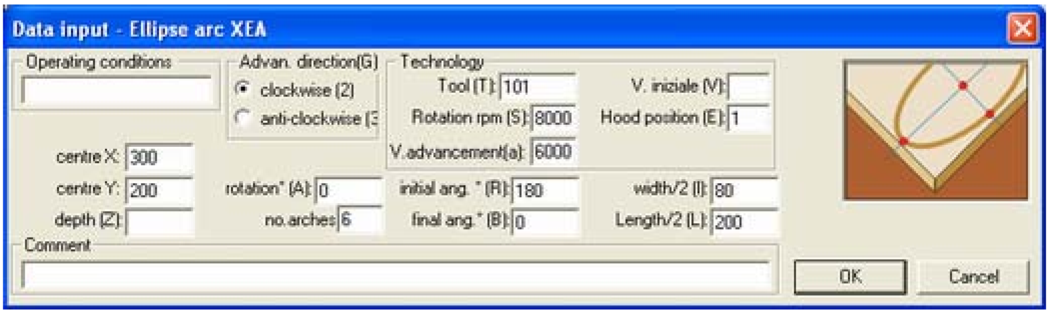
►*Front machine origin*

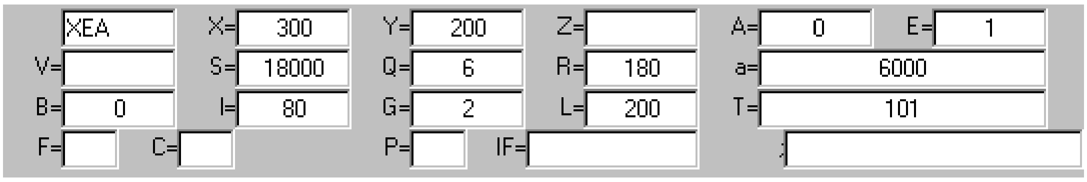


►*Rear machine origin*

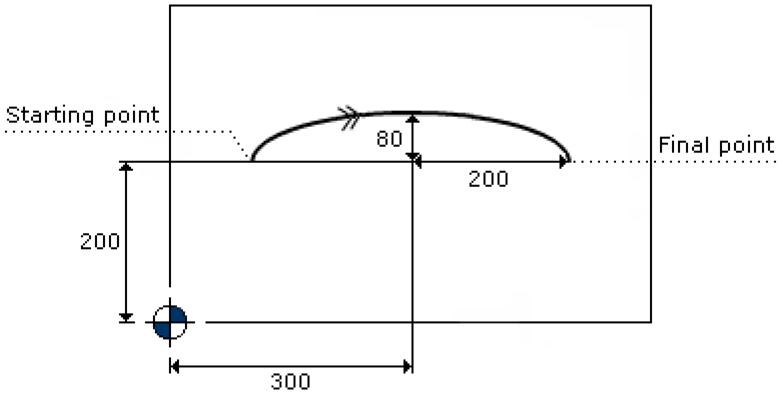


3) Programming an ellipse arc in clockwise direction with starting point towards negative X and arrival point toward positive X.

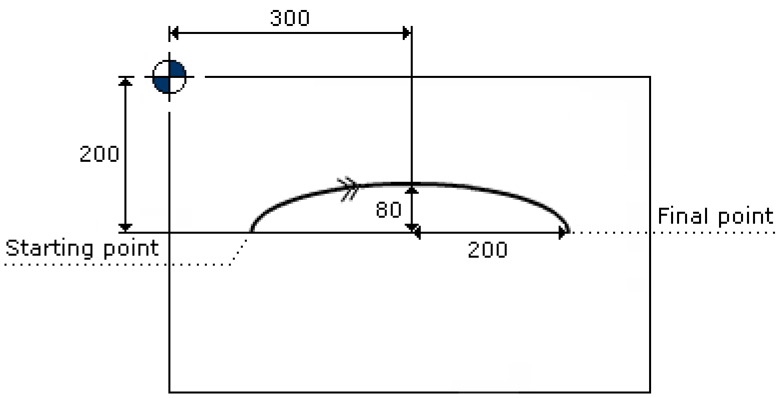




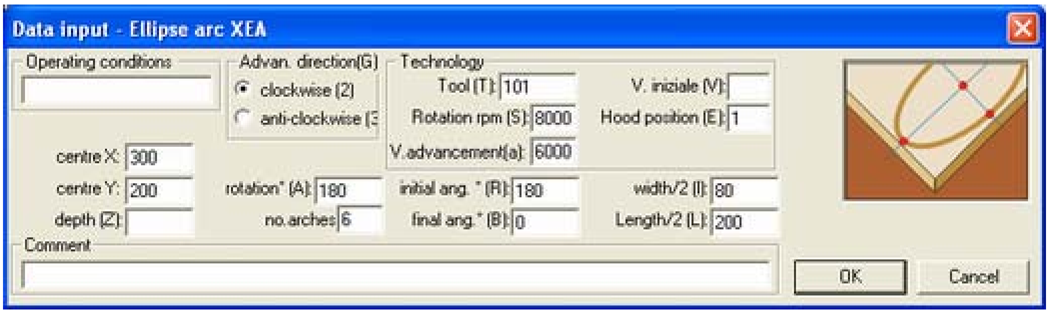
►*Front machine origin*

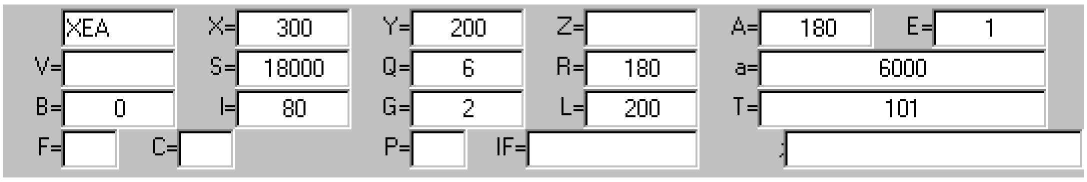


►*Rear machine origin*

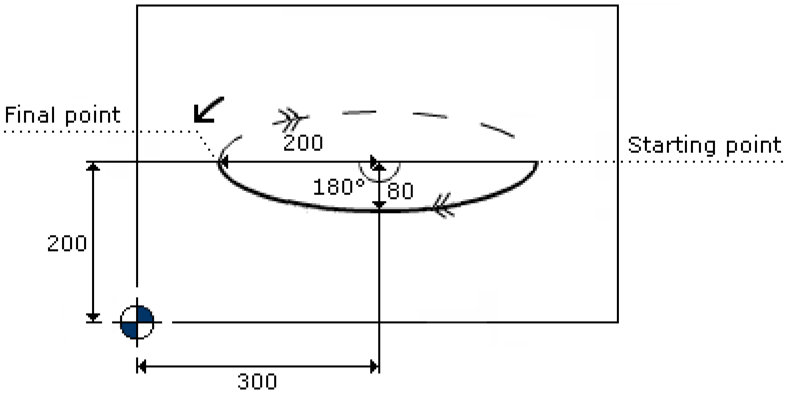


4) Programming an ellipse arc in clockwise direction with starting point towards positive X and arrival point toward negative X, but rotated 180 degrees. Program horizontally, then rotate with the “rotation (A)” parameter.

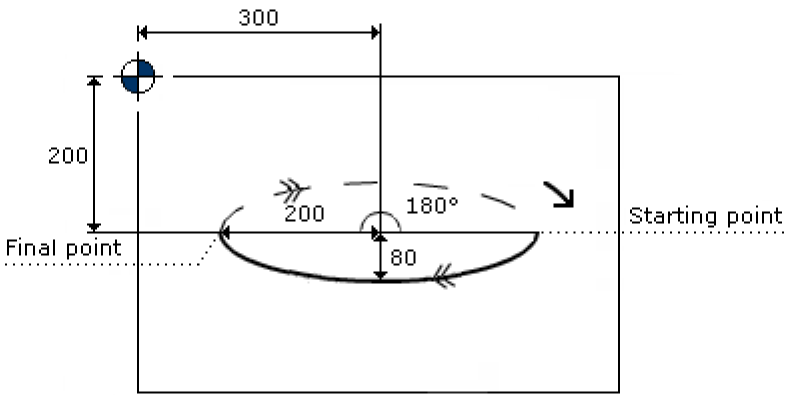




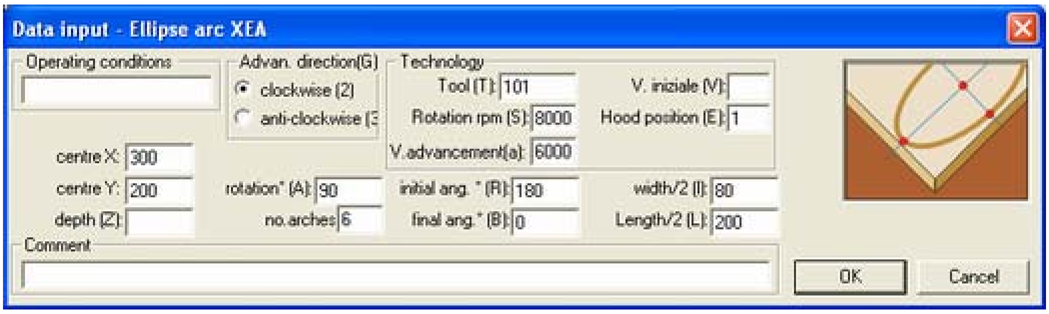
► *Front machine origin*

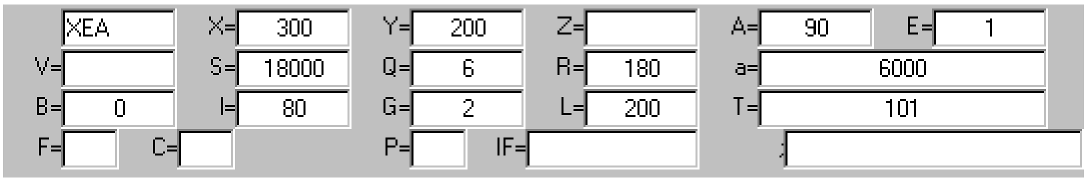


►*Rear machine origin*

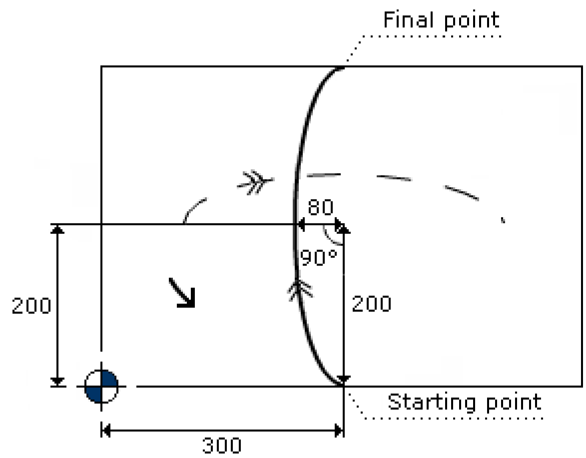


5) Programming an ellipse arc in clockwise direction with starting point towards negative Y and arrival point toward positive Y, but rotated 90 degrees. Program horizontally, then rotate with the “rotation (A)” parameter.

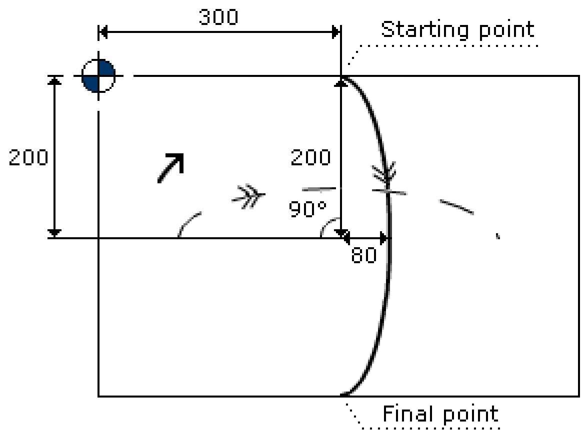




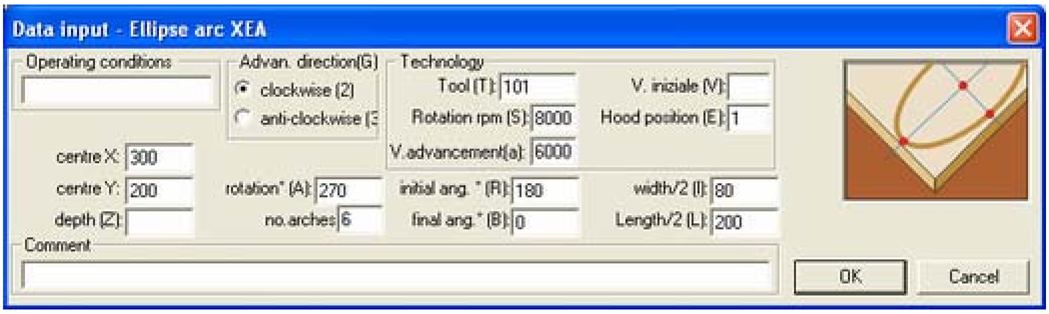
► *Front machine origin*

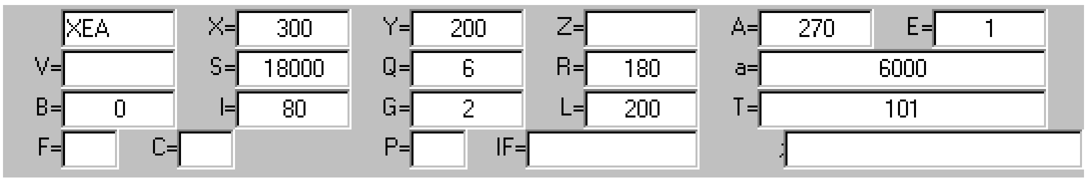


►*Rear machine origin*

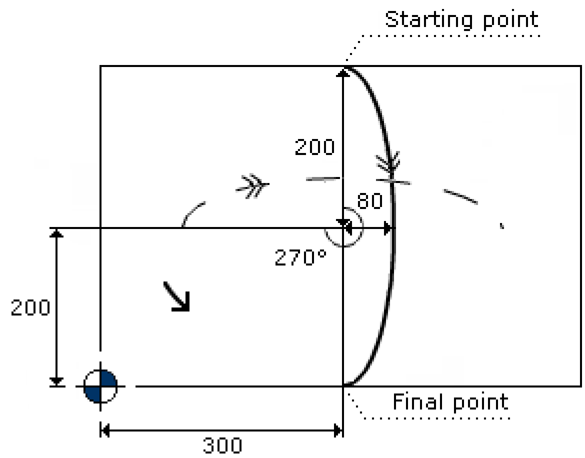


6) Programming an ellipse arc in clockwise direction with starting point towards positive Y and arrival point toward negative Y, but rotated 270 degrees. Program horizontally, then rotate with the "rotation (A)" parameter.

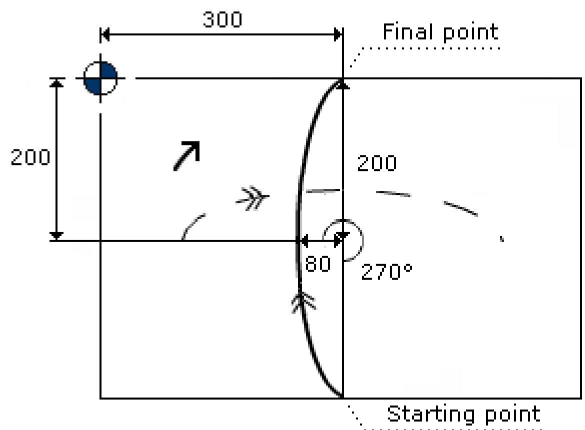




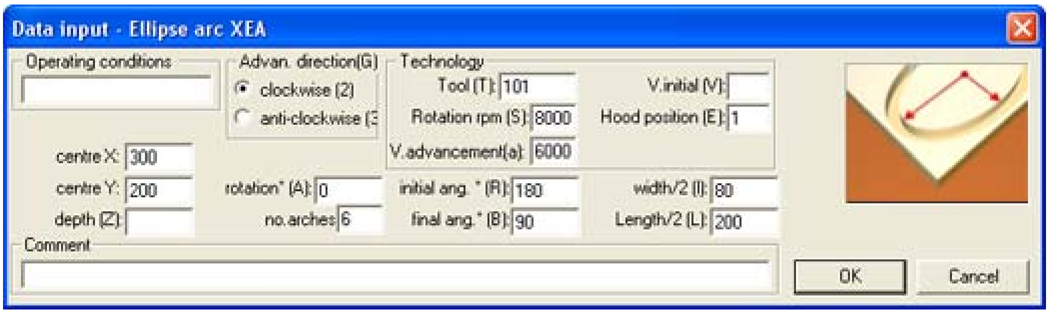
► *Front machine origin*

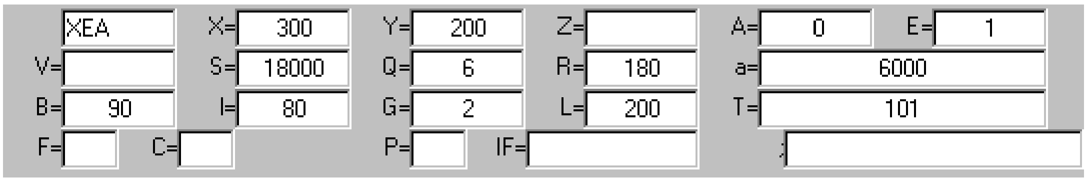


►*Rear machine origin*

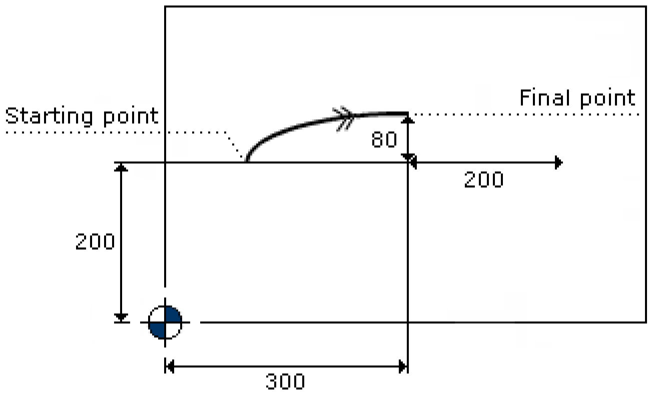


7) Programming an ellipse arc in clockwise direction with starting point towards negative X and arrival point toward positive X.

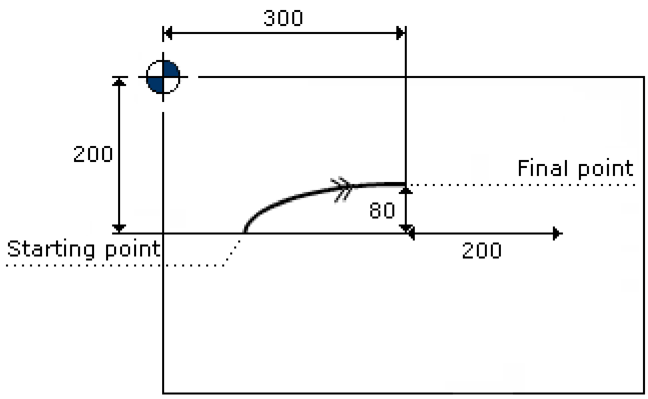




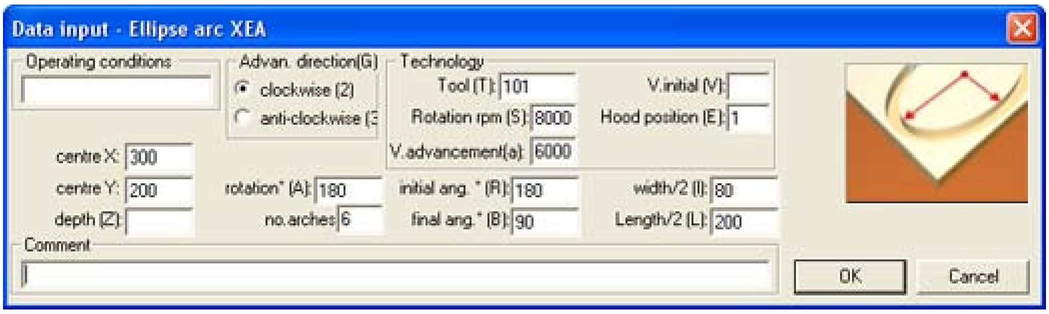
►*Front machine origin*

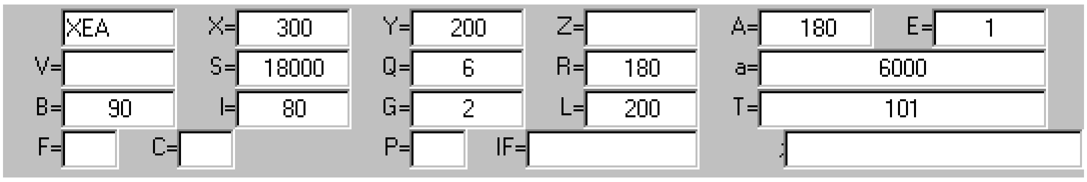


►*Rear machine origin*

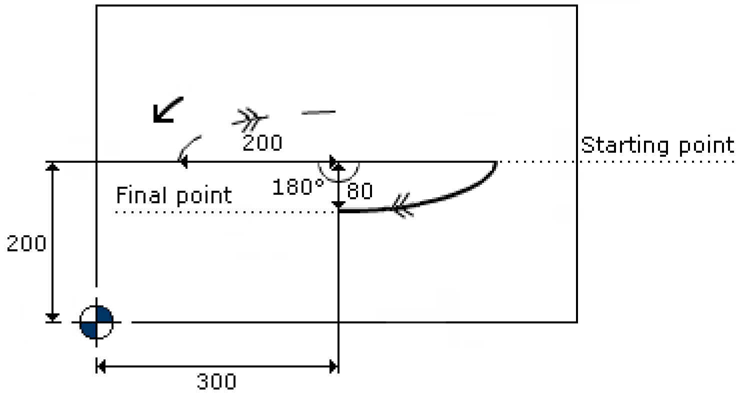


8) Programming an ellipse arc in clockwise direction with starting point towards positive X and arrival point toward negative X, but rotated 180 degrees. Program horizontally, then rotate with the “rotation (A)” parameter.

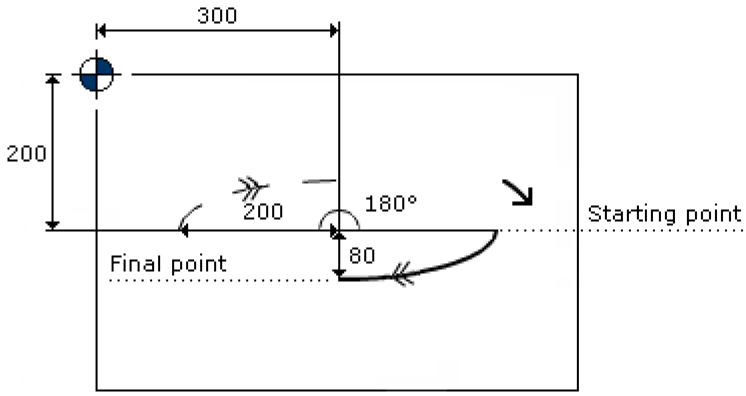




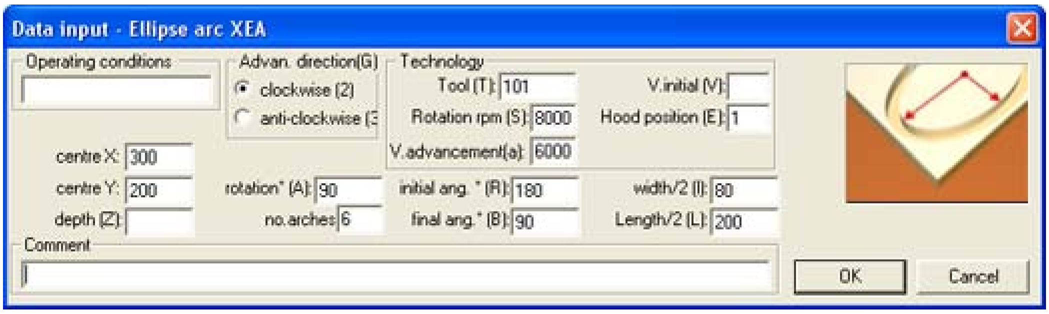
► *Front machine origin*

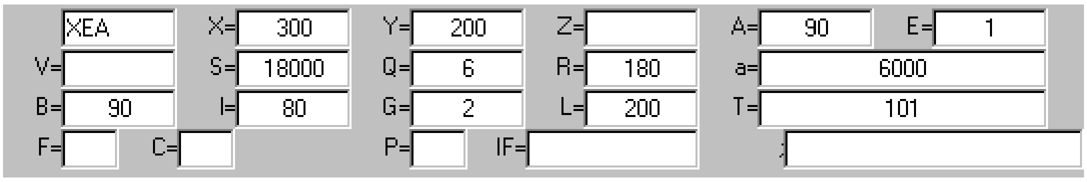


►*Rear machine origin*

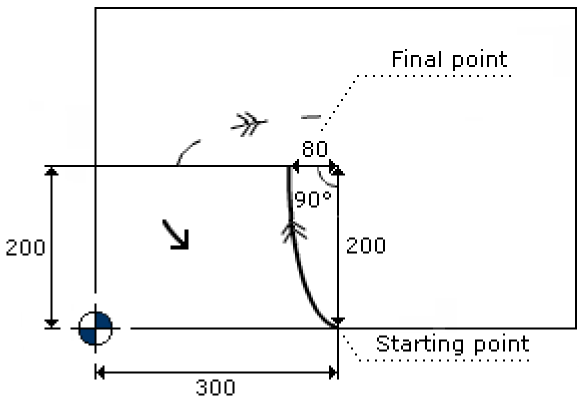


9) Programming an ellipse arc in clockwise direction with starting point towards negative Y and arrival point toward positive Y, but rotated 90 degrees. Program horizontally, then rotate with the “rotation (A)” parameter.

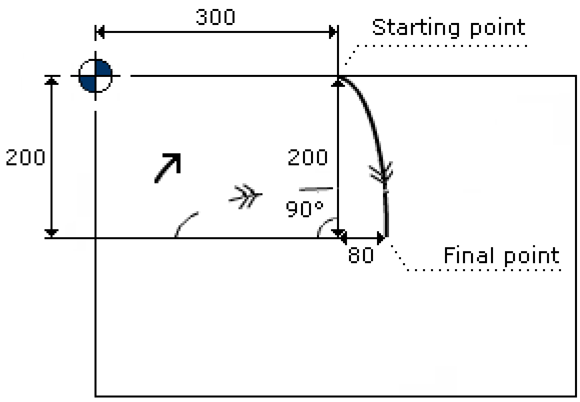




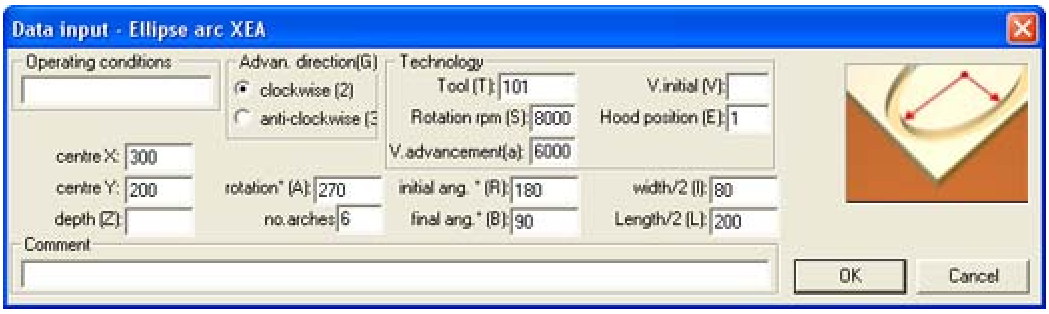
► *Front machine origin*

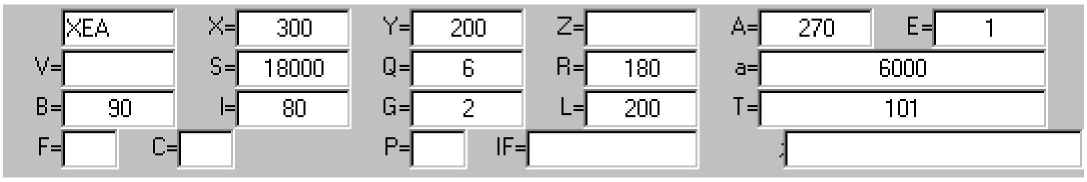


►*Rear machine origin*

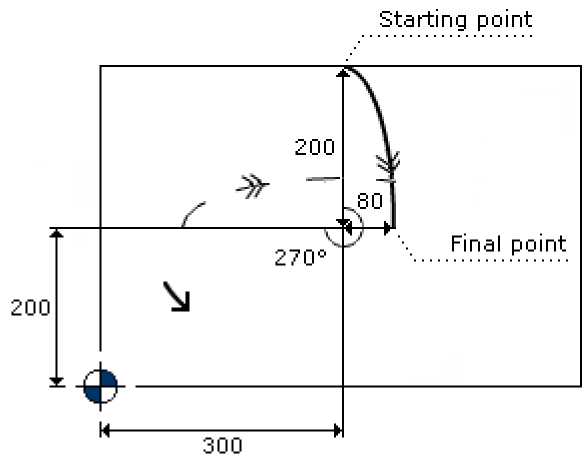


10) Programming an ellipse arc in clockwise direction with starting point towards positive Y and arrival point toward negative Y, but rotated 270 degrees. Program horizontally, then rotate with the “rotation (A)” parameter.

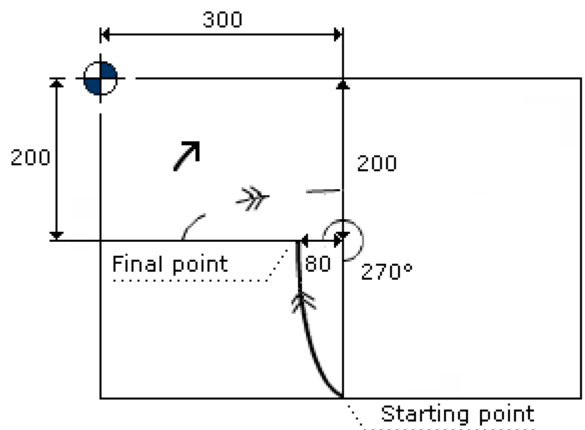




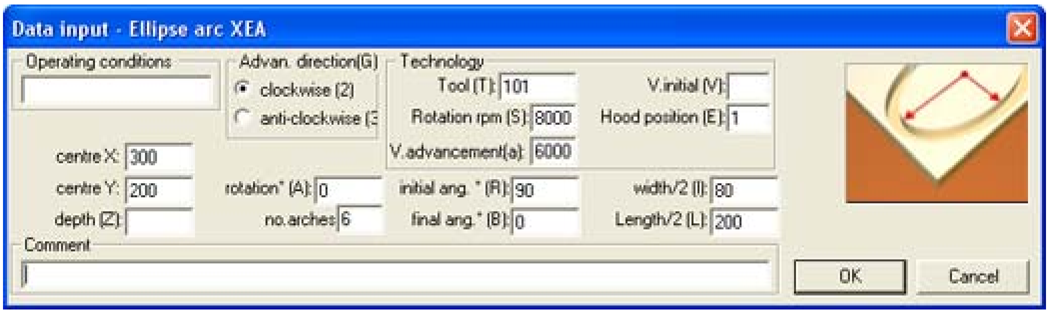
► *Front machine origin*

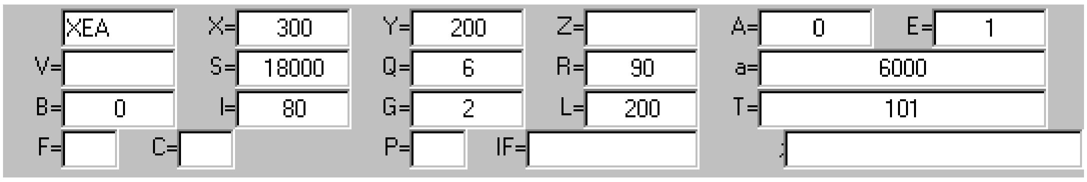


►*Rear machine origin*

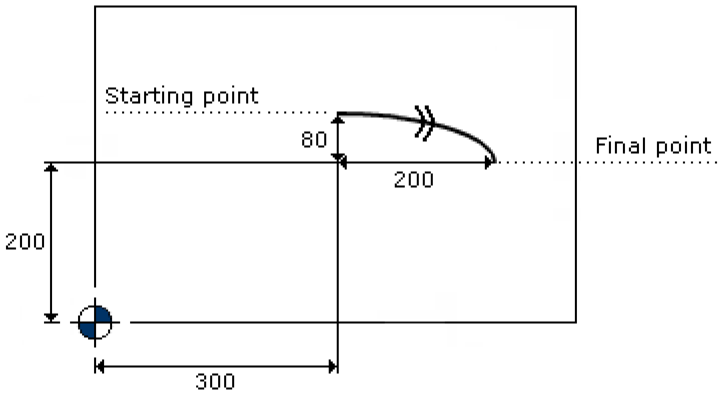


11) Programming an ellipse arc in clockwise direction with starting point towards negative X and arrival point toward positive X.

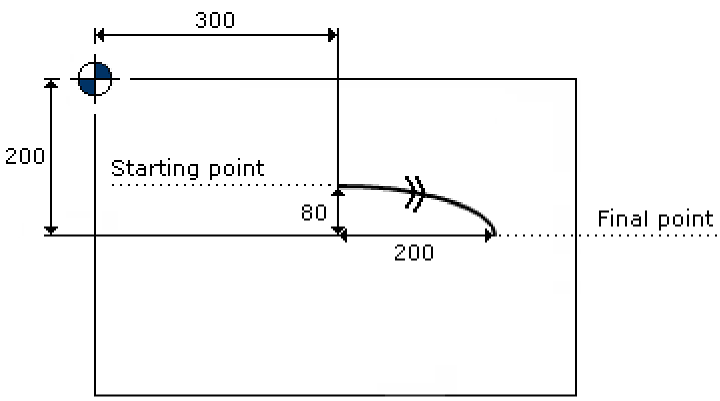




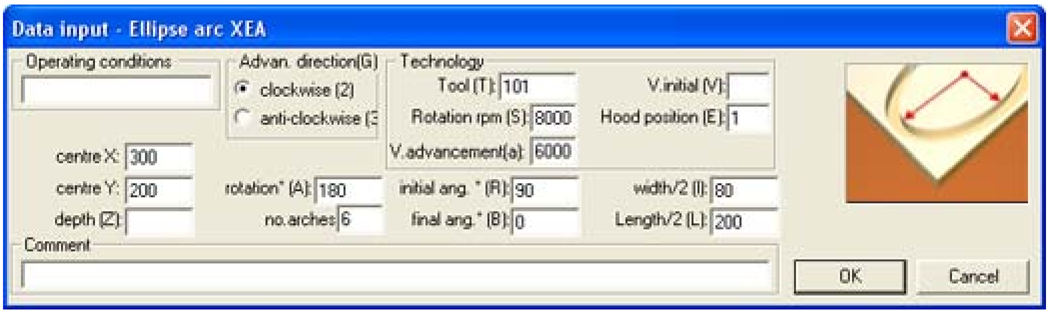
►*Front machine origin*

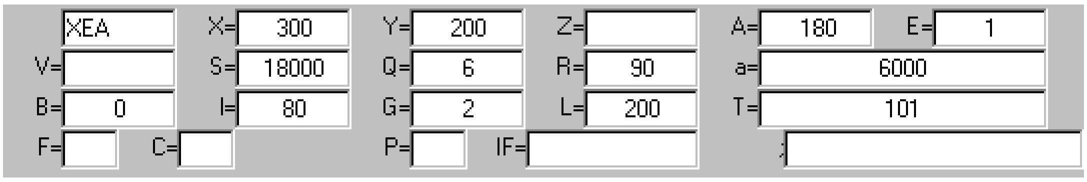


►*Rear machine origin*

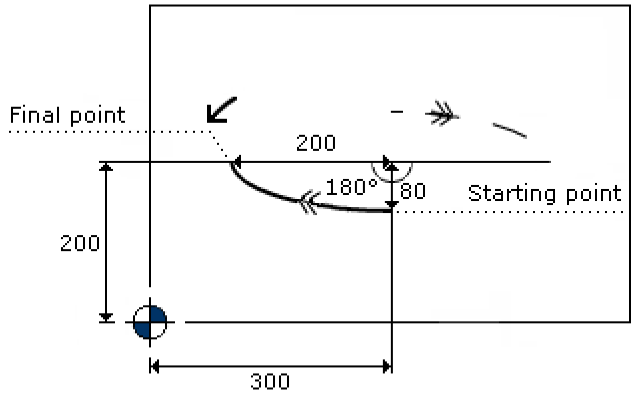


12) Programming an ellipse arc in clockwise direction with starting point towards positive X and arrival point toward negative X, but rotated 180 degrees. Program horizontally, then rotate with the "rotation (A)" parameter.

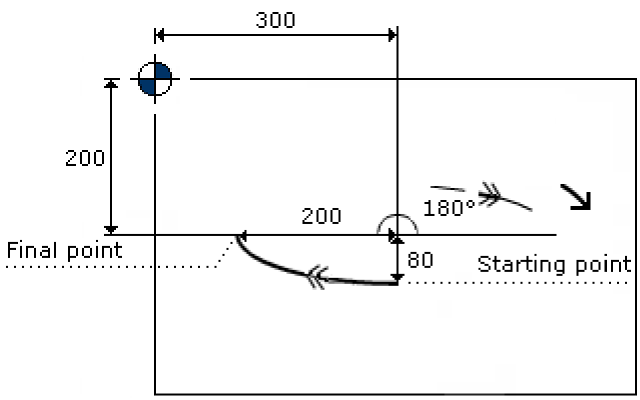




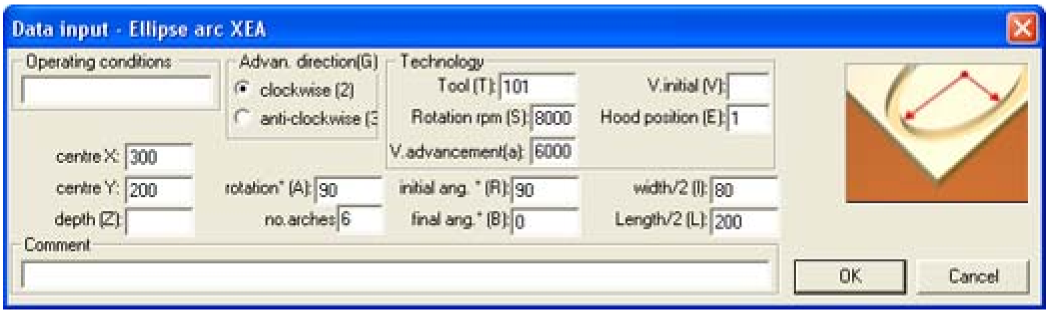
► *Front machine origin*

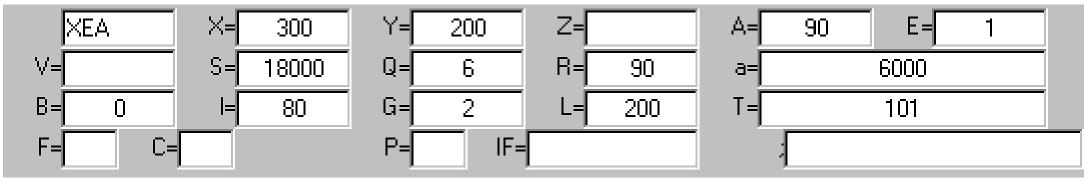


►*Rear machine origin*

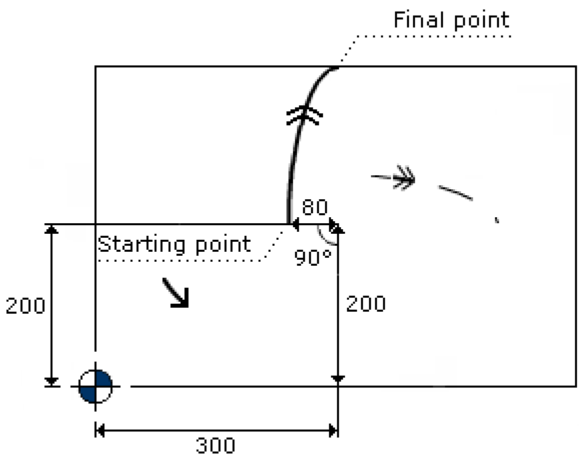


13) Programming an ellipse arc in clockwise direction with starting point towards negative Y and arrival point toward positive Y, but rotated 90 degrees. Program horizontally, then rotate with the "rotation (A)" parameter.

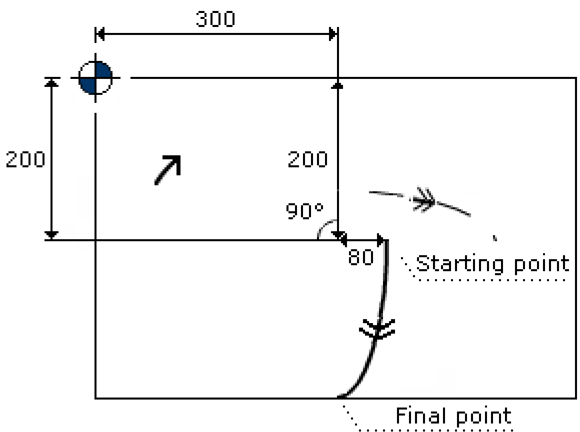




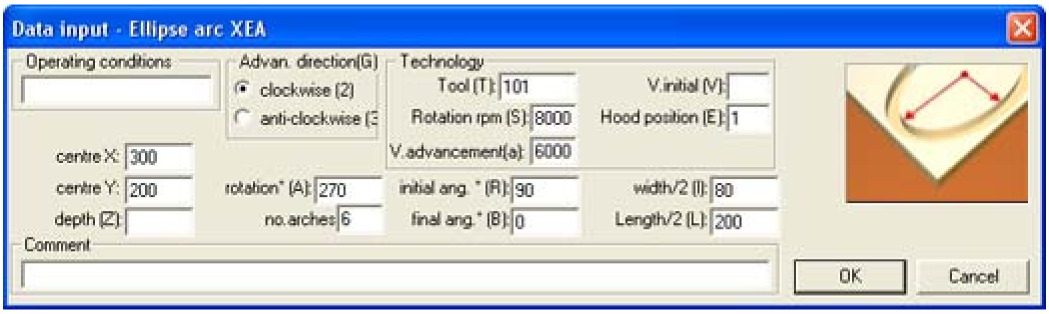
► *Front machine origin*

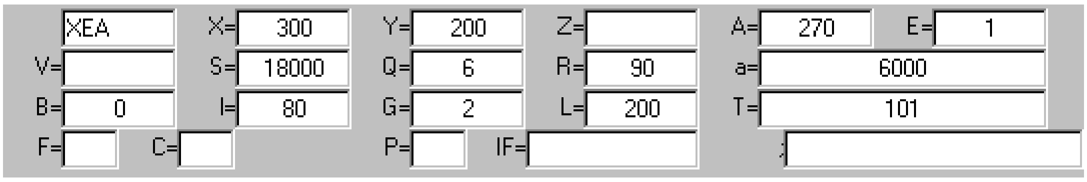


►*Rear machine origin*

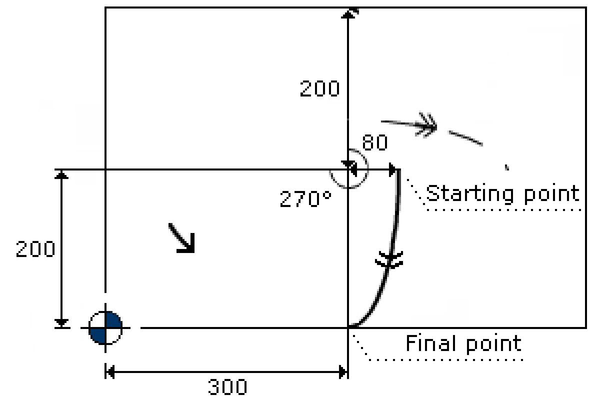


14) Programming an ellipse arc in clockwise direction with starting point towards positive Y and arrival point toward negative Y, but rotated 270 degrees. Program horizontally, then rotate with the “rotation (A)” parameter.

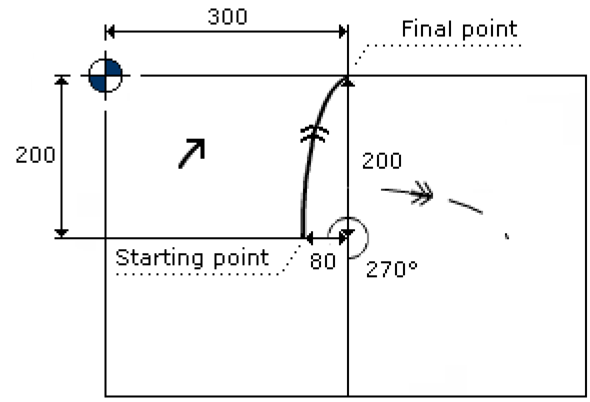




► *Front machine origin*



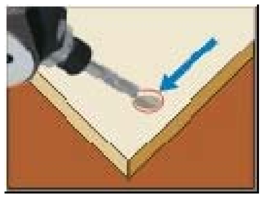
►*Rear machine origin*



Slanted Begin Milling - XG0R

Begins a milling operation with a tool slanted on a plane which is not square with the surfaces of the panel. BR is always referred to side 1. The current work side must always be side 1.

Unit:
Standard routing

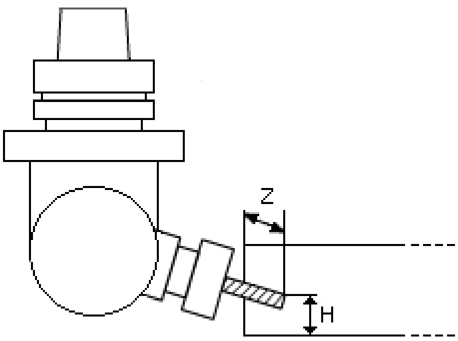


Basic parameters:

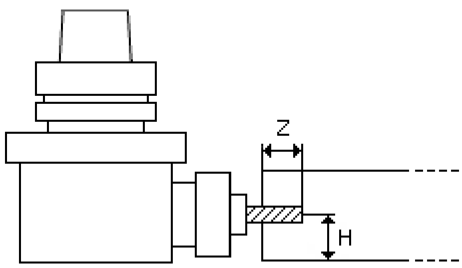
X Coordinate X for beginning of profile.
Y Coordinate Y for beginning of profile.
Z Depth of beginning of profile (measured according to the tool angle).
H Height of machining from the working table (position of the tool from the working table of the machine). Defaults to DZ.
T Tool.
B Tool angle relative to the vertical plane (only for the Prisma group).

Extended parameters:

N Name of profile (see XGREP).
I Entry side (0, 1, 2, 3, 4).
S Rotation speed of tool.
V Entry speed into piece.
E Position of vacuum hood (see: Appendix D).
D Out machining Quota.
A Milling rotation angle.
Q Angular offset relative to parameter A (for special heads Benz configured in tooling with type M, N, O; see: Configuration Manuale for the Heads).

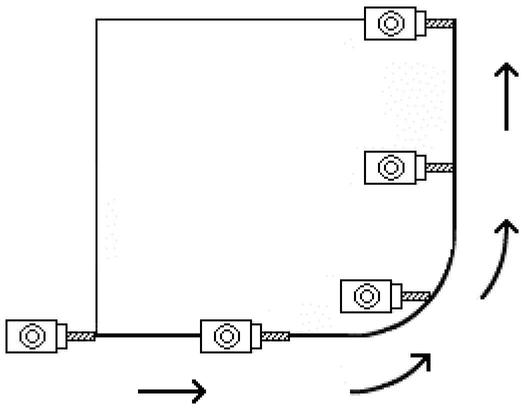


Inclined tool



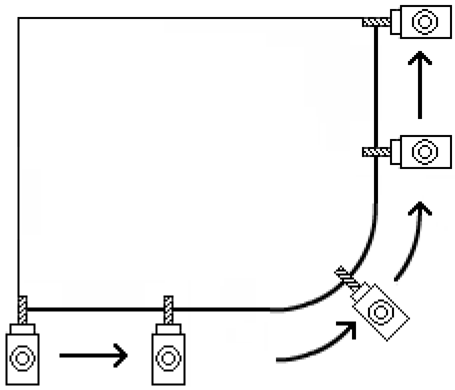
Tool at 90°

With parameter I=0.



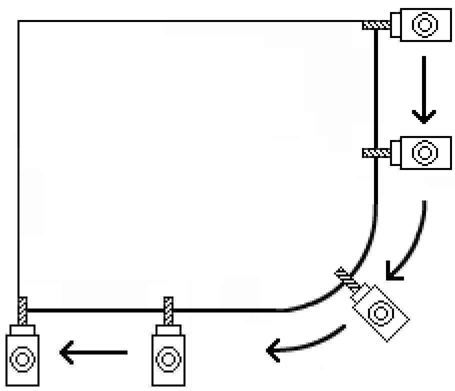
If parameter I=0 and parameter A=0 the tool is parallel with the positive X-axis. Values other than 0 for parameter A cause an angular offset in tool orientation. The angle of the tool remains fixed during machining (fixed position).

With parameter I=1.



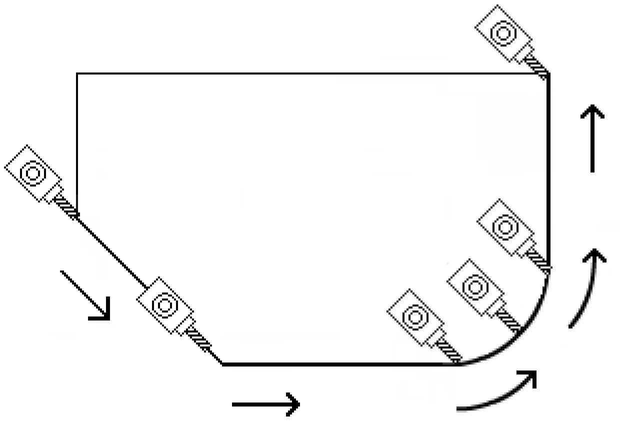
If parameter I=1 and A=0 the tool is perpendicular to the programmed trajectory on the right-hand side of the feed path. Values other than 0 for parameter A cause an angular offset in tool orientation. The tool remains perpendicular to the path during machining (interpolated position).

With parameter I=2.



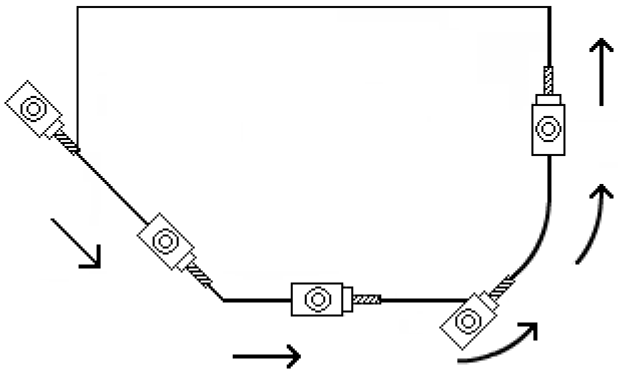
If parameter I=2 and A=0 the tool is perpendicular to the programmed trajectory on the left-hand side of the feed path. Values other than 0 for parameter A cause an angular offset in tool orientation. The tool remains perpendicular to the path during machining (interpolated position).

With parameter I=3.



If parameter I=3 and A=0 the tool is parallel with the programmed trajectory. Values other than 0 for parameter A cause an angular offset in tool orientation. The angle of the tool remains fixed during machining (fixed position).

With parameter I=4.

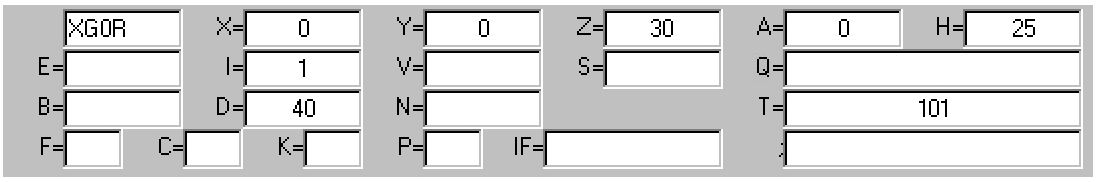


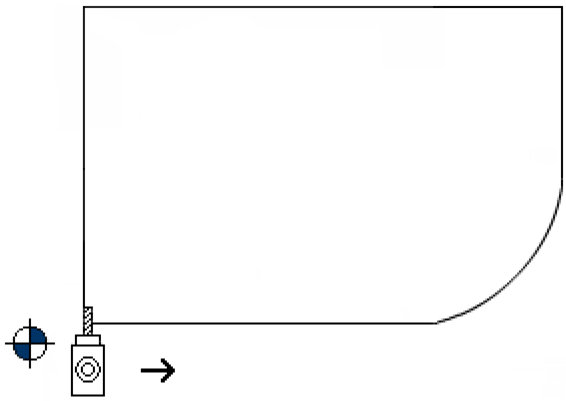
If parameter I=4 and A=0 the tool is parallel with the programmed trajectory. Values other than 0 for parameter A cause an angular offset in tool orientation. The tool remains parallel with the path during machining (interpolated position).

Example:

►*Front machine origin*

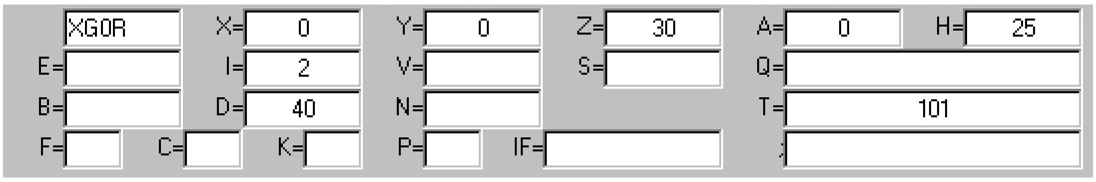
Programming a slanted begin milling with perpendicular tool and on the right-hand side of the trajectory to be followed.

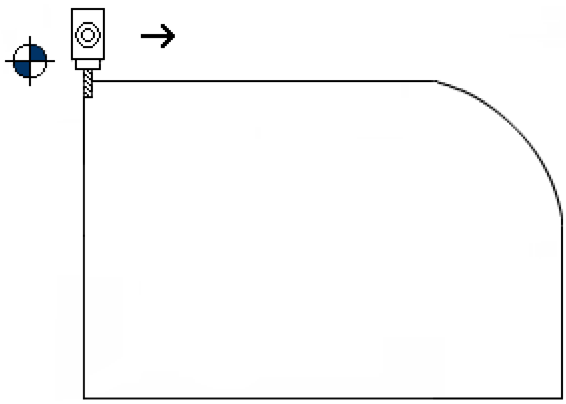




►*Rear machine origin*

Programming a slanted begin milling with perpendicular tool and on the left-hand side of the trajectory to be followed.





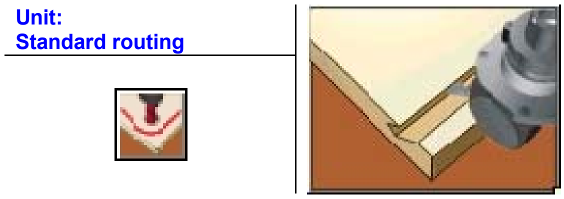
N.B. Automatic entry and exit are not envisaged for inclined routing.
The instructions that follow XG0R can only be XG1R, XG2R, XG3R, XG5R.

WARNING

Incorrect programming of machining with slanted tools may cause damages to the structure of the machine. It is advisable that a SIMULATION of this type of operation be executed first.

Slanted Linear Milling - XG1R

Makes it possible to perform linear milling on a surface that is slanted compared to the orthogonality of the panel surfaces; it must be used with slanted tools. Instruction XG1R always refers to face 1; for this reason the current work surface must always be face 1.
This instruction must be preceded by a begin milling with slanted tool or by another milling instruction with slanted tool.



Basic parameters:

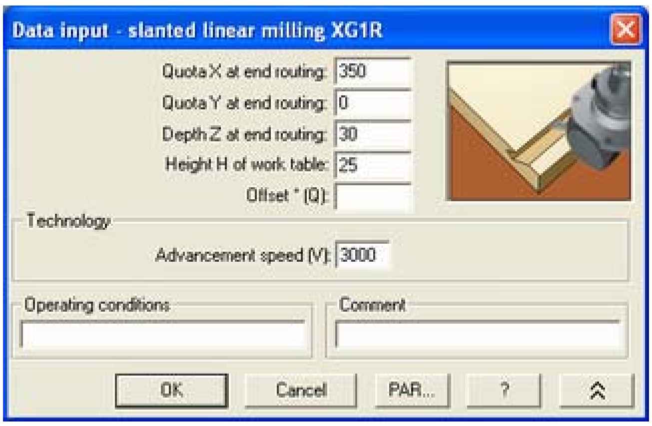
X Coordinate X for end of milling.
Y Coordinate Y for end of milling.
Z Depth of end of milling (measured according to the tool angle).
H Height of end point of machining from working table (position of the tool from the working table of the machine).
Q Angular offset relative to parameter A (for special heads Benz configured in tooling with type M, N, O; see: Configuration Manuale for the Heads).

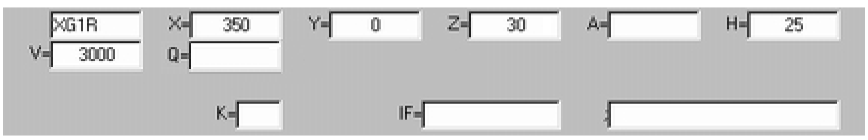
Extended parameters:

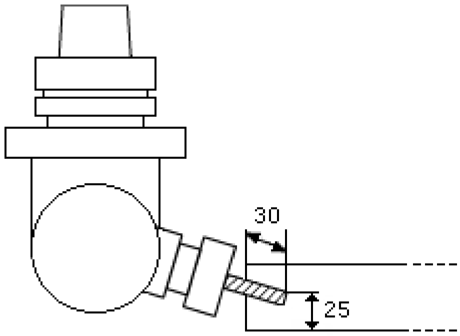
A Angle of rotation for milling.
V Milling speed.

Example:

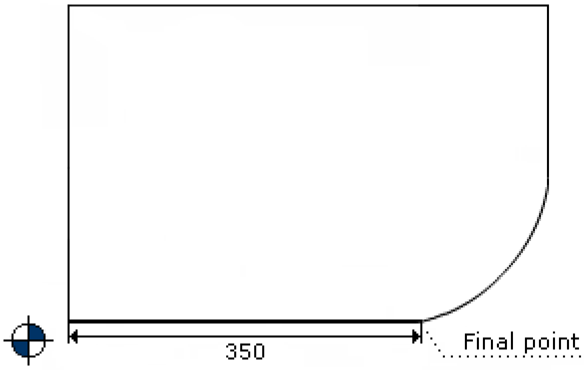
Programming slanted linear milling with starting point X=0, Y=0.



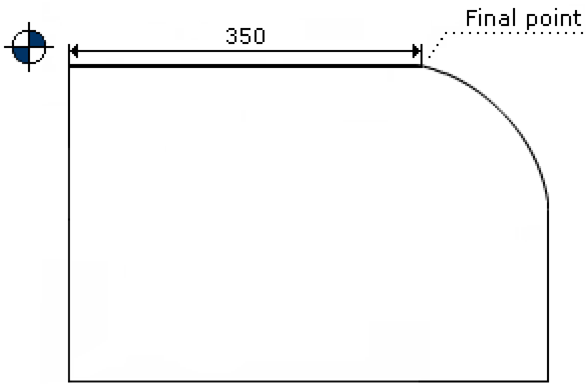




►*Front machine origin*



►*Rear machine origin*



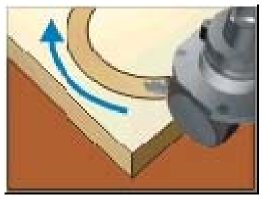
WARNING

Incorrect programming of machining with slanted tools may cause damages to the structure of the machine. It is advisable that a SIMULATION of this type of operation be executed first.

Slanted Clockwise Circular Milling - XG2R

Defines a section of circular milling (or circle arc) on surface that is inclined compared to the orthogonality of the panel surfaces, with a clockwise approach. In alternative to the definition of the center, it is possible to define the radius of the arc according to the following convention: if the radius is positive, the center falls to the left of the imaginary line going from the beginning to the end point of the arc, otherwise, the center falls to the right of this line. The instruction XG2R is always referred to as face 1; for this reason the face of the current work must always be face 1.
This instruction must be preceded by a begin milling with slanted tool or by another milling instruction with slanted tool.

Unit:
Standard routing



Basic parameters:

X Coordinate X for end of milling.
Y Coordinate Y for end of milling.
Z Depth of end of milling (measured according to the tool angle).
I Coordinate X of center of arc.
J Coordinate Y of center of arc.
H Height of end point of machining from working table (position of the tool from the working table of the machine).
Q Angular offset relative to parameter A (for special heads Benz configured in tooling with type M, N, O; see: Configuration Manuale for the Heads).

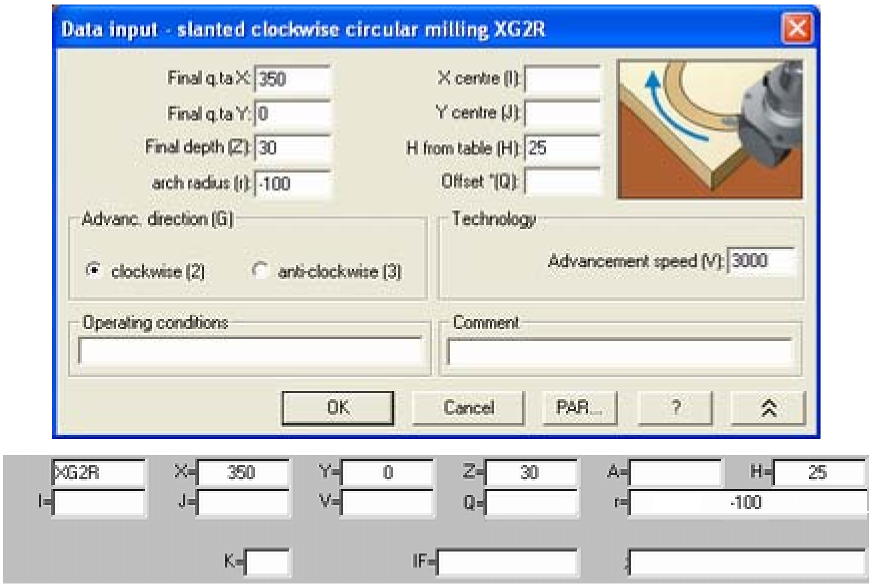
Extended parameters:

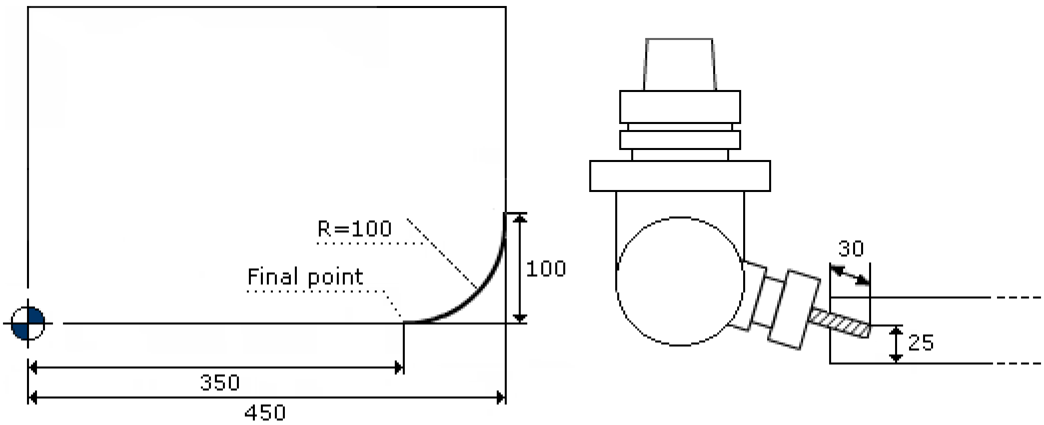
r Arc radius.
A Angle of rotation for milling.
V Milling speed.

Example:

►*Front machine origin*

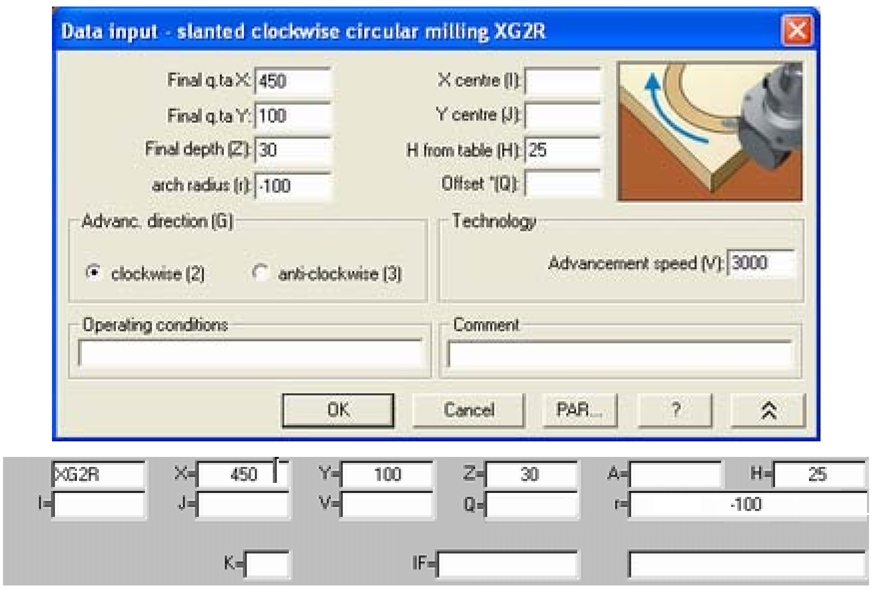
Programming slanted clockwise circular milling with starting point X=450, Y=100.

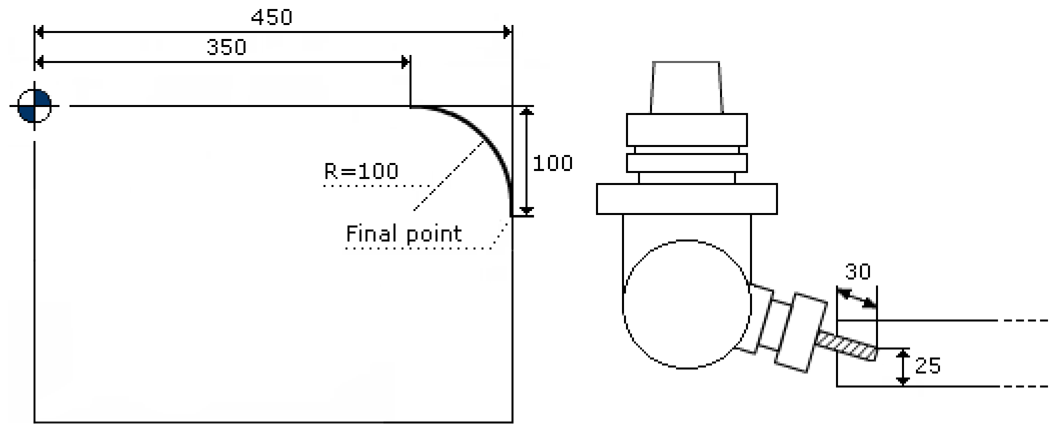




►*Rear machine origin*

Programming slanted clockwise circular milling with starting point X=350, Y=0.





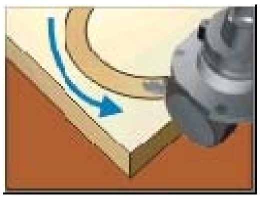
WARNING

Incorrect programming of machining with slanted tools may cause damages to the structure of the machine. It is advisable that a SIMULATION of this type of operation be executed first.

Slanted Counter-Clockwise Circular Milling - XG3R

Defines a section of circular milling (or circle arc) on surface that is inclined compared to the orthogonality of the panel surfaces, with a counter-clockwise approach. In alternative to the definition of the center it is possible to define the radius of the arc according to the following convention: if the radius is positive, the center falls to the left of the imaginary line going from the beginning to the end point of the arc, otherwise, the center falls to the right of this line. The instruction XG3R is always referred to as face 1; for this reason the face of the current work must always be face 1.
This instruction must be preceded by a begin milling with slanted tool or by another milling instruction with slanted tool.

Unit:
Standard routing



Basic parameters:

X Coordinate X for end of milling.
Y Coordinate Y for end of milling.
Z Depth of end of milling (measured according to the tool angle).
I Coordinate X of center of arc.
J Coordinate Y of center of arc.
H Height of end point of machining from working table (position of the tool from the working table of the machine).
Q Angular offset relative to parameter A (for special heads Benz configured in tooling with type M, N, O; see: Configuration Manuale for the Heads).

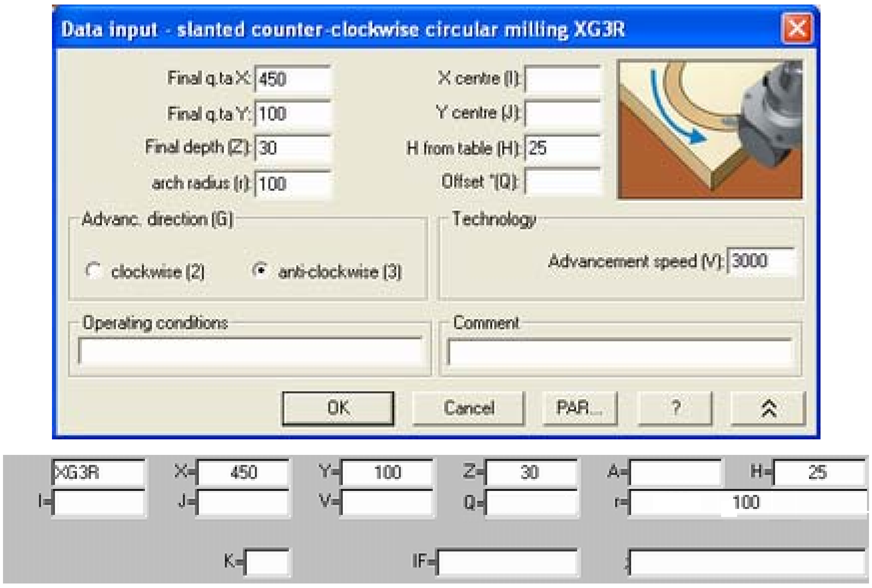
Extended parameters:

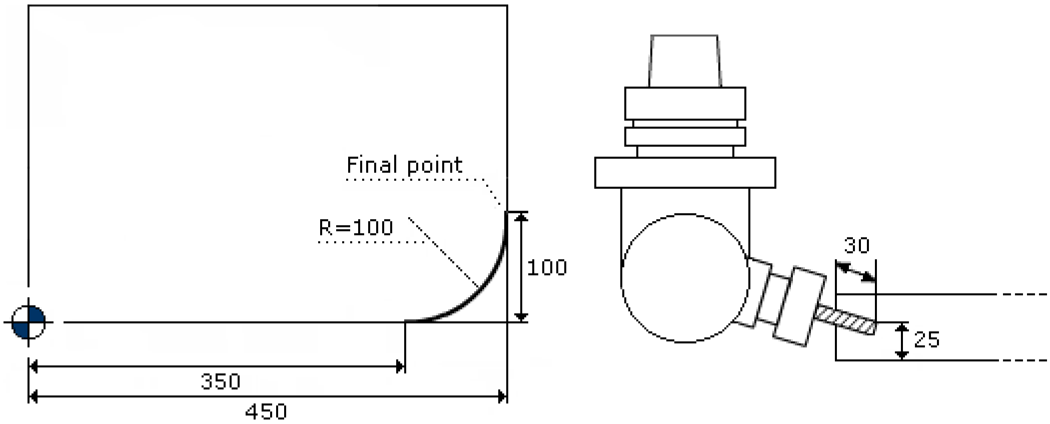
r Arc radius.
A Angle of rotation for milling
V Milling speed.

Example:

►*Front machine origin*

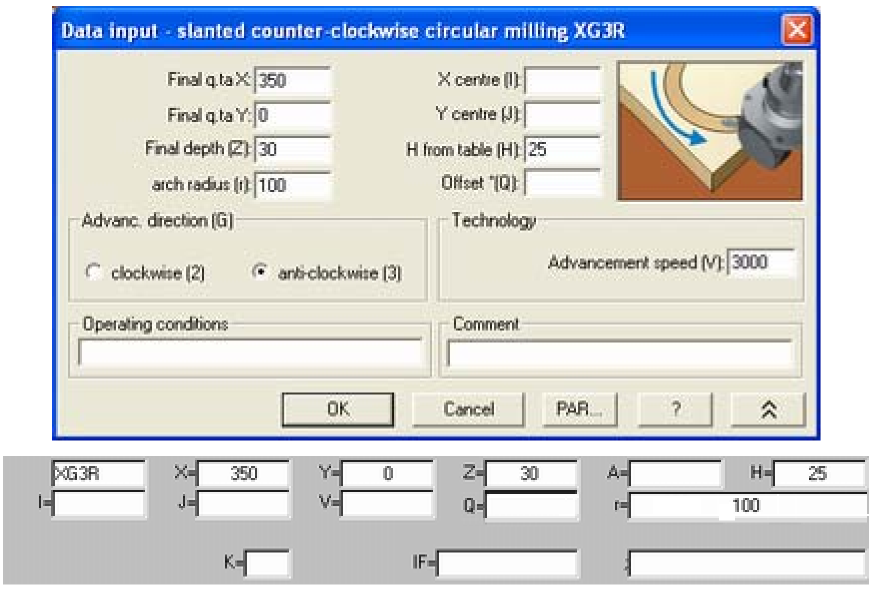
Programming slanted counter-clockwise circular milling with starting point X=350, Y=0.

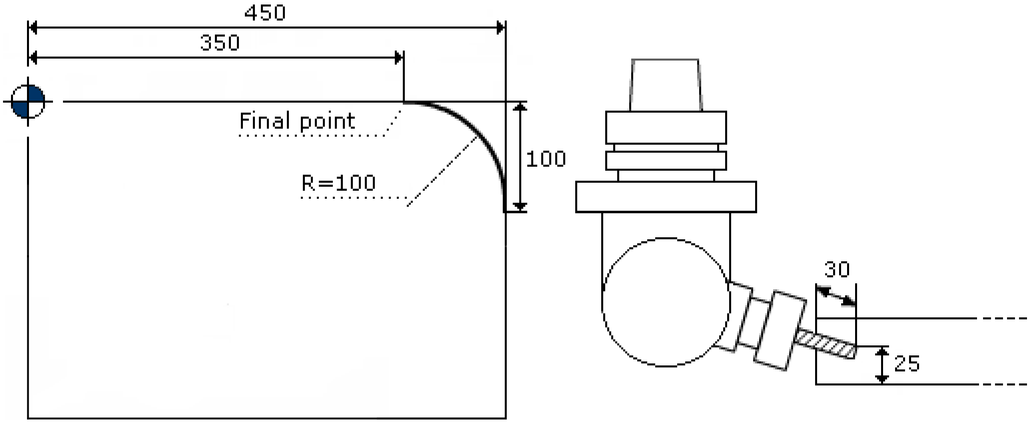




► *Rear machine origin*

Programming slanted counter-clockwise circular milling with starting point X=450, =100.





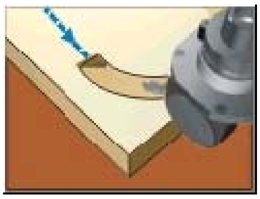
WARNING

Incorrect programming of machining with slanted tools may cause damages to the structure of the machine. It is advisable that a SIMULATION of this type of operation be executed first.

Section Tangent to Previous Inclined Section - XG5R

Defines a section of milling tangent to previous section with slanted tool. The instruction XG5R always refers to face 1; for this reason the current work surface must always be face 1.
This instruction must be preceded by a begin milling with slanted tool or by another milling instruction with slanted tool.

Unit:
Standard routing



Basic parameters:

X Coordinate X of end of milling (only if G=2 or G=3).
Y Coordinate Y of end of milling (only if G=2 or G=3).
Z Depth of end of milling (measured according to the tool angle).
H Height of end point of machining from working table (position of the tool from the working table of the machine).
Q Angular offset relative to parameter A (for special heads Benz configured in tooling with type M, N, O; see: Configuration Manuale for the Heads).

Extended parameters:

G Type of tangency:

1 Segment in the same running direction as the previous section.
-1 Segment in the opposite running direction from the previous section.
2 Clockwise circular milling.
3 Counter-clockwise circular milling.

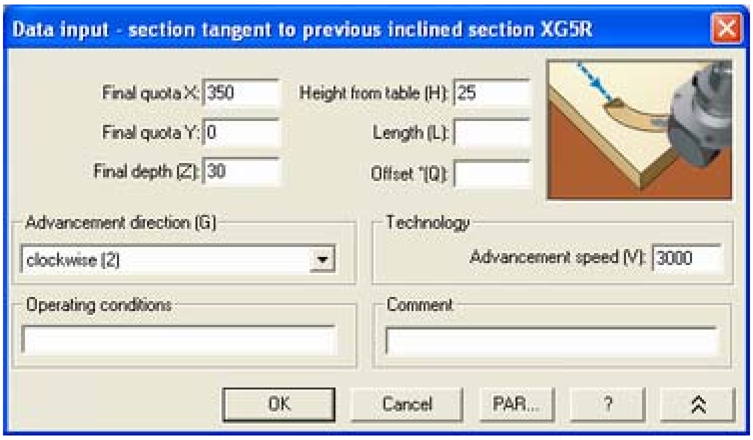
V Milling speed.
A Angle of rotation for milling.
L Length of segment (only if G=1 or G=-1).

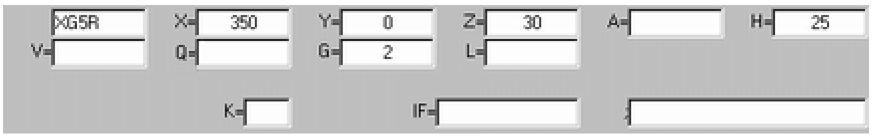
Parameters X, Y, H, G, L are mandatory.

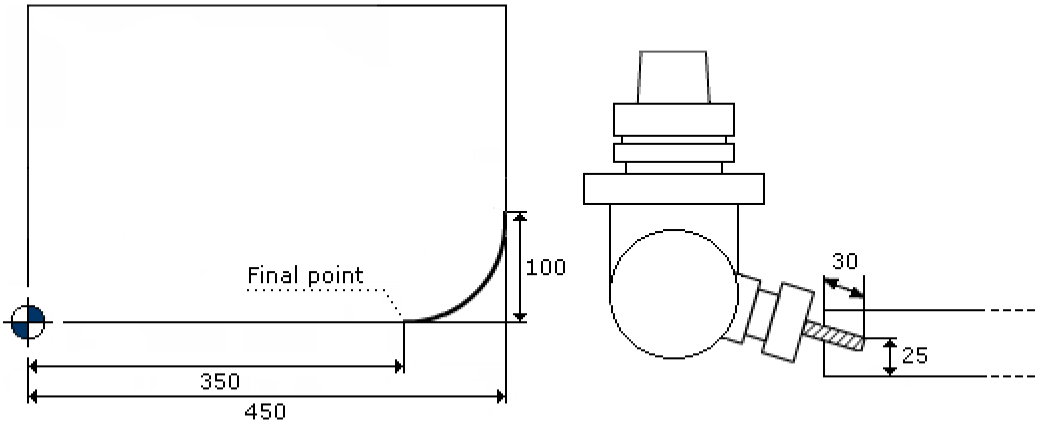
Examples:

►*Front machine origin*

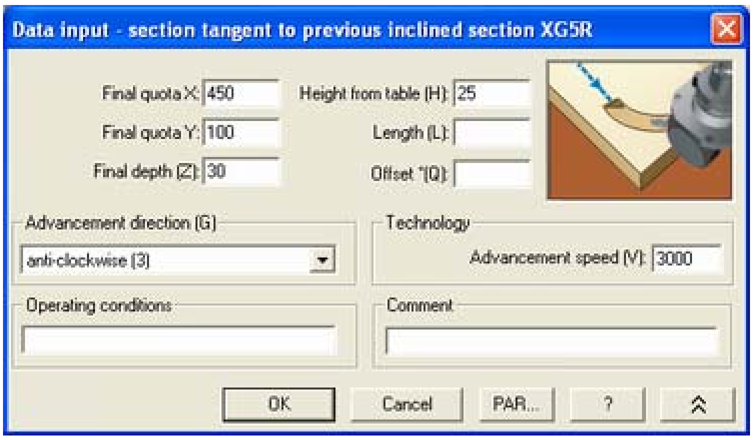
1) Programming a section of inclined milling tangent to a segment with starting point X=450, Y=100.

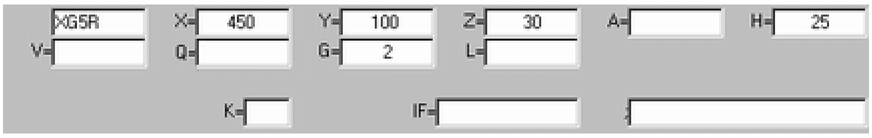


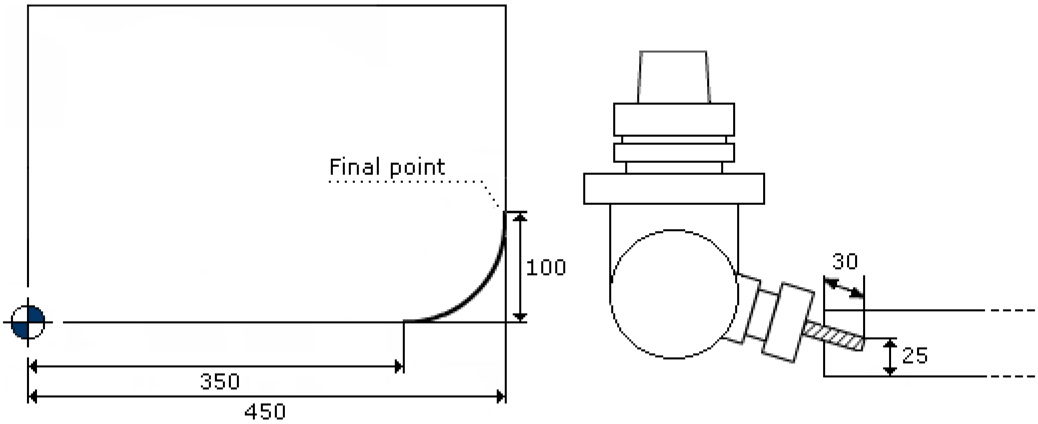




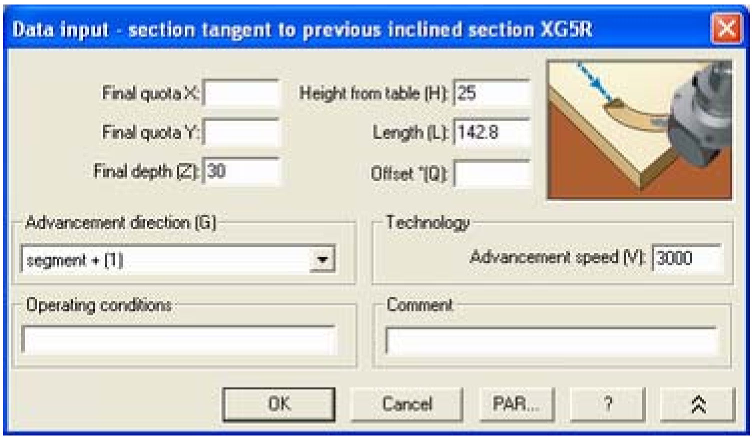
2) Programming a section of inclined milling tangent to a segment with starting point X=350, Y=0.

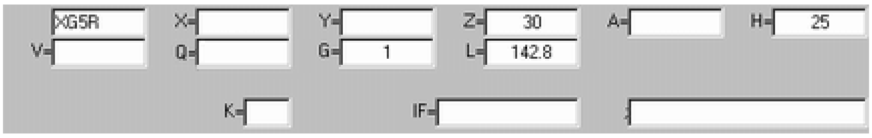


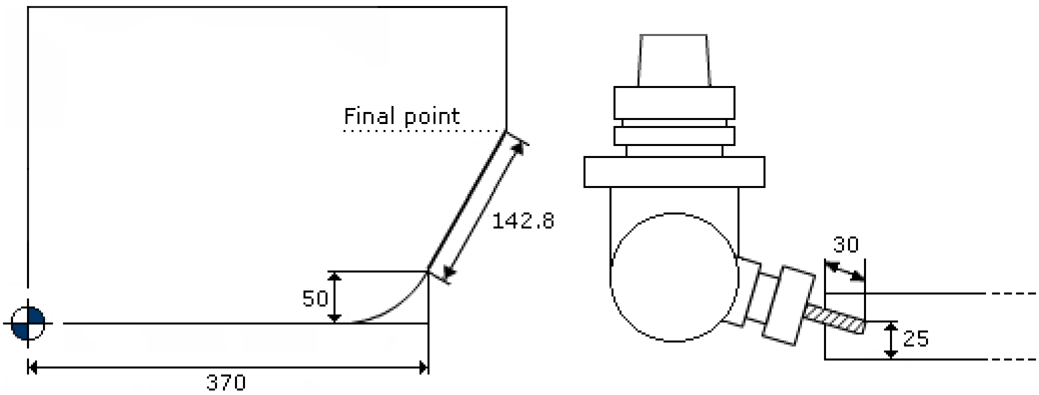




3) Programming a section of inclined milling tangent to an arc with starting point X=370, Y=50.

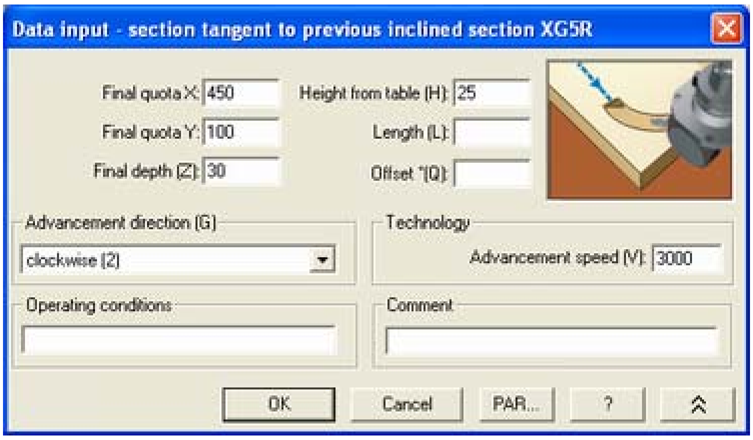


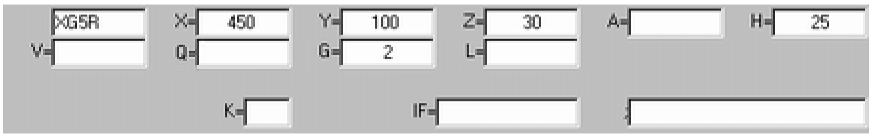


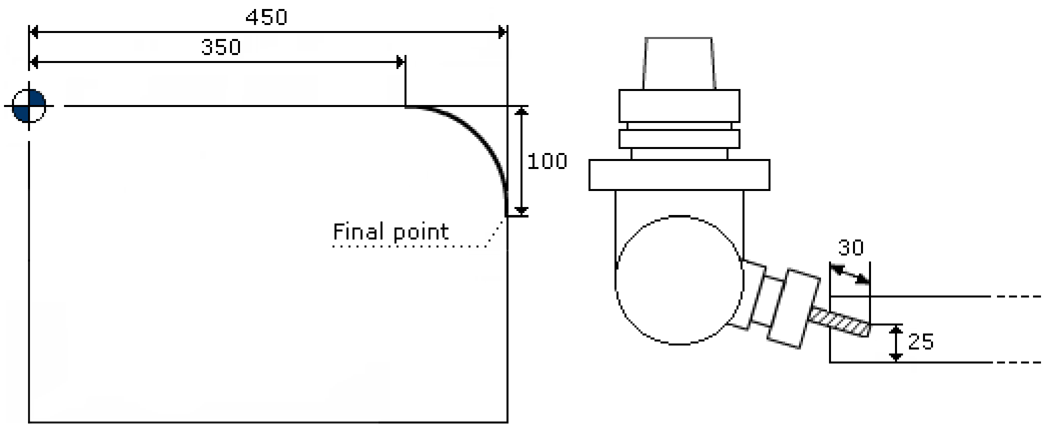


►*Rear machine origin*

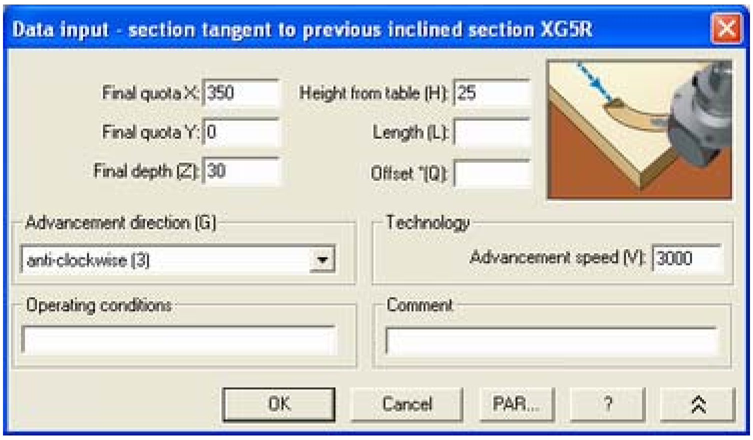
1) Programming a section of inclined milling tangent to a segment with starting point X=350, Y=0.

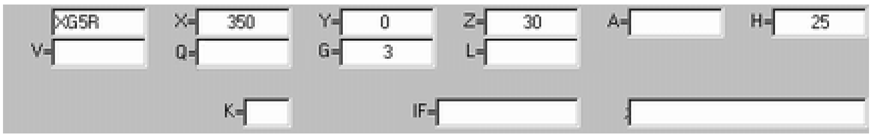


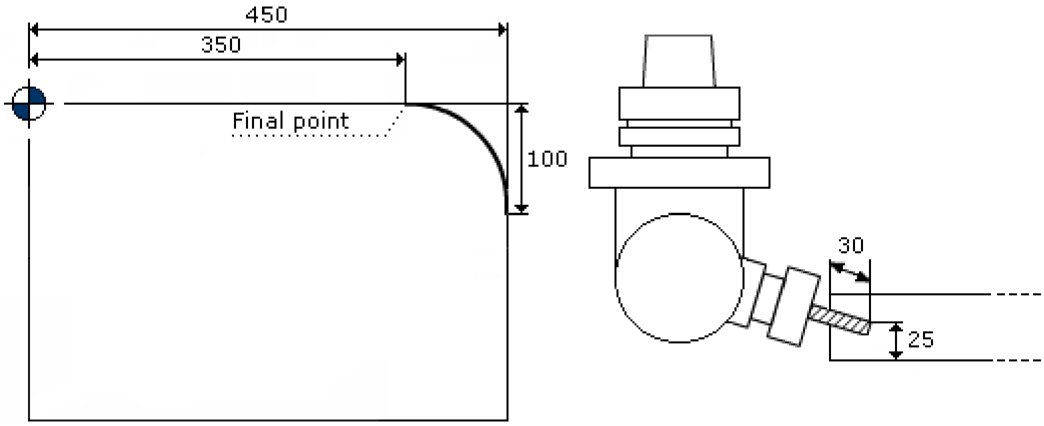




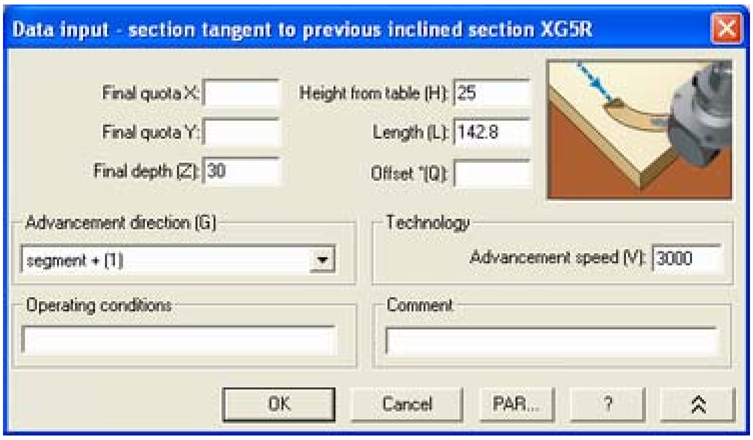
2) Programming a section of inclined milling tangent to a segment with starting point X=450, Y=100.

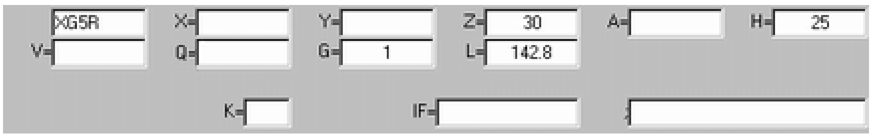


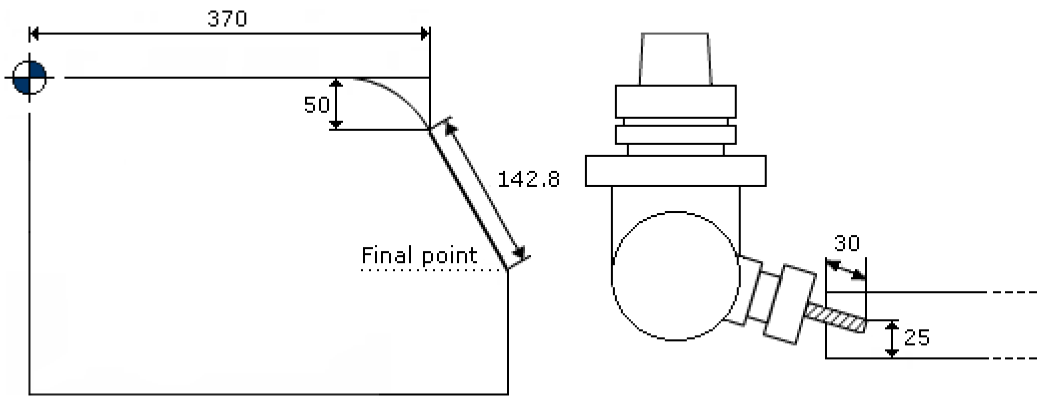




3) Programming a section of inclined milling tangent to an arc with starting point X=370, Y=50.







WARNING

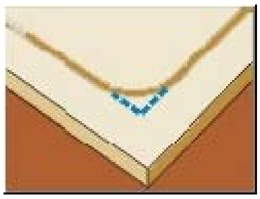
Incorrect programming of machining with slanted tools may cause damages to the structure of the machine. It is advisable that a SIMULATION of this type of operation

be executed first.

Link Between Millings - XGFIL

Makes a circular connecting milling between the milling programmed before this instruction and the one programmed after this instruction.. This instruction connects any linear or circular milling to any other linear or circular milling.

Unit:
Complex routing



Parameters:

r Radius of connecting arc.
V Milling speed.

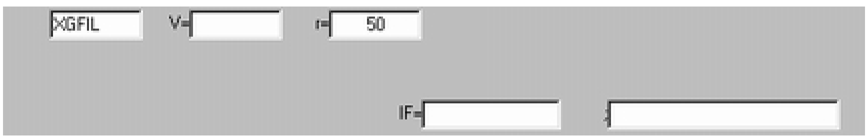
Use the following programming sequence:

1) digit the first section;
2) digit the second section;
3) enter a line with XGFIL between the two previous.

The system automatically calculates the starting and end points of the fillet.

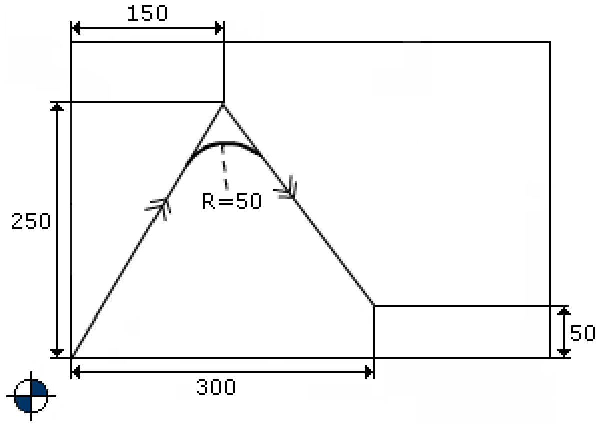
Examples:

1) Programming a link with 50 mm radius between the two linear millings

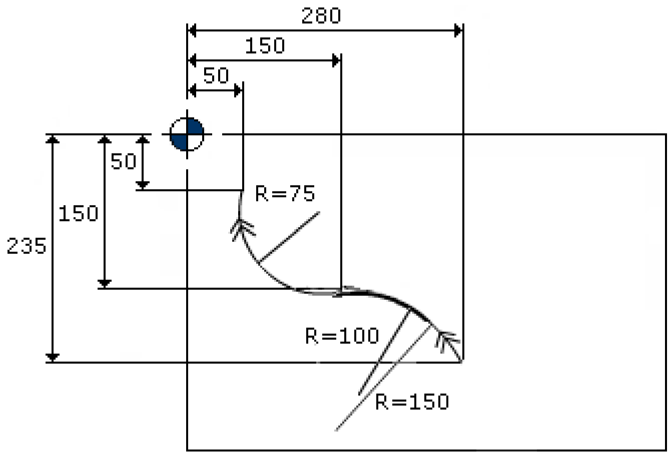


Header	H DX=450 DY=300 DZ=20 -A C=0 T=0 R=1 *MM /"test"
Start milling	XG0 X=0 Y=0 Z=10 T=101 V=5000 S=16000 E=1 D=30
First segment	XL2P X=150 Y=250
Fillet	**XGFIL r=50**
Second segment	XL2P X=300 Y=50

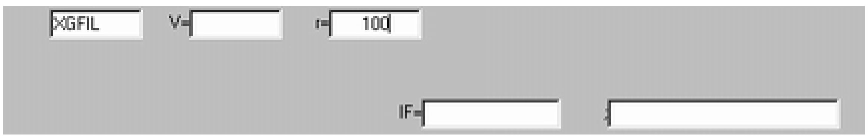
►*Front machine origin*



►*Rear machine origin*

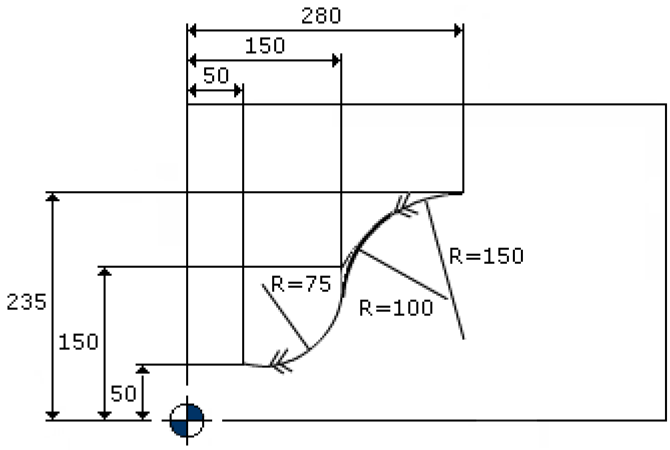


2) Programming a link with 100 mm radius between the two circular millings

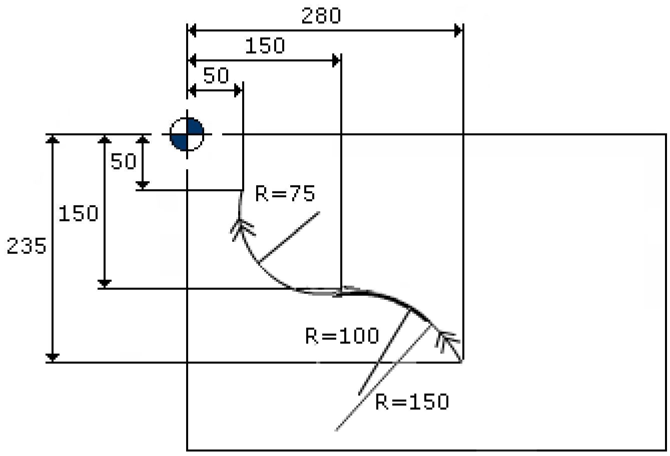


Header	H DX=450 DY=300 DZ=20 -A C=0 T=0 R=1 *MM /"test"
Start routing	XG0 X=280 Y=235 Z=10 T=101 V=5000 S=16000 E=1 D=30
First segment	XAR2 X=150 Y=150 r=150 G=3
Fillet	**XGFIL r=100**
Second segment	XAR2 X=50 Y=50 r=75 G=2

►*Front machine origin*



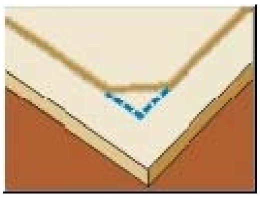
►*Rear machine origin*



Bevel Between Millings - XGCHA

Makes a circular rounding milling between the milling programmed before this instruction and the one programmed after it. The previous and following rounding instruction can be any linear milling.

Unit:
Complex routing

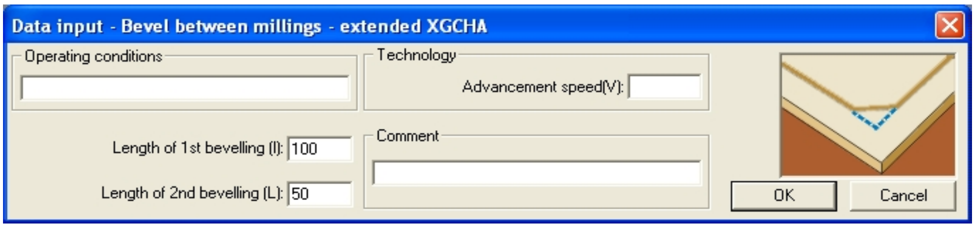


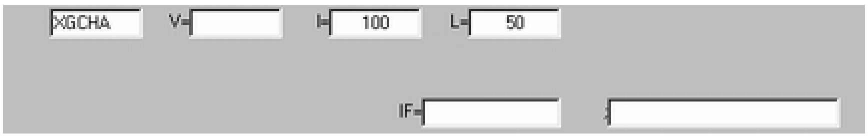
Parameters:

I Length of the first section to be rounded.
L Length of the second section to be rounded.
V Milling speed.

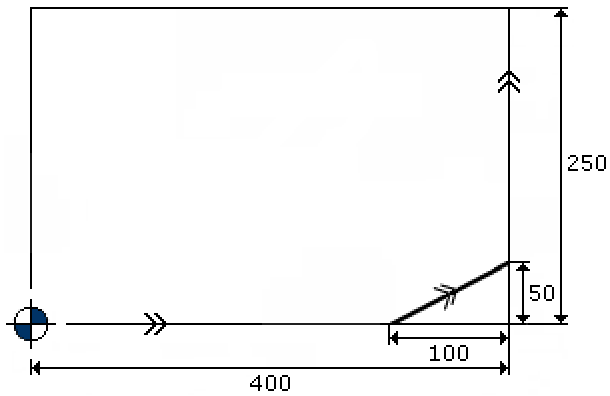
Example:

Programming a bevel between two linear millings.

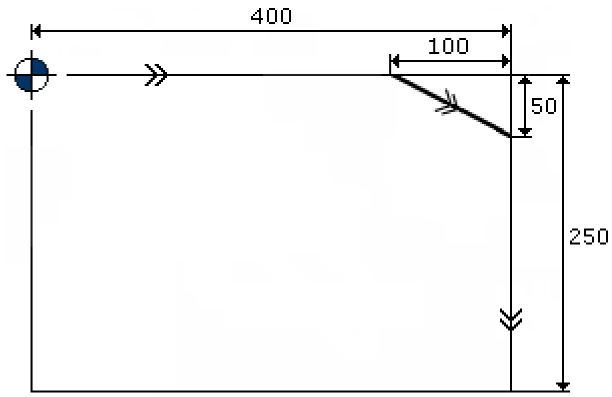




►*Front machine origin*



►*Rear machine origin*



Use the following programming sequence:
1) digit first section;
2) digit second section;
3) enter line with XGCHA between two previous.

The system automatically calculates the starting and end points of the fillet.

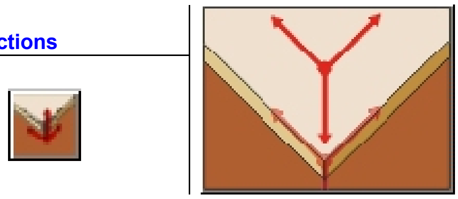
Header	H DX=400 DY=250 DZ=20 -A C=0 T=0 R=1 *MM /"ROUTE"
Start routing	XG0 X=0 Y=0 Z=10 T=101
Segment	XL2P X=400 Y=0
Chamfer	**XGCHA I=100 L=50**
Segment	XL2P X=400 Y=250

5.3.2 Modal Instructions

Origin Change - XO

Moves the origin of the panel to the programmed position. All the following instructions will refer to the new origin.

Unit:
Main functions



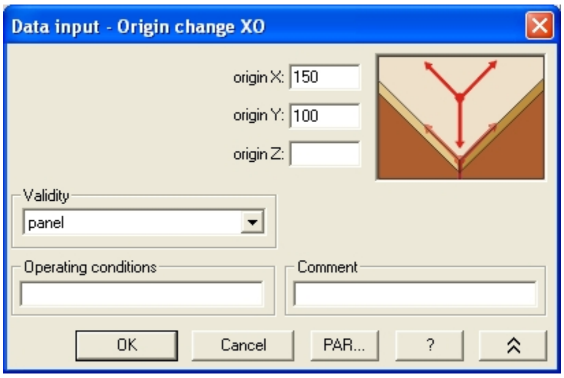
Basic parameters:

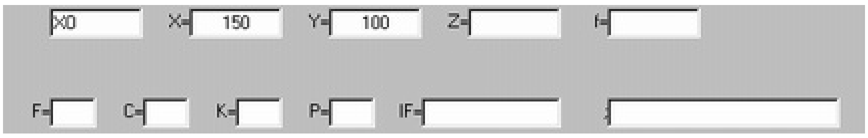
X Origin in X.
Y Origin in Y.
Z Origin in Z.

Extended parameters:

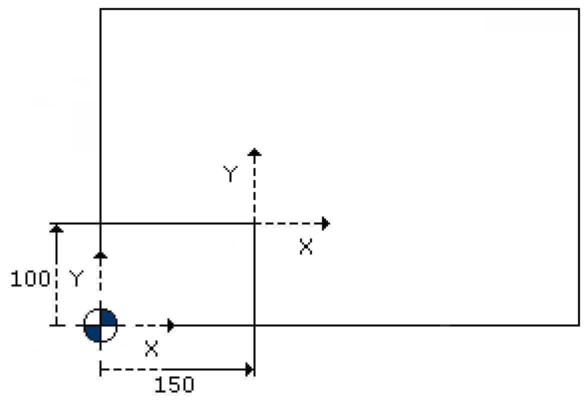
f If programmed with the number of one face (1-5), it enables the instruction only for the origin of the set face.

Example:

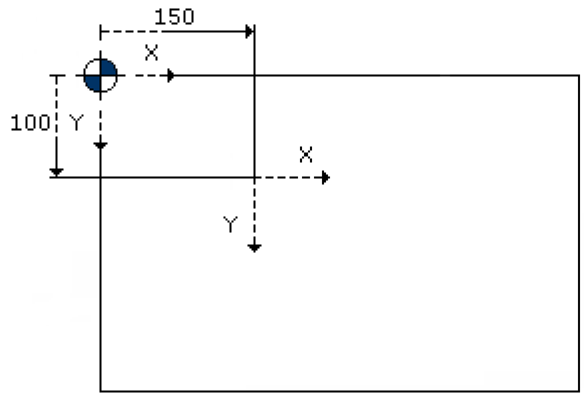




►*Front machine origin*

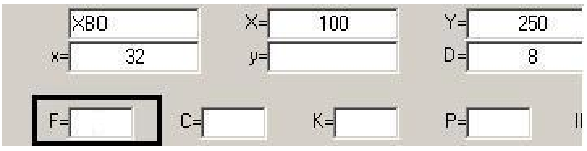


►*Rear machine origin*



F (Machining Side)

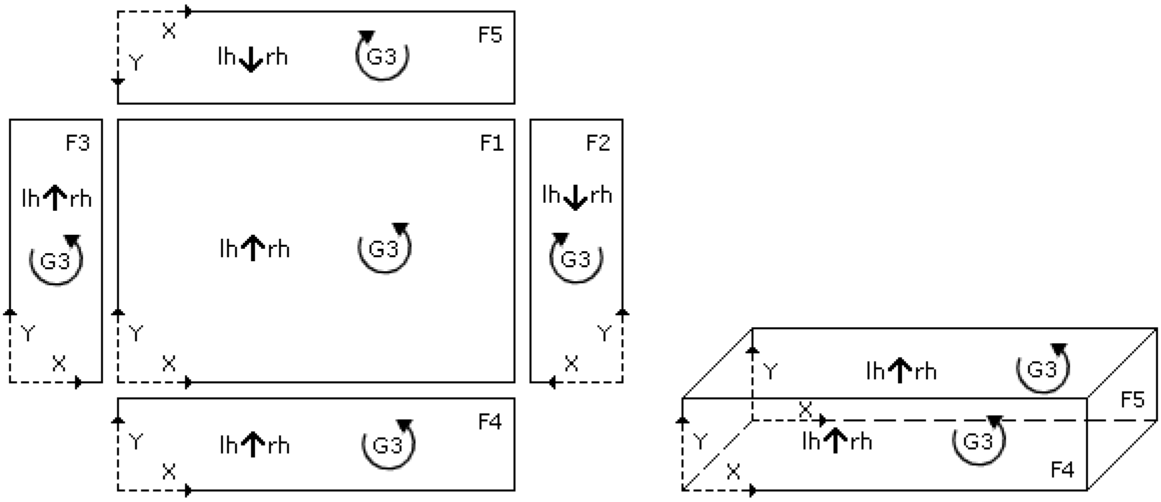
Defines the current machining face



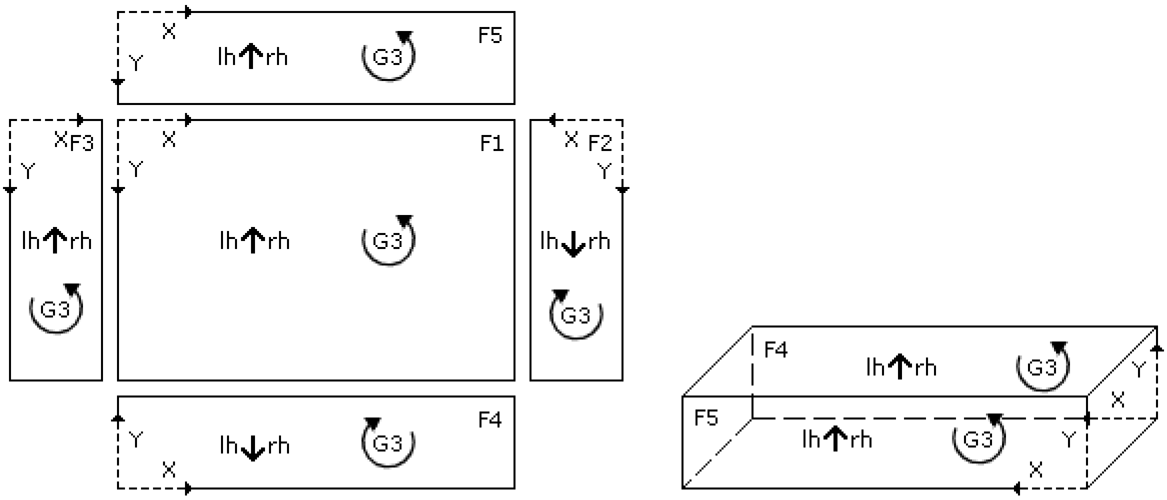
F Admitted values:

1 Upper face.
2 Right face.
3 Left face.
4 Front face.
5 Rear face.

References for front machine origin:

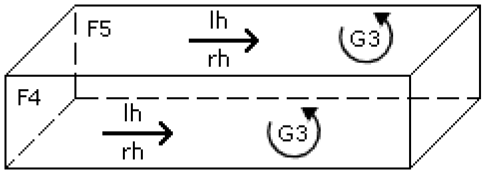


References for rear machine origin:



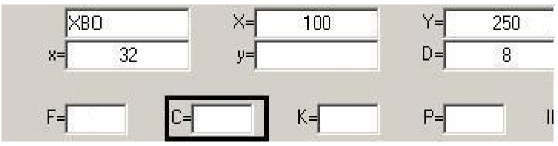
N.B. For interpolations on the side faces (F2 F3 F4 F5) Xilog Plus sees the panel as if it were transparent from the front face (F4) to the rear face (F5) and from the left-hand face (F3) to the right-hand face (F2), thus inverting the circular arches (if on F4 it is clockwise, on F5 it becomes anti-clockwise, if on F3 it is clockwise, on F2 it becomes anti-clockwise) and the radius correction (if on F4 it is LH, on F5 it becomes RH, if on F3 it is LH, on F3 it becomes RH).

Example:



C (Tool Adjustment)

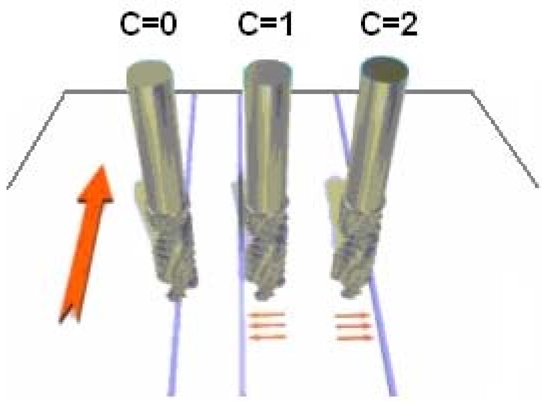
Makes it possible to correct the trajectory of the spindle based on the characteristics of the cutter mounted. If the cutter is candle-type (type F) the correction is equally to the radius stated n the tooling file under "actual diameter"; if the cutter s disk-type (type D) the correction is equally to half the thickness of the blade stated in the tooling file.

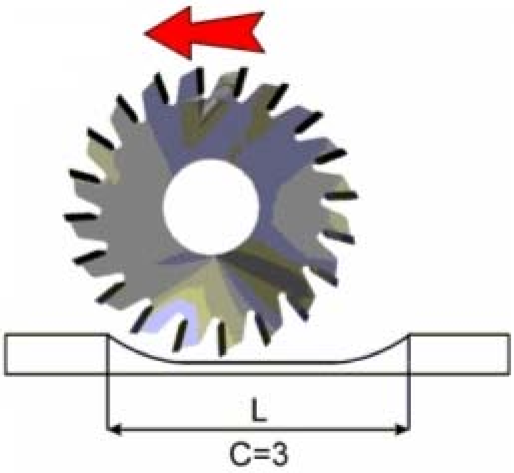


C Admitted values:

0 Void correction.
1 Right correction.
2 Left correction.
3 In depth correction (only for disk millers).
13 Correction 1 + correction 3 (only for disk millers).
23 Correction 2 + correction 3 (only for disk millers).

Examples:





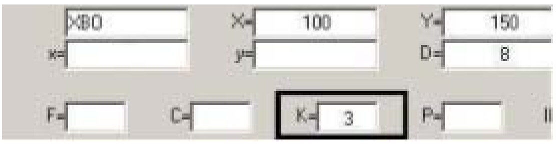
K (Incremental)

When incremental programming is enabled, the reading enabled as incremental (X or Y or both), programmed in an operative instruction, no longer refers to positioning with respect to the origin of the panel but to a shift of the value that the reading itself had undergone after carrying out the previous operative instruction.

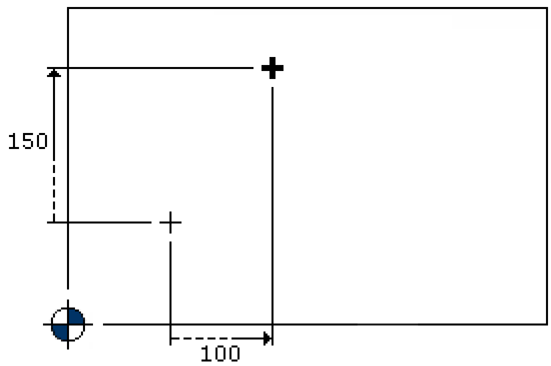
K Admitted values:

0 No incremental.
1 Incremental in X, corresponding to instruction IX=1 in basic language.
2 Incremental in Y, corresponding to instruction IY=1 in basic language.
3 Incremental in X and Y, corresponding to the sequence of the two instructions IX=1 and IY=1 in basic language.

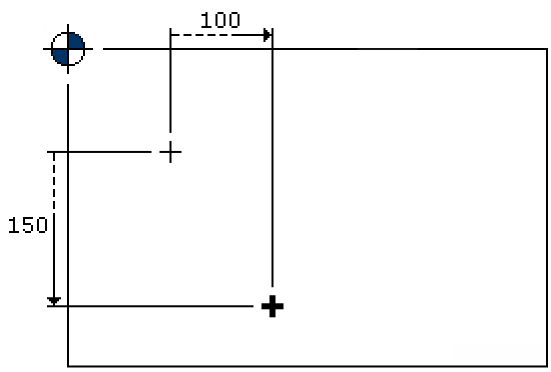
Example:



References for front machine origin:

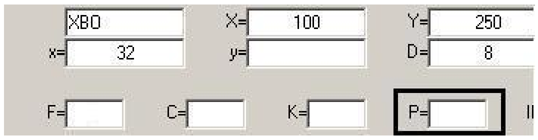


References for rear machine origin:



P (Reference Origin)

It enables the specular machining with an eventual inversion of the direction of round millings.

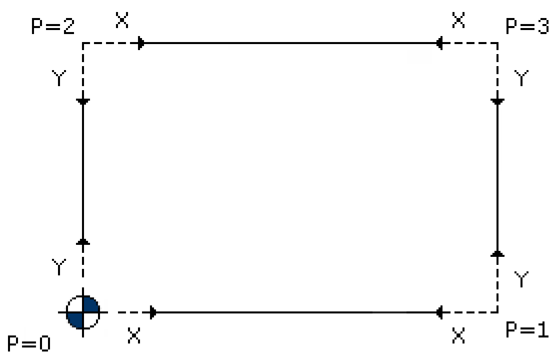


P Admitted values:

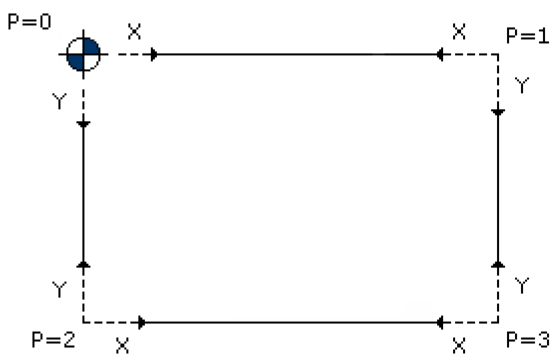
- 0 No specular.
- 1 Specular in X, corresponding to instruction SX=1 in basic language.
- 2 Specular in Y, corresponding to instruction SY=1 in basic language.
- 3 Specular in X and Y, corresponding to the sequence of the two instructions SX=1 e SY=1, in basic language.
- 11 As 1 reversing the direction of arches of circle, corresponding to instruction SX=1 M=1 in basic language.
- 12 As 2 reversing the direction of arches of circle , G2 <-> G3, corresponding to instruction SY=1 M=1 in basic language.
- 13 As 3 reversing the direction of the arches of circle G2 <-> G3, corresponding to the sequence of two instructions SX=1 M=1 and SY=1 M=1 in basic language.

The next figure show the new reference points of the X and Y readings with the enabled specular:

References for front machine origin:



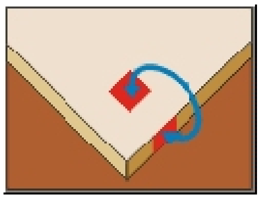
References for rear machine origin:



Inclined Plane - XPL

Defines a surface that is different from the 5 faces which Xilog Plus creates in automatic. It can be rotated around axis Z and around axis X. An inclined surface is used to create geometries that are perpendicular to the surface itself.

Unit:
Main functions



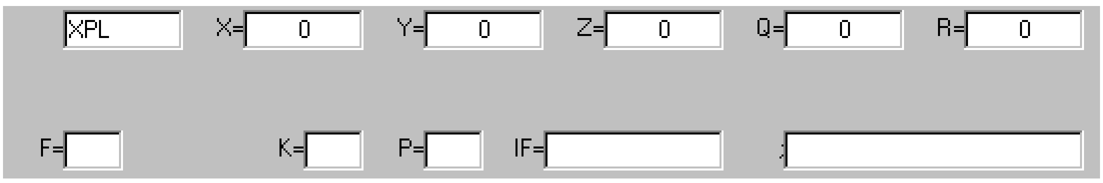
Parameters:

X Coordinate X of the origin of the plane (referred to the origin of the panel).
Y Coordinate Y of the origin of the plane (referred to the origin of the panel).
Z Coordinate Z of the origin of the plane (in relation to panel base).
Q Angle of rotation around axis Z (front machine origin; rear machine origin: positive in clockwise direction).
R Angle of rotation around axis X (front machine origin; rear machine origin: positive in clockwise direction).

Parameter F must always be set at 1 with modal instruction F. Parameters X, Y, Q, R are mandatory.
A XPL instruction is cancelled by:

- another instruction XPL;
- the retrieval of a standard face.

To reset and inclined surface it is necessary to enter another XPL instruction with all the parameters on 0, as shown in the following example.

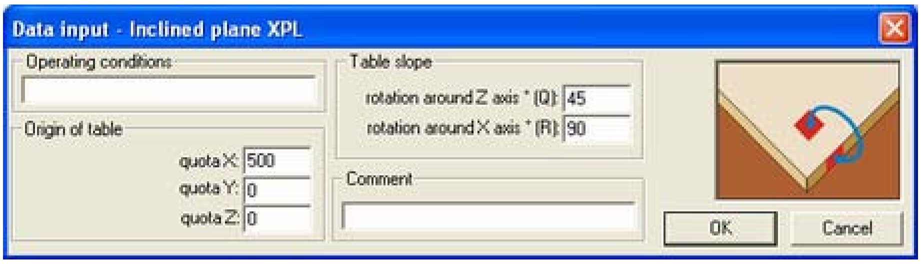


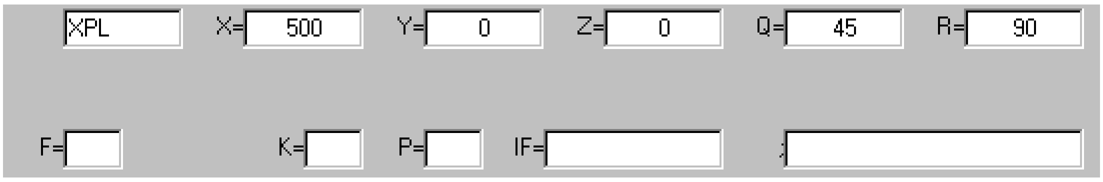
If all parameters have a value of zero face 1 is reset.

Example:

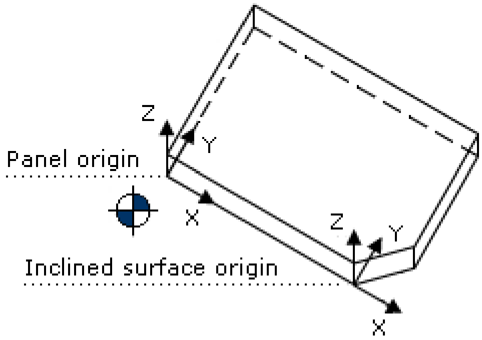
Programming an inclined surface with origins X=500, Y=0, Z=0, rotated 45° about the Z-axis and 90° about the X-axis.

►*Front machine origin*

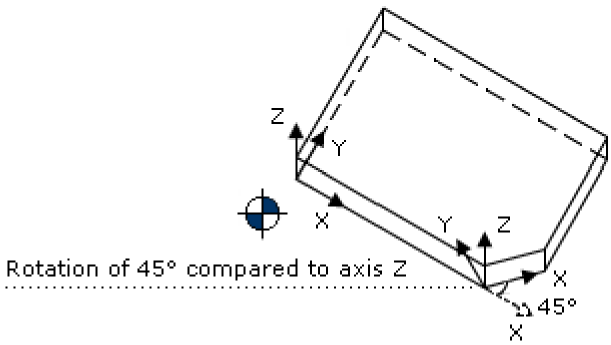




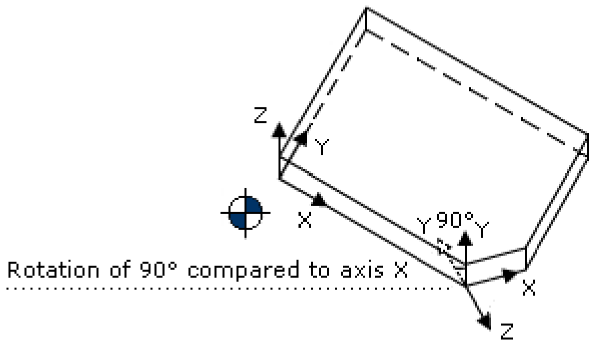
X=500, Y=0 and Z=0 are origin coordinates for the inclined surface.



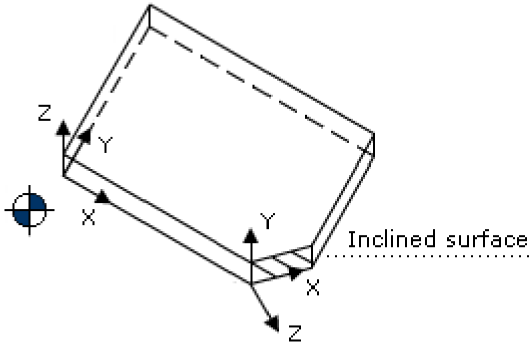
After changing origin the axis system must be rotated around axis Z by entering parameter Q=desired angle (Q=45).



At this point, the axis system must rotate around axis X by entering the parameter R=desired angle (R=90).

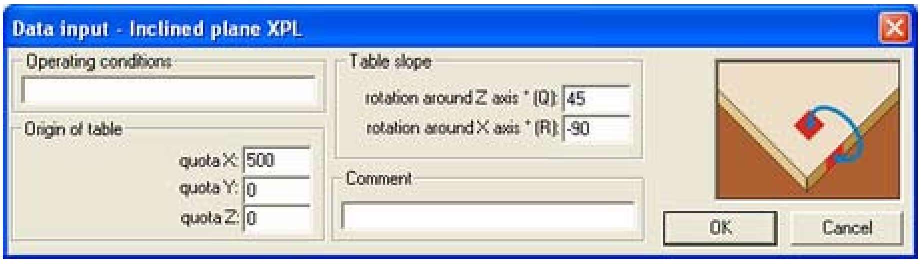


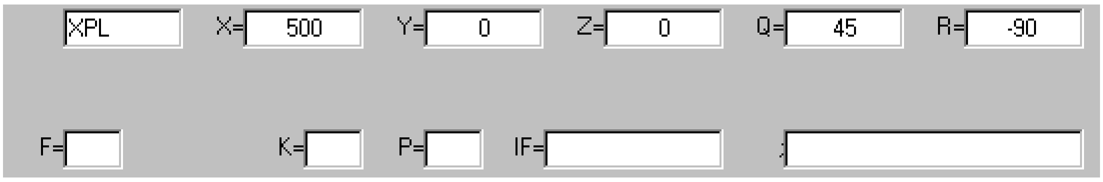
The final result will be an inclined surface, as shown in the following figure.



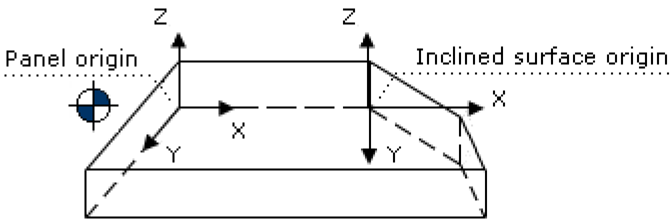
N.B. The direction of the Z-axis must be always **negative towards the outside** of the panel.

►*Rear machine origin*

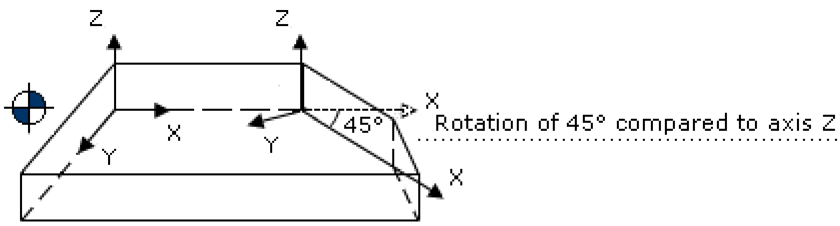




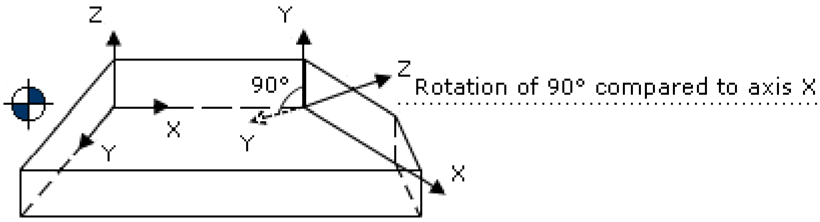
X=500, Y=0 and Z=0 are origin coordinates for the inclined surface.



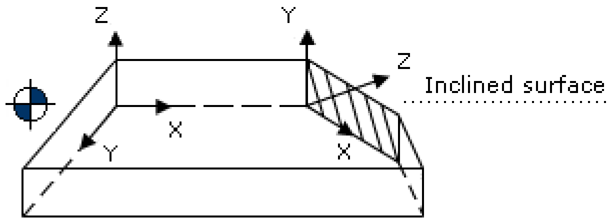
After changing origin the axis system must be rotated around axis Z by entering parameter Q=desired angle (Q=45).



At this point, the axis system must rotate around axis X by entering the parameter R=desired angle (R=-90) as shown in the following figure.



The final result will be an inclined surface, as shown in the following figure.

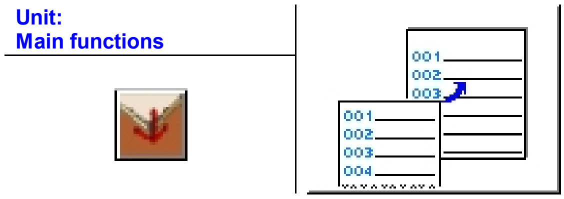


N.B. The direction of the Z-axis must be always **negative towards the outside** of the panel.

5.3.3 Instructions for Managing Subprograms

Call Sub-Program - XS

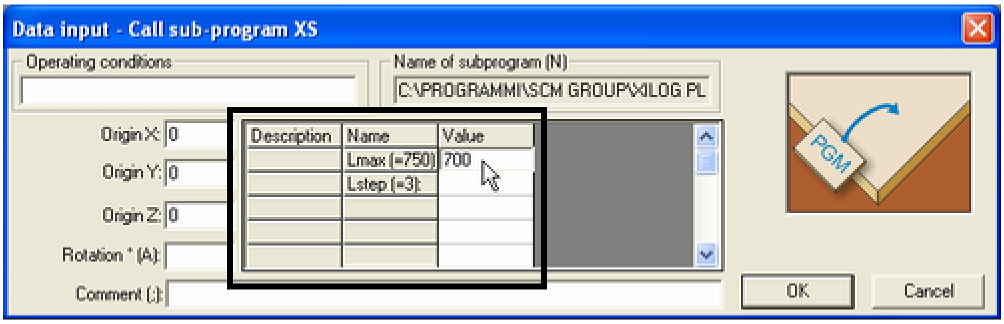
Recalls a normal program as subprogram.



Parameters:

X Origin in X of subprogram.
Y Origin in Y of subprogram.
Z Origin in Z of subprogram.
A Angle of rotation of subprogram (front machine origin; rear machine origin: positive in clockwise direction).
N Name of subprogram.

The parameters of the called up subroutine/macro can be directly modified in the respective boxes: click on the box, enter the new value and confirm with the **[ENTER].**



For unprogrammed X, Y, Z the current values pertain (for Z the last work quota pertains)

An open program stores subprograms in its memory: in order for changes to graphics in the subprograms to become effective the main program must be closed down and reopened.

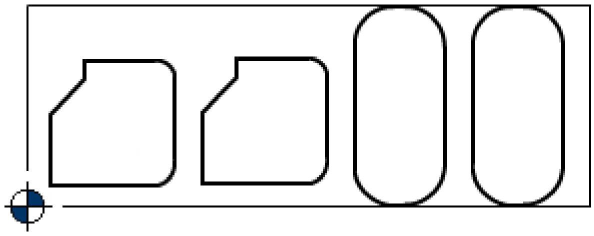
Caution! The XS instruction is not suitable for retrieving programs containing angled surfaces (XPL instruction), since the translation is only applied to machining and not to the origin of the angled surface. Use the SO instruction for this.

Warning! If the subprogram is dependent on a file of ambient variables, in order for the main program to function properly (the program which calls up the file), the same file of ambient variables in the subprogram must also be programmed in the heading of the main program.

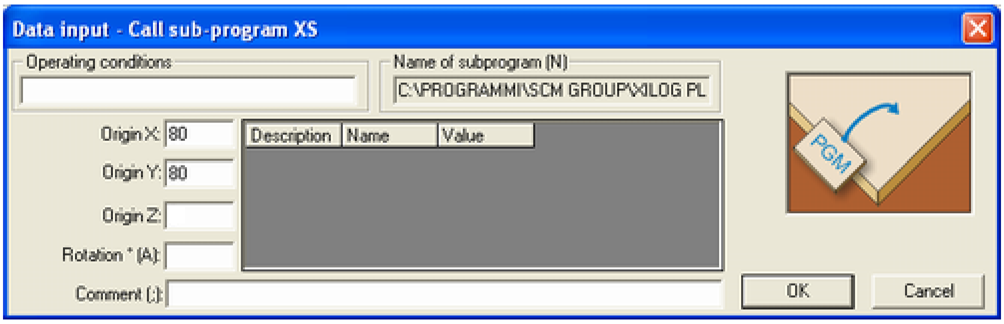
Example:

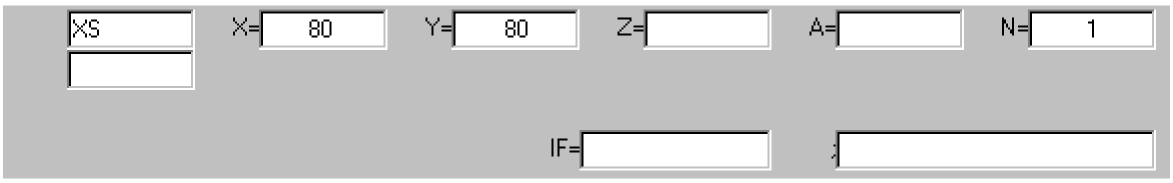
► *Front machine origin*

Obtaining four work pieces (sub-programs) from a large panel, as illustrated:

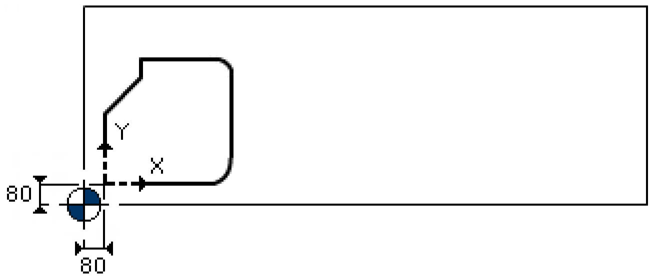


Proceed as follows:
In the programs editor, create a new program, fill in the Header with the panel information.
At this point we will recall the first subprogram with instruction XS.

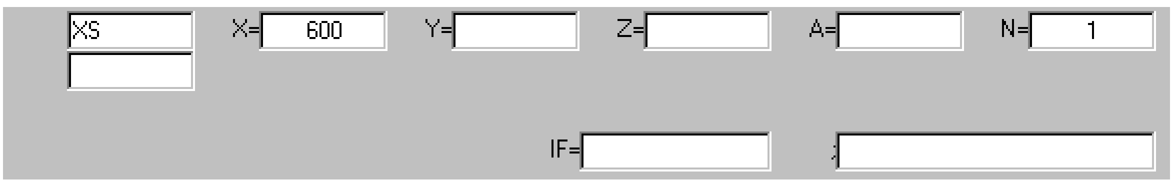


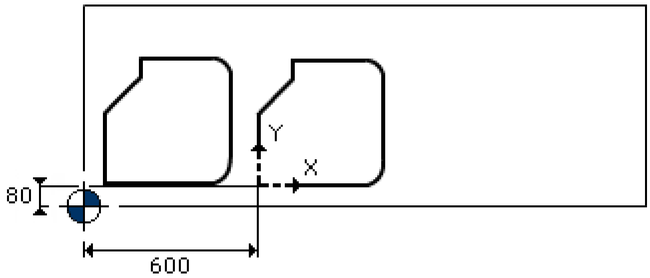


NOTE. By inserting instruction XS with the instructions bar it is possible to select the program to be inserted directly with the mouse from the **Open** window.

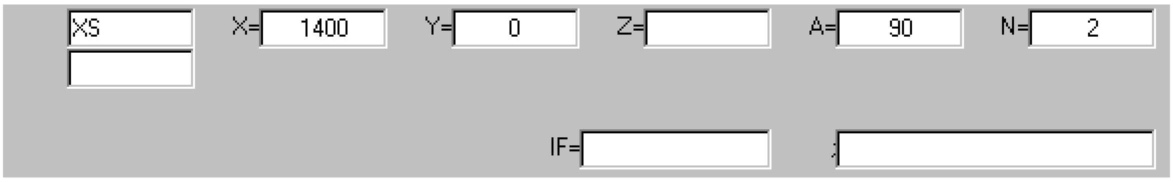


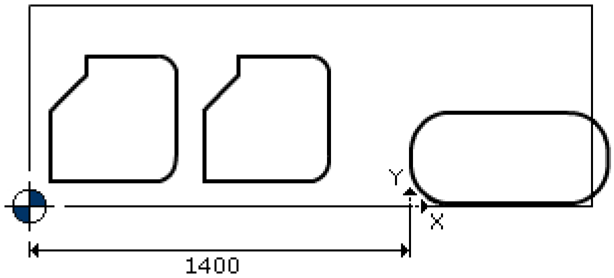
We then recall the first subprogram a second time, inserting it into the second position.

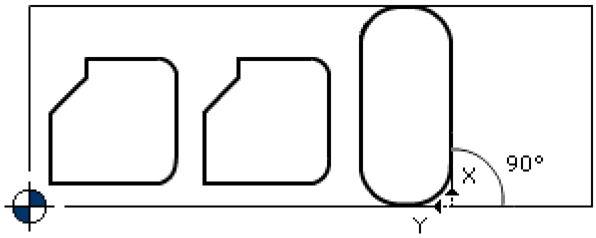




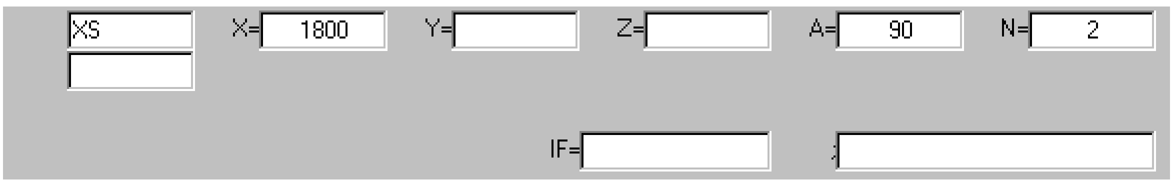
We recall the second subprogram and insert it with origin, as shown in the figure, and then rotate it by 90°.

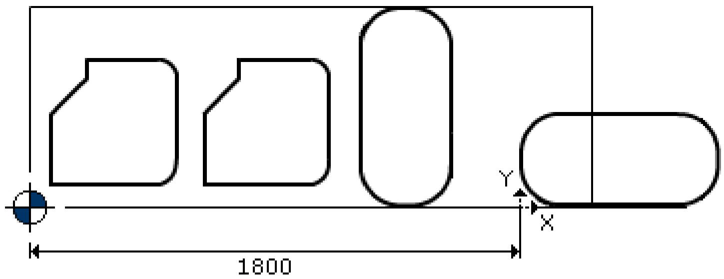


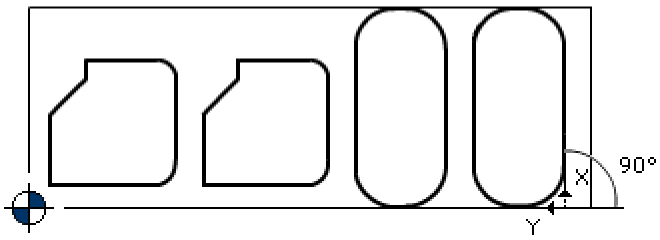




We repeat the operation for inserting the last subprogram.

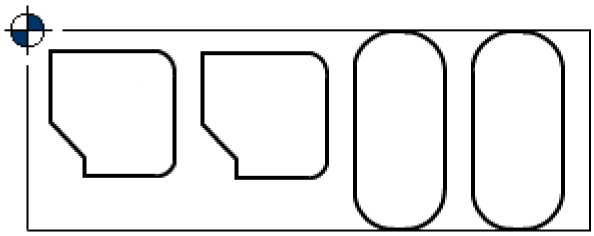




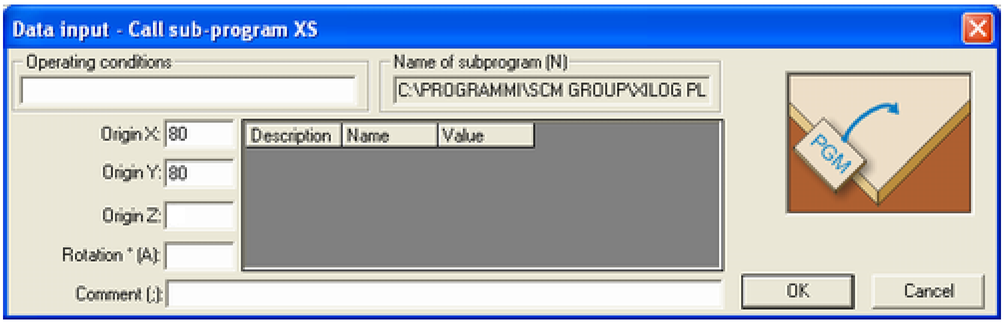


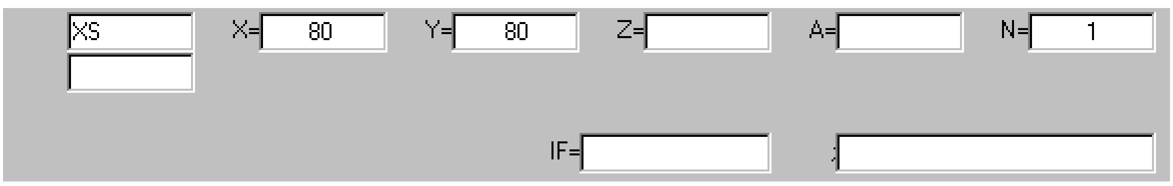
►*Rear machine origin*

Obtaining four work pieces (sub-programs) from a large panel, as illustrated:

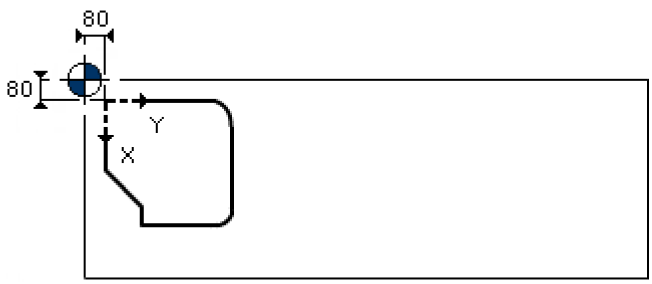


Proceed as follows:
from the programs editor create a new program, fill in the Header with the panel information.
At this point we recall the first subprogram with instruction XS.

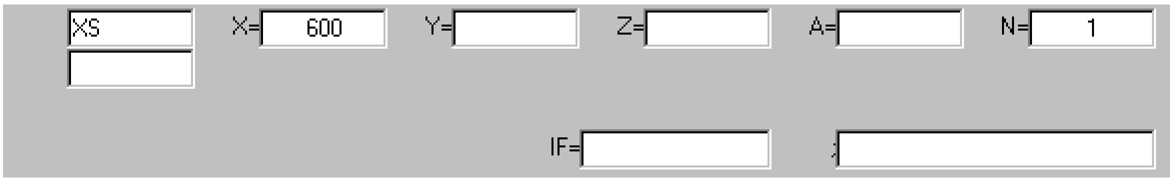


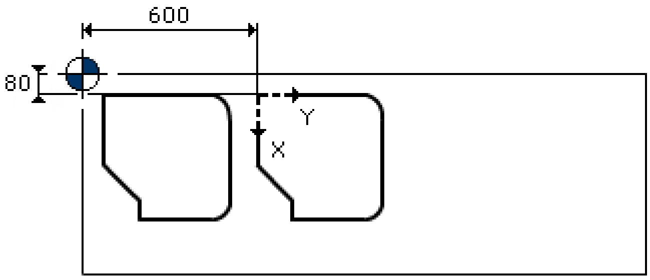


NOTE. By inserting instruction XS with the instructions bar it is possible to select the program to be inserted directly with the mouse from the **Open** window.

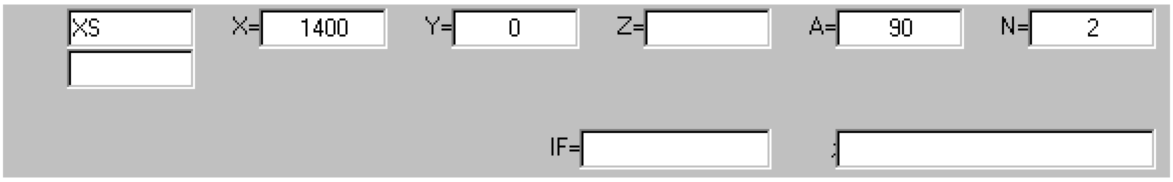


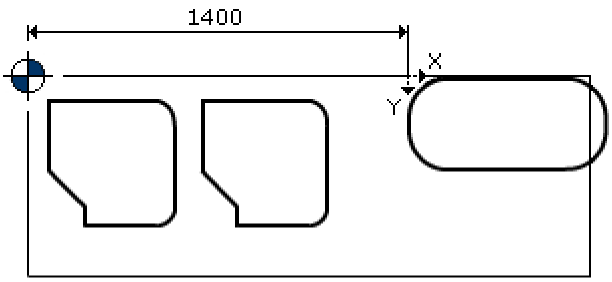
We then recall the first subprogram a second time, inserting it into the second position.

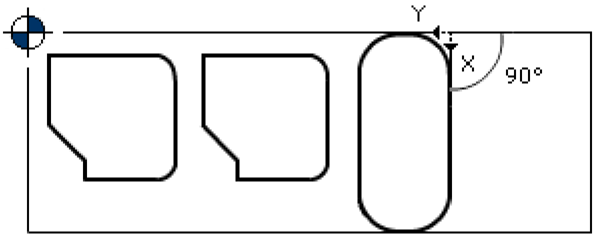




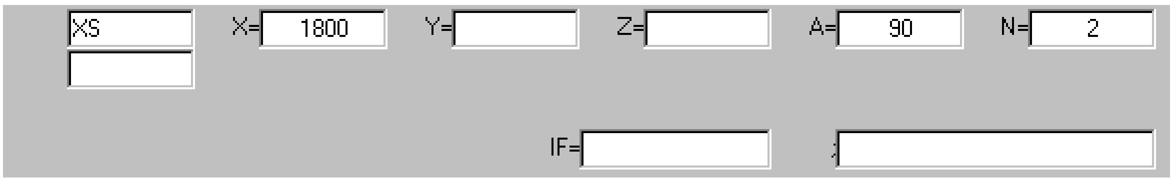
We recall the second subprogram and insert it with origin, as shown in the figure, and then rotate it by 90°.

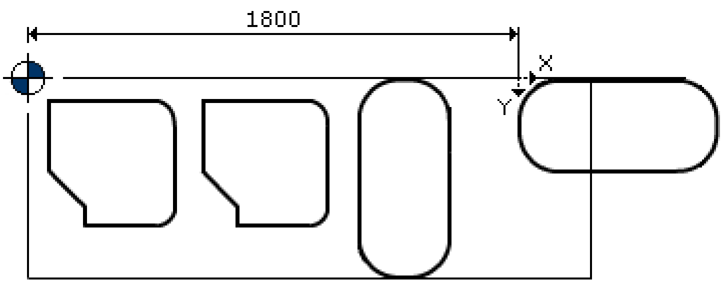


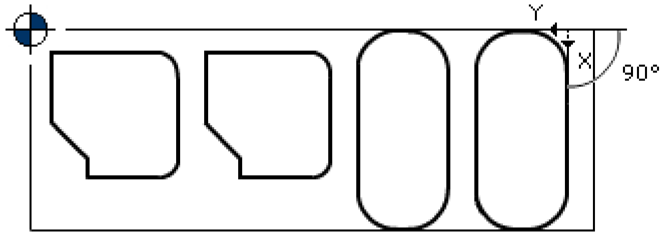




We repeat the operation for inserting the last subprogram.



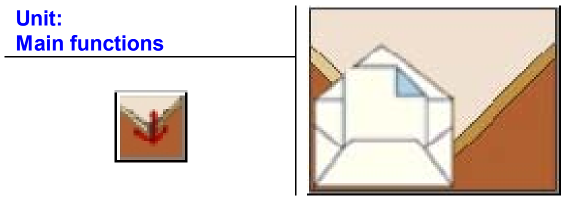



Y
X
90°

5.3.4 Sending Messages to the Operator

Print Message - XMSG

The messages to the operator contain information sent to the operator while the piece is being machined.

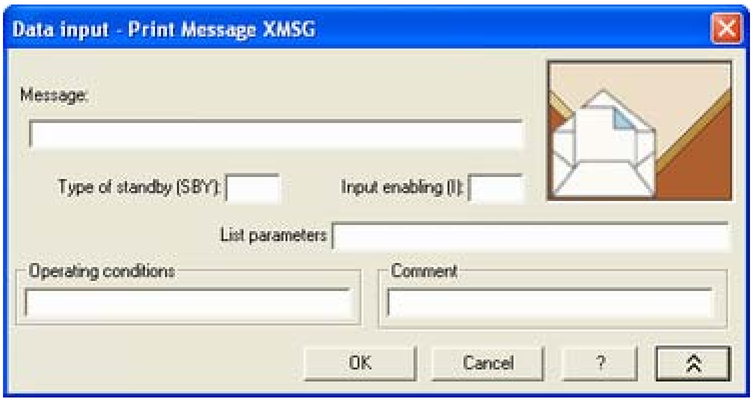


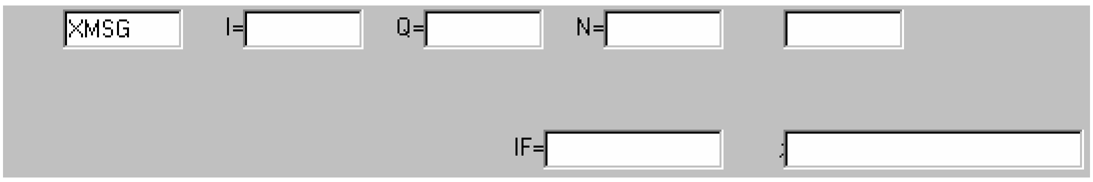
Basic parameter:

Message/N Message.

Extended parameters:

SBY/Q Type of standby to be carried out.
I Data entry request enabling.
List parameters/(empty box) Insert parameters.





An XMSG without parameters cancels the previous message.
The message may contain an indication of variable values (marked by the key ?); each of these values must correspond to a parameter entered in the list of parameters. A maximum of 7 parameters are allowed, called P1, P2 etc.: P1 is always associated with the variable value on the far left of the message, P2 with the next value on the right, and so on.
A request to input data can be associated to every message (parameter I=1; in this case, the value entered is copied in the reserved variable xMSGInVal). After a message is sent the piece program may carry on or stop waiting for the operator to input required data and/or press start cycle pushbutton.

The standby indication (parameter SBY/Q at 1 or 2) stops the piece program waiting for the cycle start pushbutton to be pressed. When data is requested the piece program stops simulating the indication SBY/Q=1.
The instructions XMSG are treated individually and displayed with logic according to the type:

XMSG without INPUT
The message is displayed in the message window; when in standby mode the message is cancelled by pressing the **Start** button, otherwise the message is cancelled when a new XMSG arrives, even without messages; in both cases the cancellation also occurs when a Reset is accepted.

XMSG with INPUT
The message is displayed in the messages window and an icon is displayed to signal the need to insert data (**CONTROLS/INPUT OPERATOR MESSAGE** menu). The message is cancelled when the **Start** button is pressed and when a Reset is accepted.

Example:

```
…
XMSG "Wait completion of unload cycle"
…
XMSG
…
XMSG N="Remove rejects and press start-cycle!" Q=1
…
XMSG N="Input right cutting angle" I=1
L ADX=xMSGInVal
XMSG N="Input left cutting angle" I=1
L ASX=xMSGInVal
IF ADX > 90 AND ASX=0 THEN
…
FI
…
L Min = 0
L Max = 4
XMSG N="Position of vacuum hood (from ?d to ?d)" P1=Min P2=Max I=1
L Hood=xMSGInVal
IF Hood >=Min AND Hood <= Max THEN
ISO "M?d" 110+ Hood
ELSE
PRINT "Position Hood=?d error!" Hood
FI
```

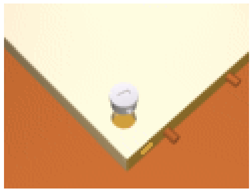
5.4 User Macros

5.4.1 User Macros for Machining for Furniture (for Windows XP)

Join Boring with Camlock - CAMLOCK

Performs boring on right, upper and lower faces to join the sides and base.

Unit:
Assembly



Minimum requirements:

- Left face and right face: flat hole tip diameter 8 mm; depth max. 23 mm.
- Upper face: flat hole tip diameter 15 mm; depth 13 mm.

Parameters:

X X coordinate of the camlock hole (default*: 32).
Y Y coordinate of the camlock hole (default*: 50).
Z Camlock hole depth (default*: 13).
H X coordinate of the horizontal holes (default*: DZ/2).
I Depth of the hole for the pin.
J Depth of the hole for the dowels (default*: 18).
Q Presence of central hole on the right and left faces (1 = yes, 0 = no) (default*: 1).
D Camlock hole diameter (default*: 15).
S Centre-to-centre between the horizontal holes (default*: 32).
G Diameter of the horizontal holes (default*: 8).

*in graphic editor.

Fitting Barrier - FITTING_32

Makes a line of holes at intervals for shelf insertion. The line of holes is centred according to the panel length.

Unit:
Assembly

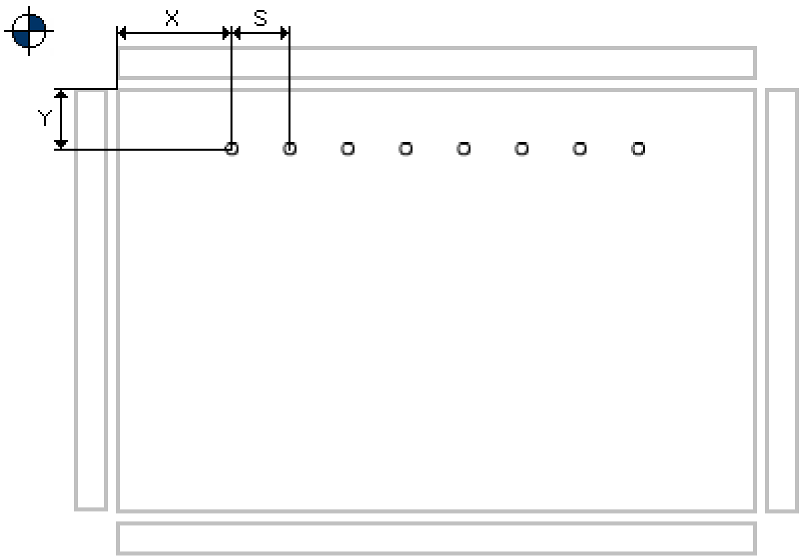
Minimum requirements:

- Upper face.
- Flat hole tip diameter 5 mm.
- Depth 10 mm.

Parameters:

X X coordinate of the first hole (default*: 80).
Y Y coordinate of the holes (default*: 30).
Z Depth of the holes (default*: 10).
S Centre-to-centre between holes (default*: 32).
D Diameter of the holes (default*: 5).

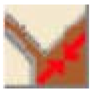
*in graphic editor.

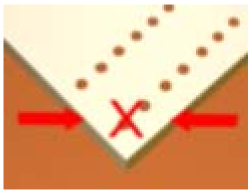


Double Fitting Barrier - FITTING_32_D

Makes a line of holes at double intervals for shelf insertion. The line of holes is centred according to the panel length.

Unit:
Assembly





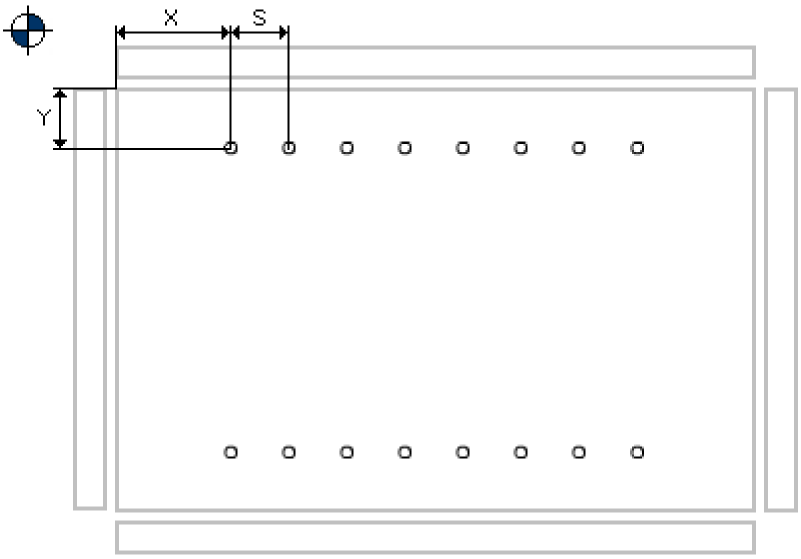
Minimum requirements:

- Upper face.
- Flat hole tip diameter 5 mm.
- Depth 10 mm.

Parameters:

X X coordinate of the first hole (default*: 80).
Y Y coordinate of the holes (default*: 30).
Z Depth of the holes (default*: 10).
S Centre-to-centre between holes (default*: 32).
D Diameter of the holes (default*: 5).

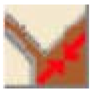
*in graphic editor.



Toe Kick - TOEKICK

Performs routing to remove an edge.

Unit:
Assembly



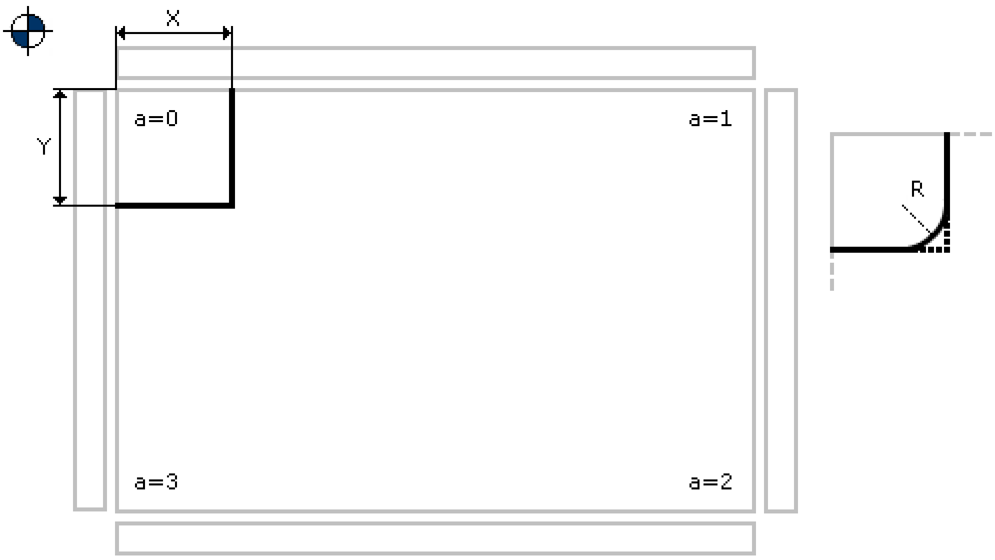
Minimum requirements:

- Upper face.
- Mill T=101.
- Depth max. DZ+2mm.

Parameters:

X Routing length in X (default*: 50).
Y Routing length in Y (default*: 50).
Z Routing depth (default*: DZ+2).
H Type of lead out (1 = straight, 2 = curved) (default*: 1).
I Type of lead in (1 = straight, 2 = curved) (default*: 1).
T Tool number (default: 101).
a Edge to be removed (0, 1, 2, 3) (default*: 0).
R Fillet radius (default*: 0).
D Tool radius multiplication factor (default*: 2).
s Type of tool approach at lead in and lead out (0 = at height, 1 = descending) (default*: 0).
G Direction of travel (2 = clockwise, 3 = anti-clockwise) (default*: 2).

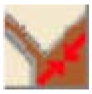
*in graphic editor.

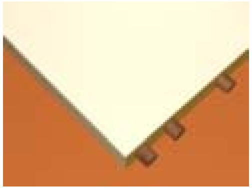


Horizontal Connection Boring - UNIONE_O

Performs horizontal boring to make holes for connecting the side to the base.

Unit:
Assembly





Minimum requirements:

- Left face and right face: tip diameter 8 mm; depth max. 15 mm.

Parameters:

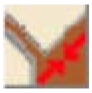
Z Depth of the holes (default*: 15).
I Centre-to-centre between the holes centred in DY (default*: 32).
R Number of holes centred in DY (default*: 1).
x X coordinate of holes (default*: DZ/2).
y Y coordinate of the initial hole (default*: 20).
B Diameter of the holes centred in DY (default*. 8).
r Number of external holes per side (default*: 2).
D Diameter of the external holes (default*: 8).
s Centre-to-centre between the external holes (default*: 32).

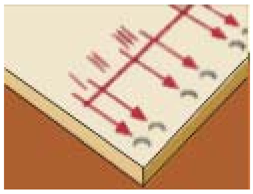
*in graphic editor.

Vertical Connection Boring Face_1 - UNIONE_V

Performs vertical boring to makes holes for connecting the side to the base.

Unit:
Assembly





Minimum requirements:

- Upper face.
- Tip diameter 8 mm.
- Depth max. 12 mm.

Parameters:

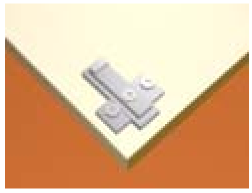
Z	Depth of the holes (default*: 12).
I	Centre-to-centre between the holes centred in DY (default*: 32).
Q	Left/right double boring (1 = yes, 0 = no) (default*: 1).
R	Number of holes centred in DY (default*: 1).
x	X coordinate of holes (default*: DZ/2).
y	Y coordinate of the initial hole (default*: 20).
B	Diameter of the holes centred in DY (default*: 8).
r	Number of external holes per side (default*: 2).
D	Diameter of the external holes (default*: 8).
s	Centre-to-centre between the external holes (default*: 32).

*in graphic editor.

Type 1 Mounting Plate Boring - BASET_1

Performs horizontal boring for mounting plates.

Unit:
Hinges/mounting plates



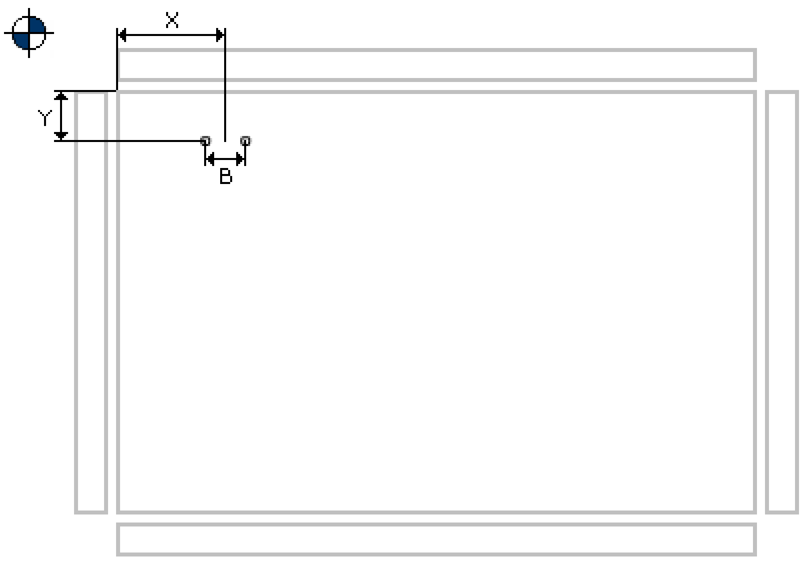
Minimum requirements:

- Upper face.
- Tip diameter 5 mm.
- Depth max. 12 mm.

Parameters:

X X coordinate of the centre of the centre-to-centre between holes (default*: 110).
Y Y coordinate of the holes (default*: 37).
Z Depth of the holes (default*: 12).
V Boring speed (default*: 3).
B Centre-to-centre between holes (default*: 32).
D Diameter of the holes (default*: 5).

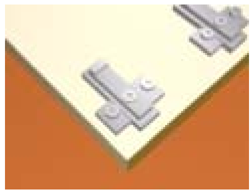
*in graphic editor.



Type 1 Mounting Plate Double Boring - BASET_1_D

Performs symmetrical double horizontal boring for mounting plates.

Unit:
Hinges/mounting plates



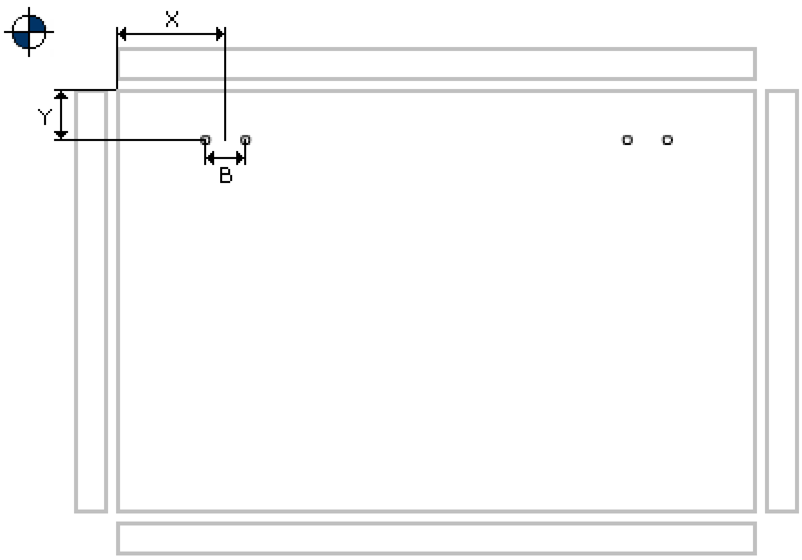
Minimum requirements:

- Upper face.
- Tip diameter 5 mm.
- Depth max. 12 mm.

Parameters:

X X coordinate of the centre of the centre-to-centre between holes (default*: 110).
Y Y coordinate of the holes (default*: 37).
Z Depth of the holes (default*: 12).
V Boring speed (default*: 3).
B Centre-to-centre between holes (default*: 32).
D Diameter of the holes (default*: 5).

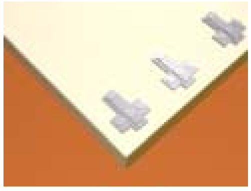
*in graphic editor.



Type 1 Mounting Plate Triple Boring - BASET_1_T

Performs triple horizontal boring for mounting plates.

Unit:
Hinges/mounting plates



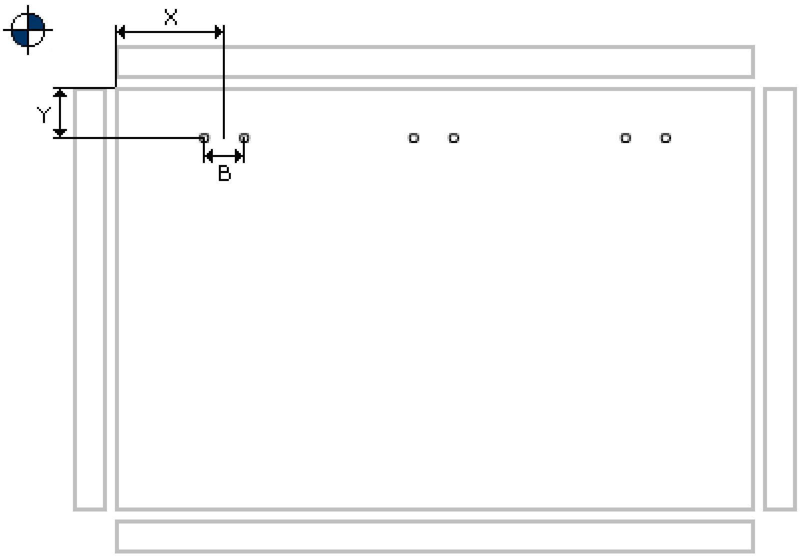
Minimum requirements:

- Upper face.
- Tip diameter 5 mm.
- Depth max. 12 mm.

Parameters:

X X coordinate of the centre of the centre-to-centre between holes (default*: 110).
Y Y coordinate of the holes (default*: 37).
Z Depth of the holes (default*: 12).
V Boring speed (default*: 3).
B Centre-to-centre between holes (default*: 32).
D Diameter of the holes (default*: 5).

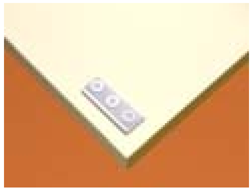
*in graphic editor.



Type 2 Mounting Plate Boring - BASET_2

Performs vertical boring for mounting plates.

Unit:
Hinges/mounting plates



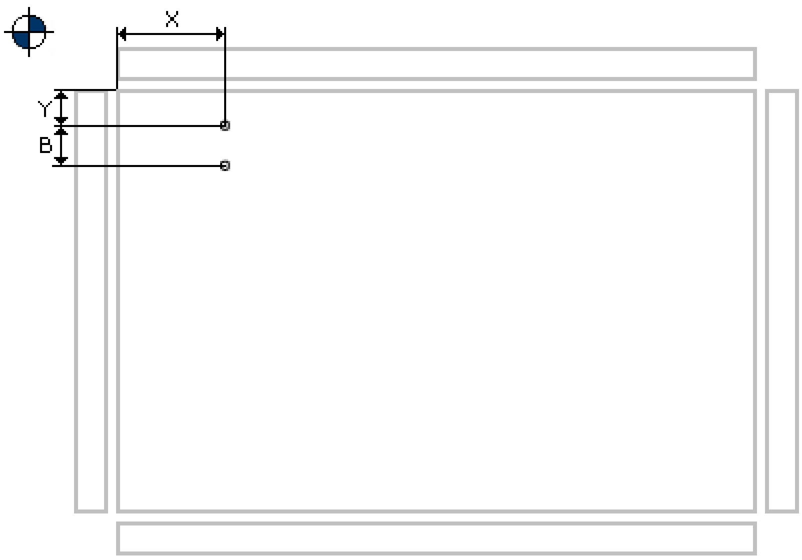
Minimum requirements:

- Upper face.
- Tip diameter 5 mm.
- Depth max. 12 mm.

Parameters:

X X coordinate of the holes (default*: 110).
Y Y coordinate of the first hole (default*: 21).
Z Depth of the holes (default*: 12).
V Boring speed (default*: 3).
B Centre-to-centre between holes (default*: 32).
D Diameter of the holes (default*: 5).

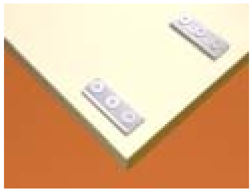
*in graphic editor.



Type 2 Mounting Plate Double Boring - BASET_2_D

Performs symmetrical double vertical boring for mounting plates.

Unit:
Hinges/mounting plates



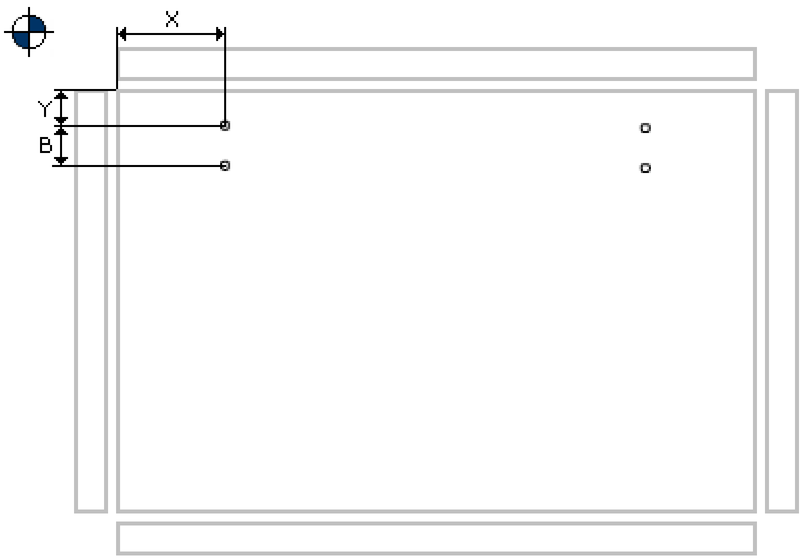
Minimum requirements:

- Upper face.
- Tip diameter 5 mm.
- Depth max. 12 mm.

Parameters:

X X coordinate of the holes (default*: 110).
Y Y coordinate of the first hole (default*: 21).
Z Depth of the holes (default*: 12).
V Boring speed (default*: 3).
B Centre-to-centre between holes (default*: 32).
D Diameter of the holes (default*: 5).

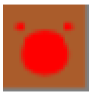
*in graphic editor.

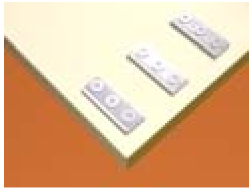


Type 2 Mounting Plate Triple Boring - BASET_2_T

Performs triple vertical boring for mounting plates.

Unit:
Hinges/mounting plates





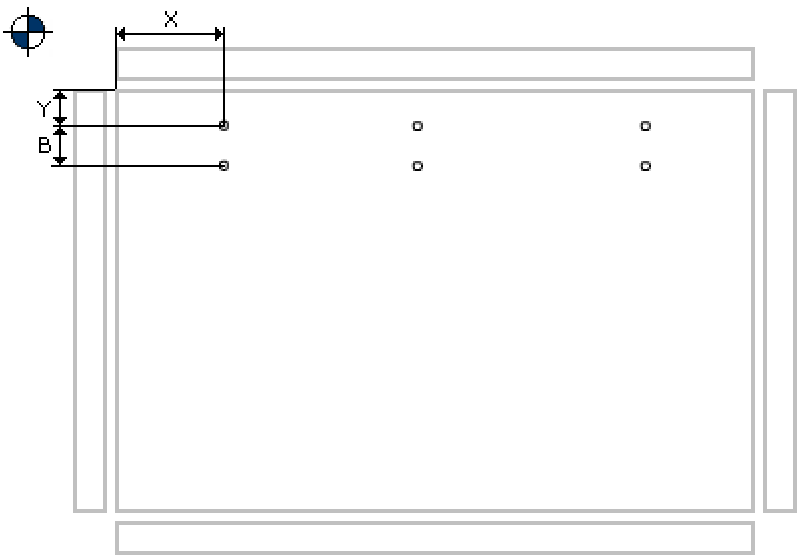
Minimum requirements:

- Upper face.
- Tip diameter 5 mm.
- Depth max. 12 mm.

Parameters:

X X coordinate of the holes (default*: 110).
Y Y coordinate of the first hole (default*: 21).
Z Depth of the holes (default*: 12).
V Boring speed (default*: 3).
B Centre-to-centre between holes (default*: 32).
D Diameter of the holes (default*: 5).

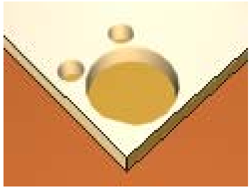
*in graphic editor.



Type 1 Hinge Boring - CERNIE_1

Performs boring for hinges.

Unit:
Hinges/mounting plates



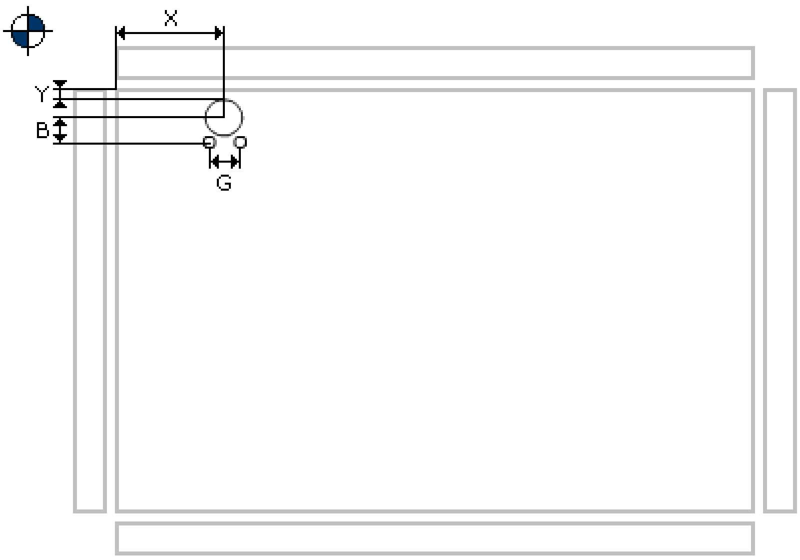
Minimum requirements:

- Upper face.
- Tip diameter 35 mm, depth max. 13 mm.
- Tip diameter 10 mm, depth max. 13 mm.

Parameters:

X	X coordinate of the centre of the biggest hole (default*: 110).
Y	Minimum distance in Y between the edge of the panel and the biggest hole (default*: 4).
Z	Depth of the biggest hole (default*: 13).
V	Boring speed for the biggest hole (default*: 1).
S	Depth of the smaller holes (default*: 13).
a	Diameter of the smaller holes (default*: 10).
B	Centre-to-centre between the biggest hole and the smaller holes (default*: 21.5).
D	Diameter of the biggest hole (default*: 35).
G	Centre-to-centre between the smaller holes (default*: 24).
L	Boring speed for the smaller holes (default*: 3).

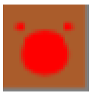
*in graphic editor.

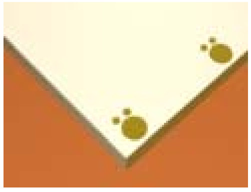


Type 1 Hinge Double Boring - CERNIE_1_D

Performs symmetrical double boring for hinges.

Unit:
Hinges/mounting plates





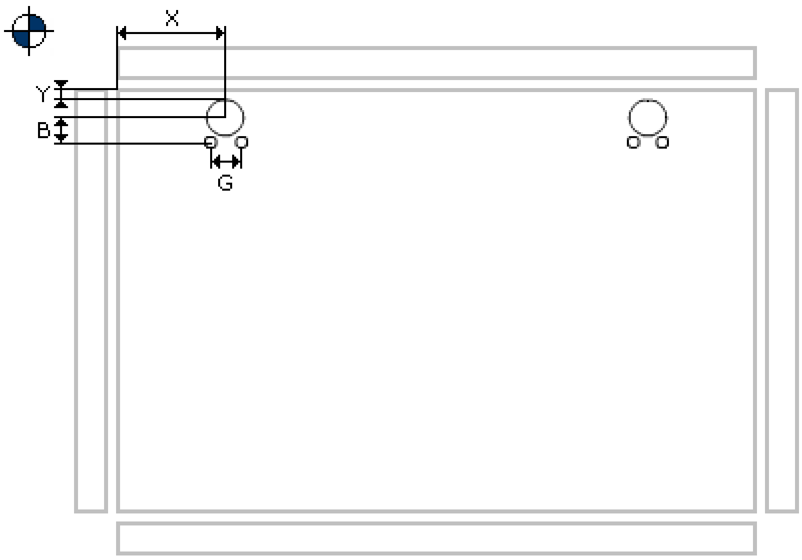
Minimum requirements:

- Upper face.
- Tip diameter 35 mm, depth max. 13 mm.
- Tip diameter 10 mm, depth max. 13 mm.

Parameters:

X X coordinate of the centre of the biggest hole (default*: 110).
Y Minimum distance in Y between the edge of the panel and the biggest hole (default*: 4).
Z Depth of the biggest hole (default*: 13).
V Boring speed for the biggest hole (default*: 1).
S Depth of the smaller holes (default*: 13).
a Diameter of the smaller holes (default*: 10).
B Centre-to-centre between the biggest hole and the smaller holes (default*: 21.5).
D Diameter of the biggest hole (default*: 35).
G Centre-to-centre between the smaller holes (default*: 24).
L Boring speed for the smaller holes (default*: 3).

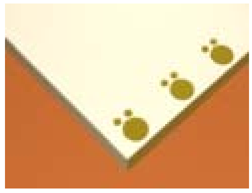
*in graphic editor.



Type 1 Hinge Triple Boring - CERNIE_1_T

Performs triple boring for hinges.

Unit:
Hinges/mounting plates



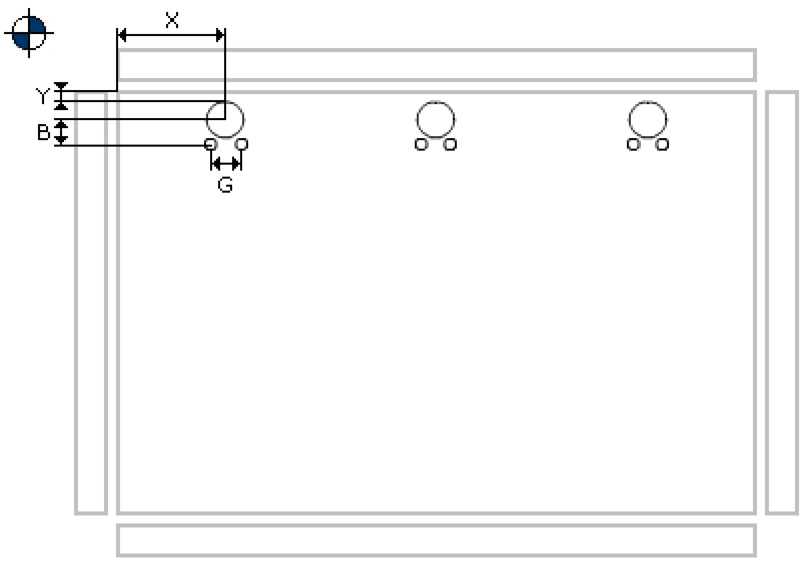
Minimum requirements:

- Upper face.
- Tip diameter 35 mm, depth max. 13 mm.
- Tip diameter 10 mm, depth max. 13 mm.

Parameters:

X X coordinate of the centre of the biggest hole (default*: 110).
Y Minimum distance in Y between the edge of the panel and the biggest hole (default*: 4).
Z Depth of the biggest hole (default*: 13).
V Boring speed for the biggest hole (default*: 1).
S Depth of the smaller holes (default*: 13).
a Diameter of the smaller holes (default*: 10).
B Centre-to-centre between the biggest hole and the smaller holes (default*: 21.5).
D Diameter of the biggest hole (default*: 35).
G Centre-to-centre between the smaller holes (default*: 24).
L Boring speed for the smaller holes (default*: 3).

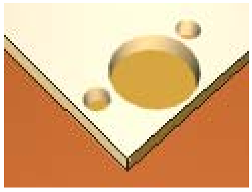
*in graphic editor.



Type 2 Hinge Boring - CERNIE_2

Performs boring for hinges.

Unit:
Hinges/mounting plates



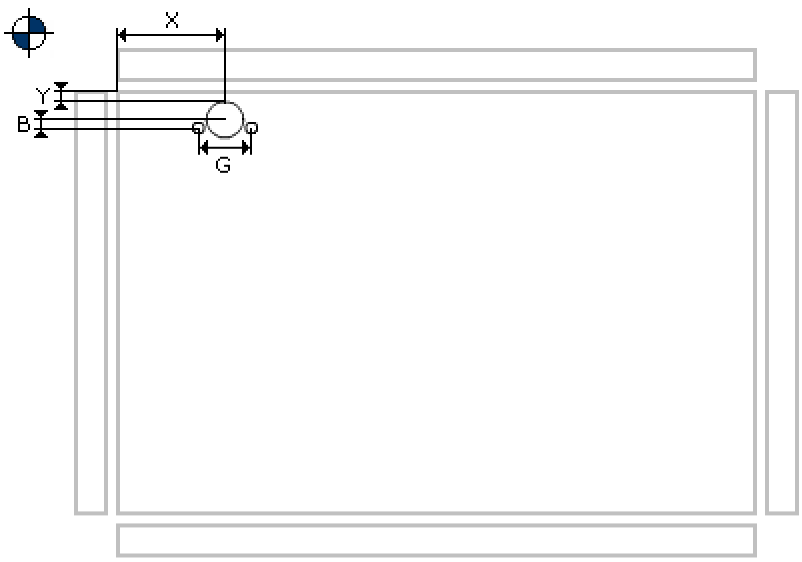
Minimum requirements:

- Upper face.
- Tip diameter 35 mm, depth max. 13 mm.
- Tip diameter 10 mm, depth max. 13 mm.

Parameters:

X	X coordinate of the centre of the biggest hole (default*: 110).
Y	Minimum distance in Y between the edge of the panel and the biggest hole (default*: 4).
Z	Depth of the biggest hole (default*: 13).
V	Boring speed for the biggest hole (default*: 1).
S	Depth of the smaller holes (default*: 13).
a	Diameter of the smaller holes (default*: 10).
B	Centre-to-centre between the biggest hole and the smaller holes (default*: 6).
D	Diameter of the biggest hole (default*: 35).
G	Centre-to-centre between the smaller holes (default*: 48).
L	Boring speed for the smaller holes (default*: 3).

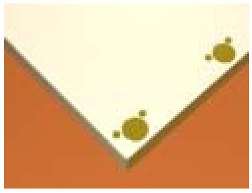
*in graphic editor.



Type 2 Hinge Double Boring - CERNIE_2_D

Performs symmetrical double boring for hinges.

Unit:
Hinges/mounting plates



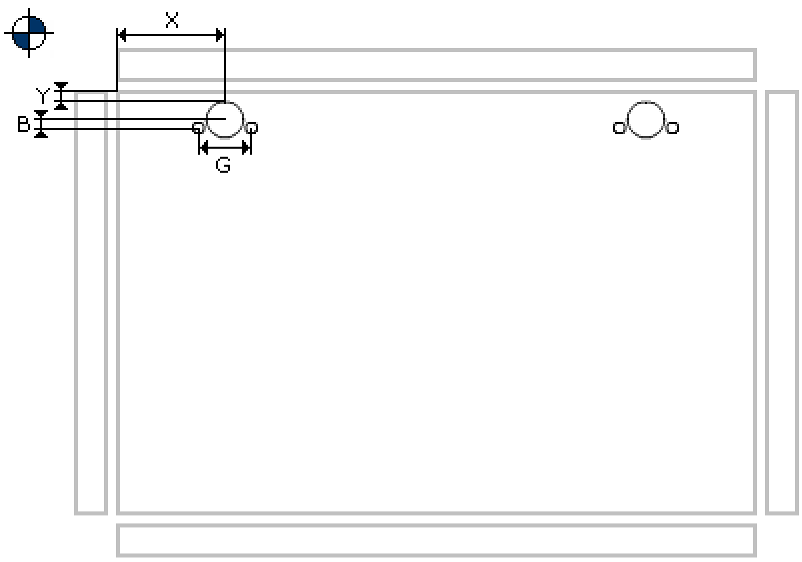
Minimum requirements:

- Upper face.
- Tip diameter 35 mm, depth max. 13 mm.
- Tip diameter 10 mm, depth max. 13 mm.

Parameters:

X	X coordinate of the centre of the biggest hole (default*: 110).
Y	Minimum distance in Y between the edge of the panel and the biggest hole (default*: 4).
Z	Depth of the biggest hole (default*: 13).
V	Boring speed for the biggest hole (default*: 1).
S	Depth of the smaller holes (default*: 13).
a	Diameter of the smaller holes (default*: 10).
B	Centre-to-centre between the biggest hole and the smaller holes (default*: 6).
D	Diameter of the biggest hole (default*: 35).
G	Centre-to-centre between the smaller holes (default*: 48).
L	Boring speed for the smaller holes (default*: 3).

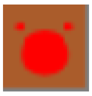
*in graphic editor.



Type 2 Hinge Triple Boring - CERNIE_2_T

Performs triple boring for hinges.

Unit:
Hinges/mounting plates



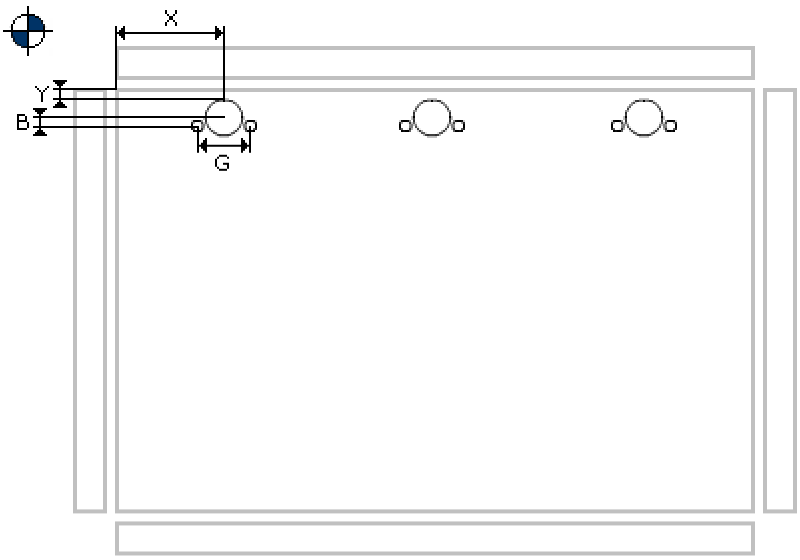
Minimum requirements:

- Upper face.
- Tip diameter 35 mm, depth max. 13 mm.
- Tip diameter 10 mm, depth max. 13 mm.

Parameters:

X X coordinate of the centre of the biggest hole (default*: 110).
Y Minimum distance in Y between the edge of the panel and the biggest hole (default*: 4).
Z Depth of the biggest hole (default*: 13).
V Boring speed for the biggest hole (default*: 1).
S Depth of the smaller holes (default*: 13).
a Diameter of the smaller holes (default*: 10).
B Centre-to-centre between the biggest hole and the smaller holes (default*: 6).
D Diameter of the biggest hole (default*: 35).
G Centre-to-centre between the smaller holes (default*: 48).
L Boring speed for the smaller holes (default*: 3).

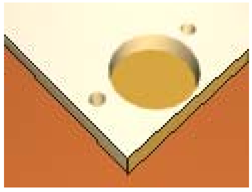
*in graphic editor.



Type 3 Hinge Boring - CERNIE_3

Performs boring for hinges.

Unit:
Hinges/mounting plates



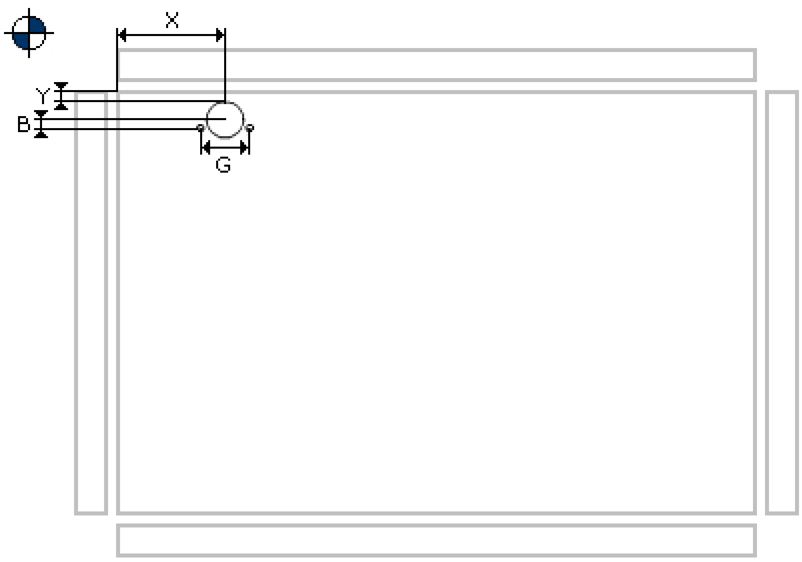
Minimum requirements:

- Upper face.
- Tip diameter 35 mm, depth max. 13 mm.
- Tip diameter 6 mm, depth max. 13 mm.

Parameters:

X	X coordinate of the centre of the biggest hole (default*: 110).
Y	Minimum distance in Y between the edge of the panel and the biggest hole (default*: 4).
Z	Depth of the biggest hole (default*: 13).
V	Boring speed for the biggest hole (default*: 1).
S	Depth of the smaller holes (default*: 13).
a	Diameter of the smaller holes (default*: 6).
B	Centre-to-centre between the biggest hole and the smaller holes (default*: 6).
D	Diameter of the biggest hole (default*: 35).
G	Centre-to-centre between the smaller holes (default*: 48).
L	Boring speed for the smaller holes (default*: 3).

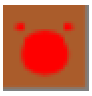
*in graphic editor.

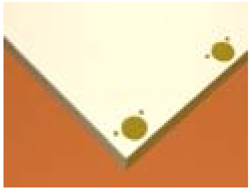


Type 3 Hinge Double Boring - CERNIE_3_D

Performs symmetrical double boring for hinges.

Unit:
Hinges/mounting plates





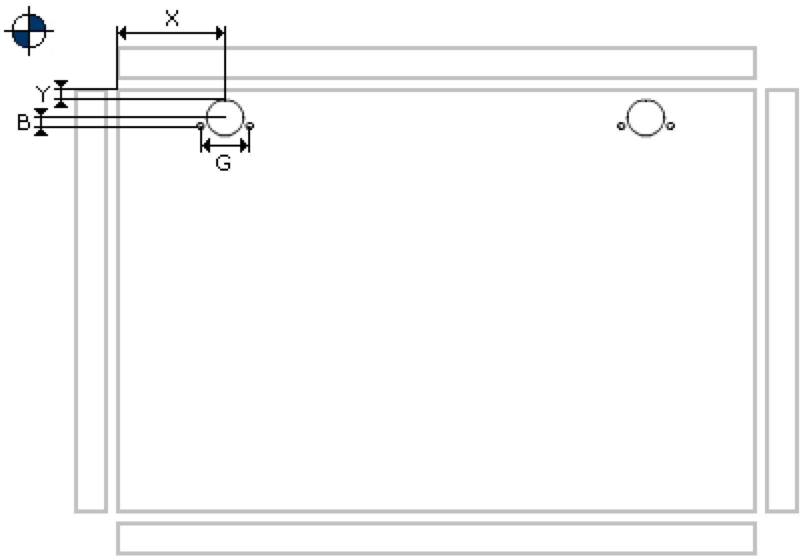
Minimum requirements:

- Upper face.
- Tip diameter 35 mm, depth max. 13 mm.
- Tip diameter 6 mm, depth max. 13 mm.

Parameters:

X	X coordinate of the centre of the biggest hole (default*: 110).
Y	Minimum distance in Y between the edge of the panel and the biggest hole (default*: 4).
Z	Depth of the biggest hole (default*: 13).
V	Boring speed for the biggest hole (default*: 1).
S	Depth of the smaller holes (default*: 13).
a	Diameter of the smaller holes (default*: 6).
B	Centre-to-centre between the biggest hole and the smaller holes (default*: 6).
D	Diameter of the biggest hole (default*: 35).
G	Centre-to-centre between the smaller holes (default*: 48).
L	Boring speed for the smaller holes (default*: 3).

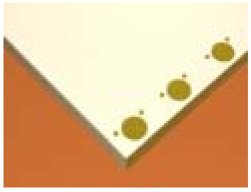
*in graphic editor.



Type 3 Hinge Triple Boring - CERNIE_3_T

Performs triple boring for hinges.

Unit:
Hinges/mounting plates



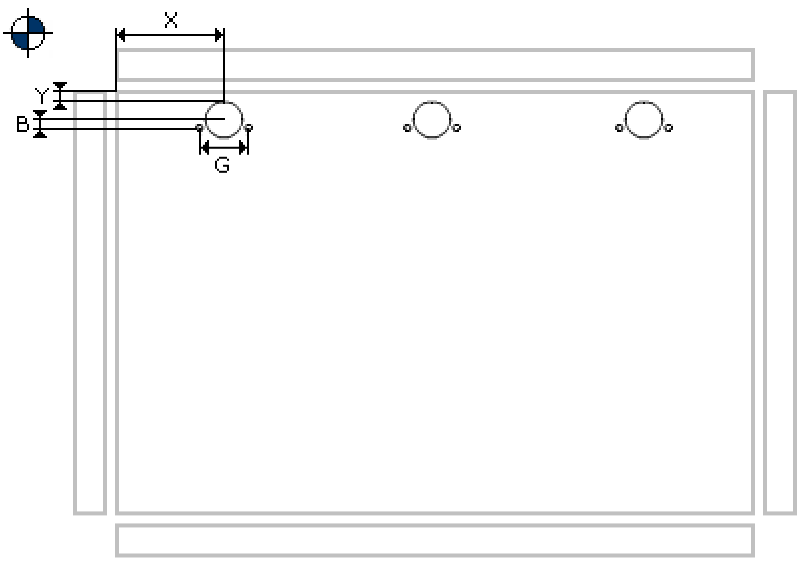
Minimum requirements:

- Upper face.
- Tip diameter 35 mm, depth max. 13 mm.
- Tip diameter 6 mm, depth max. 13 mm.

Parameters:

X	X coordinate of the centre of the biggest hole (default*: 110).
Y	Minimum distance in Y between the edge of the panel and the biggest hole (default*: 4).
Z	Depth of the biggest hole (default*: 13).
V	Boring speed for the biggest hole (default*: 1).
S	Depth of the smaller holes (default*: 13).
a	Diameter of the smaller holes (default*: 6).
B	Centre-to-centre between the biggest hole and the smaller holes (default*: 6).
D	Diameter of the biggest hole (default*: 35).
G	Centre-to-centre between the smaller holes (default*: 48).
L	Boring speed for the smaller holes (default*: 3).

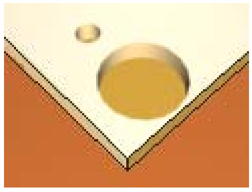
*in graphic editor.



Type 4 Hinge Boring - CERNIE_4

Performs boring for hinges.

Unit:
Hinges/mounting plates



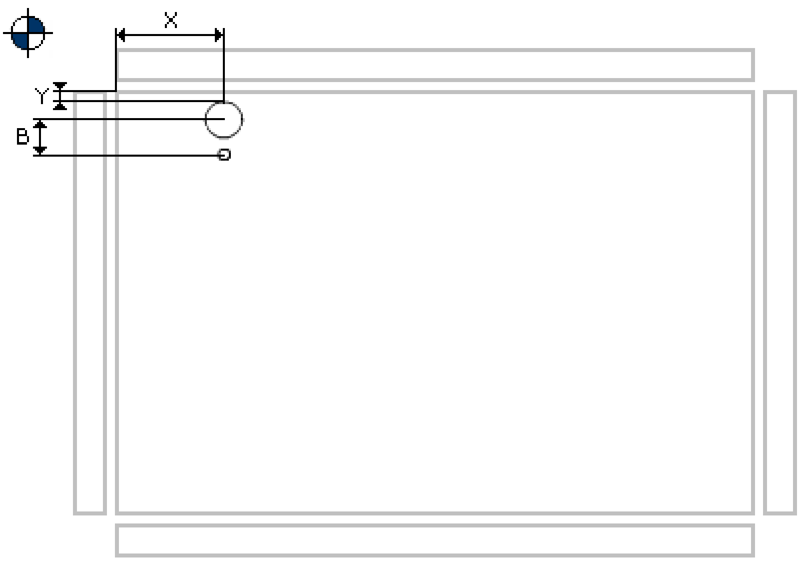
Minimum requirements:

- Upper face.
- Tip diameter 35 mm, depth max. 13 mm.
- Tip diameter 10 mm, depth max. 13 mm.

Parameters:

X X coordinate of the centre of the biggest hole (default*: 110).
Y Minimum distance in Y between the edge of the panel and the biggest hole (default*: 4).
Z Depth of the biggest hole (default*: 13).
V Boring speed for the biggest hole (default*: 1).
S Depth of the smaller hole (default*: 13).
a Diameter of the smaller hole (default*: 10).
B Centre-to-centre between the biggest hole and the smaller hole (default*: 32).
D Diameter of the biggest hole (default*: 35).
L Boring speed for the smaller hole (default*: 3).

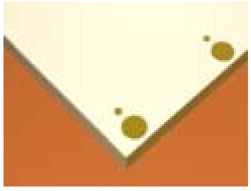
*in graphic editor.



Type 4 Hinge Double Boring - CERNIE_4_D

Performs symmetrical double boring for hinges.

Unit:
Hinges/mounting plates



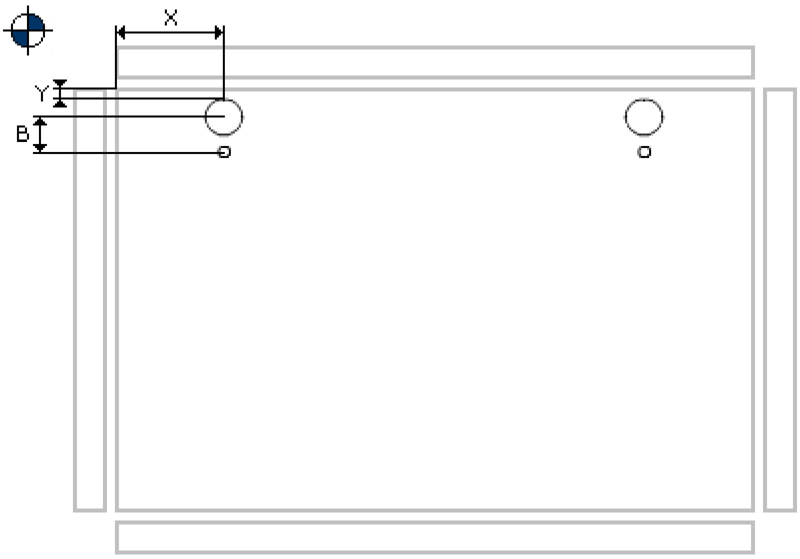
Minimum requirements:

- Upper face.
- Tip diameter 35 mm, depth max. 13 mm.
- Tip diameter 10 mm, depth max. 13 mm.

Parameters:

X X coordinate of the centre of the biggest hole (default*: 110).
Y Minimum distance in Y between the edge of the panel and the biggest hole (default*: 4).
Z Depth of the biggest hole (default*: 13).
V Boring speed for the biggest hole (default*: 1).
S Depth of the smaller hole (default*: 13).
a Diameter of the smaller hole (default*: 10).
B Centre-to-centre between the biggest hole and the smaller hole (default*: 32).
D Diameter of the biggest hole (default*: 35).
L Boring speed for the smaller hole (default*: 3).

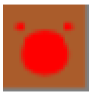
*in graphic editor.

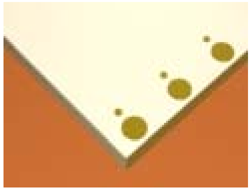


Type 4 Hinge Triple Boring - CERNIE_4_T

Performs triple boring for hinges.

Unit:
Hinges/mounting plates





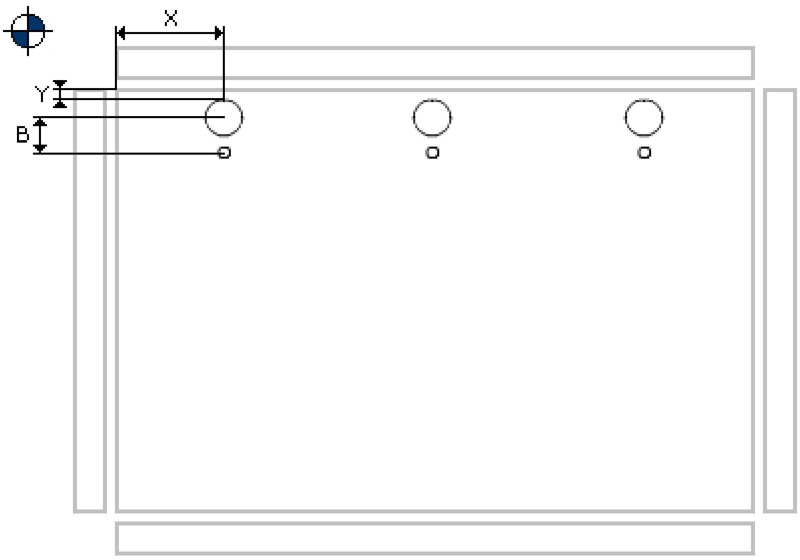
Minimum requirements:

- Upper face.
- Tip diameter 35 mm, depth max. 13 mm.
- Tip diameter 10 mm, depth max. 13 mm.

Parameters:

X X coordinate of the centre of the biggest hole (default*: 110).
Y Minimum distance in Y between the edge of the panel and the biggest hole (default*: 4).
Z Depth of the biggest hole (default*: 13).
V Boring speed for the biggest hole (default*: 1).
S Depth of the smaller hole (default*: 13).
a Diameter of the smaller hole (default*: 10).
B Centre-to-centre between the biggest hole and the smaller hole (default*: 32).
D Diameter of the biggest hole (default*: 35).
L Boring speed for the smaller hole (default*: 3).

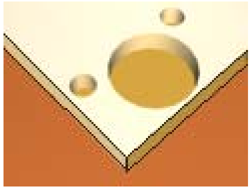
*in graphic editor.



Type 5 Hinge Boring - CERNIE_5

Performs boring for hinges.

Unit:
Hinges/mounting plates



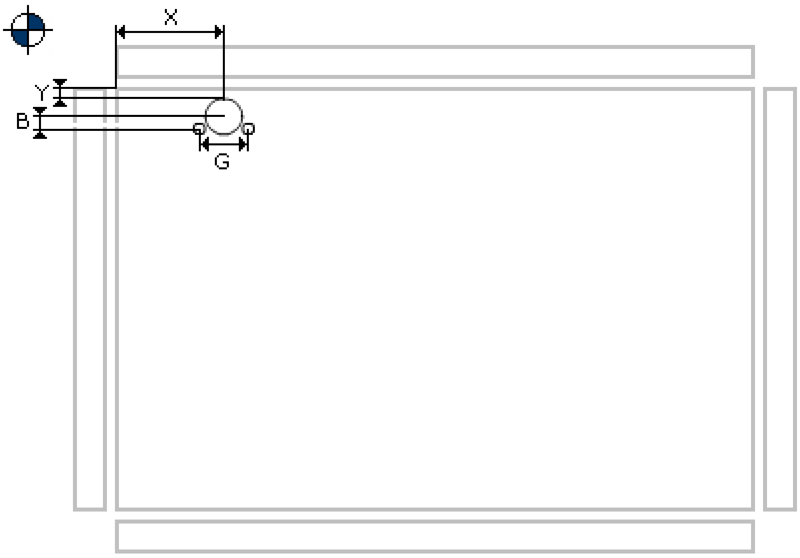
Minimum requirements:

- Upper face.
- Tip diameter 35 mm, depth max. 13 mm.
- Tip diameter 10 mm, depth max. 13 mm.

Parameters:

X X coordinate of the centre of the biggest hole (default*: 110).
Y Minimum distance in Y between the edge of the panel and the biggest hole (default*: 4).
Z Depth of the biggest hole (default*: 13).
V Boring speed for the biggest hole (default*: 1).
S Depth of the smaller holes (default*: 13).
a Diameter of the smaller holes (default*: 10).
B Centre-to-centre between the biggest hole and the smaller holes (default*: 9.5).
D Diameter of the biggest hole (default*: 35).
G Centre-to-centre between the smaller holes (default*: 45).
L Boring speed for the smaller holes (default*: 3).

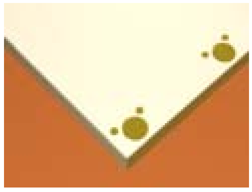
*in graphic editor.



Type 5 Hinge Double Boring - CERNIE_5_D

Performs symmetrical double boring for hinges.

Unit:
Hinges/mounting plates



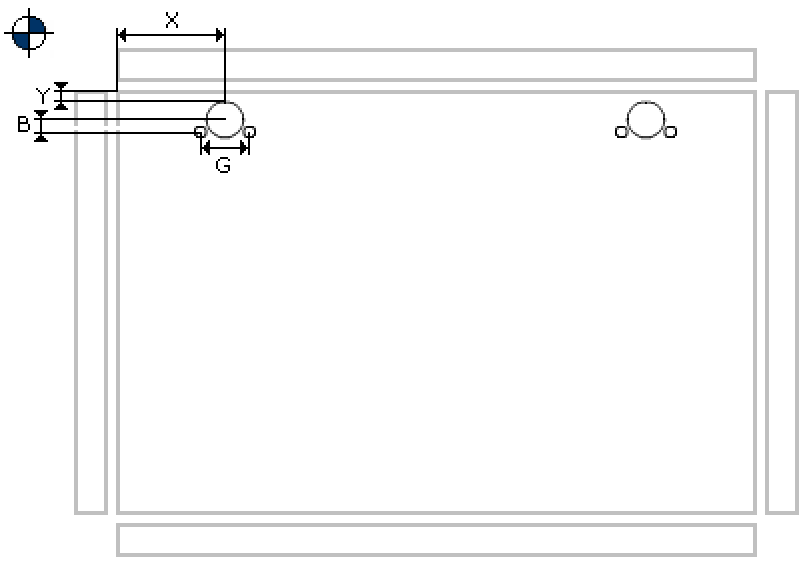
Minimum requirements:

- Upper face.
- Tip diameter 35 mm, depth max. 13 mm.
- Tip diameter 10 mm, depth max. 13 mm.

Parameters:

X	X coordinate of the centre of the biggest hole (default*: 110).
Y	Minimum distance in Y between the edge of the panel and the biggest hole (default*: 4).
Z	Depth of the biggest hole (default*: 13).
V	Boring speed for the biggest hole (default*: 1).
S	Depth of the smaller holes (default*: 13).
a	Diameter of the smaller holes (default*: 10).
B	Centre-to-centre between the biggest hole and the smaller holes (default*: 9.5).
D	Diameter of the biggest hole (default*: 35).
G	Centre-to-centre between the smaller holes (default*: 45).
L	Boring speed for the smaller holes (default*: 3).

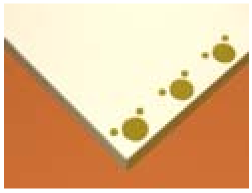
*in graphic editor.



Type 5 Hinge Triple Boring - CERNIE_5_T

Performs triple boring for hinges.

Unit:
Hinges/mounting plates



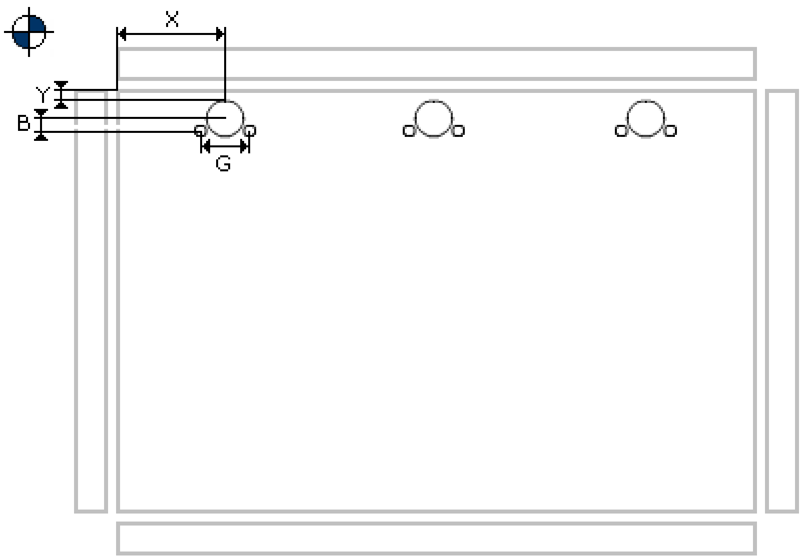
Minimum requirements:

- Upper face.
- Tip diameter 35 mm, depth max. 13 mm.
- Tip diameter 10 mm, depth max. 13 mm.

Parameters:

X X coordinate of the centre of the biggest hole (default*: 110).
Y Minimum distance in Y between the edge of the panel and the biggest hole (default*: 4).
Z Depth of the biggest hole (default*: 13).
V Boring speed for the biggest hole (default*: 1).
S Depth of the smaller holes (default*: 13).
a Diameter of the smaller holes (default*: 10).
B Centre-to-centre between the biggest hole and the smaller holes (default*: 9.5).
D Diameter of the biggest hole (default*: 35).
G Centre-to-centre between the smaller holes (default*: 45).
L Boring speed for the smaller holes (default*: 3).

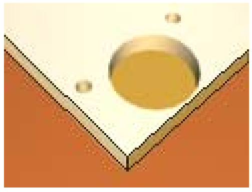
*in graphic editor.



Type 6 Hinge Boring - CERNIE_6

Performs boring for hinges.

Unit:
Hinges/mounting plates



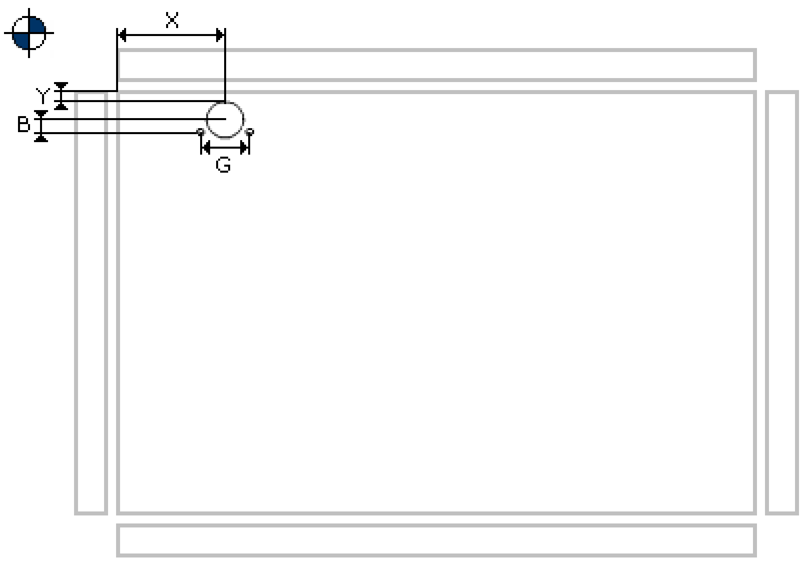
Minimum requirements:

- Upper face.
- Tip diameter 35 mm, depth max. 13 mm.
- Tip diameter 6 mm, depth max. 13 mm.

Parameters:

X X coordinate of the centre of the biggest hole (default*: 110).
Y Minimum distance in Y between the edge of the panel and the biggest hole (default*: 4).
Z Depth of the biggest hole (default*: 13).
V Boring speed for the biggest hole (default*: 1).
S Depth of the smaller holes (default*: 13).
a Diameter of the smaller holes (default*: 6).
B Centre-to-centre between the biggest hole and the smaller holes (default*: 9.5).
D Diameter of the biggest hole (default*: 35).
G Centre-to-centre between the smaller holes (default*: 45).
L Boring speed for the smaller holes (default*: 3).

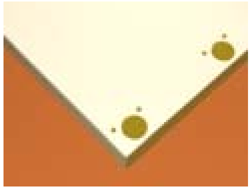
*in graphic editor.



Type 6 Hinge Double Boring - CERNIE_6_D

Performs symmetrical double boring for hinges.

Unit:
Hinges/mounting plates



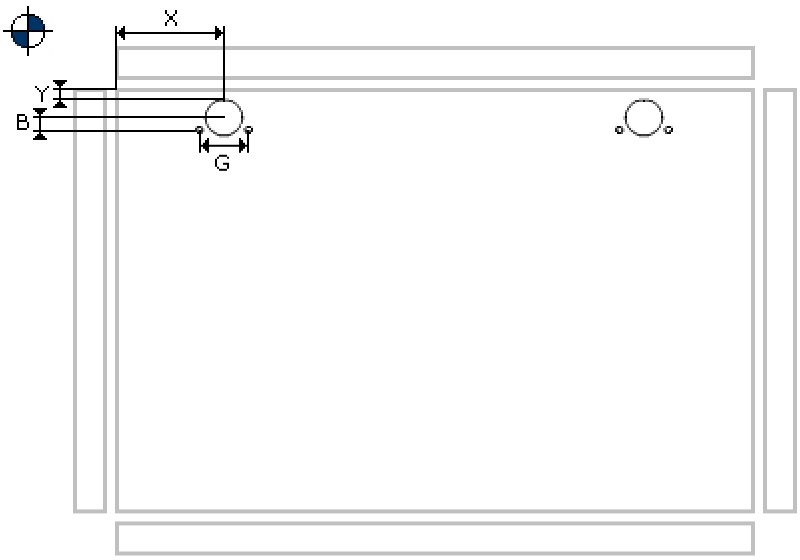
Minimum requirements:

- Upper face.
- Tip diameter 35 mm, depth max. 13 mm.
- Tip diameter 6 mm, depth max. 13 mm.

Parameters:

X X coordinate of the centre of the biggest hole (default*: 110).
Y Minimum distance in Y between the edge of the panel and the biggest hole (default*: 4).
Z Depth of the biggest hole (default*: 13).
V Boring speed for the biggest hole (default*: 1).
S Depth of the smaller holes (default*: 13).
a Diameter of the smaller holes (default*: 6).
B Centre-to-centre between the biggest hole and the smaller holes (default*: 9.5).
D Diameter of the biggest hole (default*: 35).
G Centre-to-centre between the smaller holes (default*: 45).
L Boring speed for the smaller holes (default*: 3).

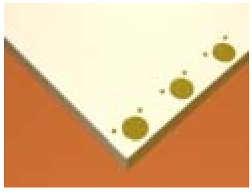
*in graphic editor.



Type 6 Hinge Triple Boring - CERNIE_6_T

Performs triple boring for hinges.

Unit:
Hinges/mounting plates



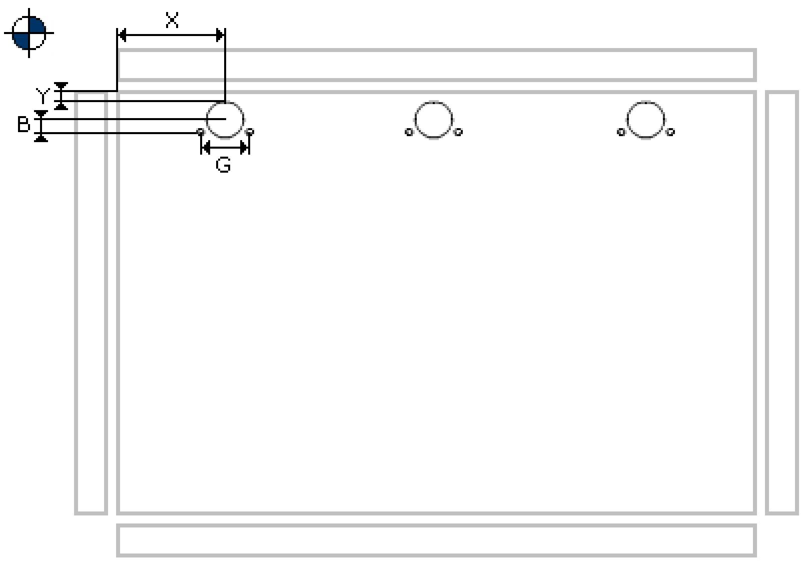
Minimum requirements:

- Upper face.
- Tip diameter 35 mm, depth max. 13 mm.
- Tip diameter 6 mm, depth max. 13 mm.

Parameters:

X X coordinate of the centre of the biggest hole (default*: 110).
Y Minimum distance in Y between the edge of the panel and the biggest hole (default*: 4).
Z Depth of the biggest hole (default*: 13).
V Boring speed for the biggest hole (default*: 1).
S Depth of the smaller holes (default*: 13).
a Diameter of the smaller holes (default*: 6).
B Centre-to-centre between the biggest hole and the smaller holes (default*: 9.5).
D Diameter of the biggest hole (default*: 35).
G Centre-to-centre between the smaller holes (default*: 45).
L Boring speed for the smaller holes (default*: 3).

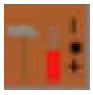
*in graphic editor.

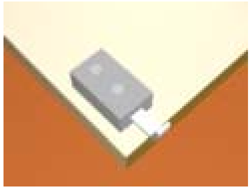


Connection for Mobile Unit - AGGANCIO_PP

Performs boring for wall unit brackets.

Unit:
Hardware/miscellaneous





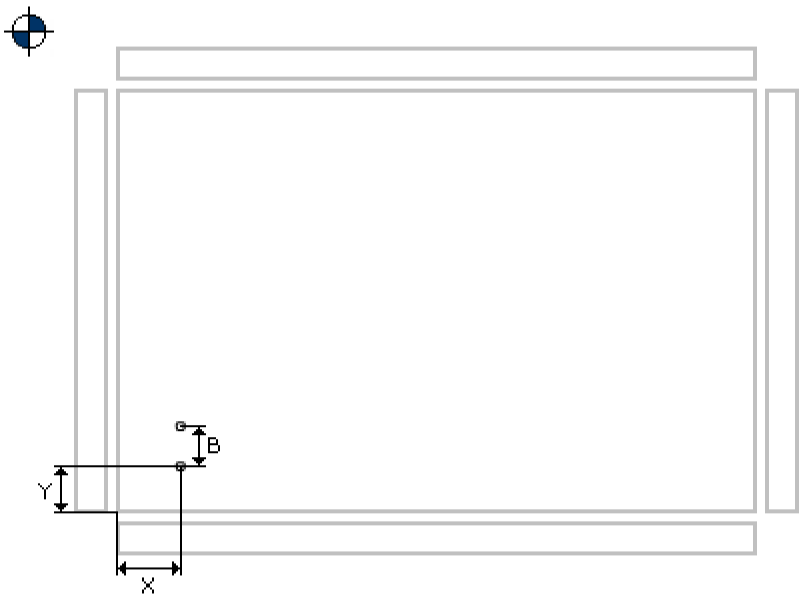
Minimum requirements:

- Upper face.
- Tip diameter 8 mm, depth max. 12 mm.

Parameters:

X X coordinate of the holes (default*: 41).
Y Distance from the edge in Y to be subtracted from the DY value (default*: 31).
Z Depth of the holes (default*: 12).
V Boring speed (default*: 3).
B Centre-to-centre between holes (default*: 32).
D Diameter of the holes (default*: 8).

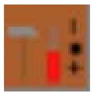
*in graphic editor.



Circular Spotlight Fixing Holes - FORI_FARETTO

Makes a set of holes in a circle for application of a spotlight.

Unit:
Hardware/miscellaneous



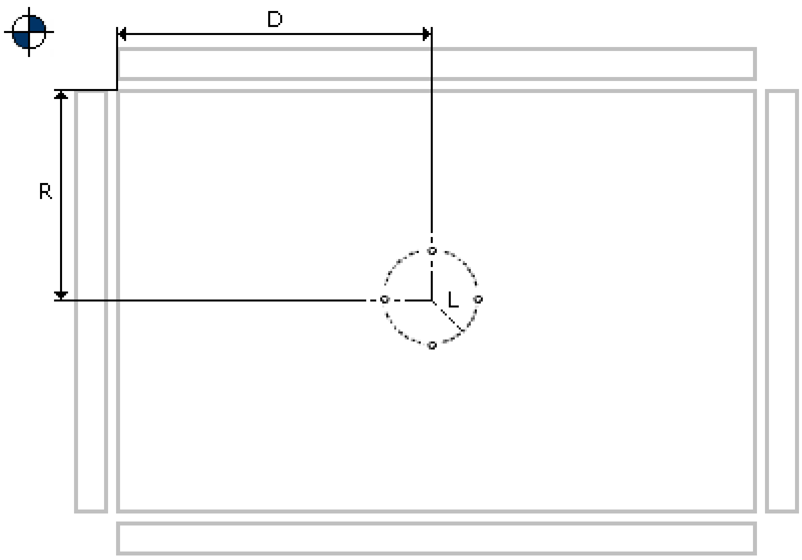
Minimum requirements:

- Upper face.
- Default tool 101.
- Depth max. 13 mm.

Parameters:

Z Depth of the holes (default*: 13).
H Number of holes (default*: 4).
T Tool number (default: 101).
R Y coordinate of the centre of the circle (default*: DY/2).
D X coordinate of the centre of the circle (default*: DX/2).
L Radius of the circle (default*: 25).

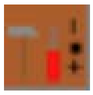
*in graphic editor.



Hole for Spotlight - FORO_FARETTO_C

Makes a circular recess for a spotlight.

Unit:
Hardware/miscellaneous



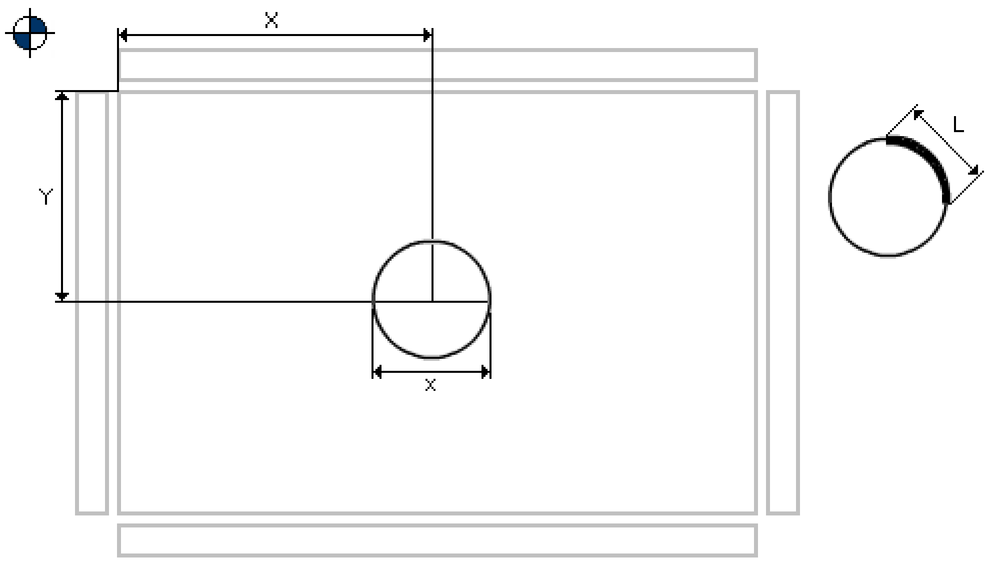
Minimum requirements:

- Upper face.
- Default tool 101.
- Depth max. DZ-1 mm.

Parameters:

X X coordinate of the centre of the recess (default*: DX/2).
Y Y coordinate of the centre of the recess (default*: DY/2).
Z Recess depth (default*: DZ-1).
H Type of lead out (1 = straight, 2 = curved) (default*: 1).
I Type of lead in (1 = straight, 2 = curved) (default*: 2).
T Tool number (default: 101).
x Recess diameter (default*: 80).
D Tool radius multiplication factor (default*: 2).
s Type of tool approach at lead in and lead out (0 = at height, 1 = descending)
G Direction of travel (2 = clockwise, 3 = anti-clockwise) (default*: 2).
L Overlap length (default*: 0).

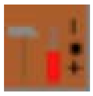
*in graphic editor.



Rectangular Spotlight Hole - FORO_FARETTO_R

Makes a rectangular recess for a spotlight.

Unit:
Hardware/miscellaneous



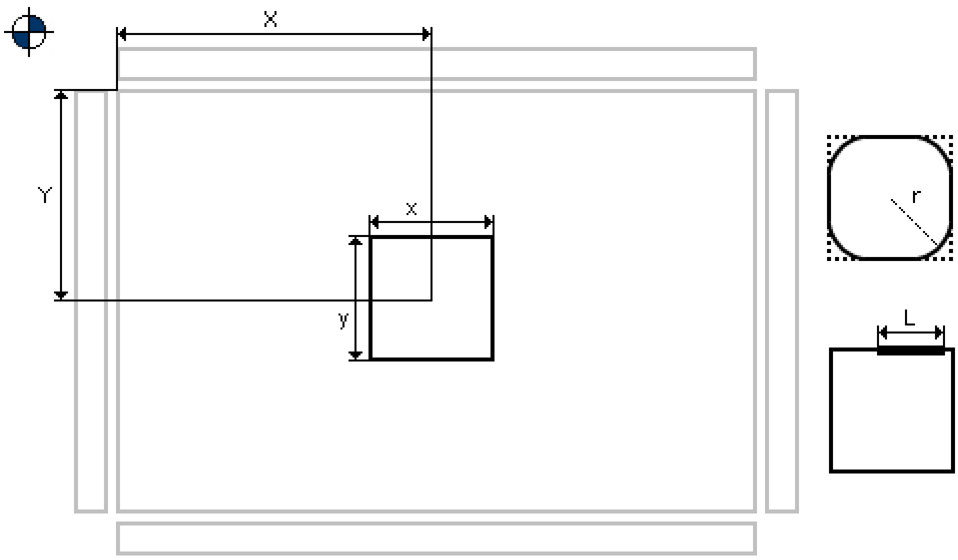
Minimum requirements:

- Upper face.
- Default tool 101.
- Depth max. DZ+2 mm.

Parameters:

X X coordinate of the centre of the recess (default*: DX/2).
Y Y coordinate of the centre of the recess (default*: DY/2).
Z Recess depth (default*: DZ+2).
H Type of lead out (1 = straight, 2 = curved) (default*: 1).
I Type of lead in (1 = straight, 2 = curved) (default*: 2).
T Tool number (default: 101).
x Recess length (default*: 80).
y Recess width (default*: 80).
r Fillet radius (default*: 0).
D Tool radius multiplication factor (default*: 2).
s Type of tool approach at lead in and lead out (0 = at height, 1 = descending) (default*: 1).
G Direction of travel (2 = clockwise, 3 = anti-clockwise) (default*: 2).
L Overlap length (default*: 2).

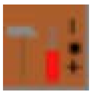
*in graphic editor.

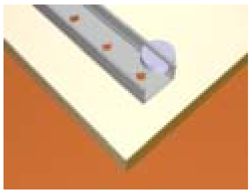


Boring for Drawer Guide - GUIDA_CASSETTI

Makes holes at intervals for a drawer guide.

Unit:
Hardware/miscellaneous





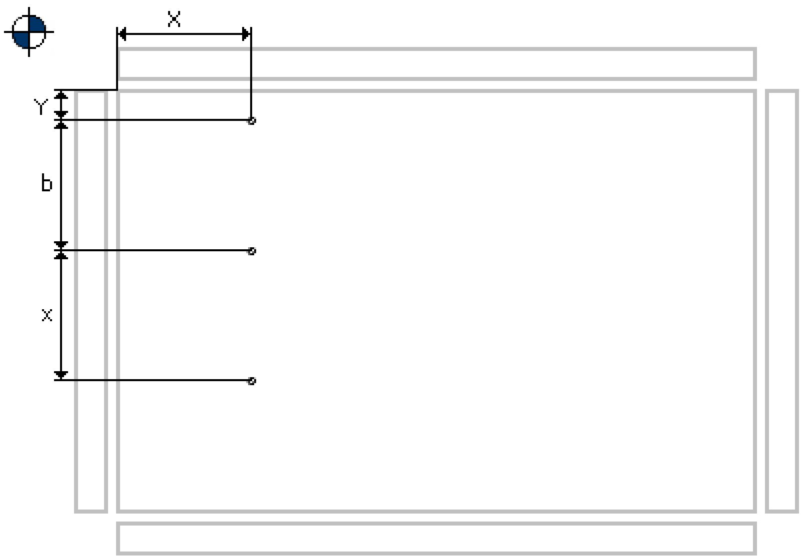
Minimum requirements:

- Upper face.
- Default tool flat hole tip diameter 5 mm.
- Depth 12 mm.

Parameters:

X X coordinate of the holes (default*: 120).
Y Y coordinate of the initial hole (default*: 28).
Z Depth of the holes (default*: 12).
V Milling speed (default*: 3).
x Centre-to-centre between the second and the third holes (default*: 128).
y Centre-to-centre between the third and the fourth holes (default*: 0).
b Centre-to-centre between the first and the second holes (default*: 128).
D Diameter of the holes (default*: 5).

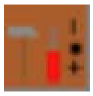
*in graphic editor.



Knob/Handle Boring - MANIGLIA_1

Makes a hole for a handle on a wing/panel.

Unit:
Hardware/miscellaneous



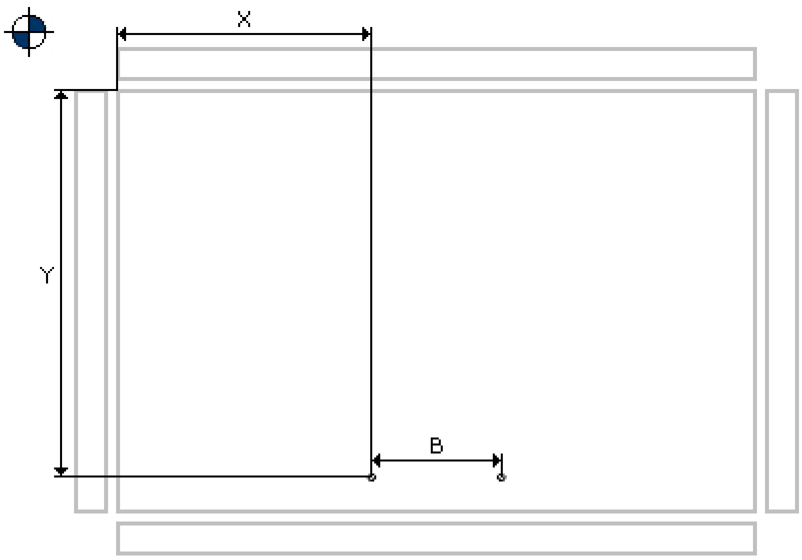
Minimum requirements:

- Upper face.
- Default tool flat hole tip diameter 5 mm.
- Depth DZ-1.

Parameters:

X	X coordinate of the first hole (default*: (DX/2)-64).
Y	Y coordinate of the holes (default*: 32).
Z	Depth of the holes (default*: DZ-1).
V	Boring speed (default*: 3).
B	Centre-to-centre between holes (default*: 128).
D	Diameter of the holes (default*: 5).

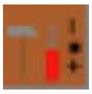
*in graphic editor.



Circular Handle Recess - MANIGLIA_C

Makes a circular recess for a handle.

Unit: Hardware/miscellaneous



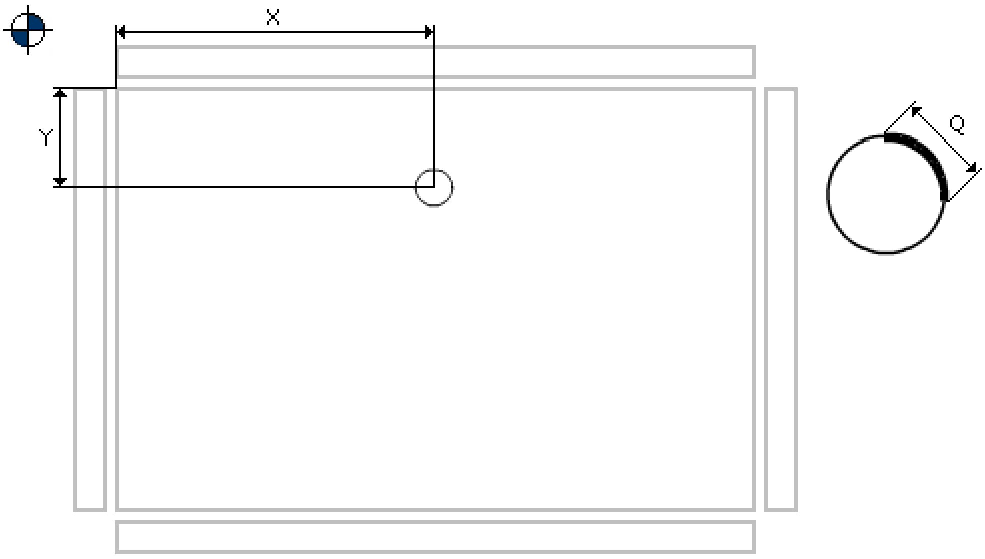
Minimum requirements:

- Upper face.
- Default tool 101.
- Depth max. 15 mm.

Parameters:

X X coordinate of the centre of the handle (default*: 300).
Y Y coordinate of the centre of the handle (default*: 1/4DY).
Z Handle depth (default*: 15).
I Overtravel distance (default*: 2).
V Operating speed (default*: 4).
T Tool number (default: 101).
Q Overlap length (default*: 2).
D Cutter diameter (default*: 12).
s Tool number of revolutions (default*: 18000).
L Handle diameter (default*: 30).

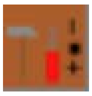
*in graphic editor.



Rectangular Handle Recess - MANIGLIA_R

Makes a rectangular recess for a handle.

Unit:
Hardware/miscellaneous



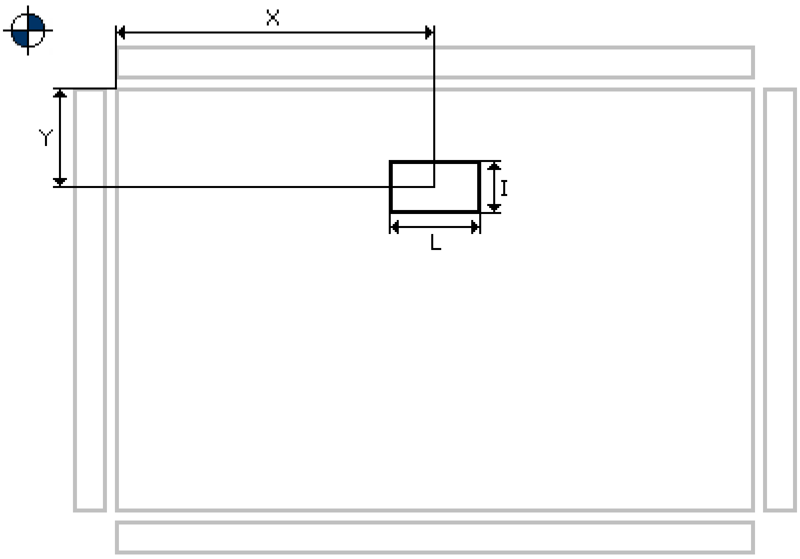
Minimum requirements:

- Upper face.
- Default tool 101.
- Depth max. 15 mm.

Parameters:

X	X coordinate of the centre of the handle (default*: DX/2).
Y	Y coordinate of the centre of the handle (default*: 1/4 DY).
Z	Handle depth (default*: 15).
I	Handle width (default*: 50).
V	Operating speed (default*: 3).
T	Tool number (default: 101).
D	Cutter diameter (default*: 12).
s	Tool number of revolutions (default*: 18000).
L	Handle length (default*: 80).

*in graphic editor.



Channel for Back Direction X - CANALE

Performs routing for back/bottom.

Unit:
Macro standard

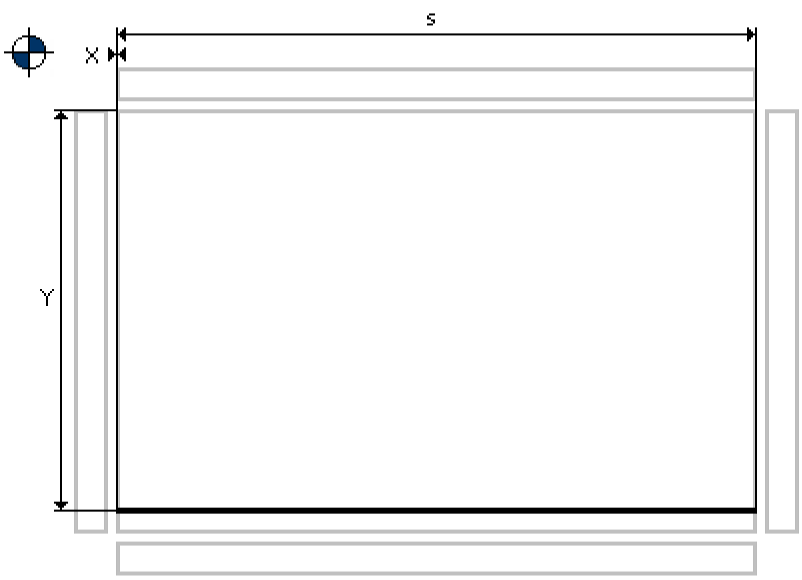
Minimum requirements:

- Upper face.
- Default tool 101.
- Depth max. 4 mm.
- It is advisable to use a disk mill.

Parameters:

X X coordinate of the start of routing (default*: 0).
Y Y coordinate of the start of routing (default*: 10).
Z Routing depth (default*: 4).
I Type of tool approach at lead in and lead out (0 = at height, 1 = descending) (default*: 0).
V Operating speed.
S Tool number of revolutions.
T Tool number (default: 101).
r Tool radius multiplication factor (default*: 2).
s X coordinate of the end of routing (default*: DX).
l Tool correction (0=no correction, 1=right correction, 2=left correction, 3=depth correction) (default*: 0).
G Tool lead in/lead out (1=yes, =no) (default*: 1).

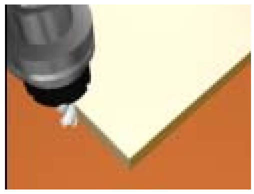
*in graphic editor.



Workpiece Moulding - SCONTORNATURA

Performs routing, trimming the rectangular workpiece.

Unit:
Macro standard

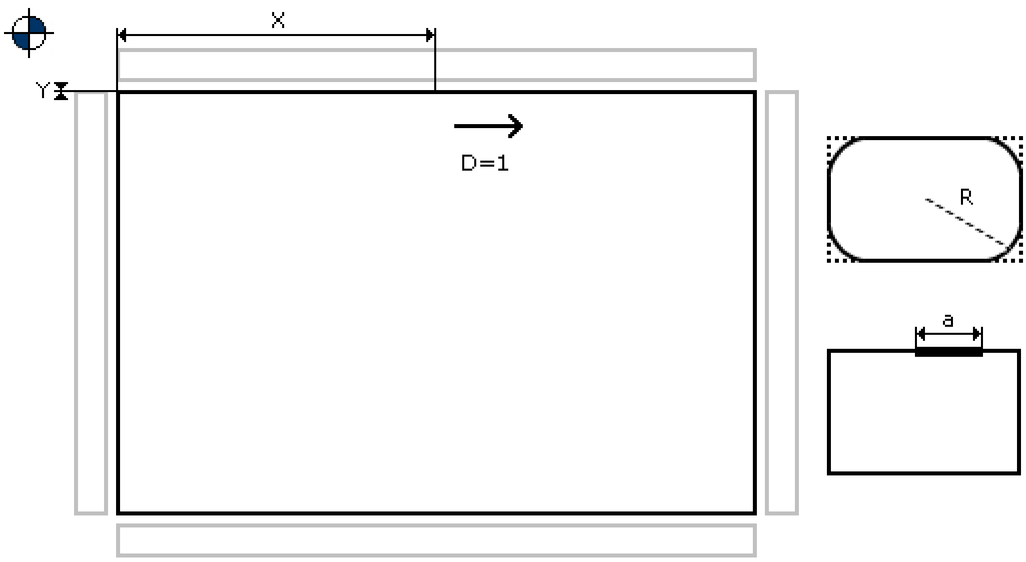


Minimum requirements:

- Upper face.
- Default tool 101.
- Depth max. DZ+2 mm.

Parameters:

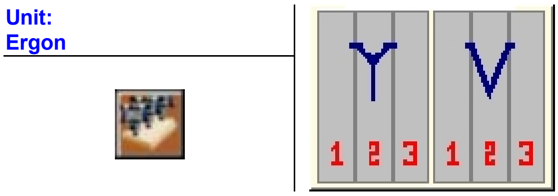
X X coordinate of the start of routing (default*: DX/2).
Y Y coordinate of the start of routing (default*: 0).
Z Routing depth (default*: DZ+2).
I Type of lead in (1 = straight, 2 = curved) (default*: 2).
V Operating speed.
S Tool number of revolutions.
T Tool number (default: 101).
R Fillet radius (default*: 0).
a Overlap length (default*: 5).
r Tool radius multiplication factor (default*: 2).
D Direction of travel (2 = clockwise, 3 = anti-clockwise) (default*: 1).
s Over-metal quota (default*: 0).



5.4.2 User Macros for Ergon Machines

Selecting Piece Block sub Areas - XSUBAREA

This macro is used to define which vacuum sub-area must be used to hold down the workpiece(s) positioned on the work table.



Parameters:

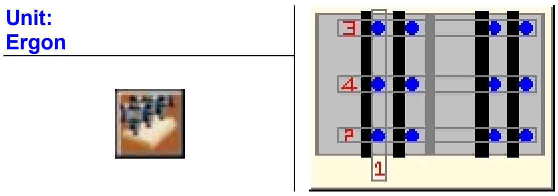
a Sub-areas combination

0	All sub-areas present.
(n)	Sub-area number n (from 1 to 3).
12, 23, ecc.	Sub-areas combinations: sub-areas number 1 and 2, sub-areas number 2 and 3, ecc.

Note. The vacuum sub-areas can also be set with parameter V of the H instruction. If you use this parameter, when the program is started from the Machine Panel you can make further changes to the selection of the sub-areas in the template for the H instruction parameters.

Selecting Stop Files for Piece Block sub Areas - XBATTON

Macro for selecting the combination of rows of stops to be activated for Ergon machines with a TV table.



Parameters:

a Workpiece hold-down sub-areas.

0	All machine table hold-down sub-areas.
1	Indicates hold-down sub-area 1.
2	Indicates hold-down sub-area 2.
3	Indicates hold-down sub-area 3.

B Selection of rows of stop.

-1	No stops required.
0	All rows of stops.

(n)	Select row of stops number n (from 1 to 4).
12, 123, ecc.	Select rows of stops: number 1 and 2, number 1, 2 and 3, etc.

Keep in mind that the system controls that the row of vertical stops (1 in the illustration) is not raised, even if requested, for the couplet area if the “a” field is programmed to “0”.

Example 1:

H … -AD
XBATTON a=0 B=12

In a machine with only one vacuum sub-area per table, rows 1 and 2 of stops are raised for table Y (AB) and only row 2 of table V (CD), because it is assumed that a long piece is to be blocked and the row of vertical stops, 1 on table V, would be in the way, if it were raised.

Example 2:

H … -AB
XBATTON a=0 B=12

In a machine with two vacuum sub-areas per table, rows 1 and 2 of stops are raised for table Y1 (A) and only row 2 of table Y2 (B), because it is assumed that a piece long enough to occupy the whole table Y is to be blocked, and the row of vertical stops n° 1 on table Y2 (B), would be in the way, if it were raised.

Example 3:

H … -AB
XBATTON a=0 B= -1

In a machine with any number of vacuum sub-areas, no rows of stops are raised since the value of the B parameter is “-1“.

To give sufficient space for programming the row of stops with the purpose of handling special cases, the desired combination of rows of stops to be raised over each configured vacuum sub-area can be forced (to date, a maximum of 4 divided into 2 per table). See the following example.

Example 4:

H … -AB
XBATTON a=1 B=12
XBATTON a=2 B=12

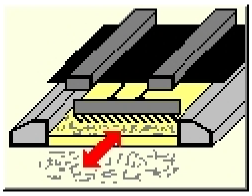
In a machine with two vacuum sub-areas per table, rows 1 and 2 of stops are raised for table Y1 (A) as well as rows 1 and 2 of table Y2 (B), because field “a” is not set to “0”, i.e. the simultaneous use of all of the vacuum sub-areas is not requested.

NOTE: Machines with just one piece-blocking sub-area can use field "T" of the header in alternative to this macro, with the advantage of being able to modify the programming of the row of stops, during program execution in Machine Panel, without having to do it in the editing phase. If more than one piece-blocking sub-areas per table are present, the macro must be used because, in the current version of Xilog Plus, the T field of the header does not permit the specification of the combination of stop rows inside each individual sub-area.

Cleaning Tables with Brushes - XCLEAN

This macro is used to request, within the workpiece program, a cleaning cycle on the work table with or without selection by means of the relative push-button.

Unit:
Ergon



Parameters:

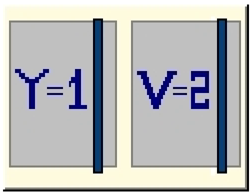
a Request for cleaning selection.

a=1 Requires selection using the push button.
a=2 Automatic cleaning (without selection).

Positioning Motorised Guides on Table - XGUIDEM

This macro is used to position the motor-driven guides (if present).

Unit:
Ergon



Parameters:

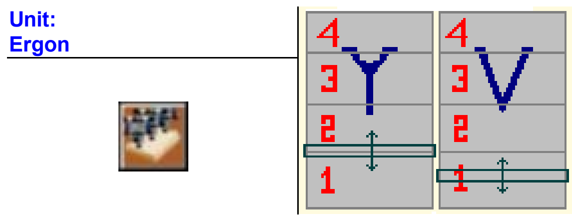
Q Positioning dimension (dimension in mm to which the guide must go).
a Guide table.

a=1 Table Y.
a=2 Table V.
a=3 Table Y and V.

s Positioning speed (speed at which the guide must move).

Horizontal Guide Dimension Programming - XGUIDEH

This macro is used for positioning the motorised guides (if present).



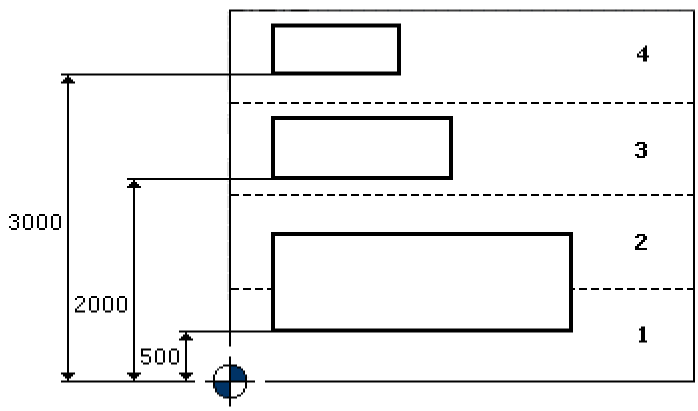
Parameters:

a Guide position. The position of the guide for the current step programmed according to the defined Y scale (origin and direction).

B Sub-area combination (combination of vacuum sub-areas affected by the current programmed guide positioning step).

This macro is programmed as many times as there are pieces to be positioned on the table along the Y direction. Each repetition of the macro represents a step in the motorised guide positioning cycle. This corresponds with blocking a piece in a given position Y on the table; the sub-areas affected by the action of blocking the piece are defined in field B.

Take, for example, the following case in which the following sequence of pieces is to be blocked: one on the pair of sub-areas 1 and 2 at the position Y=500, one on sub-area 3 at position Y=2000 and one on sub-area 4 at position Y=3000.



To program this situation, following the Header instruction, 3 XGUIDEH instructions must be inserted, constructed as follows:

H …
XGUIDEH a=500 B=12
XGUIDEH a=2000 B=3
XGUIDEH a=3000 B=4
…

Remember that, in order for the horizontal motorised guide positioning cycle to function properly, the series of keys “$” which preface the description of the same must be coherently programmed:

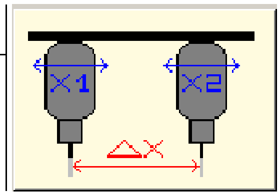
- GEN_PARK_QUOTE_GUIDE_H
- GEN_OFFSET_QUOTE_GUIDE_H
- GEN_N_MAX_POS_GUIDE_H
- GEN_SUBAREA_MIN_LIMIT
- GEN_SUBAREA_MAX_LIMIT

For detailed information on the above, see the specific section dedicated to the configuration file nci.cfg in the parameters manual.

Set Distance in X between Heads for Twin Machining - XINTAX

This macro is for defining the distance at which the two heads must be synchronised on the two independent X axes.

Unit:
Ergon



Parameters:

a Distance value on the X axis (dimension in mm for synchronisation of the two heads).

Delta between Actual Tool Length and Corrected by Feeler - XDELTAPALPATORE

This macro is used to correct, using an arbitrary value, the tool length value to which the feeler must make reference, i.e. more specifically, the system of the dynamic operators of the NC, preposed for the correction of position Z of the head-tool machining the piece.

Unit:
Ergon

Parameters:

a Delta length (value which differentiates the real tool length from that presented to the Z position run-time correction system).

If the program requires the use of the feeler (presence of a SET TAU instruction) and there is no XDELTAPALPATORE instruction present, the correction value in Z, assumed by the

dynamic operators system, will be equal to the real length of the tool. In any other case in which the macro in question is present, with a non-null value assigned to its "a" parameter, the aforementioned correction value in Z shall be determined by the algebraic sum (i.e. with sign) between the real tool length and the value of "a". This way, it is possible to machine with the same inserted tool and feeler, at different depths and, therefore, on different tool profiles.
The macro should be considered modal for the tool; therefore, after a tool change-over, the tool length correction value for the feeler is zeroed.

5.4.3 Various User Macros

Blower - XBLOWER

Blower for tool cooling and lubrication.

Unit:
User macro 5

Parameters:

E Validation.
T List of tools with relative spindle (103,145,206,…).

The combinations of authorized parameters are:

1. E=not programmed T=tools list
   E=1 T=tools list
   Validates the blower for the tools indicated.

2. E=0 T=tools list
. Disables the blower for the tools indicated

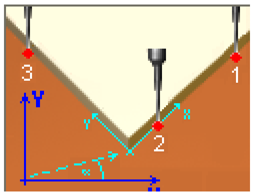
3. E=not programmed T=not programmed
   E=1 T=not programmed
   Validates generic blower.

4. E=0 T=not programmed
   Disables the general blower and/or blower for all validated tools.

Foratura multipla - XMULTIDRILL

Graphically displays the result of a boring operation with angle drive with two or more outputs. This instruction is compatible with Xilog Plus version 1.10.007 or higher. Boring on face 1, with rotation about Z, are compatible with Xilog Plus version 1.11.xx or higher.

Unit:
User macro 5



Parameters:

X X co-ordinate of the hole made with the tool programmed in field T.
Y Y co-ordinate of the hole made with the tool programmed in field T.
Z Hole depth.
R Number of holes (including the original hole).
x X step for repetitions.
y Y step for repetitions.
T Tool number.
Q Holes repetition according to specular in X/Y (0=No, 1=SX, 2=SY, 3=SXSY).
V Boring speed.
S Rotation speed of tool.
G Number of chip discharge steps.
E Position of vacuum hood (see: Appendix D).
D Out machining quota.
a Rotation of boring about the Z-axis (only for face 1).

To use this instruction, individually configure all of the tools which are part of the angle drive, assigning the same tool set number programmed in T to each. For each tool configure offsets X, Y and Z detected with the angle drive mounted on the spindle and the Vector axis at 0°.
The instruction can be entered at any point in the program.

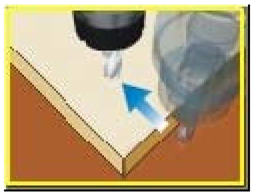
Example:

00001 H DX=450 DY=450 DZ=50 -A R=1 *MM /"DEF"
00002 **XMULTIDRILL X=100 Y=25 Z=10 T=101 F=4**
00005 …

Null Operation between Machining - XNOP

Performs a return upstroke to a skimming dimension between successive machining (bringing the Z axis to the skimming dimension). Useful on work centers with bars and suction cups to avoid collision between tools and bars for side movements.

Unit:
User macro 5



Parameters:

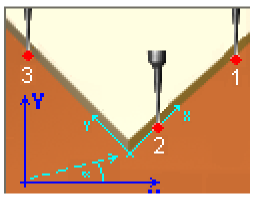
f Valori ammessi:

(n.d.) Always performs a return upstroke to a skimming dimension between successive machining (equivalent to instruction SET NOPF45 = -1).
0 Cancels the last condition set (with values greater than or less than 0) with no return upstroke between successive machining. (equivalent to instruction SET NOPF45 = 0).
23 performs a return upstroke to a skimming dimension between successive machining on face 2 or face 3 (equivalent to instruction SET NOPF45 = 23).
45 Performs a return upstroke to a skimming dimension between successive machining on face 4 or face 5 (equivalent to instruction SET NOPF45 = 1).

Acquisizione origini e rotazione - XORGACQ

Detects panel deviations in X and Y and rotation about the Z-axis relative to the origin of the field selected. The instruction performs two feeling operations on side X and side Y with a tool such as the XYZ feeler.
This instruction is compatible with Xilog Plus version 1.10.003 and higher.

Unit:
User macro 5



Parameters:

N Names of variables which will be filled with the values detected for: X origin, Y origin, rotation about the Z-axis.
T Tool (number of a feeler type tool).
QX1 Feeling dimension of first point on side X (default = DX - 10).
QX2 Feeling dimension of second point on side X (default = 10).
QX3 Feeling dimension of first point on side Y (default = DY - 10).
QZ Feeling Z dimension for all points (default = DZ/2).

Parameters N and T are obligatory. The instruction requires configuration of a XYZ feeler type tool.

The XORGACQ instruction can be entered only once, after the Header and before any other operating instruction, and must be followed by a ROT instruction, to which the origin values detected must be passed (the XORGACQ instruction only applies the translation and not the rotation to machining which follows it).

Example:

00001 H DX=450 DY=450 DZ=50 -A R=1 *MM /"DEF"
00002 **XORGACQ N="ORGX ORGY ANG" T=180**
00003 **ROT X=ORGX Y=ORGY A=ANG**
00004 XB X=100 Y=50 T=101
00005 …

xT - XT

Enables a tool.

Unit:
Macro standard

Parameters:

G Electric spindle + tool.
N Command.

1. If no tools are programmed, any enabled tools are annulled.

2. If a tool is programmed, but not the command, the tool becomes the only enabled tool

Example:

XB X=1200 Y=350 Z=10 **T=101**

can be written as follows

XT G=101
XB X=1200 Y=350 Z=10

3. If the tool is programmed and the command is **"+"**(in guided text editor enter only the character +), the tool is added to the list of active tools.

Example:

XB X=1200 Y=350 Z=10 **T=1,2,3**

can be written as follows

XT G=1
XT G=2 N="+"
XT G=3 N="+"
XB X=1200 Y=350 Z=10

4. The tool may be indicated with an expression.

Example:

XB X=1200 Y=350 Z=10 **T=145**

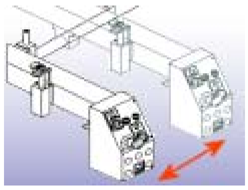
can be written as follows

PAR mySpindle = 1
PAR myTool = 45
XT G=mySpindle*100+myTool
XB X=1200 Y=350 Z=10

TvOnOff - XTVONOFF

Steps Separator.

Unit:
User macro 5



Parameters:

E Validation (0 o 1).

Before moving the bars, the macro brings the units to the home position.

The following information regarding the Steps Separator must be inserted in the configuration file nci.cfg:

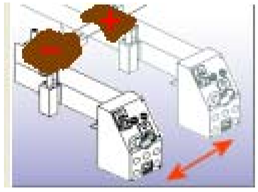
- $GEN_GENTVON$
codes for validating the Separator

- $GEN_GENTVOFF$
codes for disabling the Separator

TVSIDE - XTVSIDE

Modal macro for defining the half-workpiece being machined after step separation: moves the origin in the direction set and over the distance set.

Unit:
User macro 5



Parameters:

D Step separator stroke.
N Direction of movement in X (- or +; otherwise the current origin is confirmed).

If parameter D is omitted, the stroke distance is that set in the configuration file fields.cfg.
The new origin is cancelled by another XTVSIDE or by the O/XO instruction.

6. Advanced Programming

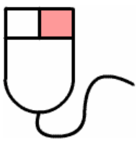
6.1 Parametric Programming

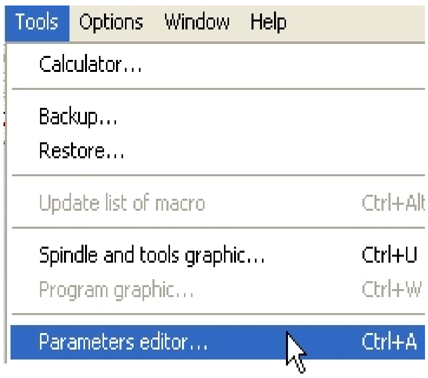
6.1.1 Introduction to Parametric Programming

With parametric programming it is possible to insert various parameters into a program (which represent, for example, length, width, thickness) the value of which can vary depending on the need. In this way, a single program allows you to control work pieces with the same profiles but of different sizes.
The instructions for parametric programming can be used in the text editor in the same manner as the other instructions. In the Graphic Editor, the PAR and D instructions can be used; they are entered using the Parameters Editor. You can enter a total of up to 512 variables (PAR, L, D) in a program.
There is a Parameters Wizard, for both the Text and Graphic Editor, to easily display the PAR parameters of a program and modify the values.
Once they are inserted into a program, the parameters can easily be changes before the program is run: the changes will only be valued for that execution. In this manner the same program can be "personalized" for each execution.

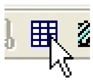
► Opening the Parameters Editor (only in graphic editor).

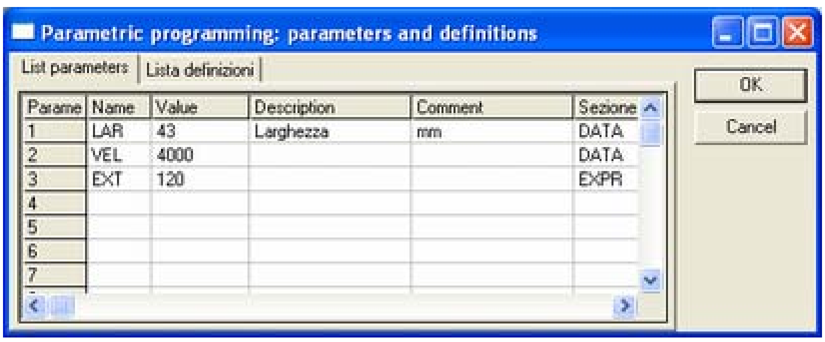
Click on the
TOOLS/PARAMETERS EDITOR
menu…





… or on the button.

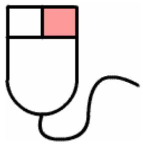


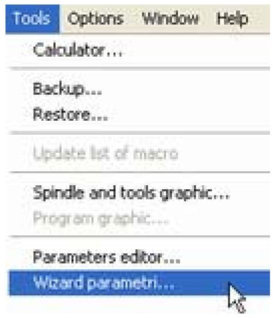


The Parameter Editor is pre-set for entering the PAR instructions (Parameters List) and D instructions (Definitions List). To enter data, click twice on a box, enter the data and confirm with the **[ENTER]** key.

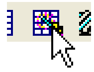
► Opening the Parameters Wizard.

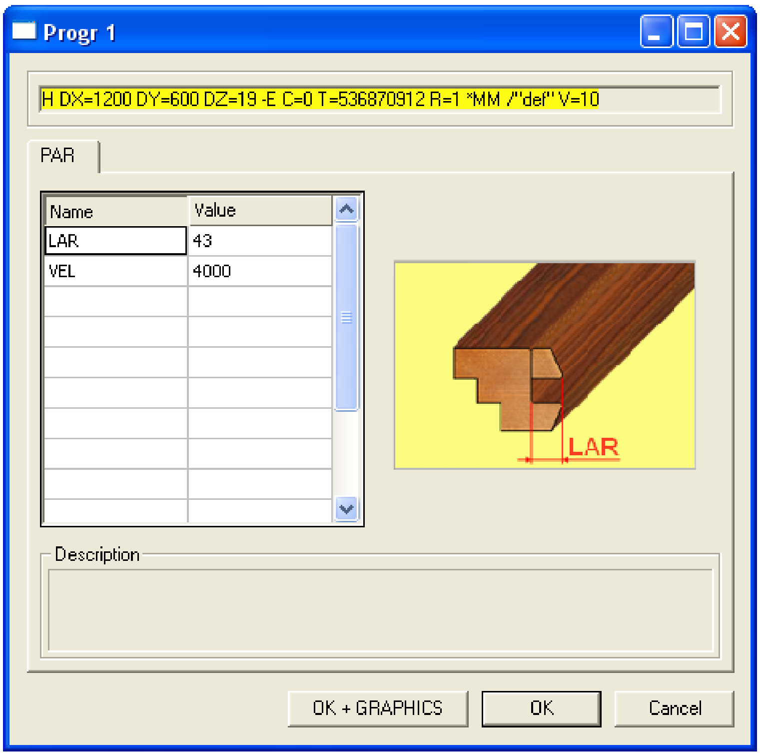
Click on the **TOOLS/PARAMETERS WIZARD** menu…





… or on the button.



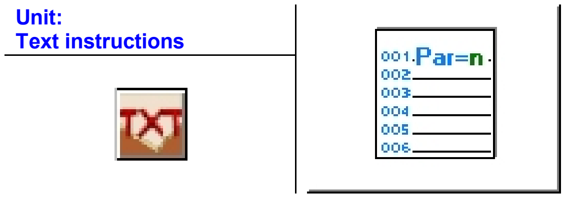


The Parameters Wizard shows the PAR parameters grouped in pages according to the section it belongs to (see: PARSECTION instruction) so that the value can be modified: to modify a value, click on the value to be modified, enter the new value and confirm with the **[ENTER]** key.
The **OK** + **GRAPHICS** button (enables both in text editor and in graphic editor) confirms the parameters and updates the editor below without exiting from the environment.
In graphic editor mode, the graphic is updated automatically, while in text editor mode the graphic display window is automatically opened.

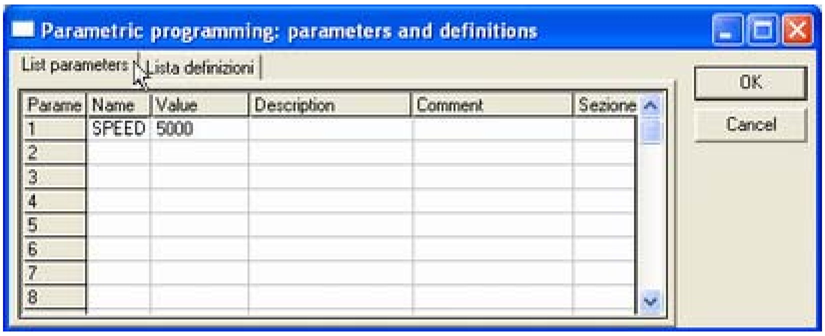
6.1.2 Parametric Instructions

PAR (Parameter Setting)

Allows you to assign a numeric value to a word (parameter). The parameter can substitute the value of any coordinate, quota, speed etc. It is possible to enter up to 256 parameters in each program, but they must all be after the Header instruction.



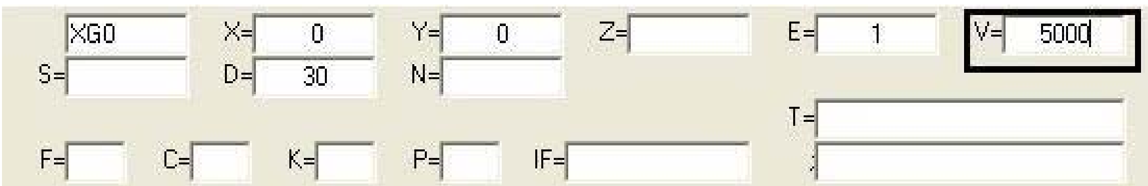
Example:



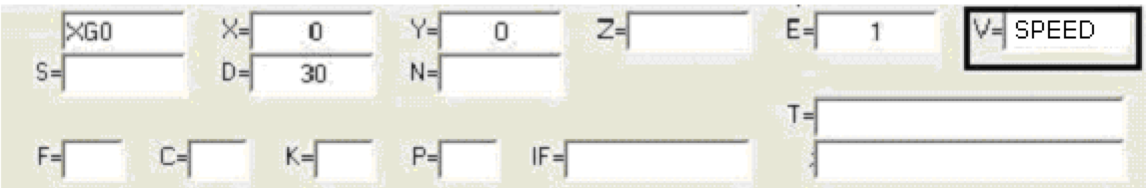
In this example, a parameter has been set with the name SPEED which is assigned a value of 5000 mm/min.
The fourth box of the Text Editor ("Description") and the "Description" and "Comment" fields of the Parameter Editor can be used to document the parameter.

The parameter can now be used in place of the number in an instruction, for example, to indicate the approach speed in instruction XG0:

In place of



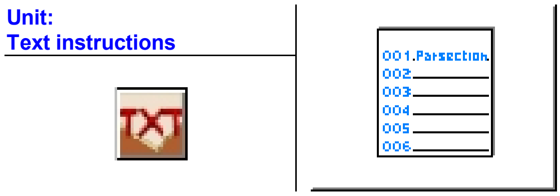
we can write



N.B. A parameter cannot be given the same name reserved for standard instruction fields: for example, a parameter cannot be called “X”, “Y”, or “Z”, however “W” or “QUOTA” are accepted.

PARSECTION (Section Setting for Parameters)

Groups the PAR parameters which follow it into a section.



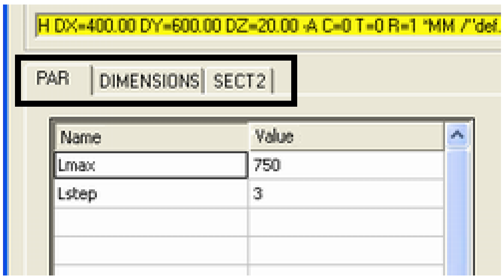
To create a section, simply assign it a name.
In the Text Editor, the section created includes the parameters entered subsequently, up to the following PARSECTION instruction.
In the Graphic Editor, the section is directly assigned to the individual parameters in the “Section“ box of the parameter Editor.

Esempio:

H DX=1200 DY=600 ...

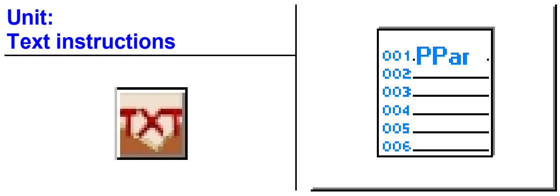
PAR LMax.=750 PAR LStep=3	Parameters which do not belong to a section.
PARSECTION / “DIMENSIONS” PAR Width=100 PAR Length=300	Parameters in the “DIMENSIONS“ section.
PARSECTION / “SECT2” PAR Par1=1 PAR Par2=2	Parameters in the “SECT2“ section.

In the Parameters Wizard, the sections appear as separate pages; the parameters which do not belong to any section are grouped under the PAR section.



PPAR (Sub-Program or Macro Parameter Setting)

Groups the parameters to be passed on to subroutines and macros.



The PPAR instruction is useful in the case where the parameters to be passed to subroutines and macros are too numerous to be written in the call instruction line of the subroutine/macro.

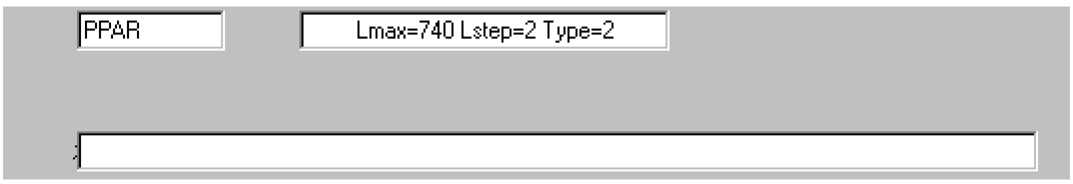
Example:

Subroutine call

...
S /"MySubroutine" X=1200 Y=350 **Lmax=740 Lstep=2 Type=2 Cycle=30**
...

can be written also as:

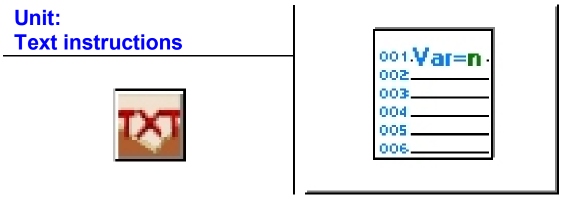
...
PPAR Lmax=740 Lstep=2 Type=2
PPAR Cycle=30
S /"MySubroutine" X=1200 Y=350
......



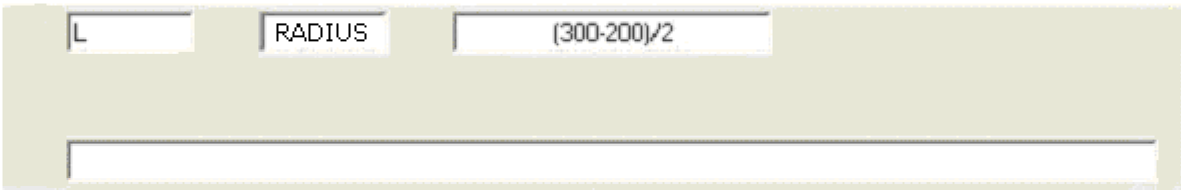
The PPAR instructions must be inserted **before** the call instruction to which they refer; if a parameter entered into an PPAR instruction is repeated in the following call instruction, the value entered in the call instruction prevails.

L (Assign a Value to Variable)

Makes it possible to attribute a numeric value to a word (variable) by using also an expression. The variable can substitute the value any coordinate, quota, speed, etc. It is possible to enter up to 512 variables in each program, at any point in the program. The variables can be redefined more than once inside the program.



Example:



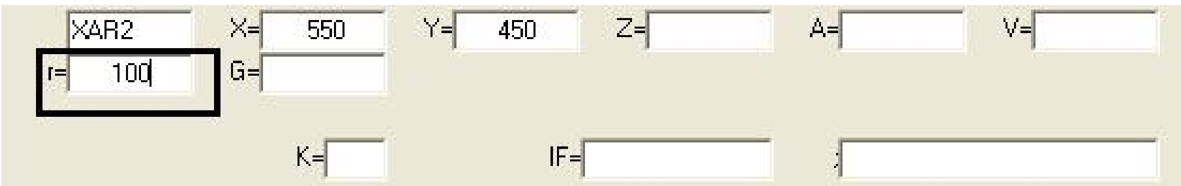
In this example, a variable has been set with the name RADIUS which will be assigned a value of 100 mm, as the result of the expression (300-200)/2. Not only can expressions use numbers but also other previously defined variables as well.

For example:

DX/2
(VARIABLE1 – DY)*2

In an instruction, the variable cannot be used in place of the number, for example, to indicate the arc of the radius in an XAR2 instruction:

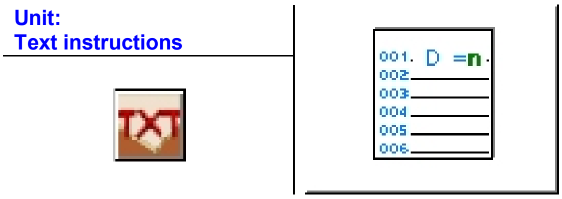
In place of



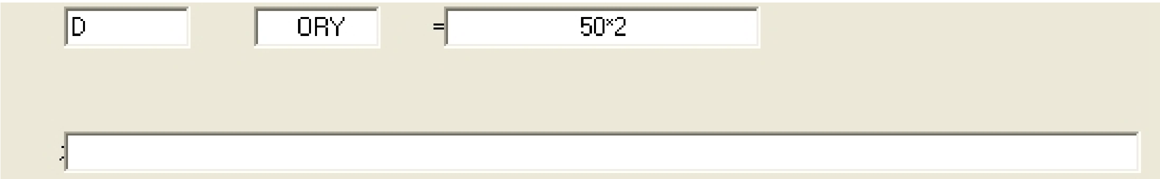
we can write

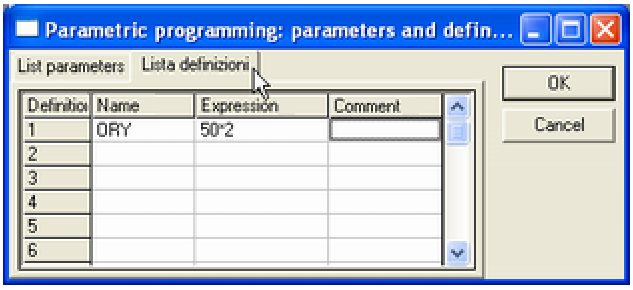
D (Alias Declaration)

The alias has the same function as a variable L: in other words it allows a numeric value to be attributed to a word using also an expression. Contrary to a variable L, an alias can however only be defined once inside a program. Furthermore, the alias must be defined immediately after the PAR parameters, and up to 64 can be used in each program.



Example:



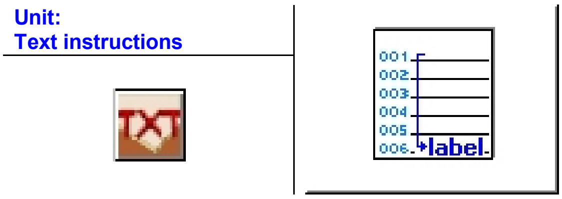


In this example, the alias ORY has been set which will be assigned a value of 50*2. The parameter can now be used in an instruction in place of the number, in the same manner as for variable L.

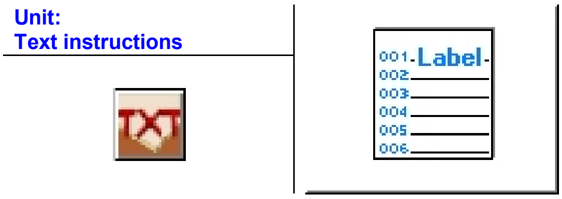
6.2 Structured Programming

6.2.1 Unconditional Jump

GOTO (Unconditional Jump)



. (Label Definition)



The unconditional jump makes it possible to jump some instructions, passing directly from one part to another of the program. The GOTO instruction tells the program the point in which to jump, defined by a label. More than one jump can be entered into a program.

Example:

Esempio:

H ………
…………

GOTO HERE ; jump to HERE label
…………
…………

. HERE ; HERE label
…………

The two instructions must always be matched.

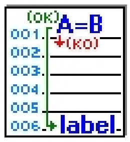
The GOTO instruction tells the program to jump to the HERE label (inserted in the second box). The instruction must be entered on the previous line, at the beginning of the jump: position yourself on the line and press the **[INS]** key.

The instruction for defining the label defines the HERE label (to be inserted in the line in which program execution must resume).

6.2.2 Conditional Jump

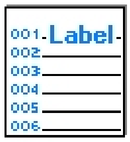
IF (Conditional Jump)

Unit:
Text instructions



. (Label Definition)

Unit:
Text instructions



With the conditional jump it is possible to jump a few instructions and pass directly from one part to the other of the program, if a condition is true. The condition control instructions are capable of telling between composed instructions, even at the same time, an constant values, parameters, variables and alias. More than one jump can be entered into a program.

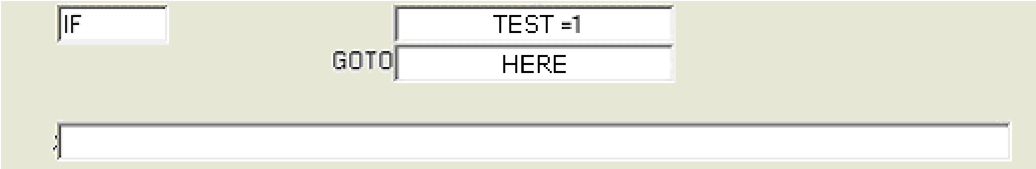
Example:

H ....
PAR TEST=1 ; setting parameter TEST=1
....
....

IF TEST=1 **GOTO HERE** ; jump to HERE label, if TEST=1
....
....

. HERE ; HERE label
....

The two instructions must always be matched.



The IF GOTO instructions tells the program to jump to the HERE label (inserted in the GOTO box) if the TEST parameter (inserted in the second box; the TEST parameter has been declared after the program header) has a value of 1. The instruction must be inserted on the previous line at the beginning of the jump: position yourself on the line and press the **[INS]** key.
The label is defined in the same manner as the unconditional jump. If the TEST parameter has a value other than 1, program execution will not jump to the HERE label.

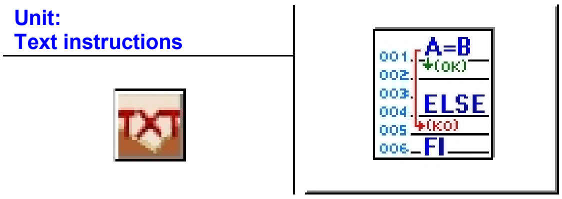
6.2.3 Producing and Executing a Block

IF THEN (Conditional Execution)

FI (End Conditional Instruction)

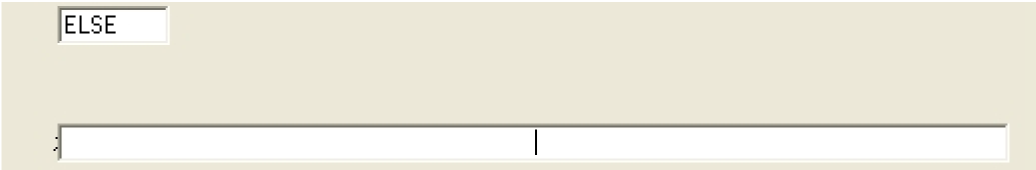
ELSE (Alternative of the Conditional Execution)

With the IF THEN instruction it is possible to verify a condition. Use the ELSE command to divide the block into two parts and make them alternatives. In a program, more than one condition can be entered and it is possible to insert one into the other for up to 16 blocks.



To insert the IF THEN instruction type IF THEN in the text box and press the **[ENTER]** key. The following screen will appear, where the word THEN appears in front of the second box in which the condition for execution is to be entered (for example, that parameter DX is less than 2000).

On the other hand, the FI and ELSE instructions do not require that parameters be inserted.



1) Example with true condition (without ELSE instruction).

H DX=1900 …. ; setting the parameter DX=1900 in the Header
………..
………..

IF DX<2000 **THEN** ; start block. If the DX parameter is less then 2000 (like for example the Header), a block is placed also on instructions that depart from this point and end up with instruction FI.

…………
…………

FI ; end of conditional block
…………

2) Example with false condition (without ELSE instruction).

H DX=2100 …. ; setting the parameter DX=2100 in the Header
…………
…………

IF DX<2000 **THEN** ; start block. If the DX parameter is greater or equal to 2000 (like for example the Header), the instruction block that starts from this point and end up with instruction FI **is not** carried out. The program flow is interrupted and starts again from the instruction that follows the end of block FI instruction.

…………
…………

FI ; end of conditional block
…………

3) Example of true condition and with ELSE instruction

H DX=1900 ….; setting the parameter DX=1900 in the Header
…………
…………

IF DX<2000 **THEN** ; start block. If the DX parameter is less then 2000 (like for example the Header), only the first part of the block is carried out, from the IF THEN instruction to the ELSE instruction.

…………
…………

ELSE ; the second part of the block, that starts from the ELSE instruction and terminates at the FI instruction, is not carried out. The program flow is interrupted and starts again from the instruction that follows the end of block FI instruction.

…………
…………

FI ; end of conditional block
…………

4) Example with false condition and ELSE instruction

H DX=2100 ….; setting the parameter DX=2100 in the Header
…………
…………

IF DX<2000 **THEN** ; start block. If the DX parameter is greater or equal to 2000 (like for example the Header), the instruction block that starts from this point and end up with the ELSE instruction **is not** carried out. The program flow is interrupted and starts again from the instruction that follows the ELSE instruction.

…………
…………

ELSE ; the second part of the block, that starts from the ELSE instruction and terminates with the FI instruction, is carried out.

…………
…………

FI ; end of conditional block.
…………

5) Example with various conditions and ELSE instruction:
case with first true condition, second false condition.

H DX=1400 ….; setting the parameter DX=1400 in the Header
…………..
………….

IF DX<2000 **THEN** ; start block. If the DX parameter is less then 2000 (like for example the Header), the first block is carried out that starts from the first IF THEN and terminates at the second ELSE.

………….
………….

IF DX>1500 **THEN** ; start of second block. If the DX parameter is less than or equal to 1500 (like for example the Header), the program flow is interrupted and starts again from the instruction the follows the ELSE instruction.

……………
……………

ELSE ; the second part of the second block, that starts from the ELSE instruction and terminates at the FI instruction, is carried out.

……………
……………

FI ; end of second conditioned block
……………
……………

FI ; end of first conditioned block
……………

6) Example with various conditions and ELSE instruction: case with both the first and second true condition.

H DX=1800 …. ; setting the parameter DX=1800 in the Header
………….
………….

IF DX<2000 **THEN** ; start of first block. If the DX parameter is less then 2000 (like for example the Header), the first block is carried out that starts from the first IF THEN and terminates at the second IF THEN.

………….
………….

IF DX>1500 **THEN** ; start of second block. If the DX parameter is greater than 1500 (like for example the Header), the first part of the second block is also carried out, that starts from the second IF THEN and terminates at ELSE.

……………
……………

ELSE; the second part of the block, that starts from the ELSE instruction and terminates at the FI instruction, **is not** carried out. The program flow is interrupted and starts again from the instruction that follows the end of block FI instruction.
……………
……………

FI ; end of second conditioned block

……………
……………

FI ; end of first conditioned block
……………

7) Example with various conditions and ELSE instruction: case with first false condition.

H DX=2100 ….; setting the parameter DX=2100 in the Header
………….
………….

IF DX<2000 **THEN** ; start of first block. If the DX parameter is greater then 2000 (like for example the Header), the first block **is not** carried out. The program flow is interrupted and starts again from the instruction that follows the end of block FI instruction. (at the end of the example).

………….
………….

IF DX>1500 **THEN** ; start of second block. Since the second block is hidden in the first, it is jumped.

……………
……………

ELSE ; the second part of the first block is also jumped.

……………
……………

FI ; end of second conditioned block
……………
……………

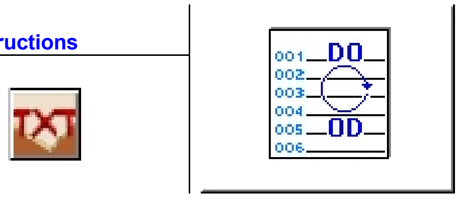
FI ; end of first conditioned block. Program execution starts again from here.
……………

6.2.4 Producing and Executing a Cycle

DO (Begin Cycle)

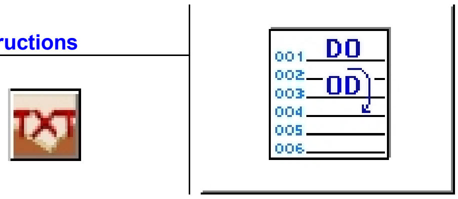
OD (End Cycle)

Unit:
Text instructions



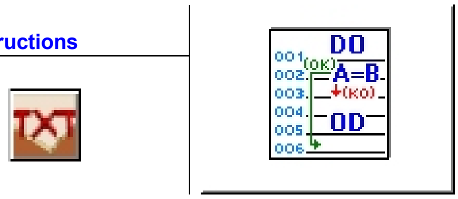
EXIT (Exit a Cycle)

Unit:
Text instructions



IF EXIT (Conditional Exit from a Cycle)

Unit:
Text instructions

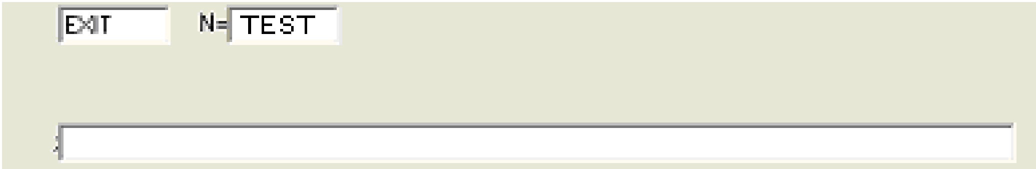


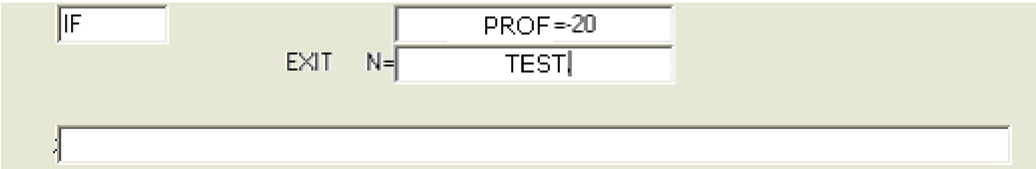
A cycle is a set of commands between the DO (start cycle) command and the OD (end cycle) command which the machining center continuously repeats until it encounters the EXIT (unconditioned cycle exit) or the IF EXIT (conditioned cycle exit) commands. If the EXIT or IF EXIT instruction is not entered, the cycle is repeated indefinitely. More than one cycle can be inserted into a program.

The cycle starts with a DO instruction. The cycle can be given a name in box N (it is not mandatory to assign a name).

After having entered all the instruction that compose the cycle, it is necessary to enter the O instruction that marks the end of cycle.

To exit a cycle, insert an EXIT or IF EXIT instruction **before the end of cycle OD instruction.**





The IF EXIT command (activated by writing IF EXIT in the text box) will exit the TEST cycle if the condition PROF=-20 is true.
If the name of the cycle is not specified, the EXIT and IF EXIT instructions will exit from the inner cycle.

1) Example of a cycle with unconditional exit.

H ...………
………..

DO N=TEST ; start TEST cycle
…………
………....

EXIT N=TEST ; exit TEST cycle

OD ; end of TEST cycle
……………

In this example the cycle called TEST is only repeated once.

2) Example of a cycle with conditional exit.

H ...…....
………..

L PROF=-1; variable for depth to be used conditional exit

………..
………..

DO N=TEST ; start TEST cycle

………...

XG0 X=… Y=… Z=PROF

```
        …………
        …………

        L PROF=-1+PROF; at each passage add the value of –1 to the PROF variable

        IF   PROF=-20   EXIT   N=TEST ; condition for exiting TEST cycle

OD     ; end of TEST cycle.
…………
```

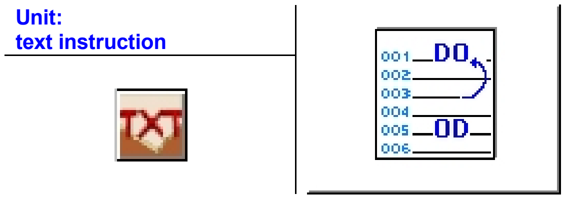
In this example, the cycle called TEST is repeated 19 times because the condition for exiting the cycle requires that the "PROF" (depth) variable arrive at -20mm and the decrement is 1mm for each cycle reading. The "PROF" variable set before DO has the value -1 and before the IF EXIT command the L PROF=-1+PROF is set, meaning that "each time this line is read, the value -1 is added to the "PROF" variable". Once out of the cycle, the program flow starts again from the line that follows the OD command.

3) Example of a cycle inside another cycle.

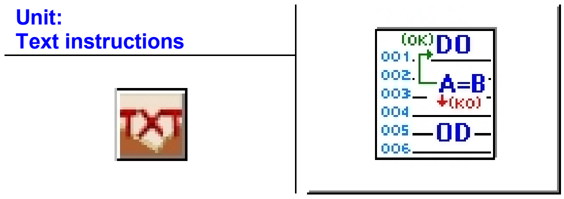
```
H ...………
…………..

DO     N=1; start cycle 1
        ……………
        ……………

        DO N=2 ; start cycle 2
            ……………
            ……………

            EXIT N=2    ; exit cycle 2

        OD ; end of cycle 2
        ……………
        ……………

        EXIT N=1 ; exit cycle 1

OD     ; end of cycle 1
……………
```

6.2.5 Repeating a Cycle

REPEAT (Repeat a Cycle)



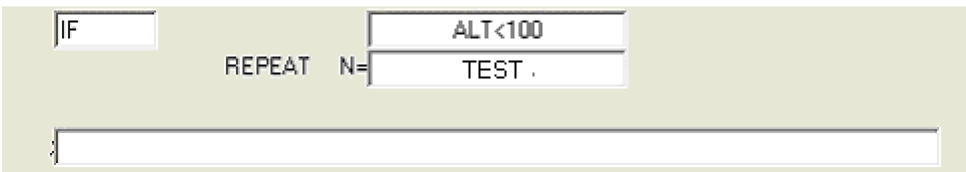
IF REPEAT (Conditional Repeat of a Cycle)



With the REPEAT instruction it is always possible to return to the point in which the start cycle DO instruction is inserted. With the IF REPEAT instruction it is possible to return to the point in which the start cycle DO instruction is inserted, if the set condition is true. Both instructions allow the cycle to be completely or partially repeated.

Examples:

The REPEAT instruction sets the repeat of the TEST cycle. (the name of which is entered in box N).



The IF REPEAT instruction (activated by writing IF REPEAT in the text box), sets the repeat for the TEST cycle, if the ALT condition (inserted in the second box) is less than 100.

Example:

H………

L CX=0

L CY=0

…………
…………

DO N=TEST1; start TEST1 cycle

IF CY=3*50 EXIT N= TEST1; exit TEST1 cycle if variable CY is equal to 3*50

DO N= TEST2 ; start TEST2 cycle

…………

…………

L CY=CY+50 ; at each passage the variable CY increases by 50.

REPEAT N=TEST1; repeat TEST1 cycle

…………………

OD ; end TEST2 cycle

OD ; end TEST1 cycle

In this example, when the REPEAT N=TEST1 command is read the flow restarts from the instruction DO N=TEST1 (start of cycle called TEST1) and is repeated until the conditions established in line IF CY=3*50 EXIT N=TEST1 have been met, that is, “exit from TEST1 cycle when variable CY is equal to 150”. Increase variable CY by 50 at each pass.

6.3 Macros

A macro is a special type of *subprogram*, the name of which is a true and real language instruction. The same instructions available in Xilog Plus are macros: for example, the boring B instruction which, even though it produces complex functions, is viewed by the program as a simple command. The macros are collected into groups in the instructions toolbar; both the group macros and individual macros are represented graphically by bitmap images.

► To write a macro it is possible to use the macro editor which can be selected from the **OPTIONS/MACRO EDITOR** menu. The macro editor can be used as text editor, but some functions that are available in the text editor are not active (for example, displaying suction cup editor).

► After having selected macro editor, proceed with creating a new parametric type program. Once the program is created, when it is time to save, Xilog Plus will ask for the program to be associated with a bitmap image (BMP extension), that must measure 120 x 90 pixel. The bitmap, which can be produced with any type of drawing program, must graphically represent the functions performed by the macro. The macro will be saved with a PGM extension (for example, HINGE.PGM).

► To properly integrate the macro in the programming environment thereby making it available in the instructions toolbar, it is necessary to define a macro group to which it must belong. For example, the Xilog Plus instructions XB, XBO and XBR are macros that belong to the boring group, which is represented in the instructions toolbar with a bitmap image. To define a new macro group and activate a new macro as its element, the following operations must be performed:

1. create a new folder inside folder FXC (that is located inside folder SCM GROUP/XILOG PLUS), and the name must not contain extensions (for example, “DOOR”);
2. produce a bitmap image that measures 32 x 32 pixel, which graphically represent the group and has the same name as the folder (for example, “DOOR.BMP”) and leave it inside the folder FXC;
3. copy the PGM file that represents the macro previous created (like, for example, “HINGE.PGM”) into the relative folder of the group to which it belongs (for example, the folder DOOR created inside the folder FXC).
4. restart Xilog Plus editor.

Macro groups created in this manner can include up to 100 macros; the maximum number of groups is 40.

WARNING

The contents of the folder …\FXC\PRIVATE must not be changed!

The programming of a macro follows the same rules that govern a normal parametric program, including the possibility or recalling subprograms or other macros; however, it is necessary to observe some special directives described below:

Header

The fields **DX**, **DY** and **DZ** must be set to zero, while field /, normally indicating the tooling, must have the following format:

"<shortcut>,<parameters>,<description index>,<description>"

- <shortcut> is a pair of characters used for quickly selecting the Fixed Cycle from the general instruction list. The first character must be a number between 5 and 9, and the second character must be a letter of the alphabet.
- <parameters> is the list of parameters for the macro.
- <description index> must be set to zero.
- <description> is the string that describes the macro.

Examples:

H DX0 DY0 DZ0 *MM /"9p,XYZDT,0,DOOR TYPE 08"

Macro parameters

Parameters are data that the user must pass to the macro in order for it to perform the function for which it was written.
The parameters are to be selected among those contained in the following list. Some have a set meaning, which must be maintained, others are free, but may have a recommended meaning.

X	Free (coordinate X)
Y	Free (coordinate Y)
Z	Free (coordinate Z)
A	Angle of rotation
H	Free (coordinate H)
E	Free (hood position)
I	Free (coordinate I)
J	Free (coordinate J)
V	Approach speed
S	Spindle rotation speed
T	Tools list
F	Machining face
C	Tool correction
K	Incremental
P	Panel reference
Q	Free (quadrant)
R	Free (repetitions)
x	Free (final position X, step in X, ...)
y	Free (final position Y, step in Y, ...)
a	Free
B	Free (angle, ...)
r	Free (radius, ...)
D	Free (diameter, out machining range, ...)
s	Free (step, ...)

l Free (length, ...)
G Free (direction of circular milling, direction of multiple tool, ...)
L Free (length, ...)
N Subprogram name

The parameters may be indicated in any order. If the value for a parameter is missing in macro call assigns the corresponding parameter the value foreseen by the macro.

Using the parameters

From within the Fixed Cycle, it is possible to read the value of the parameters using the reserved variables pX, pY, pZ, pA, pH, pE, pI, pJ, pV, pS, pT, pF, pC pK, pP, pQ, pR, px, py, pa, pB, pr, pD, ps, pl, pG, pL, pN, called **p variables**.
All p variables (except pT) may also be used in expressions, as long as their status has been defined, otherwise the result is not evaluated. To know the status of a p variable, you can use the **NDEF** operator, which returns *True* (1) if the variable has not been defined, and *False* (0) otherwise.

Examples:

```
H DX0 DY0 DZ0 *MM /"9p,XYZDT,0,DOOR TYPE 08"
L FOOT=100 ;the foot to use if the value of parameter X has not been specified
IF NDEF pX GOTO TESTpY
L FOOT=pX ;foot passed as value of parameter X
.TESTpY
    ...
```

Special Variables

In the macro there are some special read-only variables that can be used also in the expressions:

X Current Coordinate X
Y Current Coordinate Y
Z Current Coordinate Z
C Current radius correctio
Cp Programmed radius correction
F Current face

Examples:

```
L oldF = F
L curC = C
L QY = Y+32
B X=X Y=QY Z=Z
```

Special instructions

Some special instructions are available:

PRINT "message" displays the message
C=<variable> assigns the radius correction type
F=<variable> assigns the face

Examples:

```
F=oldF
C=curC

IF NOT NDEF pX GOTO Beginning
PRINT "value of parameter X missing"
GOTO End
.Beginning
...
.End
```

Tool management and ^ operator

The special variable T contains the current tools. Using the ^ operator, is possible to:

1. cancel the tools in T
2. add a tool to T
3. copy pT and T in normal variables that acquire the significance of tool variables;
4. copy pT and the tool variables to T
5. read the number of tools in pT or in the tool variables
6. read one at the time all the tools in pT or in the tool variables

Canceling current tools

To cancel the current tools, add the value zero to T.

Examples:

```
L T = T+0
```

Adding a current tool

To add a current tool, add to T the number of the tool.

Examples:

```
L T = T+6
L T = T+101
```

Copying tools

To copy tools, follow the ^ operator with -2.

Examples:

```
L CopyOfT = T^-2          ;copies the current tools

L CopyOfpT = pT^-2        ;copies pT

L T = T + 0
L T = CopyOfT^-2          ;assigns a tool variable to T

L T = T + 0
L T = CopyOfpT^-2

L T = T + 0
L T = pT^-2               ;assigns pT to T
```

Reading the number of the tools

To read the number of the tools, follow the ^ operator with -1.

Examples:

```
L NumTool = pT^-1
L NumT = CopyOfT^-1
```

Reading individual tools

To read the individual tools, follow the ^ operator with the position of the required tool (beginning with 0).

Examples:

```
L FirstTool = pT^0
L SecTool = CopyOfT^1
L ToolPos = 3
L NextTool = pT^ToolPos
```

6.4 Passing Parameters to Subprograms and Macros

There are three ways of passing a set of parameters from a main program to a subprogram or to a macro.

1) Programming a sequence of PAR instructions (up to a maximum of 256) inside a program.

If, when calling the subprogram (in main program), the name of the parameters defined inside the subprogram (they can be composed of numbers or letters, to a maximum of 15 characters) is not known, the values set using the syntax:

PAR1=*value1* PAR2=*value2* ,.., PAR256=*value256*

are passed to the parameters present inside the subprogram. The value defined for PAR1 is associated to the first parameter of the subprogram, the one defined for PAR2 is associated to the second, and so forth.
Otherwise, in case the names of the parameters of the subprogram are know, the parameters can be programmed directly with their names. For example, if in the subprogram DOOR.PGM the parameter has been programmed:

PAR length = 3

The call:

S /DOOR length=10

Will execute the subprogram DOOR.PGM with the length parameter set to 10.
The same is true for all nestling levels of the subprograms (or macro), in other words, it is true for a subprogram call made inside another subprogram, etc.
See also: PPAR instruction.

2) Programming global variables.

All variables defined in the main program before a subprogram (or a macro) call, can be recalled inside the subprogram itself (or the macro).In the subprogram (or fixed cycle), these variables can be handled as local variables (those set internally), with the difference that they are automatically initialized with the value they had in the main program. In the subprogram, any modification of the value of the overall variables is only effective until the system exits from the subprogram.
In this case as well, the method is true for all nestling levels of the subprograms (or the macros): a local variable of a subprogram becomes overall for all its subprograms, and so on.
The overall variables at the highest levels are those set in the **environment variables file**. The name of this file, which is a program that only contains the definitions for the variables (see: instruction L), must be declared in the Header of a program.

3) Programming global variables with value returned to caller.

A main program has an *array* (vector) of 128 variable values that can be viewed and modified also from inside all the subprograms and macros called. This means that it is possible to transmit values from subprograms or macros to the main program. This is true for all subprograms and macros nestled at all levels.
Two functions are available to read and write these values: **HeapGet** and **HeapPut**.

L VAR = HeapGet(0 < *index* <= 128),
copies the value contained in the *index* position of the vector in VAR;

L VAR = HeapPut(0 < *index* <= 128, *value*),
writes the *value* value in the *index* position of the vector and copies it in VAR.

The following regulations apply:

- *index* and *value* can be expressions;
- the *array* of variables is zeroed after each program interpretation (graphic or execution);
- HeapGet can also be used inside expressions.

Example:

In the main program:

```
…
L BEAM = 0
S "CALCULATEBEAM.PGM"
L BEAM = HeapGet(1)
…
```

In the CALCULATEBEAM.pgm program or macro:

```
…
L R = HeapPut(1,5)
…
```

In the main program, after the HeapGet function, BEAM is worth 5.

WARNING

When using an overall variable, always make sure that it has been set at a higher level. If this is not so, the value of this variable is 0, if used in expressions, "undefined" if used as quota.

As an alternative to programming a subprogram which is passed values in the three methods described above, it is possible to program a macro using the passage for standard parameters for the macro.

4) Parameter passage to macro by name.

As well as serving as the name of a sub-program or generic message, parameter N of a macro may also be used to pass parameters to the macro by name. In this case, the value of parameter N must be a list of names of variables separated by a comma or by one or more spaces. The names must belong to variables used in the sub-program/macro calling.

Example:

```
...
xMyMacro X=.. Y=.. N="Alfa Beta Gamma Delta"
xMyMacro X=.. Y=.. N="Alfa,Beta,Gamma,Delta"
...
```

Within the macro the values of the variables may be read and/or edited using the ^ operator applied to pN.

Reading the number of variables passed

To read the number of variables passed place -1 after the ^ operator.

L NumVar = pN^-1

In the previous example the number 4 is obtained.

Reading the value of a single variable

To read the value of a single variable, make sure that the ^ operator is followed by the position, starting at 0, of the variable required. Reading the value of a variable not present in the list produces the value 0.

L FirstVarValue = pN^0	Reads the value of "Alfa" in the previous example.
L ThirdVarValue = pN^2	Reads the value of "Gamma" in the previous example.
L FifthVarValue = pN^4	Reads 0 in the previous example, since the fifth variable was not passed.

Editing the value of a single variable

You can edit the value of a single variable in the sub-program/macro calling by applying the RETV operator to the variable required.

L Res = RETV(pN^1,123)	The "Beta" variable takes the value 123.
L Res = RETV(pN^3,456)	The "Delta" variable takes the value 456.

If you use the comma as a separator between the names in parameter N, you can also omit the passage of one or more variables.

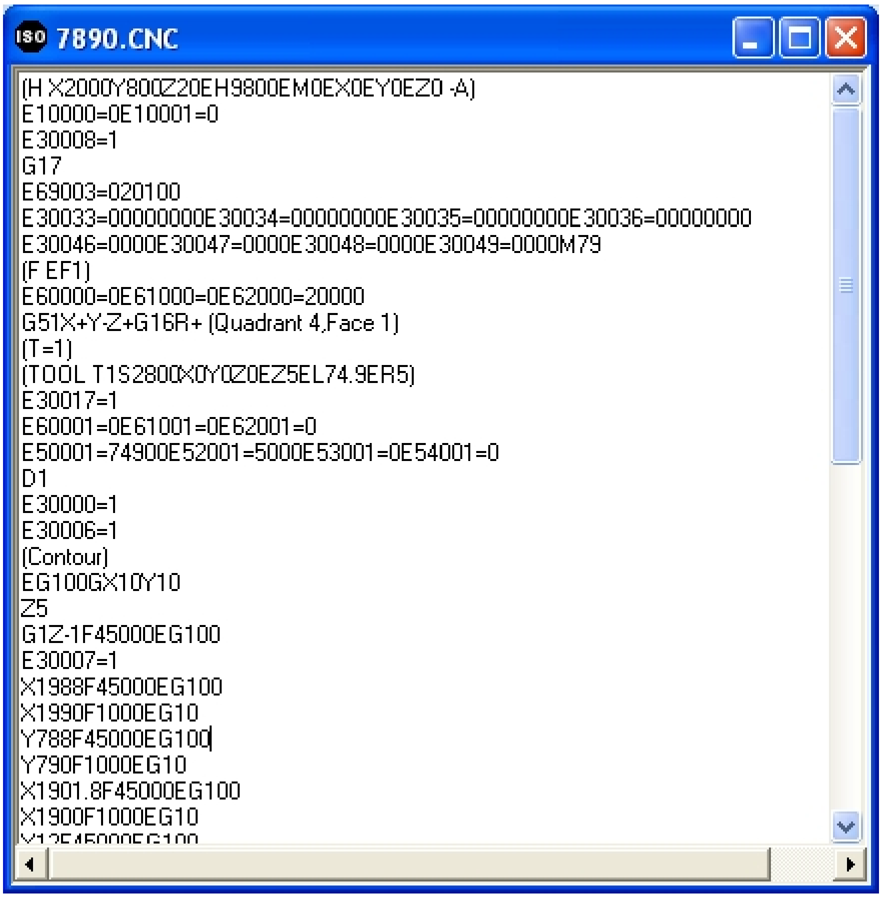
xMyMacro X=.. Y=.. N="Alfa,Beta,Gamma,"	Variable 4 missing.
xMyMacro X=.. Y=.. N=",Beta,,"	Variables 1, 3 and 4 missing.
xMyMacro X=.. Y=.. N="Alfa,,Gamma,Delta"	Variable 2 missing.

To check if a variable has been omitted, in the sub-program/macro called you can use the STRCMP operator, applied to the variable and to the null string.

```
...
IF STRCMP(pN^3,"") = 0 THEN
PRINT "The fourth variable was not passed"
FI
...
```

6.5 ISO Program

Xilog Plus provides a text editor for creating programs is ISO format (with CNC extension). The editor for ISO programs presents a simple editing window. Editing of CNC files is not assisted, and is done by writing and changing the instructions in a simple text editor.



6.6 Importing a DXF File

6.6.1 Importing Procedure

With the Xilog Plus editor it is possible to convert a DXF file, produced with a CAD program, into a program with PGM extension, that can be executed on the bore-milling machine. The method for importing geometries from the DXF file and the standard routing characteristics are defined in the configuration file cad.cfg. Below are the descriptions of the various parameters defined in this file and the corresponding options:

Code xISO/xISOE (0,1).
0 = "basic" Xilog Plus instructions are generated (ex. G0, G1, etc.).
1 = "full" instructions are generated (XG0, XL2P, etc.).

Number standard tool.
Xilog Plus will assign by default this tool number to all of the imported machining operations.

Empty inputs/outputs (0=NO).
0 = All of the input instructions on the in / out of the profile (XGIN, XGOUT) are generated with the values defined in the successive parameters.
1 = The above instructions are generated with all of the parameters at zero (disabled).

Outputs = inputs (0=NO).
0 = The above instructions (xGIN, xGOUT) are generated with the parameters defined below, which may be different.
1 = The out instructions from the profile (xGOUT) are generated with the same parameters defined for the in ones in the same (xGIN).

Values for the parameters C, G, Q, R of the xGIN and xGOUT instructions.
See: Cap. 5 – Programming Instructions.

Max. distance joints (for Auto-Join).
If two imported graphic elements (ex. lines, arcs, etc.) end at a distance less than this parameter, Xilog Plus considers them to be joined in the geometry grouping phase ("Autojoin") and therefore assigns them the same machining operation. Instead, if the distance is greater than the set value, they are considered part of distinct profiles for which the corresponding XGIN and XGOUT instructions will be generated.
The meaning of this parameter is that of compensating any inaccuracies in the generation of the profiles, within the CAD: if the set value is too small, importation may be incorrect, generating too many unconnected profiles which must then be joined "by hand". To the contrary, if the set value is too large, profiles which must remain distinct may be incorrectly "glued".
Normally the standard value [0.01mm] is adequate.

Panel values DX, DY, DZ .
Pre-set dimensions of the panel to be machined, which will be given in the header of the Routolink program. These values are the default values used when the DXF file does not contain this information. If the DXF file contains the "panel" layer (see below), the dimensions defined therein replace the values of this parameter.

Standard depth of routing (Z).
Standard value for field Z generated for all of the rounting instructions. This value is the default which may be modified in the import phase. This parameter is ignored if the value of "Z routing" is not 0..

Routing speed (m/min) e **Motor rotation speed (rpm).**
These are the default values assigned to the routing operations in the import phase (they can be modified, if necessary)

Max diameter of borings.
In the import from DXF phase, the primitive graphs of CIRCLE types are compared with this value: if the diameter of the circle is less than or equal to the parameter, it is managed as HOLE (an XBO optimised boring instruction is generated); otherwise, it is handled as routing.

Standard depth of boring (Z).
As for routing, this parameter defines the default depth of the holes. This parameter is ignored when the value of "Z borings" is not 0 or when the "Vertical holes layer" is programmed.

Boring speed (m/min).
Tool advance speed during boring (default).

Type of optimization (0=A., 1=X, 2=Y).
0 = ABSOLUTE. The path of the drilling head is minimised by following the diagonal
1= AXIS X. The path of the boring head is minimised first in function of axis X then according to axis Y.
2 = AXIS Y. As above, but the path is first optimised in function of axis Y.

Prec. of optimization (%).
Control parameter for the optimisation algorithm: the larger this value is, the longer the calculation time and the better the optimisation. Normally a value of 80% gives good results.

Type hole tip (0=P, 1=L, 2=S).
This indicates the type of bit to be used in default in generating the boring instructions: 0 = flat; 1 = lance; 2 = countersink.

Height countersink.
To be used only if the hole tip type is set on 2.

No. arches per quadrant in ellipse interpolation.
Used to generate ellipses, the larger the value, the better the interpolation of "ellipse" type geometries (ellipses). Normally, values from 3 to 6 are considered acceptable, but in order to achieve greater interpolation precision, it may be necessary to increase this value.

Directory.
Not used.

Panel dimensions layer.
"Root" of the layer name which defines the panel dimensions.

Copy machining layer.
Layer name which defines all of the routing related geometries.

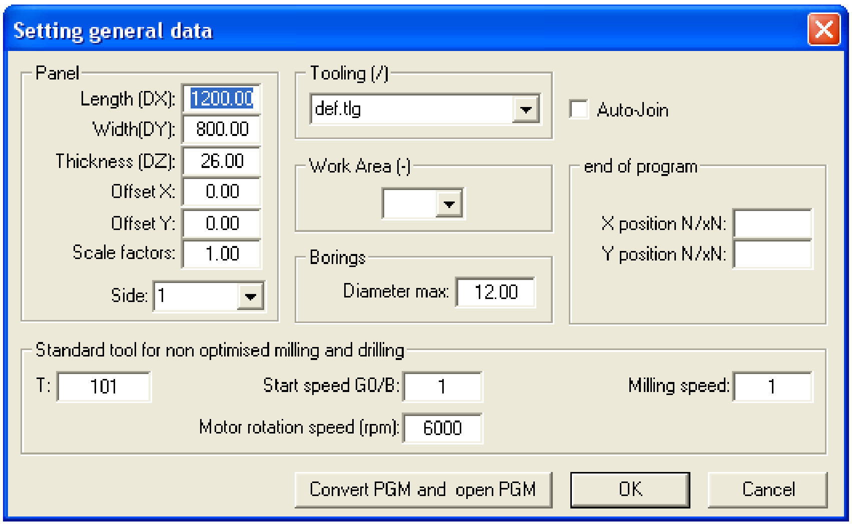
Vertical holes layer.
"Root" of the layer names which define "vertical" boring operations (face 1).

Horizontal holes layer.
“Radice” dei nomi di layer che definiscono operazioni di foratura “orizzontali”.

Z borings (0=CFG, 1=THK, 2=ELEV).
Criterion for defining the boring depth. This parameter is ignored when the "Vertical holes layer" is defined.
0 = Boring depth is deduced from the configuration parameter “Standard depth of boring (Z)”.
1 = Boring depth is deduced from the THICKNESS property of the geometry, as defined in the DXF file.
2 = Boring depth is deduced from the ELEV property of the geometry, as defined in the DXF file.

Z routing (0=CFG, 1=THK, 2=ELEV).
Criterion for defining the machining depth for milling operations. If this parameter is set to 1 or 2, the Z position of the machining operations and profile entry points can no longer be modified (these settings may be modified only after editing the imported PGM file).
0 = Boring depth is deduced from the configuration parameter “Standard depth of routing (Z)”.
1 = Boring depth is deduced from the THICKNESS property of the geometry, as defined in the DXF file..
2 = Boring depth is deduced from the ELEV property of the geometry, as defined in the DXF file.

► The DXF file can be opened like any other file, with the **FILE/OPEN** command. After the file is opened a window appears for setting general data, in which to insert or change data that is needed for the Header of the PGM program.



Length, width, thickness. Panel measurements (fields DX / DY / DZ).
Offset X/Y. Possible shift of panel origin (fields BX / BY).

Scale factors. Factor for which all coordinates X and Y present in the DXF file are multiplied.
Side. Panel side to which the imported geometries refer.
Diameter max. All circles with a diameter that is less than or equal to the one set here will be converted to boring (with the same diameter as that of the circle: XBO), while those with a larger diameter will be converting into milling (XG0, XA2P).
Tooling. Name of the tooling file that must be used.
Work area. Program work area.
Auto-Join. Makes it possible to join the disjointed segments before conversion in PGM.
End program. Makes it possible to set the X and Y positions for the N/XN end program instruction.
The **CONVERT IN PGM AND OPEN PGM** button automatically converts in PGM format, using both standard data and data programmed and displayed in the General Data Settings window and then opens the obtained PGM program.
In any moment of DXF file management, it is possible to call up this window and use the above command.

► After having set the general data, the graphic representation of file DXF will appear and will activate all specific conversion functions, that are available in the **EDIT** menu and in the relative buttons of the functions toolbar.
The geometries can be selected with the mouse. Like in the graphic editor of the programs, the elements selected are highlighted with the color: green, for profiles (after the auto-join operation: see further on), red, for individual sections of a profile; a small light blue square marks the final point of the section. To select profiles and sections it is possible to also use the arrow keys on the keyboard: the horizontal arrows to pass from one profile to another; the vertical arrows, to move between the sections of a profile.

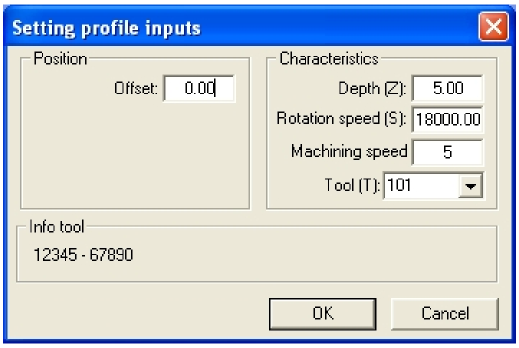
►The first operation to perform is to cancel all sections that do not correspond to panel machining (for example, dimensioning and various trimmings) and any sections that overlap and/or cross over, that could lead to an erroneous interpretation of the geometries. To delete a geometry it is sufficient to select it and click on the **EDIT/DELETE** menu.

► At this point it is necessary to group the geometries of the DXF file into tool routes: in other words, profiles are created that are characterized by a continuous route between all contiguous sections. This operation takes place automatically for all geometries present in the drawing by clicking on the **EDIT/AUTO JOIN** menu.

►Furthermore, for all geometries it is possible to:

- change the approach direction (clockwise - counter-clockwise) with the **EDIT/INVERTING PROFILE ROUTE** menu.
- change the entry point on a profile and the characteristics of a hole with the **EDIT** menu, under **INPUT/HOLE DATA.**

Setting profile inputs

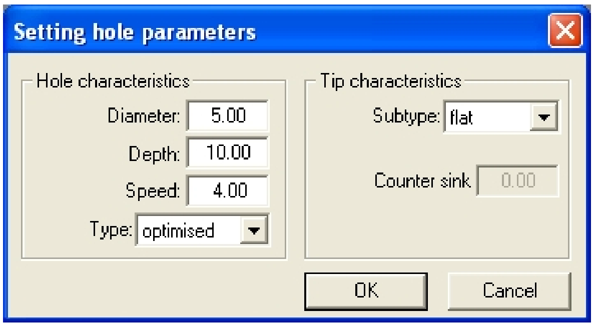


Offset. Defines the entry point in the profile, compared to the starting point of the selected geometry.
If the value is *positive* (but not greater than the length of the geometry itself), the geometry is divided into two distinct geometries with a distance from the initial point equal to the specified offset. The result will differ based on the type of original tool route:

- For an "open" route (in other words, with different entry and exit points) it will be "broken down " into two distinct routes that will respectively terminate and start at the "separation" point. These new routes are fully independent and can be managed separately one from the other.
- For a "closed" tool route (in other words, the initial and final points coincide) its geometries will be reorganized so that the entry/exit point of the tool corresponds to the "separation" point programmed; the result will however be only one tool route.

Depth. Machining quota.
Rotation speed and **Machining speed**. Rotation and machining speed for the selected tool.
Tool. Tool selected for machining.

Setting hole parameters



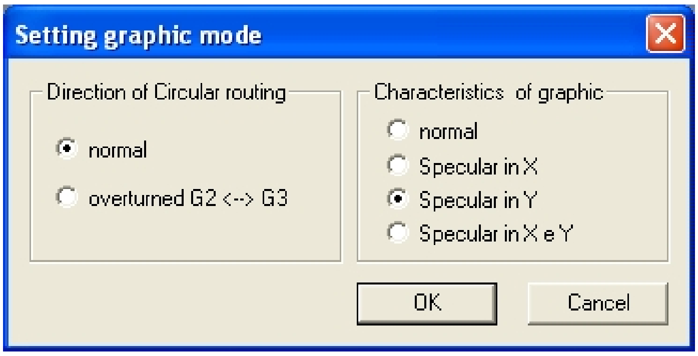
► It is also possible to change the order of sequence for machining. By pressing **EDIT/GEOMETRY ORDER** menu, a number will appear near the starting point of each machining to indicate the sequence currently set. To change the machining sequence it is necessary to:

1. Select the machining to be done first
2. Press the **[ENTER]** key to confirm the choice: the number next to the selected profile now becomes No. 1
3. Repeat points 1 and 2 for all subsequent machining up to the last one.

NOTE. All machining present must necessarily be reconfirmed, even if they are already in the desired order.

To exit the geometries arranging function click on the **EDIT/EXIT ORDER** menu.

►With the **EDIT/REFERENCE SYSTEM** menu it is possible to change the direction of the circular milling and to program the specularity of axis X and Y.



► At this point the file can be saved in PGM format (or XXL). The same DXF file can be saved various times in different files. The PGM files that result from the conversion can be further changed with the text or graphic Xilog Plus editor.

6.6.2 Layers

6.6.2.1 Panel Dimensions Layer

Indicates the overall dimensions of the rough panel from which the finished panel will be cut (parameters DX, DY, DZ).

- In the DXF, this layer must contain 4 lines (LINE) joined to form the rectangle of the panel (the LINE elements of this layer only define the panel dimensions in X and Y and are not interpreted as linear millings).
- The layer name must be followed by a number which represents the size of the panel in Z.

If the parameter “Panel dimensions layer” is not programmed, the panel sizes programmed in the parameters DX, DY, DZ (window **Setting general data**) are valid.

Example:

cad.cfg configuration	DXF file	DZ value
“Panel dimensions layer” = PANN		
	PANN25	25 mm
	PANN25_5	25.5 mm

6.6.2.2 Copy Machining Layer

Indicates the profile to be cut from the rough panel. In this case you do not need to specify a numeric parameter as lines, arches and circles are converted into millings. The milling and boring depth is determined by the parameter “Z routing” of the configuration file cad.cfg.

WARNING. All geometric elements not belonging to other layers are considered elements of this layer.

6.6.2.3 Vertical Holes Layer

Defines the vertical boring as an alternative to the standard based on programming the parameter “Diameter max.” (in **Setting general data**), which refers to **all** the “CIRCLE” present in the DXF file. If programmed, the vertical boring layer establishes that **only** the circles in this layer are interpreted as vertical holes, their diameter is the diameter of the circle programmed in the DXF file.
In the DXF file the layer name must be followed by a number representing the dimension in Z of the holes (in relation to the panel surface) belonging to the layer. If you drill a different diameter you have more layers.

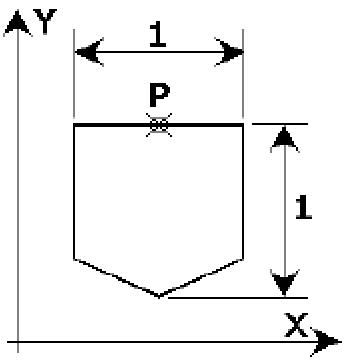
Example:

cad.cfg configuration	DXF file	Z value in XBO
“Vertical holes layer” = FV		
	FV-25	-25 mm
	FV25_5	25.5 mm

6.6.2.4 Horizontal Holes Layer

Horizontal holes are defined by the instructions BR or XBR.
To program these holes characterise DXF file (through CAD) as follows:

1. Input the drawing of a point with closed lines and **unitary sides (1 x 1)** in block named HORIZ.



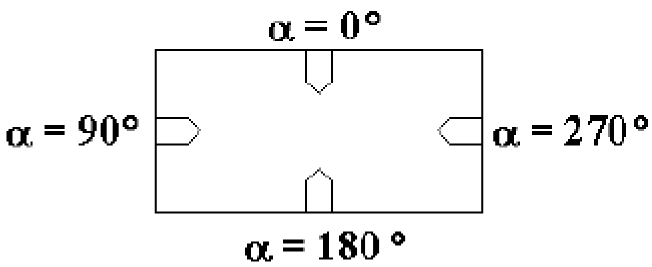
2. Create a layer named HO. In the DXF file the layer name must be by a number which corresponds to the distance from the centre of the hole to the work table (under side of panel - see parameter H of XBR instruction).

Example:

cad.cfg configuration	DXF file	H value in XBR
"Horizontal holes layer" = HO		
	HO25	25 mm
	HO25_5	25.5 mm

3. Insert the block HORIZ in the layer HO specifying:

- **X coordinate of P (block inserting point – see figure above).** Equivalent to X quote of XBR instruction.
- **Y coordinate of P (block inserting point – see figure above).** Equivalent to Y quote of XBR instruction.
- **Z coordinate of P (block inserting point – see figure above).** Enter 0.
- **Scale in X**. Enter the hole diameter.
- **Scale in Y**. Enter the hole depth.
- **Rotation angle** (see figure).



6.6.3 Importable DXF Sections

The following sections are recognized:

- **Header**
- **Tables**
- **Entities**
- **Blocks** (only for horizontal boring through layer)

The following entities are imported:

- **Line**
- **Arc**
- **Circle**
- **Polyline**
- **Ellipse**
- **Lwpolyline (Rectangle, Polygon)**

- **Insert** (only for horizontal boring through layer)

The **$EXTMIN** and **$EXTMAX** values in the **HEADER** section establish the working range on the basis of which the drawing of the processes on the panel is made. If the DXF file does not contain these values, there is no guarantee that the geometries will be displayed correctly. All the geometries produced by CAD **except for those listed above** are ignored by the DXF importer.

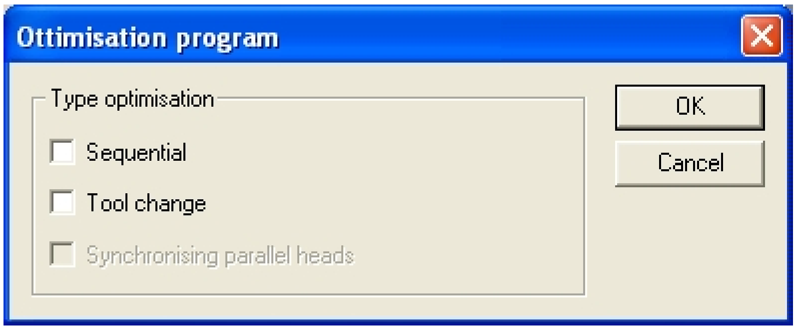
WARNING

The DXF file must always contain the Header section
(for further information consult the CAD Manuals)

7. Optimization Program

With the Xilog Plus optimization program it is possible to optimize machining programs in order to reduce tool changing operations to a minimum.

► Once the program to be optimised is opened, the optimisation can be activated by clicking on the **TOOLS/PROGRAM OPTIMIZER** menu. The file save window will appear to select an existing program or a new program which will be the program resulting from the optimisation of the current program. Click on the OK button to open the following window, used to select from two types of optimisation: sequential and tool change (see also: Appendix F – Ergon Machines).

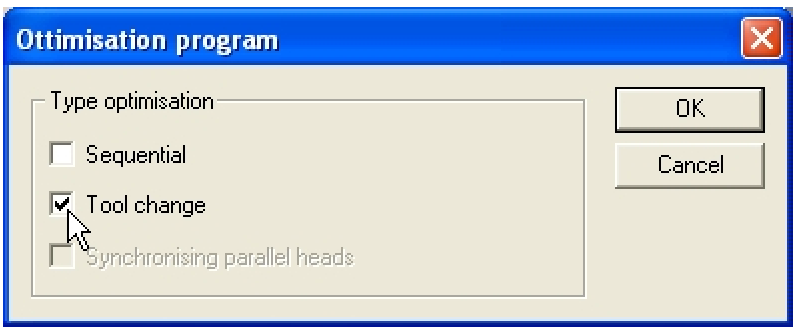


► The result is a new optimised program (previously established) which can be saved in the memory like a normal program. The original program is not modified.

Program optimization responds to two distinct requirements:

1) Machining that required a precise sequence in the use of the tools. For example, block wood, doors, fixtures. In this case:

- The program must be composed only of SO instructions (Optimized subprogram call), that serve to recall programs as subprograms;
- using the mouse, select the **TOOL CHANGE** box.



Optimization of tool change has two effects:

- it optimizes the tool change operation;
- it optimizes the routes (the machine will perform the machining foreseen with each tool carrying out in succession that nearest).

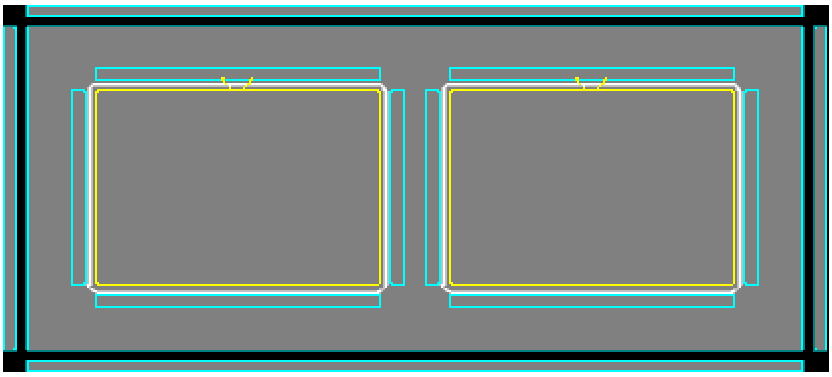
Example:

The TEST_1 program trims a panel in two phases: roughing using tool E1 and finishing using tool E2.

```
H DX=400 DY=300 DZ=20 -A C=0 T=0 R=1 *MM /"def"
; Roughing
C =2
GIN G=2
G0 X=DX/2 Y=0 N="PROF" T=101
G1 X=DX
G1 Y=DY
G1 X=0
G1 Y=0
G1 X=DX/2
GOUT
; Finishing
GREP N="PROF" T=102
```

Lets create the MAIN_1 program that calls up the TEST_1 program twice to trim two distinct panels.

```
H DX=1100 DY=500 DZ=20 -A C=0 T=0 R=1 *MM /"def"
SO /"test_1" BX=100 BY=100
SO /"test_1" BX=600 BY=100
```



Carried out without optimization, the MAIN_1 program machines the left panel first with tool E1 and then with tool E2; it then machines the right panel first with tool E1 and then with tool E2. A total of 4 tool change requests (3 if E1 is already in the spindle when it starts).

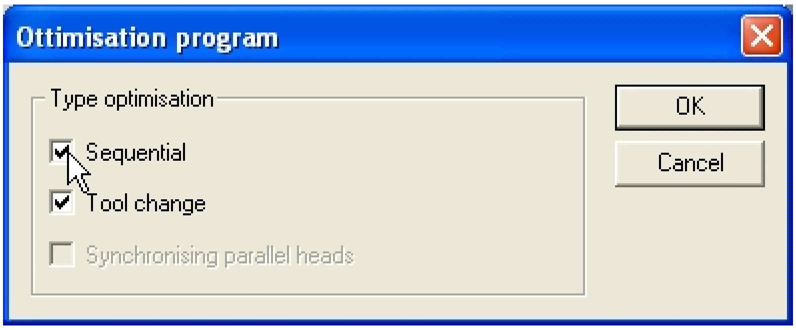
The program optimized with tool change however, performs the machining in the following sequence:

1. Roughing left panel with tool E1
2. Roughing right panel with tool E1
3. Finishing right panel with tool E2
4. Finishing left panel with tool E2

requiring a total of 2 tool changes (1 if E1 is already in the spindle when it starts); furthermore, thanks the optimization of the route, the first finishing work (point 3) is carried out on the closest panel, that is, the one on the right.

2) Machining that allows a change in the sequence in which the tools must be used. For example, repeated cyclic machining (decorations), nesting. In this case:

- the program must not be composed of SO instructions, but XS instructions, or macro, etc.
- it is necessary to select the **SEQUENTIAL** box: it is suggested that the **TOOL CHANGE** box be selected as well.

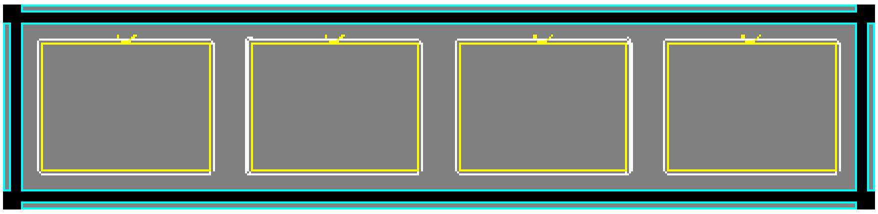


Sequential optimization reorders the sequence of machining, by optimizing the tool change operations. If it the **TOOL CHANGE** box is activated as well, the routes will be optimized too.

Example:

The TEST_2 program trims the four panels in two phases: roughing using tool E1 and finishing using tool E2.

```
H DX=2000 DY=400 DZ=20 -AB C=0 T=0 R=1 *MM /"def"
PAR Num =4
PAR OffsetY =0 " "
L MyDX =400
L MyDY =300
L Cnt  =0
DO
  O X=50+500*Cnt Y=50+OffsetY
  ; Roughing
  C =2
  GIN G=2#
  G0 X=MyDX/2 Y=0 N="PROF" T=101
  G1 X=MyDX
  G1 Y=MyDY
  G1 X=0
  G1 Y=0
  G1 X=MyDX/2
  GOUT
  ; Finishing
  GREP N="PROF" T=102
  ;
  L Cnt =Cnt+1
  IF Cnt=Num EXIT
OD
```



If carried out without optimization TEST_2 program machines the panels from left to right using for each panel first tool E1 and then E2; requiring a total of 8 tool changes (7 if E1 is already in the spindle at the start).
The program optimized only with sequential optimization performs machining in the following sequence:

1. Roughing all panels from left to right with tool E1
2. Finishing all panels from left to right with tool E2

Requiring a total of 2 tool changes (1 if E1 is already in the spindle at the start).

If optimization also includes optimization of the tool change, panel finishing will start from the right instead of restarting from the left.

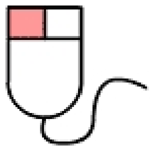
8. Worktables Editor

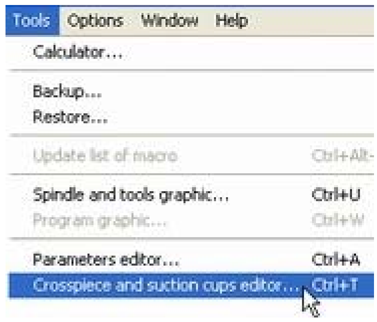
8.1 Description of Interface

With the worktables editor it is possible to graphically program the position of the supports used to fasten the panels to the machines' work table.

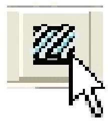
► Opening the worktables editor.

Open or create a program, then click on the **TOOLS/CROSSPIECE AND SUCTION CUPS EDITOR** menu.

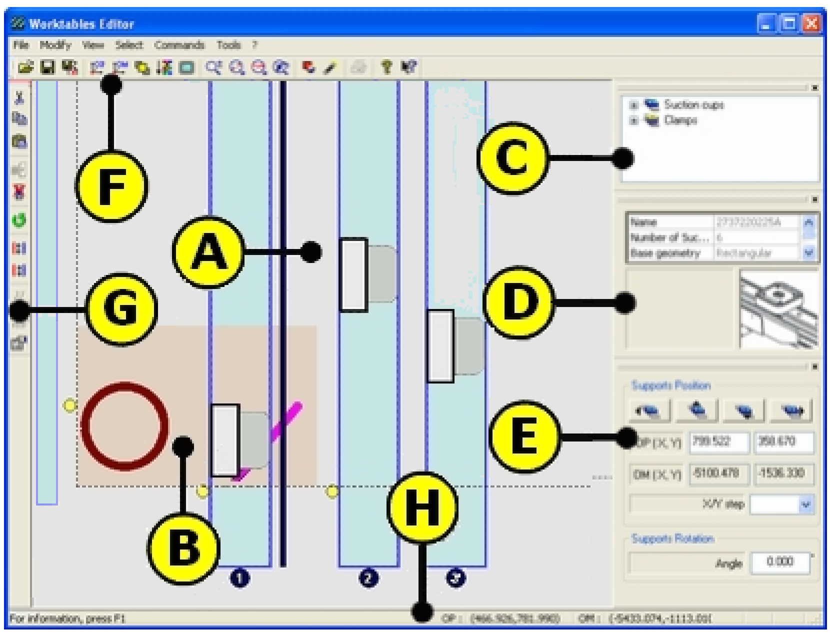




… or on the button.



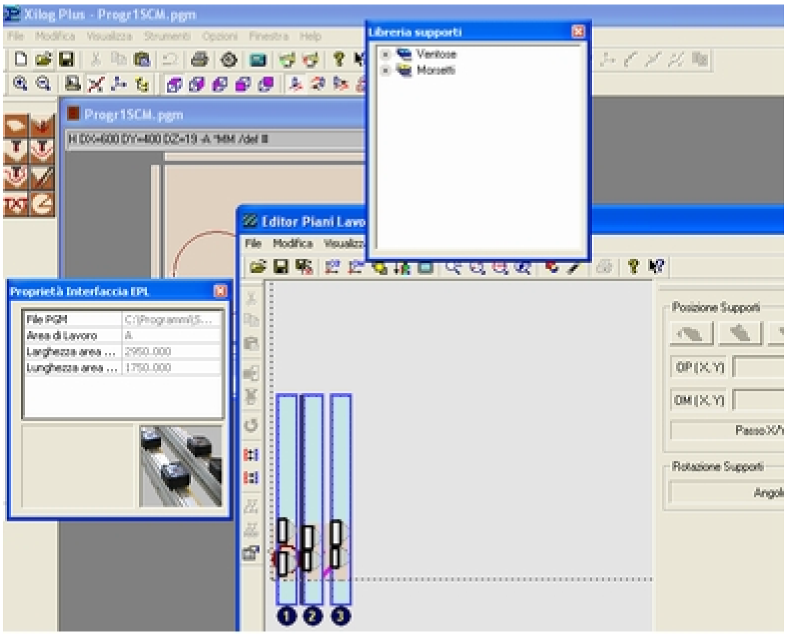
The worktables editor opens on a separate screen to that of Xilog Plus.



A) Work table layout.
B) Panel and programmed machining layout.
C) Support library toolbar: contains the list of available supports and table elements.
D) Properties toolbar: displays the characteristics for the support or table element selected.
E) Supports movement toolbar: makes it possible to move and rotate the supports by entering information from the keyboard.
F) Toolbar (see: Appendix A).

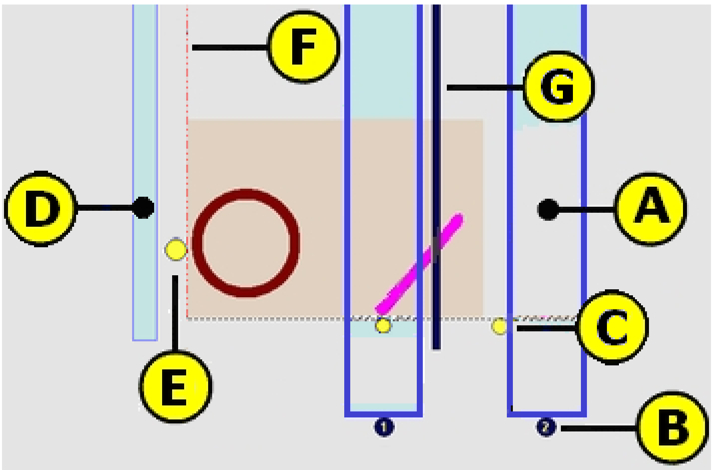
G) Special toolbar (see: Appendix A).
H) Status toolbar (see: Appendix A).

Each of the toolbars is contained in a separate screen that can be moved into any position by dragging the upper edge of the screen with the mouse.



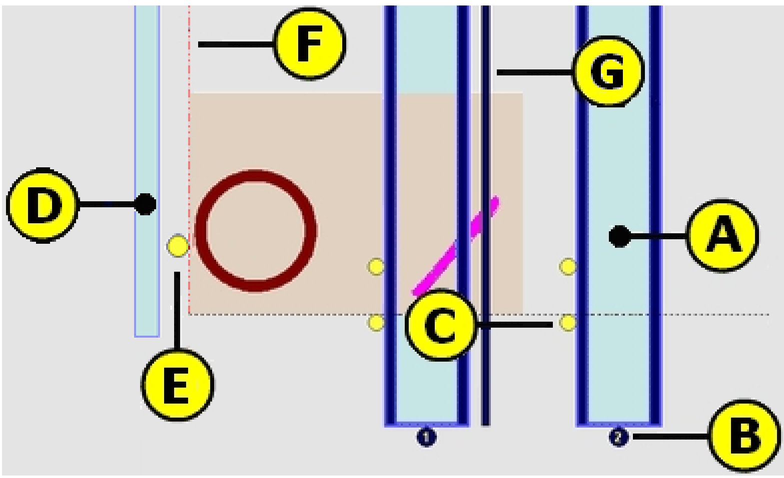
8.2 Types of Tables

8.2.1 Automatic Crossbeams and Suction Cups Table



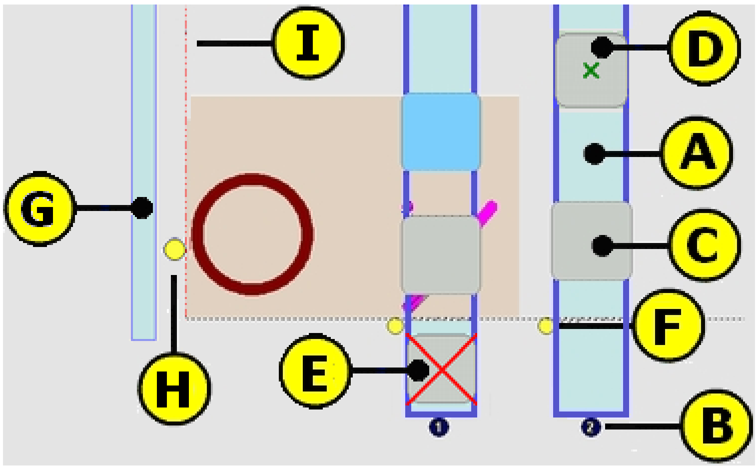
A) Crossbeam.
B) Crossbeam number.
C) Bottom stop residing on the crossbeam.
D) Movable bar for insertion of side stops.
E) Side stop.
F) Piece reference stop.
G) Load lifting toolbar.

8.2.2 Manual Crossbeams and Suction Cups Table



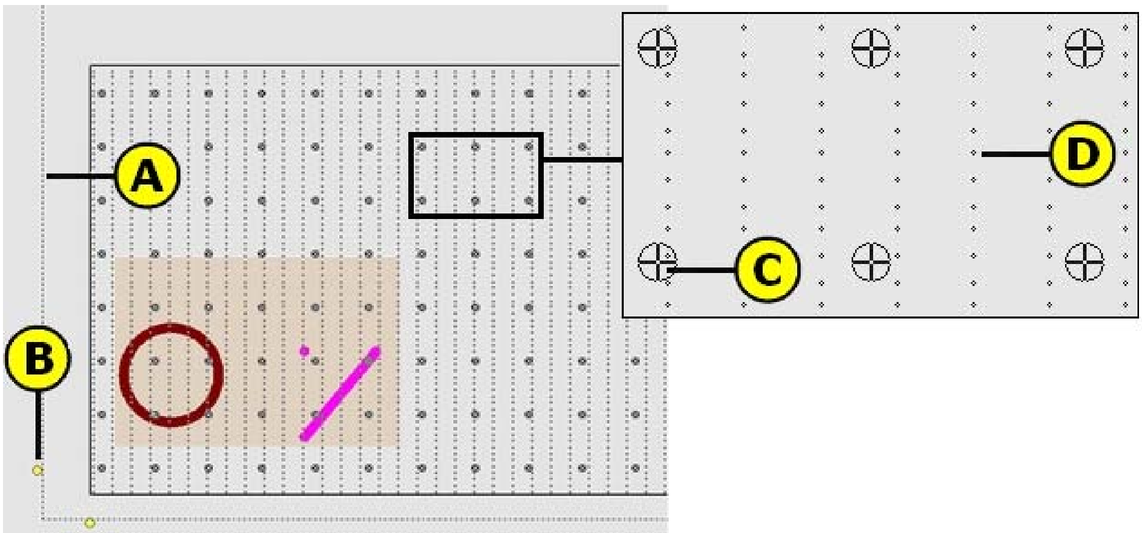
A) Crossbeams (with side tracks).
B) Crossbeam number.
C) Bottom stop residing on the crossbeam.
D) Movable bar for insertion of side stops.
E) Side stop.
F) Piece reference stop.
G) Load lifting toolbar.

8.2.3 Motorized Crossbeams and Suction Cups Table



A) Crossbeam.
B) Crossbeam number.
C) Motorized base.
D) Manual base. May be moved between the minimum and maximum limit switch limits established in the configuration file supports.cfg.
E) Fixed base. Positioned at the parking dimension and cannot be moved.
F) Bottom stop residing on the crossbeam.
G) Movable bar for insertion of side stops.
H) Side stop.
I) Piece reference stop.

8.2.4 Multipurpose Table



A) Piece reference stop.
B) Piece support stop.
C) Suction screws.
D) Support passage point.

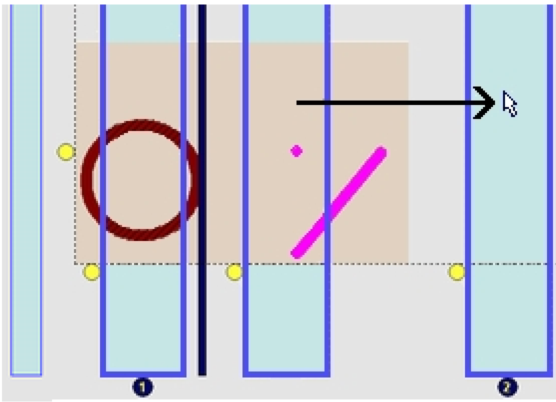
8.3 Table Programming

8.3.1 Table Programming with the Mouse

8.3.1.1 Crossbeams and Suction Cups Tables

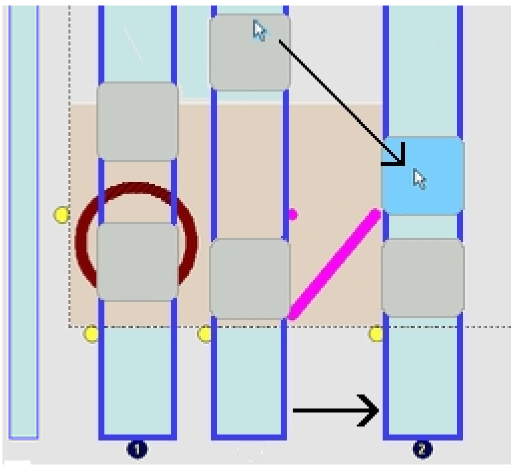
► Positioning a crossbeam.

Crossbeams can be positioned by dragging them with the mouse: place the cursor over one of the crossbeams, keep the left mouse button pressed and drag.



► Positioning a base (motorized crossbeam and suction cups table).

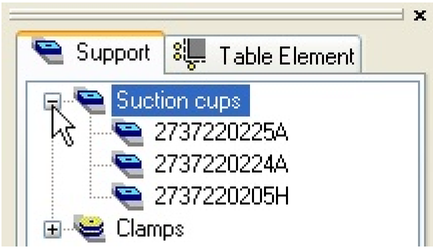
To move a base over a motorized crossbeam table, move the cursor over the base, keep the left mouse button pressed and drag. Together with the base, it is also possible to move the crossbeam upon which the base is mounted.



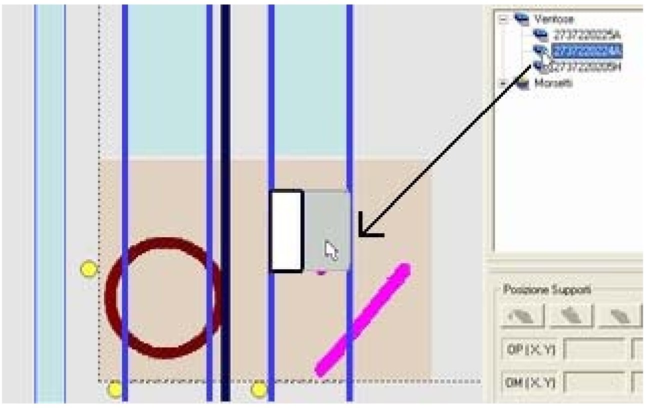
► Inserting a support or a table element.

1. Select the supports or the table elements page.

2. Click on the + sign to open the list of supports or of table elements.



2. Move the cursor over the support or the table element to be inserted then, keeping the left mouse button pressed, drag the support into the desired position.



On motorized crossbeams and suction cups, the support must be inserted onto one of the bases present on the crossbeams.

The piece support stops can only be inserted if the movable bar is present.

► Eliminating a support or a table element.

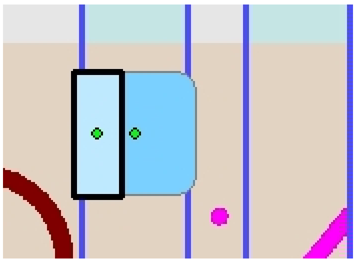
Select the support or the table element and click on the **MODIFY/CLEAR** menu.

► Moving a support or a table element.

Place the cursor over the support or the table element, keep the left mouse button pressed and drag.

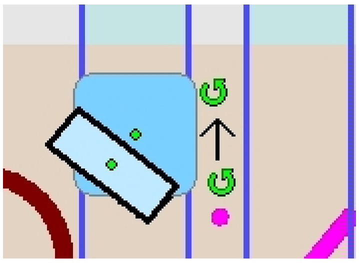
► Rotating a support (suction cups).

1. Select a suction cup to be rotated and click on the **COMMANDS/ROTATE SUPPORT** menu.



The center of the suction cups rotation is highlighted graphically by a green circle.

2. Keeping the left mouse button pressed, drag to position the suction cup.

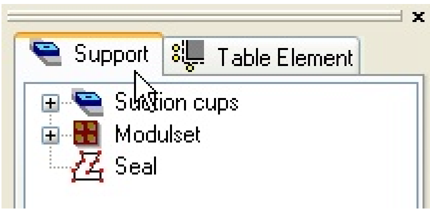


To exit the rotating mode, click on the **COMMANDS/ROTATE SUPPORT** *menu again or press the* **[ESC]** *key on the keyboard.*

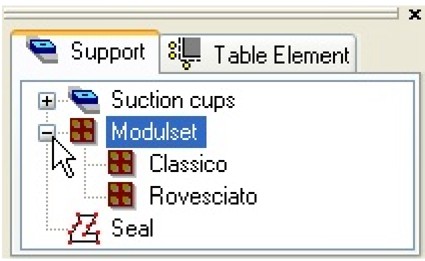
8.3.1.2 Multipurpose Table

► Inserting a support (modulset or suction cup) or a table element (support stop).

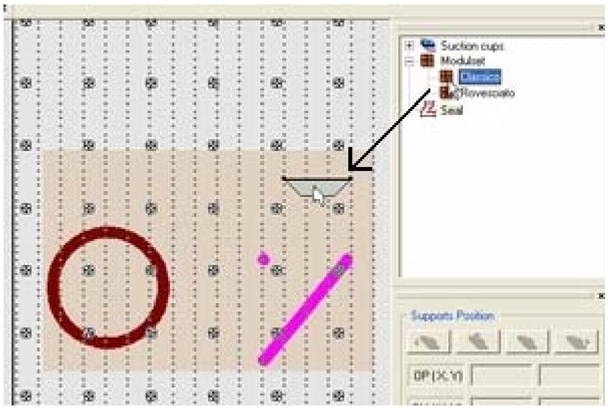
1. Open the list of supports or table element.



1. Click on the + sign to open the list of supports or table element.



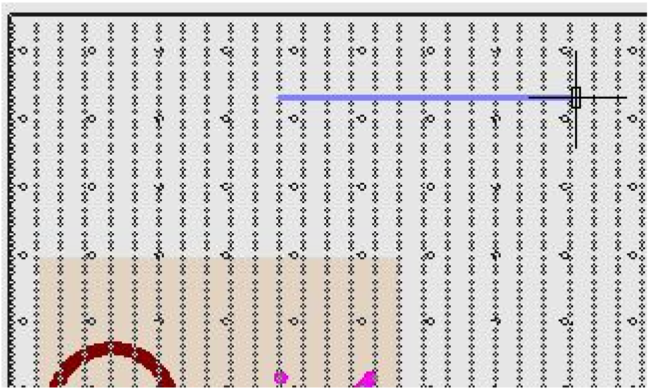
2. Move the cursor over the support or support stop to be inserted then, keeping the left mouse button pressed, drag the support or the support stop into the desired position.



The program will automatically position the support in the proper position as established by the table grid.

► Inserting a seal.

1. Click on the **TOOLS/POSITION SEAL** menu to activate the function that inserts the seal.
2. To insert the seal, click on the starting point and move the mouse to trace the first line.



The line just traced can be moved in intervals of 90° by simply moving the mouse. To delete, press the **[ESC]** *on the keyboard.*

3. To trace the next line, click and then move the mouse in the direction of the new line.
4. To stop inserting the seal, click twice on the final point of the route.

Each line terminated with a double click is considered a separate line. If the separate lines are next to each other (in other words they form a continuous profile) they can be united by clicking on the **TOOLS/JOIN ADJACENT POLYLINES** *menu. All separate adjacent lines however, are joined automatically by the program when the file is saved.*

To exit the insert seal mode, click on the **TOOLS/POSITION SEAL** *menu or press the* **[ESC]** *key on the keyboard. Other types of support cannot be inserted while in seal mode.*

► Eliminating a support or a support stop.

Select a support or a support stop and click on the **MODIFY/CLEAR** menu.

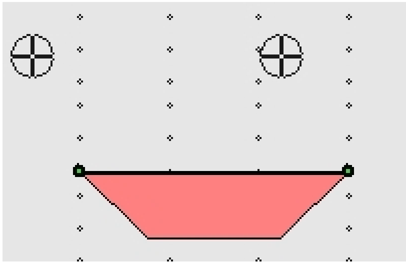
To contemporarily select all the lines for a profile that is joined by a seal, click on the seal while keeping the ***[CTRL]*** *key on the keyboard pressed.*

► Moving a support (suction cup or modulset) or a support stop.

Select the support or the support stop and drag it to the desired position using the mouse.

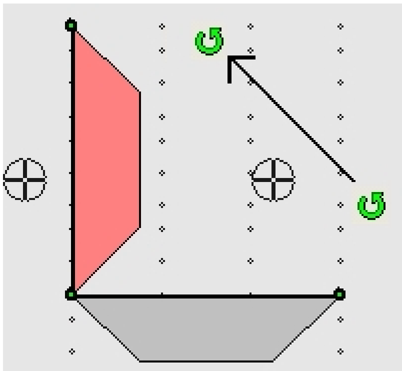
► Rotating a support (suction cup or modulset).

1. Select a support that can be rotated and click on the **COMMANDS/ROTATE SUPPORT** menu.



The modulset rotates around the tongue joint.

2. Hold the left mouse button pressed and drag to position the support.

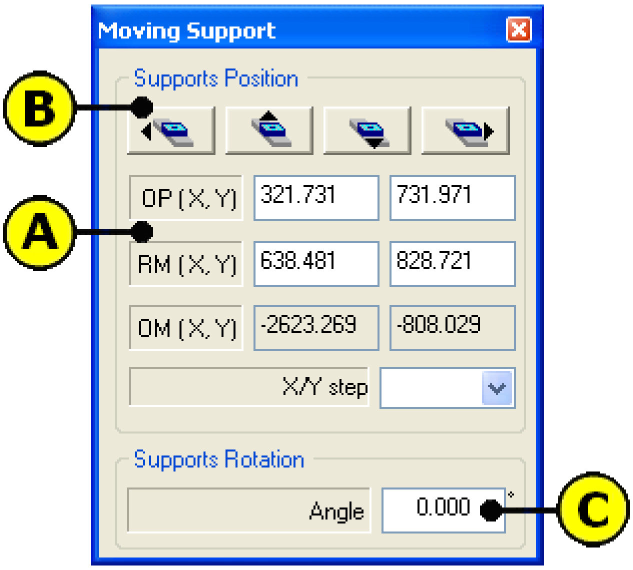


The modulset can only be rotated at intervals of 90 degrees.

To exit the rotation mode, click on the **COMMANDS/ROTATE SUPPORT** *menu again or press the* **[ESC]** *key on the keyboard.*

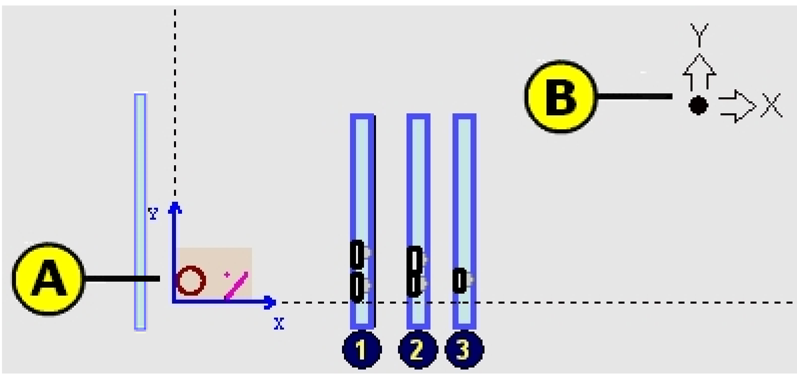
8.3.2 Table Programming Using the Keyboard

Inserting, moving and rotating a support (except for seals on multipurpose tables) or inserting a table element can be also programmed by entering the data using the keyboard. A support movement toolbar is provided to move and rotate a support inserted on a work table.



A) Box for positioning selected support:
- **OP** boxes **(X, Y)**: show the X and Y coordinates for the support selected compared to the panel starting point. In these boxes it is also possible to enter the coordinates for the point in which the support is to be moved.
- **RM** boxes **(X, Y)**: show the X and Y co-ordinates of the "sight" of the selected support relative to the metric rulers on the table. The sight represents an alternative reference point relative to the centre of the support. The co-ordinates of the sight coincide (default values) with those of the centre of the support base, but can be configured differently for the individual supports. On tables with crosspieces and suction cups, you can use these boxes to enter the co-ordinates of the point you want to move the support to, according to the position of the sight.
- **OM** boxes **(X, Y)**: show the X and Y coordinates for the support selected compared to the machine starting point. New coordinates for moving the support cannot be entered.
The X and Y coordinates indicate a different support point for each type of support. For further information, refer to paragraph 8.6 (Types of supports).
B) Buttons for moving selected support. Can be used after having set the distance in the **X/Y STEP** box.
C) Box for setting rotation degrees for those supports than can be rotated.

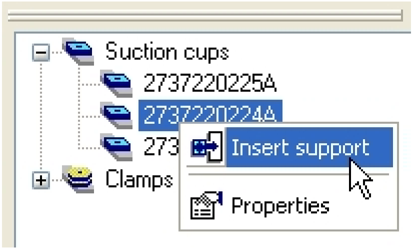
To display the panel starting point and the machine starting point click on the **VIEW/ SHOW/HIDE PGM ORIGIN** *and* **VIEW/ SHOW/HIDE MACHINE ORIGIN**.



A) Panel starting point.
B) Machine starting point.

► Inserting a support or a table element.

1. Open the list of supports or table elements and click on the support or table element to be inserted with the right mouse button.
2. Click on **INSERT SUPPORT** in the shortcut menu.

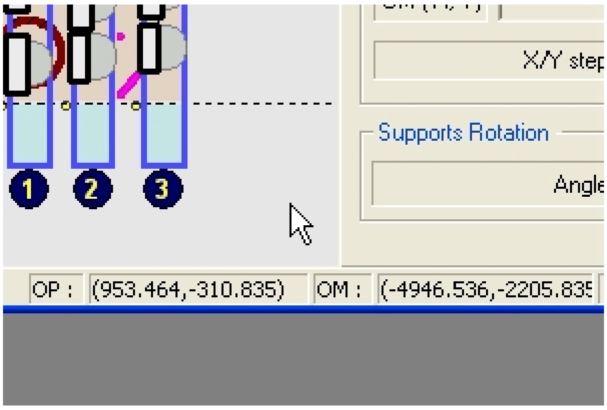


3. Set the insert position in the **Insert support** or in the **Insert table element** screen, depending on the operation to be performed.
4. Click the **OK** button to confirm the insertion of the support or table element.

► Moving a support by setting the coordinates.

1. From the work table, select the support to be moved.
2. Enter the X and Y coordinates for the new position in the **OP** boxes.

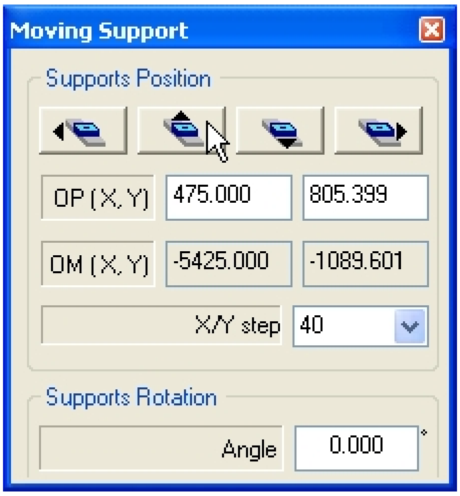
*The **OP** and **OM** boxes show the current coordinates for the selected support. To obtain the coordinates for any point on the work table, position the cursor on the required point: the coordinates will be displayed on the right of the status toolbar.*



3. Press the **[ENTER]** key on the keyboard.

► Moving a support by setting the distance.

1. From the work table, select the support to be moved.
2. Set the distance in the **X/Y step** box.
3. Move the support in the desired direction using the direction keys.



The selected support can be moved various times and in different directions, depending on the distance set.

The move can also be controlled from the keyboard using the arrow keys.

► Moving or duplicating a support.

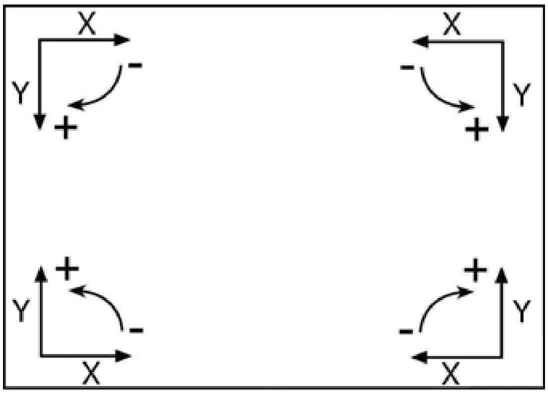
The supports already inserted can be moved or duplicated using the **CUT/COPY** and **PASTE** controls. These features are particularly useful for moving or duplicating a support that is already inserted and has been changed (for example, by rotation).

1. Select a support on the work table.
2. Click on **MODIFY/CUT** (to move the support) or **MODIFY/COPY** (to duplicate the support). The selected support, or its copy, is stored under Clipboard.
3. Click on the **MODIFY/PASTE** menu.
4. Insert the data requested in the **Paste support** screen .
5. Click on **OK**: the support that is memorized in the Clipboard will be inserted in the set position.

► Rotating a support.

1. From the work table, select a support that can be rotated.
2. Set the rotation angle in the **Angle** box.

With positive values, the rotation always turns from the X axis towards the Y axis. The direction will therefore either be clockwise or anticlockwise, depending on the user panel starting point (piece starting point - OP), as indicated in the layout below.



3. Press the **[ENTER]** key on the keyboard.

8.3.3 Parametric programming (motorised cross bar and suction cup table)

8.3.3.1 Introduction to parametric programming

Parametric programming makes it possible to insert several parameters into a program with values that can be varied as needed. This way we will have a single program which allows us to manage pieces with different dimensions.
Parametric programming of the positions of the supports can be carried out inside the **supports movement** bar by directly inserting the parametric formulas in the OP boxes relative to the X and Y coordinates of the selected supports.

There is also another value, **numeric offset,** which, when algebraically summed to the parametric value, defines an **absolute numeric value** which shall constitute the real programming position applied to the support.
The **numeric offset** has been introduced to guarantee that successive modifications to the support position (for example by moving it with the mouse) make it possible to maintain the corresponding parametric formula unaltered.

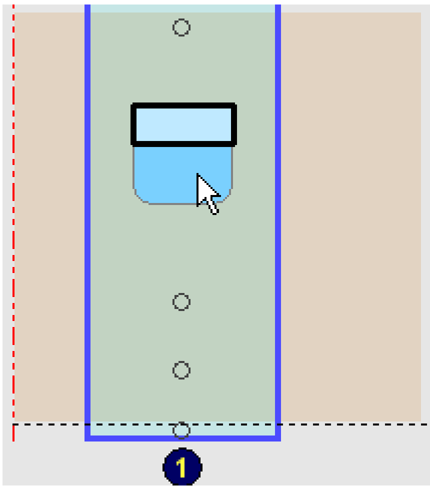
By confirming the programming, this absolute value becomes a property of the support and is displayed in the **support movement** bar as if it were a directly entered value.

Absolute numeric value = Parametric Value + Numeric Offset

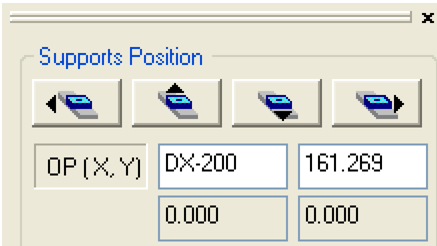
The parametering is the centre of the support rest.
When programming several supports within the same cross bar, in relation to the X axis, a modification of the coordinate X of the last parametered support modifies all of those previously set.

► Parametering a suction cup support.

1 select a suction cup support by clicking with the left mouse button.



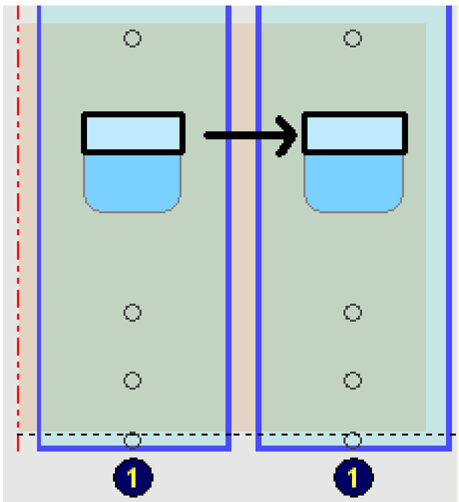
2 Select the OP box relative to the X coordinate of the suction cup support and enter the DX-200 parametric formula. Press the **[ENTER]** key from the keyboard to confirm.



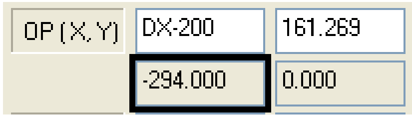
The X coordinate of the selected support has been parametered. If the panel dimensions are modified, the work table Editor will reposition the cross bar on the basis of the new panel dimensions.

► Modify the position of the previously parametered suction cup support.

1 Select and move the cross bar with the mouse.

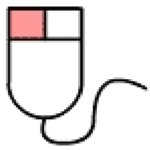


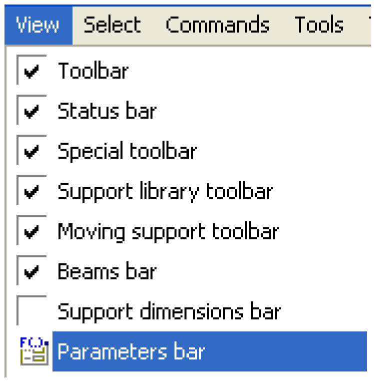
2 The work table Editor maintains the parametric formula unaltered by modifying the value of the **numeric offset**; the value will change (in positive or in negative) on the basis of the difference between the initial and final values of the support X coordinate.



► Open the program parameters bar.

Click on the **DISPLAY/PARAMETER BAR.**



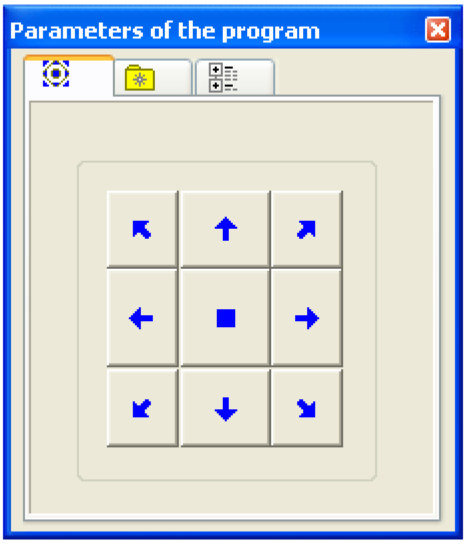


... or on the button.

The parameter Bar is composed of three windows:

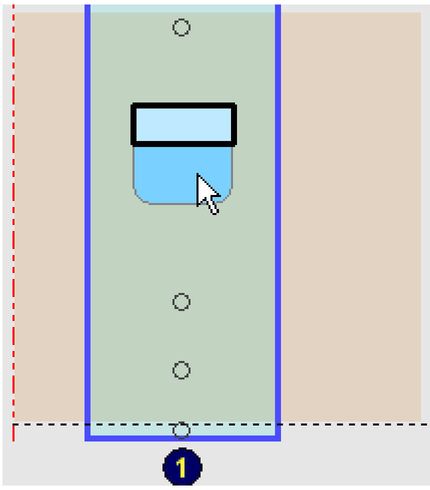
A) Magic Eye
B) Preferred
C) Category

A) The **Magic Eye** window is used to parameter the supports using formulas predefined by the work table Editor.

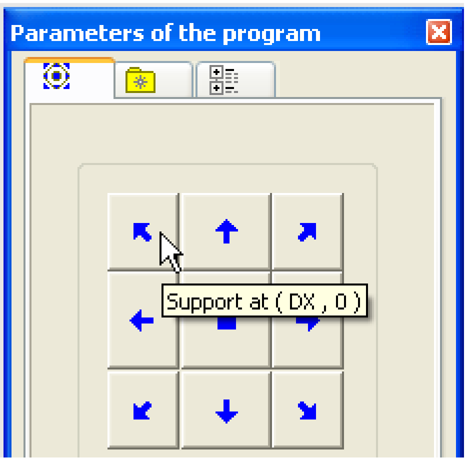


► Parameter a support using the Magic Eye.

1 select a support.

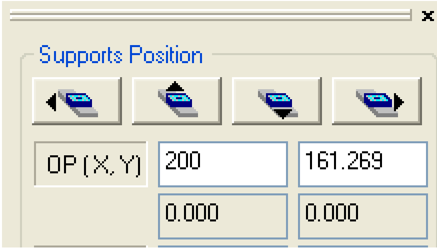


2 Insert the parametric formula (DX,0) by clicking on the proper button.
The X coordinate of the selected support has been parametered.

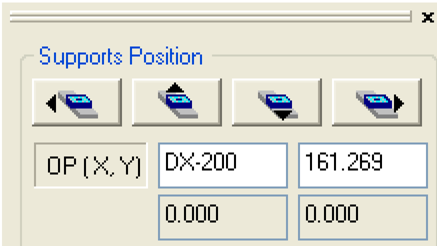


(X,Y) offsets will be automatically and suitably summed to the selected parametric formulas, allowing the parametered support to maintain its position.
These offsets can be successively modified.

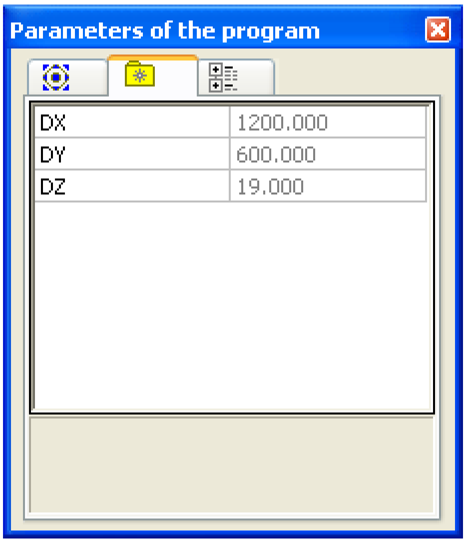
Before the parametering:



After the parametering:

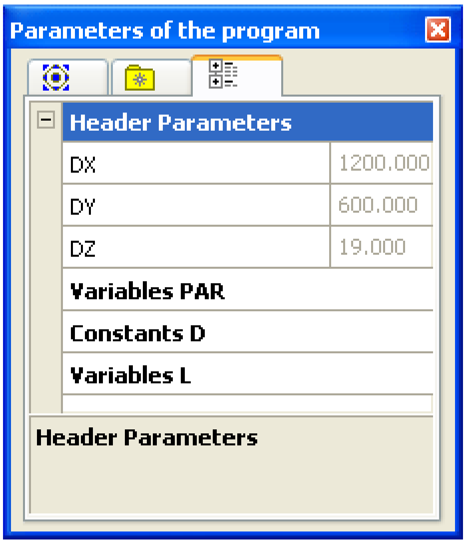


B) The **Preferred** window is used to display the parameters previously selected as preferred.



It is possible to copy the selected variable from the window inside the OP box of the (X,Y) coordinate of the support.
To copy the formula, first select the OP box in which the parametric formula is to be inserted, then double click with the left mouse button on the formula inside the **Preferred** window.

C) The **Category** window is used to display the entire list of available variables, grouped by category.



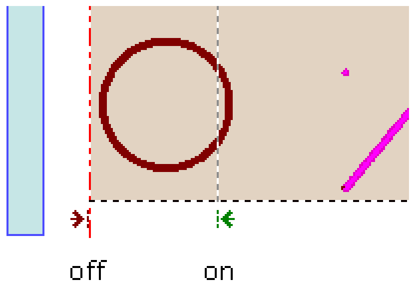
The following variables are available:

DX, DY of the main program Header.
PAR variables of the main program Header.
D constants of the main program Header.
L variables of the main program Header.

It is possible to copy the selected variable from the window inside the OP box of the (X,Y) coordinate of the support.
To copy the formula, first select the OP box in which the parametric formula is to be inserted, then double click with the left mouse button on the formula inside the **Category** window.

8.4 Movable Bar Automatic Management

If the movable bar for insertion of the side stops against which the workpiece rest is configured for automatic management, the graphics show two vertical dashed lines highlighted by arrows. These lines indicate two possible positions (**OFF** and **ON**) of the line of contact between the work table and the workpiece stops.

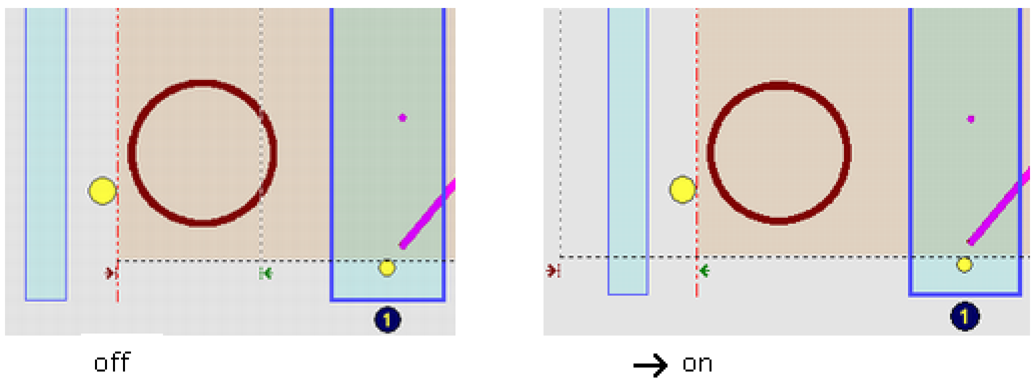


Automatic management allows you to move the movable bar between the OFF position and the ON position in two ways:

A) OFF position or ON position.
B) Continuous movement between the OFF position and the ON position (only for central bars on tables configured with double areas from the outside in; for example, AB, DC, EF, HG, etc.).

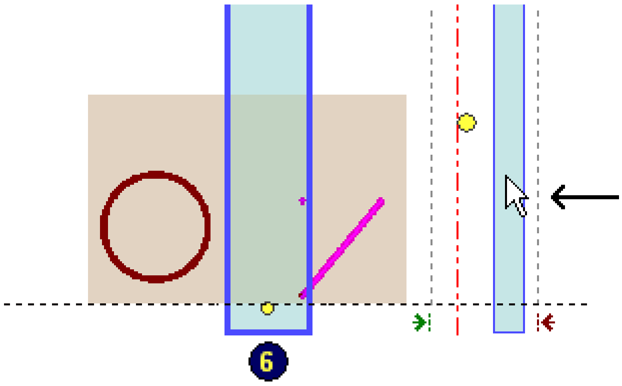
► OFF or ON position.

You can set the OFF or ON position using the **MOVABLE BAR** parameter in Header field T. In the OFF position (default) the movable bar is at the outer end of the work table; in the ON position the movable bar is moved towards the inside of the work table.



► Continuous movement between the OFF position and the ON position.

Continuous movement is only available for the central bars on tables configured on double areas from the outside in (for example, areas AB, DC, EF, HG, etc.). In this case, the right-hand bar may be dragged with the mouse into the space between the OFF position and the ON position.


6

8.5 Saving

With the editor, work table program can be stored in two ways:

a) stored inside the machining program.
Table program is stored together with the machining program.

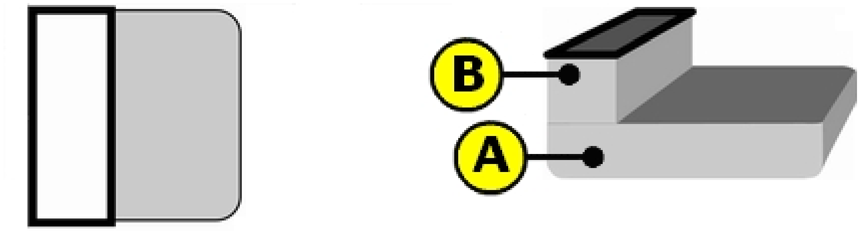
► Click on **FILE/SAVE LAYOUT**, to save without closing the editor, or **FILE/SAVE AND CLOSE**, to save and close the editor.

b) store in a separate file.
Table program is stored (by default, in the Job folder) in a separate file with .epl extension. The file can be reopened through the **FILE/OPEN LAYOUT** menu and reutilized various times.

► Click on **FILE/SAVE LAYOUT WITH NAME.**

8.6 Types of Supports

a) Suction cup (for crossbeams and suction cups tables, multipurpose tables).



A) Base.
B) Rest.

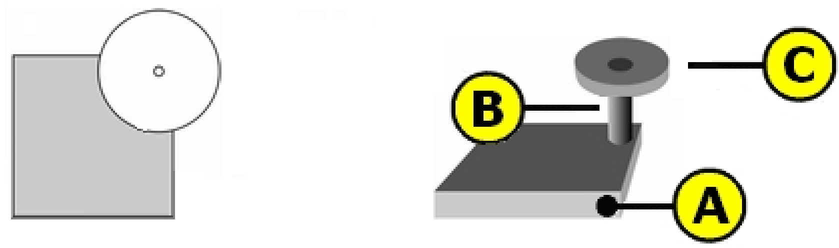
The X, Y coordinates for insertion on the work table indicate the central point of the rest.

The suction cups may have rectangular or circular bases and supports, and may have two supports. On multipurpose tables, each different base shape requires a different hooking system:

- rectangular base: joins at the points where the support passes;
- circular base: joins at the vacuum screws.

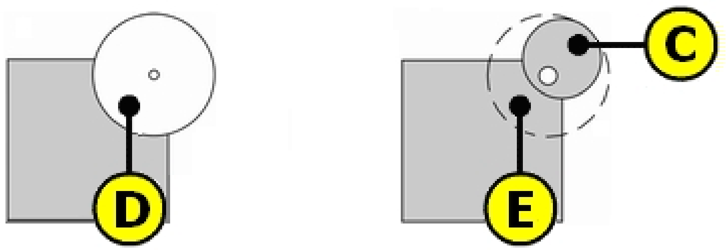
The exact position in which the suction cup bases attach to the multipurpose table is outlined in blue, and is displayed when the suction cup is inserted with the mouse.

b) Clamp (for crossbeams and suction cups tables).



A) Base.
B) Stand.
C) Plate.

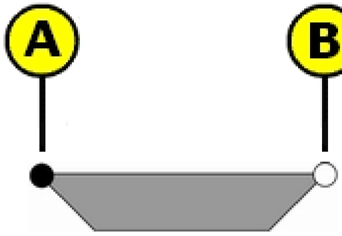
The circle (**D**) found in the graphic picture of a clamp can represent the entire plate (which cannot be rotated) of the clamp or the rotation area of a plate that can be rotated, according to the following diagram:



C) Plate.
E) Plate rotation area.

The X, Y coordinates for insertion on the work table indicate the central point of the stand.

c) Modulset (for multipurpose tables).

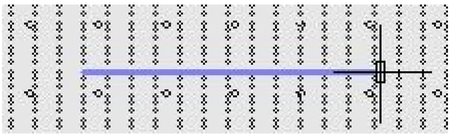


A) Tongue joint point.
B) Groove joint point.

The X, Y coordinates for insertion on the work table indicate the tongue joint point.

The exact position in which the modulset attaches to the multipurpose tables is outlined in blue and is displayed when the modulset it inserted with the mouse.

d) Seal (for multipurpose tables).



The thickness of the seal (indicated it the **Properties** screen) can be configured.

8.7 Other Functions

8.7.1 Display Modes

The various parts of the work table layout can easily be examined thanks to the display and zoom features.

a) Dragging the graphic layout.
The graphic layout of the work table can be dragged with the mouse: move the cursor over any point of the work table (not over a support or a crossbeam), keep the left mouse button pressed and drag.

b) Displaying the panel in the foreground.
The panel is displayed in the foreground of the work table.

► Click on the **VIEW/PANEL CLOSE UP** menu.

c) "Light" display
This display mode shows a simplified drawing of the work table. It can be useful for computers with low graphic capability.

► Click on the **VIEW/ACTIVATE LIGHT DRAWING**.

d) Zoom.
The **TOOLS/ZOOM** menu holds various zoom functions:

- **REAL TIME ZOOM**. Makes it possible to enlarge or reduce the image of the work table. After having activated this feature, just click on the left mouse button to enlarge or on the right mouse button to reduce the image. The zoom is centered on the clicking point.
- **ZOOM AREA**. Makes it possible to trace a rectangle by dragging the mouse over the drawing: the area inside the rectangle will be enlarged when the mouse is released.
- **WORKPIECE PROGRAM ZOOM**. Enlarges the area occupied by the panel.
- **RESTORE ORIGINAL ZOOM.** Resets the drawing to its original size.

With a *wheel mouse* the zoom can be used by turning the small wheel directly.

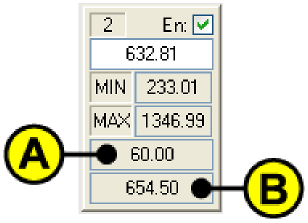
The following variables are available:

DX, DY of the main program Header.
PAR variables of the main program Header.
D constants of the main program Header.
L variables of the main program Header.

It is possible to copy the selected variable from the window inside the OP box of the (X,Y) coordinate of the support.
To copy the formula, first select the OP box in which the parametric formula is to be inserted, then double click with the left mouse button on the formula inside the **Category** window.

e) Cross bar displayers (cross bar and suction cup table)
The displayers show the identification number and the position in Y of the cross bars on the cross bar and suction cup tables.

► Click on the **DISPLAY/SHOW/HIDE DISPLAY MENU CROSS BARS.**



The check box **En** is used to enable sending the positions to the displayer on the cross bar.

The box that shows the position in Y is also used to enter a position and to move the cross bar by pressing the **[ENTER]** key. The entered value is highlighted in red if it is not compatible with the configured travel end minimum and maximum limits of the work area.

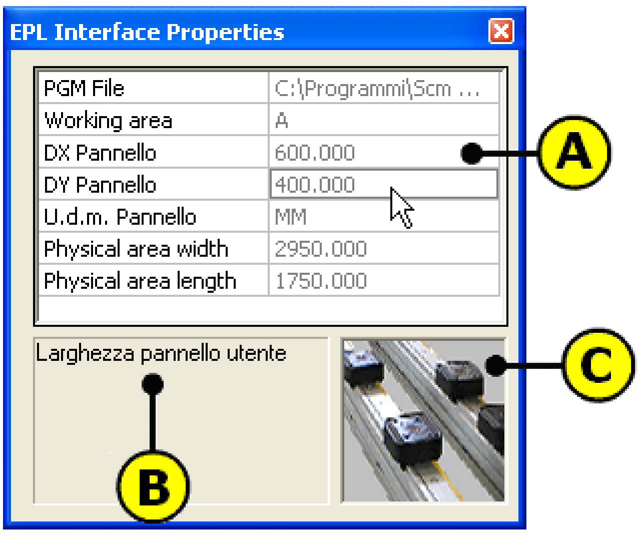
MIN: minimum travel end value of the cross bar with reference to the panel origin.
MAX: maximum travel end value of the cross bar with reference to the panel origin.

A) minimum value of the work area with reference to the panel origin.
B) maximum value of the work area with reference to the panel origin.

The displayer is also available for the mobile bar on the continuous automatic management tables.

8.7.2 Properties Toolbar

The properties toolbar can be displayed/hidden with the **VIEW/PROPERTIES BAR** menu or under **PROPERTIES** from the shortcut menu. Once activated, it is sufficient to click on a support or work table to view the relative information.



A) This area displays all the information available on the item selected: a support, a table element or, as shown in the example, the work table itself.
B) This box displays an explanation for the item selected in screen A.
C) Descriptive image of the item selected (the work table or a support).

8.7.3 Anticollision Control

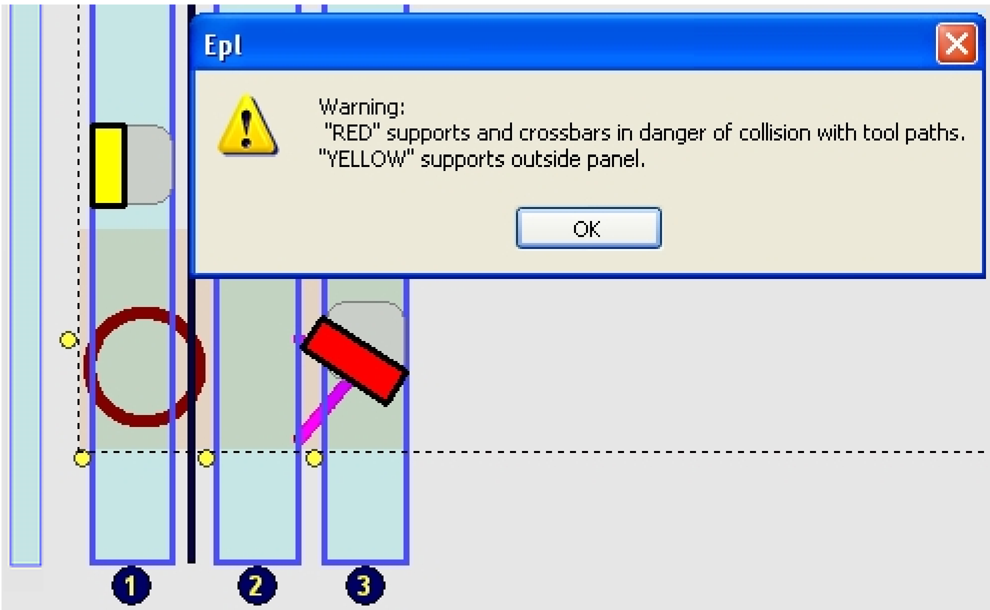
The through lines for the work programmed on the panel are shown in fuchsia. In this manner it is possible to position the supports for the work table supports in such a way as to prevent machine tools and work table from colliding.
Additionally, the worktables editor is equipped with an anticollision feature which **should always be used prior to storing the table program.**

Caution! The clamp and seal position is not checked.

► Click on the **TOOLS/ ANTI-COLLISION CONTROL** menu.

If work table program involves collisions, the control displays a warning and changes the color of the crossbeams and supports:

a) to red, for crossbeams and supports on collision;
b) to yellow, for supports positioned outside the panel.



The work table Editor can also run the collision prevention check automatically. To activate this function, click on the **TOOLS/OPTIONS** menu, then click on the **ENABLE ANTI COLLISION CONTROL FOR SAVE** box and click on **OK** to confirm: the check will be run automatically each time the table programming is saved.

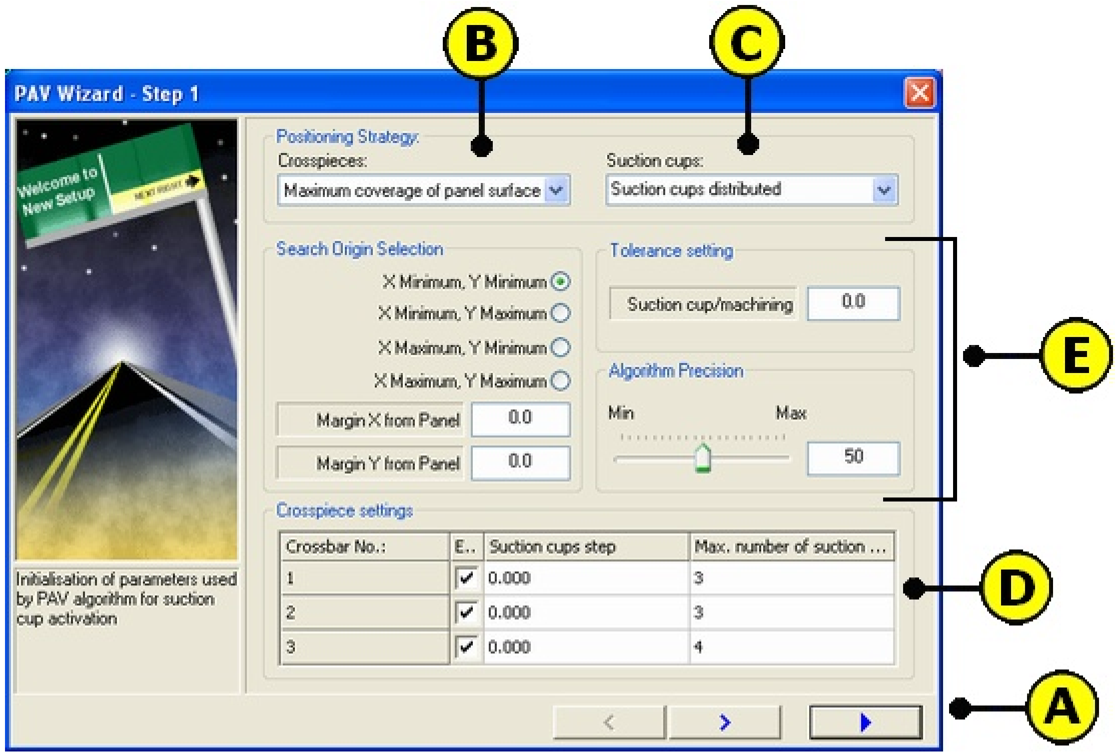
If the automatic collision prevention check is switched on, you cannot save the table programming with supports in collision or outside the panel.

8.7.4 Automatic Suction Cup Programming

The work table editor is equipped with an automatic programming function for the position of the suction cups on the cross bar and suction cup table; it positions the suction cups for optimising the programmed machining operation. This function can be customised by following a guided procedure.

► Step 1.

► Click on the **TOOLS/SUCTION CUP AUTO. PROGR./NEW SUCTION CUP AUTOMATIC PROGR.** menu.



A) Navigating buttons. The first two buttons are used to move backward/forward between the steps of the guided procedure. The third button (to the right) is used to start the automatic positioning of the suction cups, at any time: in this case, the default values will be used for the unmodified options.

B) Positioning strategy: crosspieces. Used to set a method for automatic cross bars positioning.

- Crosspieces distributed (default). The cross bars are positioned so that they are the same distance one from the other. The cross bars are positioned independently of the number of suction cups which will then be inserted (there may even be no suction cups inserted).

- Maximum coverage of panel surface. The cross bars are positioned to create the maximum suction cup coverage possible of the panel.

- Rectangles defined by users. Used to draw more rectangular areas on the panel, in which to insert the suction cups.

► Draw a rectangle for the insertion of the suction cups.

1. Click on the "forward" navigation button (A) to pass to the draw environment.
2. Keep the left button of the mouse pressed and drag the mouse to draw the rectangle.

Each suction cup is inserted only if its rest is totally within a single rectangle.

► Delete a rectangle for suction cup insertion.

1. Disable the draw function: click on the **TOOLS/SUCTION CUP AUTO. PROGR./ DRAW INSERTION RECTANGLES** menu.
2. To select the rectangle to be deleted, click on it.
3. Click on the **TOOLS/SUCTION CUP AUTO. PROGR./ DELETE INSERTION RECTANGLES** or press the **[DEL]** key.

- **Crosspieces positioned by user**. Used to manually position the cross bars.

► Manually position the cross bars.

1. Click on the "forward" navigation button (A) to display the cross bars.
2. Position the cross bars by dragging them with the mouse.

C) Positioning strategy: suction cups. Used to set an automatic positioning method for the suction cups. Positioning the suction cups on the cross bars is independent of positioning the cross bars on the panel.

- **- Suction cups distributed (default).** The suction cups are distributed uniformly. The suction cups positioned this way, but with rests that collide with through machining, are not inserted.

This option is not enabled if the "Rectangles defined by user" option for cross bar placement has been selected.

- **Step set setting between two adjacent suction cups.** The suction cups are positioned according to the minimum distance set in "Crosspiece settings" (D).

- **Maximum filling crosspiece surface.** The suction cups are positioned with the smallest distance between them possible (starting from 1 mm of distance between suction cups).

D) Crosspiece settings. There are several options available for each cross bar.

- **Enabling.** To enable/disable suction cup insertion, check/uncheck the box. The disabled crossbeams are still positioned.

The crossbeams cannot be disabled if the user has chosen the "Maximum coverage of panel surface" or the "Rectangles defined by user" options.

- **Suction cups step.** Used to set the distance at which the suction cups are placed, if the "Step set setting between two adjacent suction cups" option is chosen.

- Max. number of suction cups. Used to set the maximum number of suction cups which can be inserted on a crossbeam.

The maximum number of suction cups set is not considered if the "Maximum filling of crosspiece surface" positioning option is chosen.

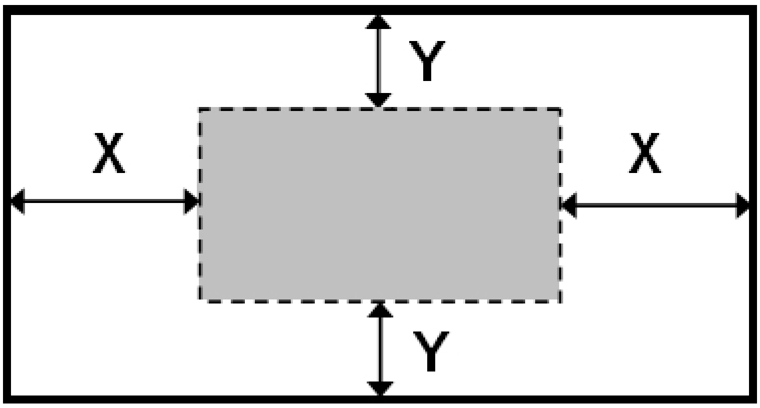
E) Advanced options.

- Search origin selection. Defines the panel corner from which suction cup placement begins. The corner is defined based on the X, Y minimum/maximum co-ordinates (the default value is "X Minimum, Y Minimum", i.e. the point of origin of the axes).

The definition of the corner depends on the axis reference system (front or rear machine origin) and on the configuration of the work area.

The choice of the search origin significantly influences crossbeams positioning (if the options "Maximum coverage of the panel surface" and "Rectangles defined by user" are selected) and suction cups positioning. In particular, in the case where not many suction cups are available, by setting different origins the type of suction cup inserted may also vary.

- Margin X from panel and **Margin Y from panel.** Used to define a rectangular area inside the panel in which the suction cups are positioned (grey, in the example). The margins are calculated on both sides of the panel.



The X and Y margins are also considered in the case where a rectangle has been defined using the "Rectangles defined by user" crossbeams positioning option. In this case the suction cups are only positioned in the area where the two rectangles intersect.

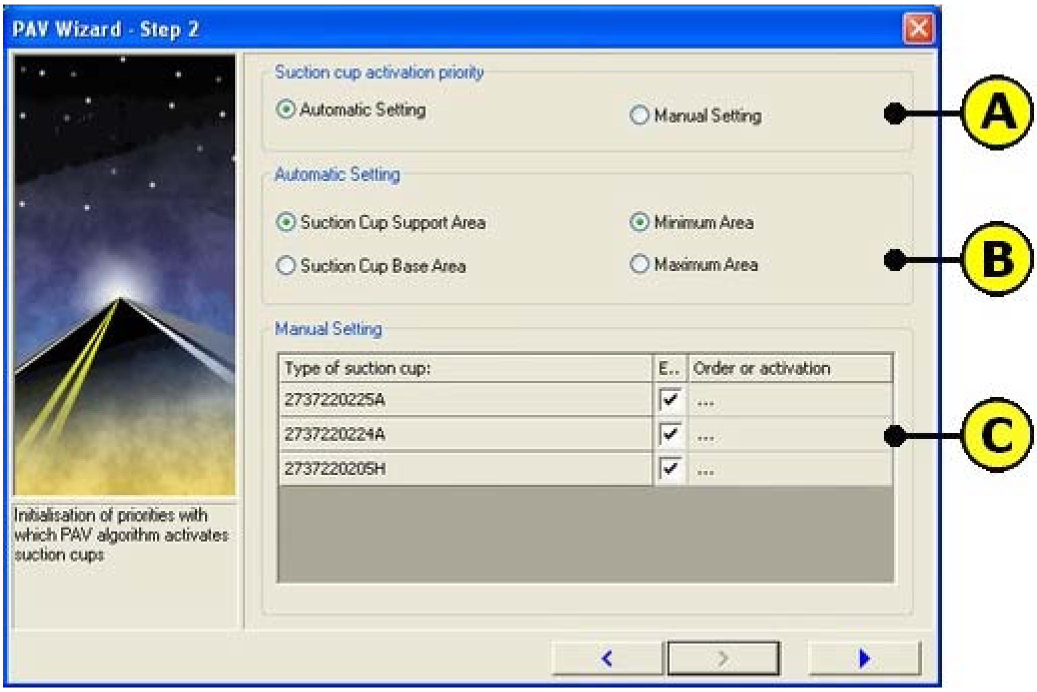
- Tolerance setting. Used to set the minimum distance between a through machining operation and the suction cup rest. This distance is valid in both X and Y; the default value is 1 mm.

- Algorithm precision. Used to adjust the precision of the calculation of the possible crossbeam and suction cup positions. The greater the precision, the small the distance used to position the crossbeams and suction cups.

A high precision lengthens the calculation times.

► **Step 2.**

► Click on the "forward" navigation button (A) to pass on to the second step of the guided procedure.



A) Suction cup activation priority. Used to set the order in which the suction cups will be placed based on the size of the rest area or of the base ("Automatic setting": B) or on the type of suction cup ("Manual setting": C).

B) Automatic setting. The suction cups are inserted based on the rest area o on the base; in both cases, the "Minimum area" or "Maximum area" options can be specified.

If a type of suction cups cannot be positioned, the program tries with the next type.

Several types of suction cups can be excluded with the "Enabling" option (C).

C) Manual setting.

- Enabling. The box to be checked/unchecked is used to include/exclude individual types of suction cups from insertion.

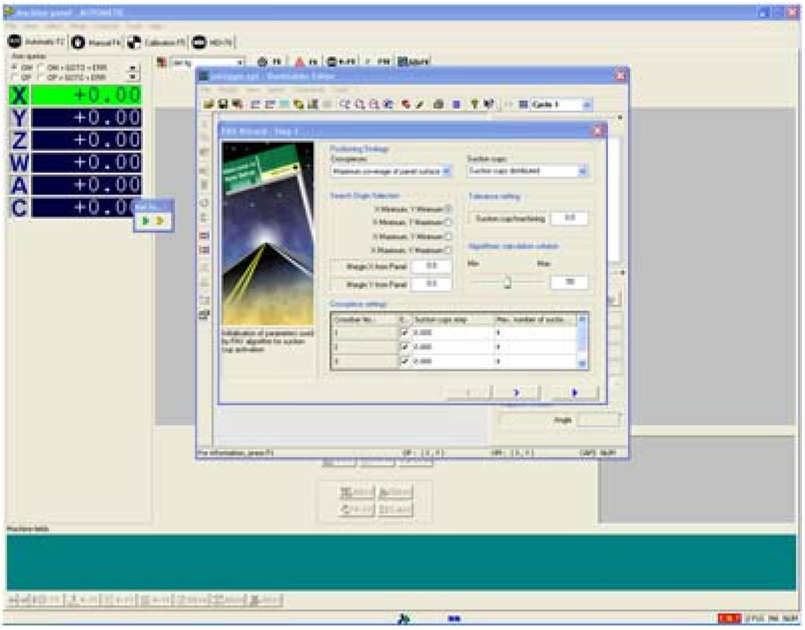
This option is also active during "Automatic setting".

- Order of activation. Used to specify the insertion order for each type of suction cup, by numbering each type in the corresponding text box. The suction cups are inserted until there are no more to be inserted, starting from the type indicated by the smallest number.

A value of 0 ("zero") disables the insertion process.

Automatic start of automatic suction cup programming.

The automatic suction cup programming function can be automatically called up from the Machine Panel in the automatic program execution mode.
At the end of automatic programming, it is possible to display, save, modify the proposed solution or load a new one.
During the execution of the automatic suction cup programming function, the work table editor always remains active.



The operator must give positive or negative confirmation of the layout before he can close it using the bar for the automatic mode of the Machine Panel, shown below.

 Save the layout and execute the program
 Exit from the session without saving the layout

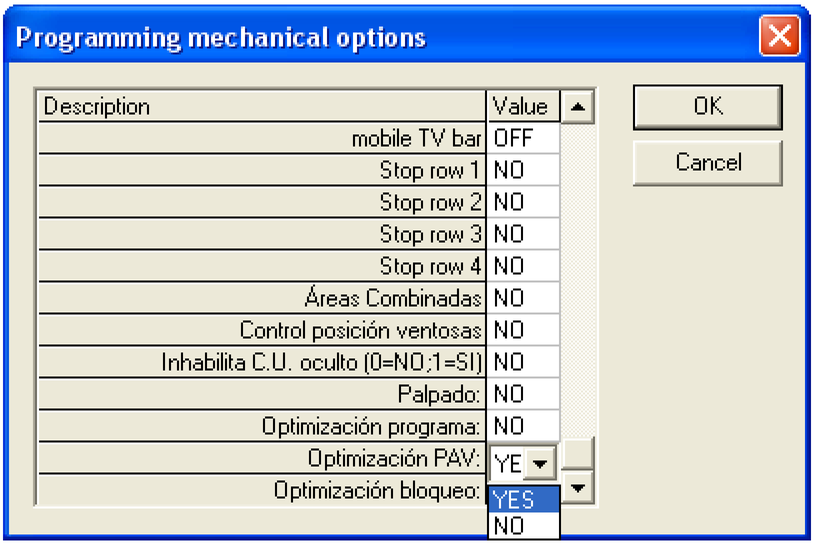
N.B.
Executing the automatic suction cup programming function in automatic may also be carried out through Bar code acquisition.
This option is not managed for the mix.

► Enable/disable the automatic start of the automatic suction cup programming.

A new field has been inserted into the program Header which specifies the option to run or not to run the PAV before executing the program.

1 Click on the **OPTIONS T** to open the program Header.

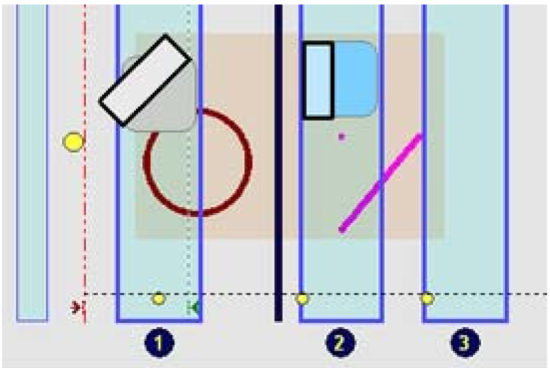
2 the **OPTIONS SETTING** window will appear. Double click on the **PAV OPTIMISATION** field and select YES from the menu options (to enable automatic start), NO (to disable automatic start).



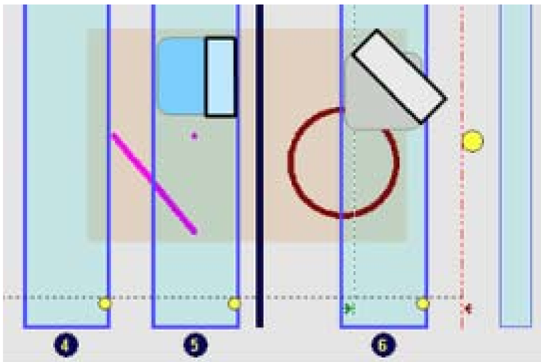
The predefined value is the one entered in the new "Run PAV" parameter in Gendata.cfg. The Options setting window can also be called up from the Machine Panel by clicking on the button inside the Header modification window.

8.7.5 Table Programming Mirror Adaptation

If you change the orientation of the work table (for example, if you swap from the front origin to the rear origin, or if you switch from area AB to area BA), the work table Editor automatically changes the orientation and position of the supports and the table element so that they mirror the original programming. For example, the following programming in area AB...



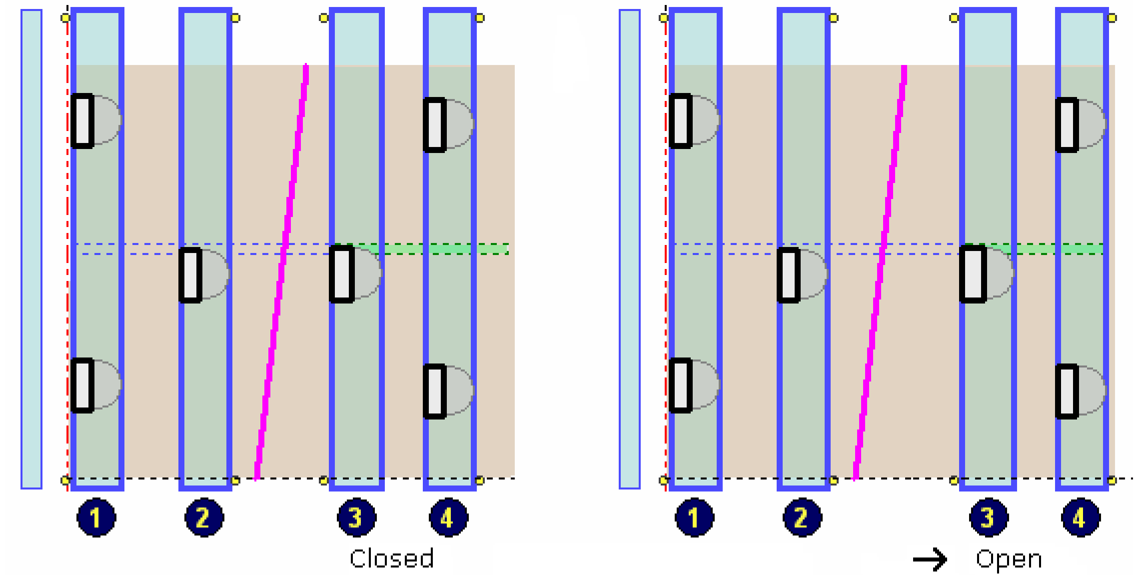
... will be transformed by mirroring if you switch to area BA...



The central suction cups and clamps are only repositioned with precise mirroring if the support or plate are symmetrical on the X axis or on the Y axis relative to the centre of the base, or if the centre of the support or the plate coincides with the centre of the base. The side clamps are simply moved to the opposite side of the crosspiece.
In the case of repositioning that does not give perfect mirroring, you can easily correct the position of the suction cup support and clamp plate (except for side clamps) with the **COMMANDS/ROTATE SUPPORT** command, which turns the support through 180 degrees.

8.7.6 Step Separators

If there is a step separator on the machine, the worktables editor can be used to display the closed and open position of the step separator using the **COMMANDS/OPEN / CLOSE THE STEP SEPARATOR** menu.



8.7.7 Prohibited Insertions

In some cases, the worktables editor prevents a support from being inserted. The prohibition is immediately displayed when moving a support with the mouse: a prohibition sign appears together with the hand icon.

If a support cannot be inserted, one of the following situations may pertain and will be signaled by the editor with a message:

1) The maximum number of available supports or table elements have been inserted.

The maximum number of available supports and table elements programmed (for each type of support) can be checked in the property toolbar screen under **Number of …**

2) You have already entered the maximum number of supports available for each crosspiece.

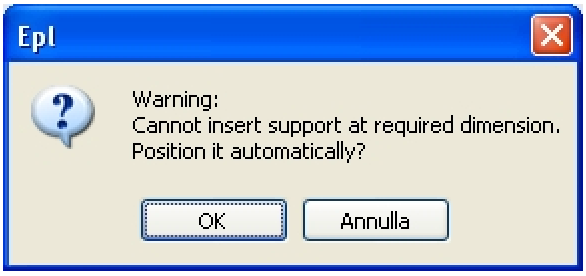
The maximum number of supports which can be inserted on each crosspiece (for each insertion position: left, central, right) is set in the configuration file supports.cfg with the **MAX. NUMBER OF SUCTION CUPS ON CROSSPIECE** parameter.

3) An attempt was made to insert the support outside the area predisposed for insertion.

4) An attempt was made to insert a support in collision with a support already inserted.

On motorized crossbeams and suction cups tables it is possible to insert a support in the same position as another support: the new support will replace the previous one.

On automatic and manual crossbeams and suction cups tables, for insertions with the keyboard or with the **CUT/COPY** and **PASTE** command, when a support is placed over an existing support, the editor will activate an *intelligent positioning* feature capable of automatically positioning the support in the first available space.



► Click on the **OK** button to allow the editor to insert the support in the first available space.

5) An attempt was made to insert a suction cup, whose base is incompatible with the table, on a multipurpose table.

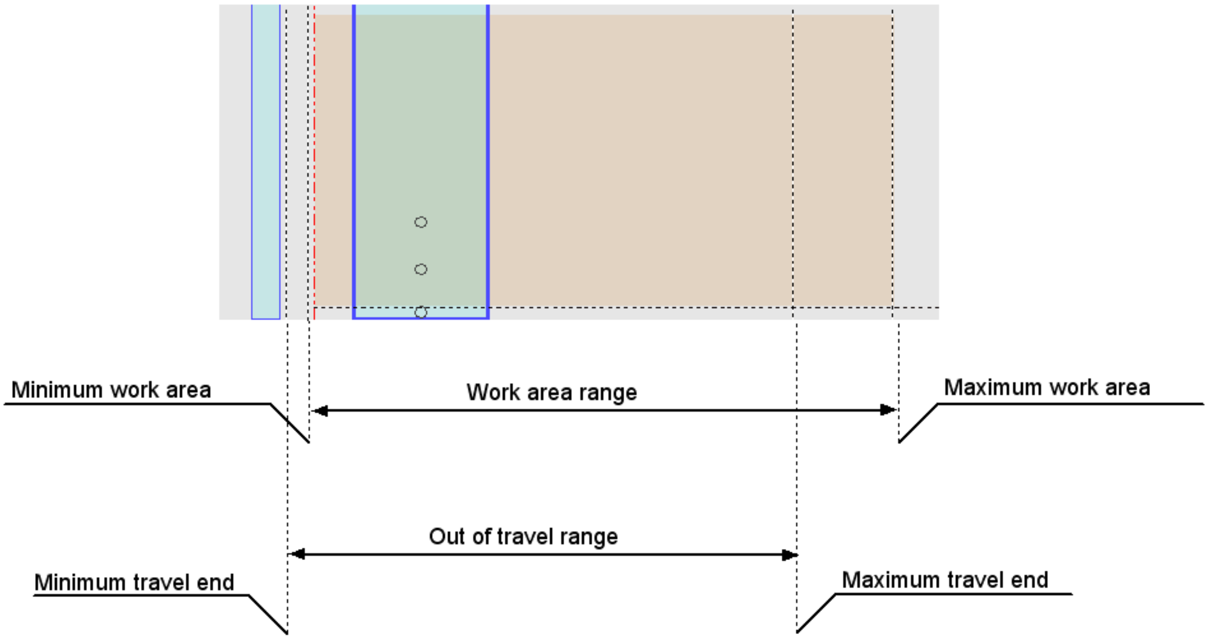
The base configuration for some types of suctions cups found in the support list may be incompatible with the multipurpose table. This means that it is physically impossible to attach the base of suction cup to the table.

8.7.8 Rules for cross bar and suction cup tables movement and specularity.

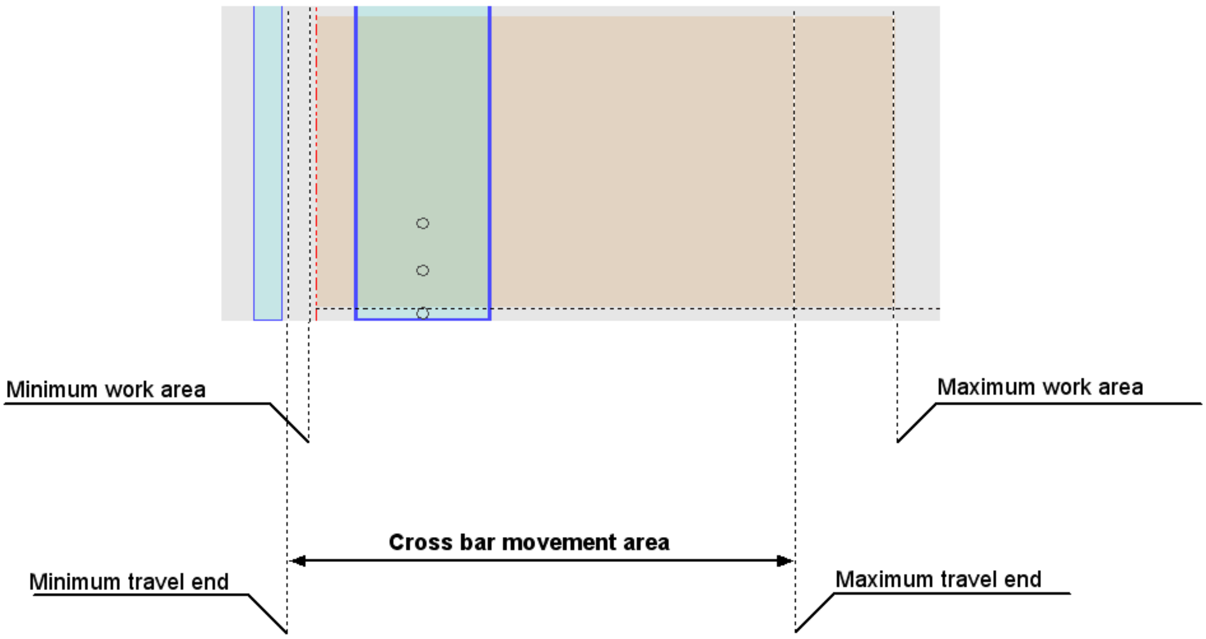
8.7.8.1 Moving the cross bars

The cross bars are moved within the work area according to the following rules:

A) **Single areas**. Within the single work areas, the cross bars may be moved within the limits of the travel end and of the work area.
During positioning, the work table Editor considers the most internal values between the travel end limits and work area limits.



B) **Double areas.** Within the double work areas, the cross bars may be moved within the limits of the travel end.



The travel end limits and the work area limits can be displayed using the cross bar displayers.

N.B.
When positioning a cross bar, the work table Editor also considers the left and right X volume.
If set, the volumes are also shown in the graphic display of the cross bar.

Motorised cross bar and suction cup table.

There is a minimum safety distance between the cross bars; during the reset phase of a layout and in the movement phase, this distance must be respected.
This space constitutes the minimum distance which must exist between one cross bar and another.

8.7.8.2 Moving the supports (motorised cross bar and suction cup table)

The suction cups, the clamps and the motorised bases have minimum and maximum travel ends; during the layout reset and movement phases, they must be respected.
These travel ends constitute the minimum and maximum positions along the Y axis of the cross bar, between which the centre of the supports must be positioned.

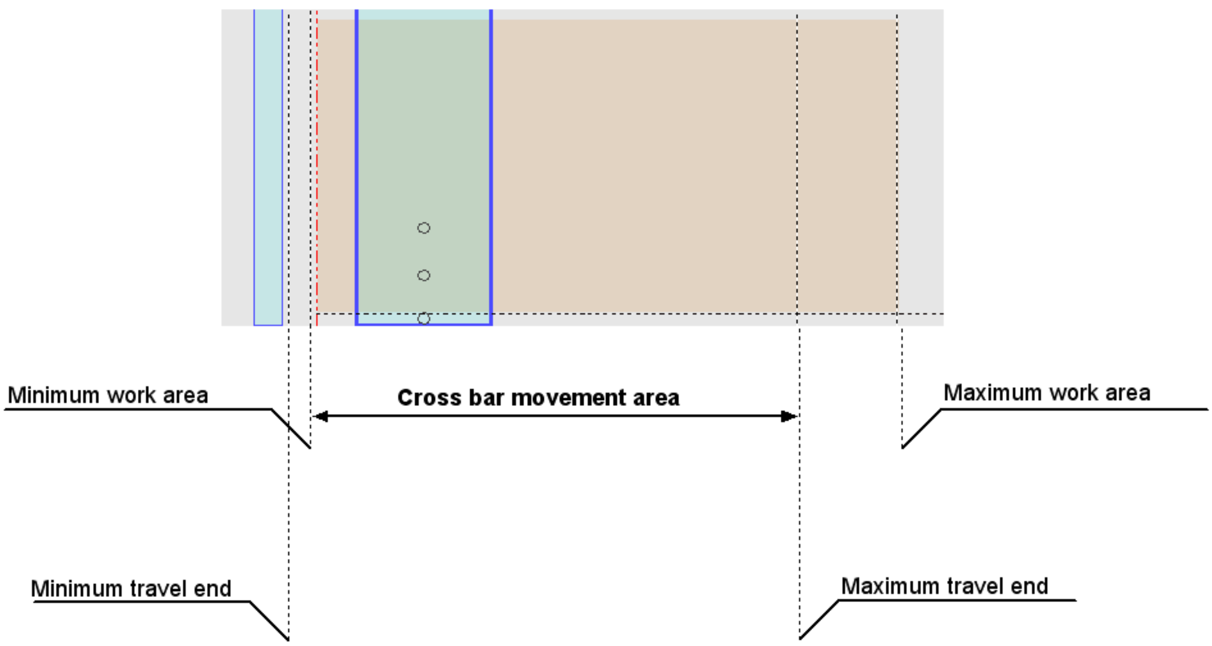
8.7.8.3 Reset of configurations on different areas

It is possible to open a configuration within a work area other than the initial one.
In this case, the work table Editor checks and signals any incompatibilities between the initial configuration and the new work area.

A) Single areas.

If we bring a configuration into a single work area, the cross bars are positioned at the same positions with respect to the initial work area.
If the position of several cross bars does not respect the limits of the work area, the work table Editor will automatically position the cross bars within the limits of the work area, signalling whether or not there are travel end limits which are not respected. If several cross bars do not respect the travel end limits of the work area, they will be positioned overlapped within the requested limits.

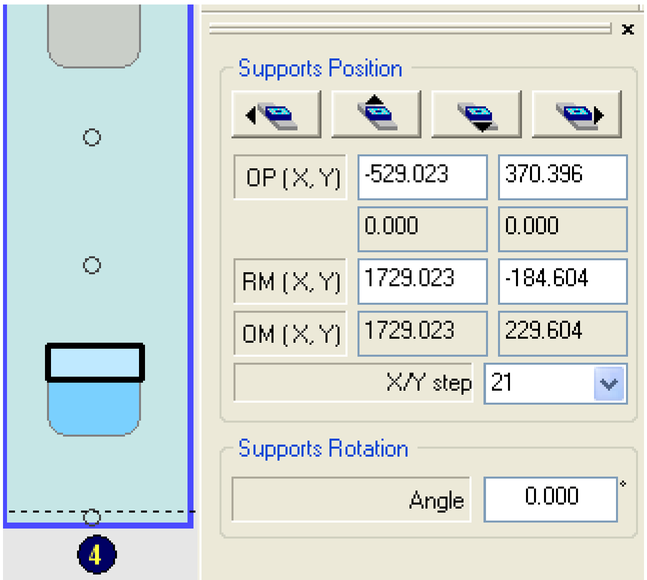
WARNING: during automatic positioning, the work table Editor considers the most internal values between the travel end limits and work area limits.



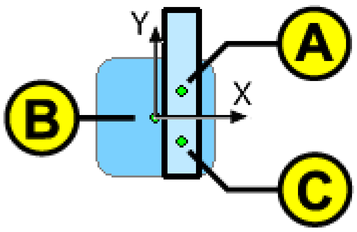
B) **Double areas.** If we bring the configuration into a double area, the cross bars are positioned at the same positions with respect to the initial work area, by signalling the presence of non respected travel end limits.

8.7.8.4 Rotating the suctions cups

Every suction cup support inserted on the base within a work area has a rotation angle set to 0°.



The rotation angle is set between the centre of rotation and the centre of the resting point regardless of type and configuration of the support.



A) Centre of suction cup resting point
B) Centre of the base
C) Centre of rotation

With positive values, the rotation is always from the X axis towards the Y axis. The direction will, therefore, be either clockwise or counter-clockwise depending on the position of the origin of the user panel (piece origin – OP).

In Libsupp.cfg these points are defined by the following keys:

[OFFX_OVR] …

[OFFY_OVR] ...
Which define the centre of rotation with respect to the centre of the Suction cup base.

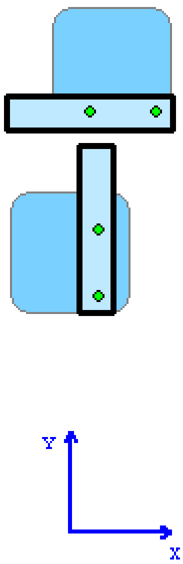
[OFFX_OVAPP] ...
[OFFY_OVAPP] ...
Which define the centre of the resting point with respect to the centre of rotation of the Suction cup. (See Appendix K)

Example of suction cup rotation.

Suction cup 1

Suction cup 2

Panel origin



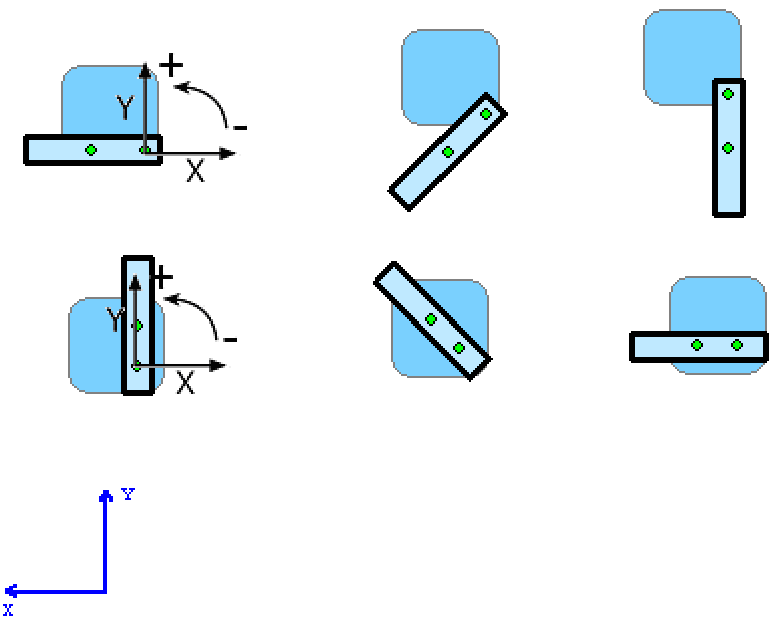
Angle of rotation

0° **45°** **90°**

Suction cup 1

Suction cup 2

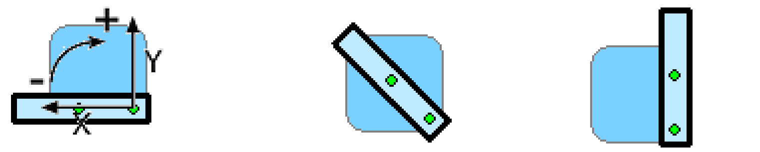
Panel origin



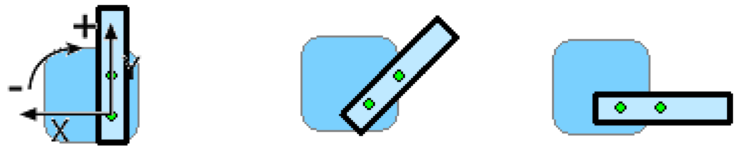
Angle of rotation

0° **45°** **90°**

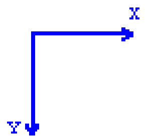
Suction cup 1



Suction cup 2



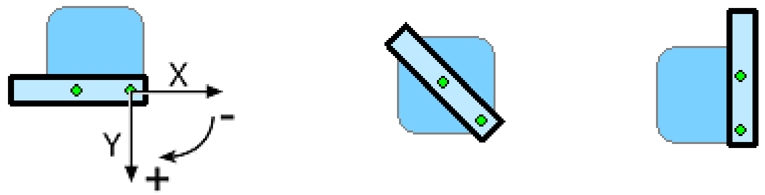
Panel origin



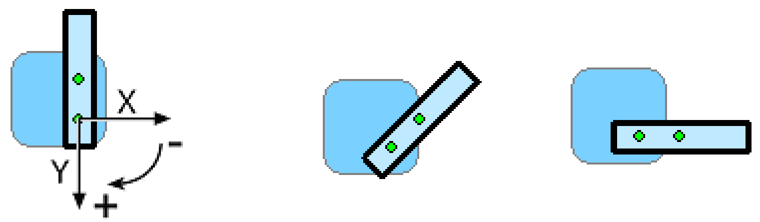
Angle of rotation

0° 45° 90°

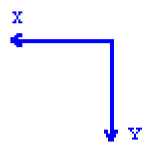
Suction cup 1



Suction cup 2



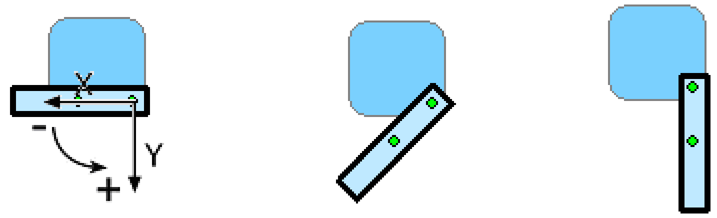
Panel origin



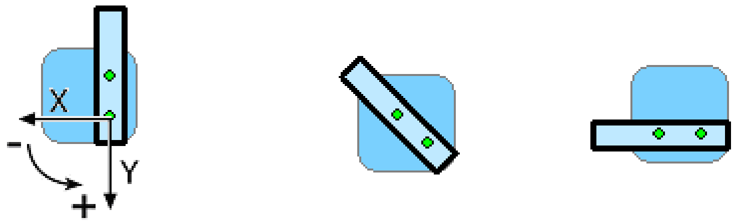
Angle of rotation

0° 45° 90°

Suction cup 1



Suction cup 2



8.7.8.5 Suction cup symmetry and specularity

8.7.8.5.1 Suction cup symmetry

The resting point of the suctions cups inserted on the table may assume the following symmetries:

- Symmetry in X (SYMX)
- Symmetry in Y (SYMY)
- Symmetry in XY (SYMXY)
- Symmetry not defined (NDEF)

The symmetry of the resting points is automatically defined by the work table Editor.
The value determined for each single Suction cup is saved in the key [SYMSUPPORT] … contained in the saved .EPL File.

The [SYMSUPPORT] key … is determined in the following ways:
In the case of suction cups with circular and rectangular bases and resting points, the position that the centre of the resting point may assume with respect to the centre of the suction cup base is considered.
To perform this operations, the following keys are tested in the Libsupp.cfg file:
[OFFX_OVR] …
[OFFY_OVR] …

(which define the centre of rotation with respect to the centre of the Suction cup base)

[OFFX_OVAPP] …
[OFFY_OVAPP] …

(which define the centre of the resting point with respect to the centre of rotation of the Suction cup)

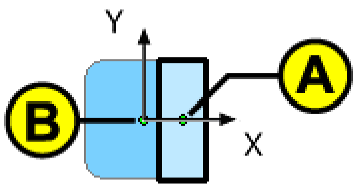
Or else, in the case of suction cups with generic type resting points, the value configured with the [SYMAPP] key in the Libsupp.cfg (See Appendix K) is considered.

For example:

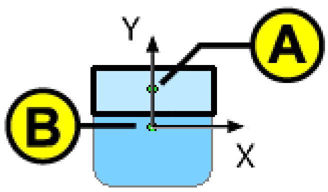
[INIT_VT]
  [GEOMBASE] …
  …
  …
  [GEOMAPP] G
  [G0] ...
  [G1] ...
  [G2] ...
  ...
...

[SIMAPP] SIMXY or **SIMX** or **SIMY** or **NDEF**
…
…
…
[END_VT]

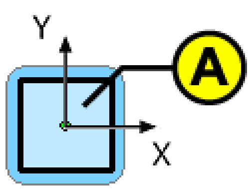
The **Symmetry in X (SYMX)** is reached when the Y of the centre of the resting point coincides with the Y of the centre of the base.

Graphic Example	Values of the keys in Libsupp.cfg	Saving the support symmetry
 A) Centre of the resting point B) Centre of the base	**[OFFY_OVR] 0** **[OFFY_OVAPP] 0** or, in the case of suction cup with generic type resting point **[SIMAPP] SIMX**	In the file (.EPL) the support in question will be saved as: **[SIMSUPPORT] SIMX**

The **Symmetry in Y (SYMY)** is reached when the X of the centre of the resting point coincides with the X of the centre of the base.

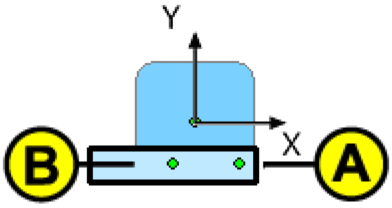
Graphic Example	Values of the keys in Libsupp.cfg	Saving the support symmetry
 A) Centre of the resting point B) Centre of the base	**[OFFX_OVR] 0** **[OFFX_OVAPP] 0** or, in the case of suction cup with generic type resting point **[SIMAPP] SIMY**	In the file (.EPL) the support in question will be saved as: **[SIMSUPPORT] SIMY**

The **Symmetry in XY (SYMXY)** is reached when the centre of the resting point coincides with the centre of the base.

Graphic Example	Values of the keys in Libsupp.cfg	Saving the support symmetry
	**[OFFX_OVR] 0** **[OFFY_OVR] 0** **[OFFX_OVAPP] 0** **[OFFY_OVAPP] 0** or, in the case of suction cup with generic type resting	In the file (.EPL) the support in question will be saved as: **[SIMSUPPORT] SIMXY**

A) Centre of the resting point point and centre of the base	[SIMAPP] SIMXY	**SIMXY**

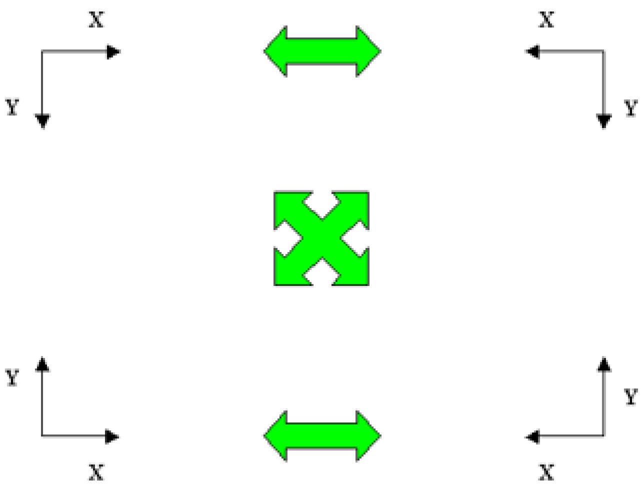
The **Undefined Symmetry (NDEF)** is reached when the centre of base does not coincide with the centre of the resting point.

Graphic Example	Values of the keys in Libsupp.cfg	Saving the support symmetry
 A) Centre of the resting point B) Centre of the base	**[OFFX_OVR]…** **[OFFY_OVR]…** **[OFFX_OVAPP]…** **[OFFY_OVAPP]…** or, in the case of suction cup with generic type resting point all different from 0 **[SIMAPP] NDEF**	In the file (.EPL) the support in question will be saved as: **[SIMSUPPORT] NDEF**

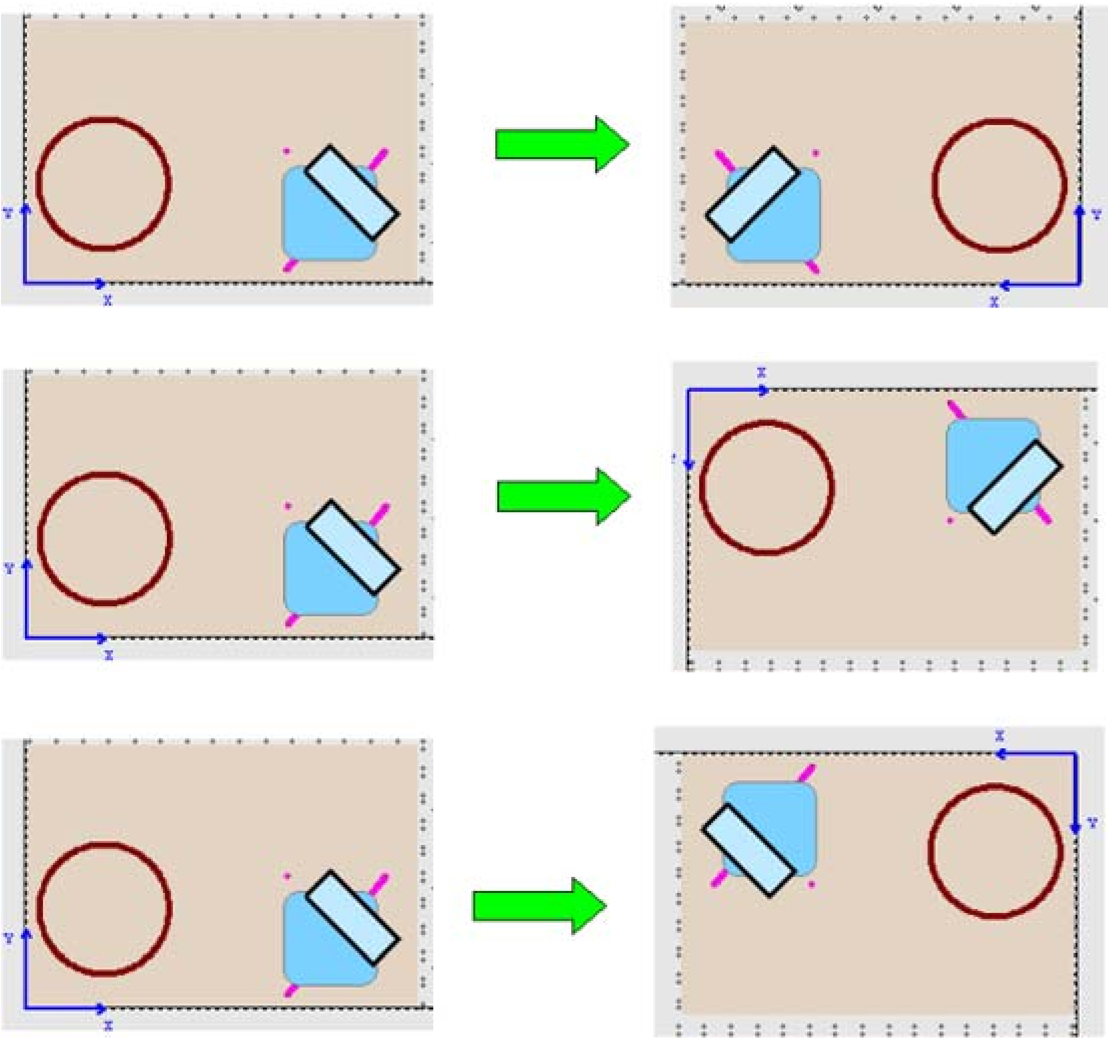
8.7.8.5.2 Suction cup specularity.

The behaviour of the suction cup supports during the passage from one work area to another is established by the type of symmetry of the resting point of the support.

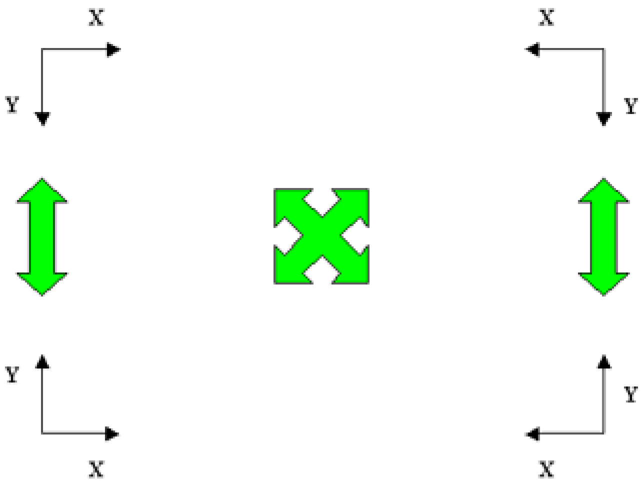
In the case of the passage from one work area to another, in which the specularity of the X axis changes, the suction cup with a resting point with **Symmetry X (SYMX)** is repositioned in an perfectly specular position.



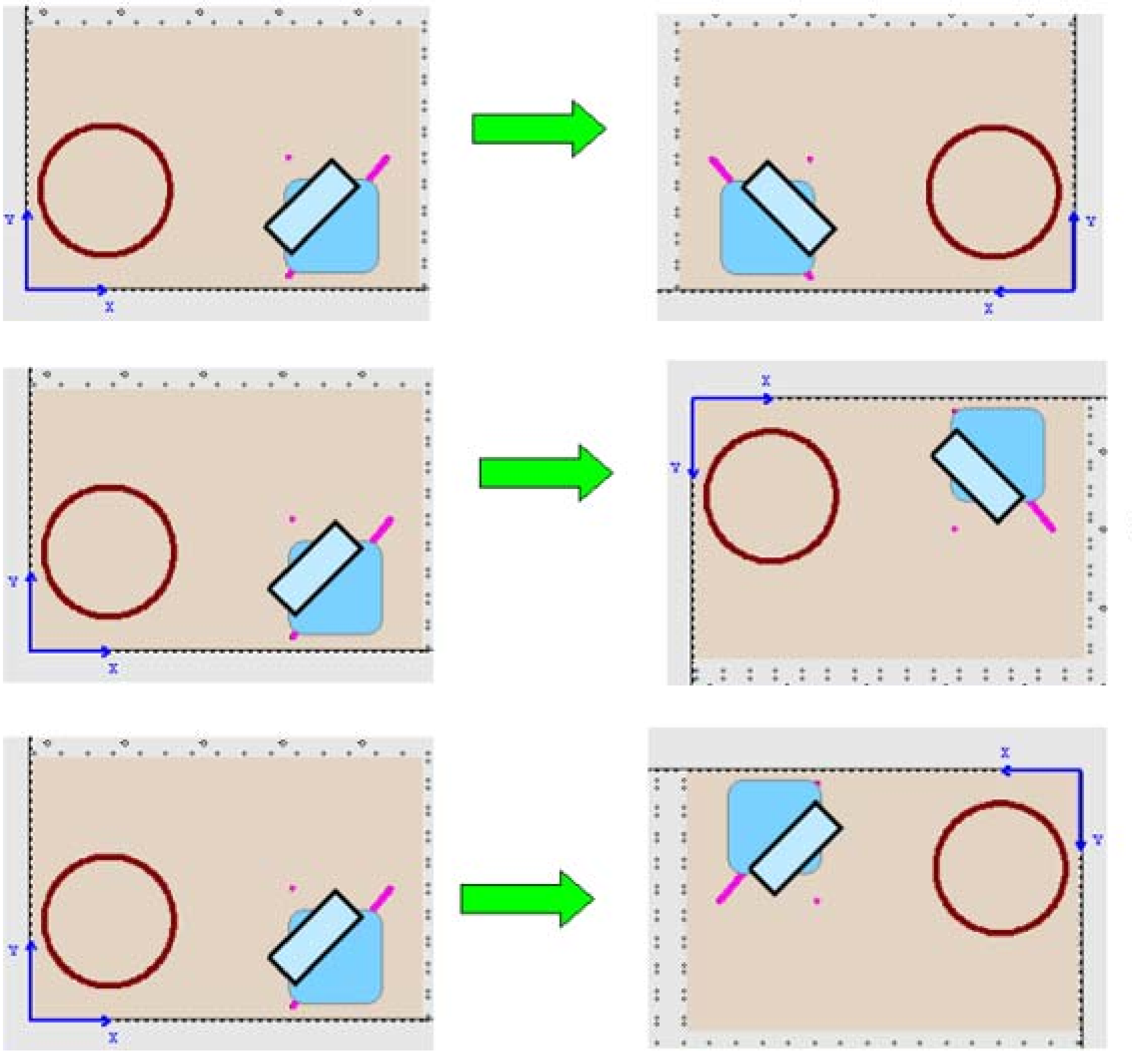
Examples.



In the case of the passage from one work area to another, in which the specularity of the Y axis changes, the suction cup with a resting point with **Symmetry Y (SYMY)** is repositioned in an perfectly specular position.

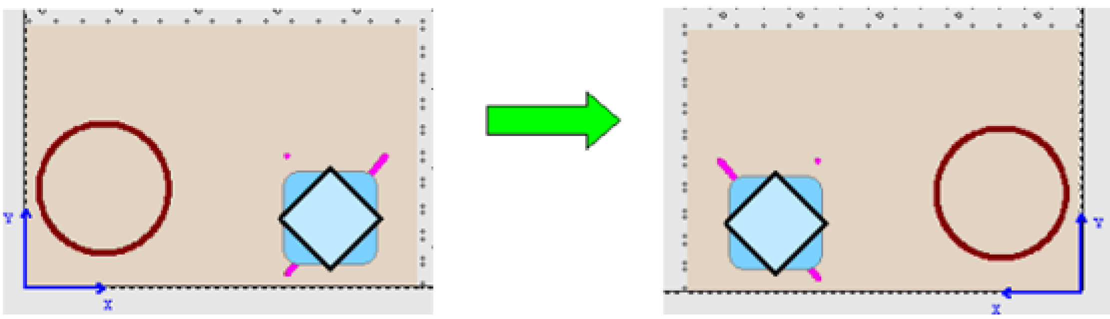


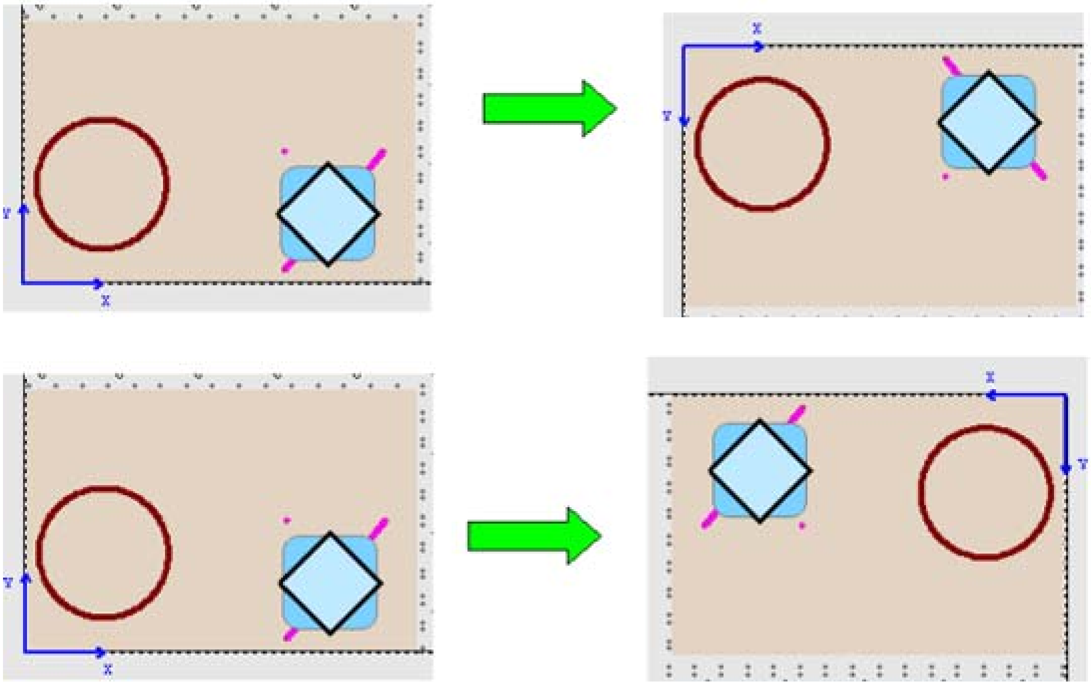
Examples.



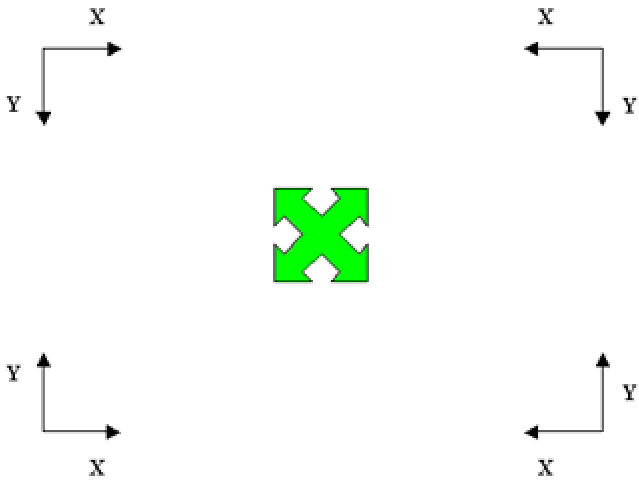
In the case of passage from a one work area to another, the suction cup with a resting point with **XY Symmetry (SYMXY)** is repositioned the way it was saved.

Examples.

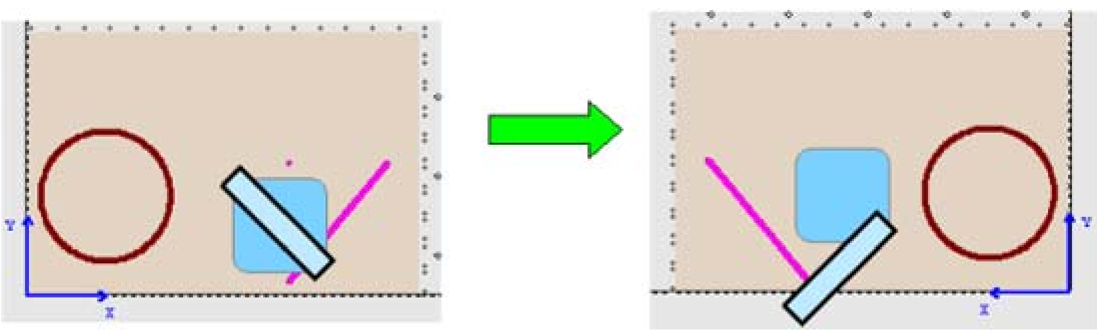


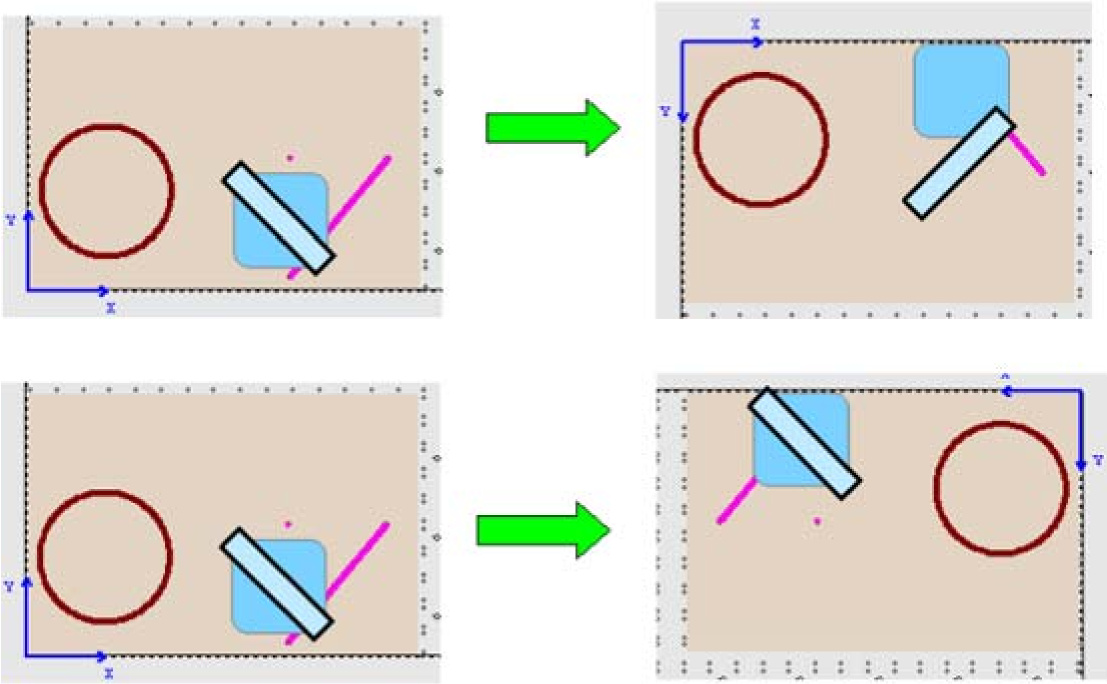


In the case of passage from one work area to another, the suction cup with a resting point with **Undefined Symmetry (NDEF)** is turned upside down based on the specularity in X and Y of the work area.



Examples.

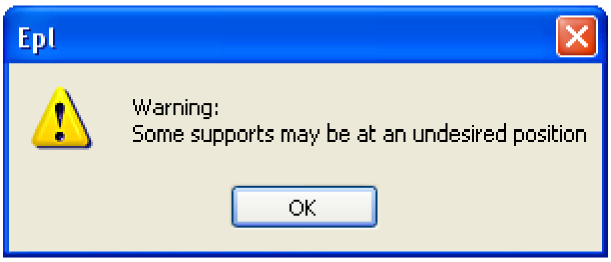




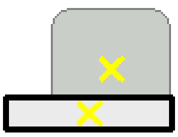
8.7.8.5.3 Unwanted positioning of the suction cup supports

It is possible to propose a suction cup support inside an area other than the initial one.
In the new configuration, it may occur that the suction cup support appears in a physically possible positioning, but which, for reasons linked to use, is inappropriate.

Upon the passage of the suction cup support from one area to another, the work table editor displays the following message:



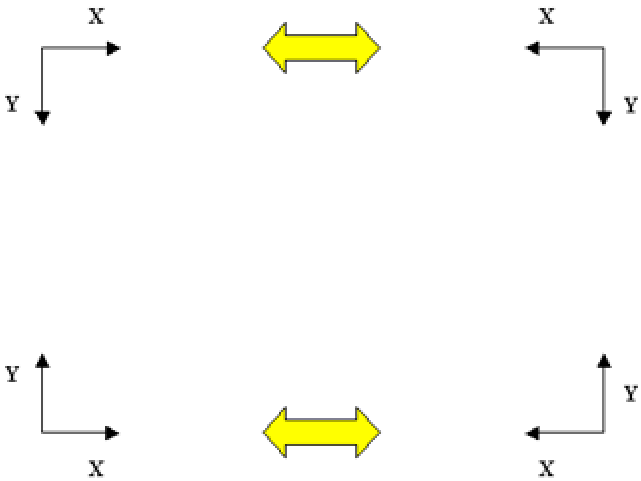
The suction cups with unwanted position will be graphically displayed with yellow x's. The suction cups remain highlighted only until the support is selected.



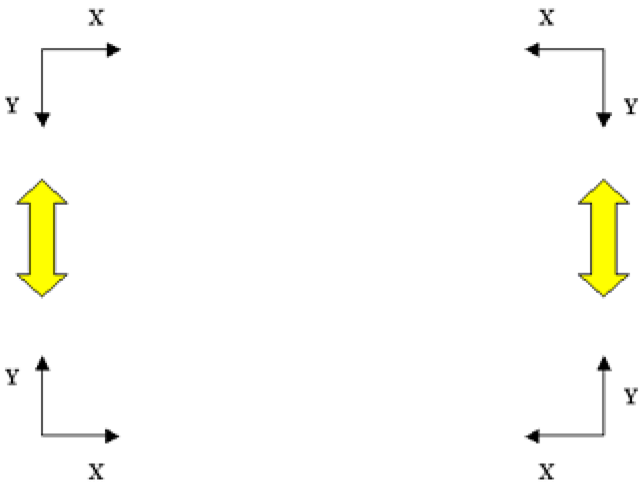
In order for the work table editor to be able to manage the unwanted positioning of the suction cups, the **[NONSPEC]** key must be configured inside the Libsupp.cfg. (See Appendix K).

The **[NONSPEC]** key may assume the following values:

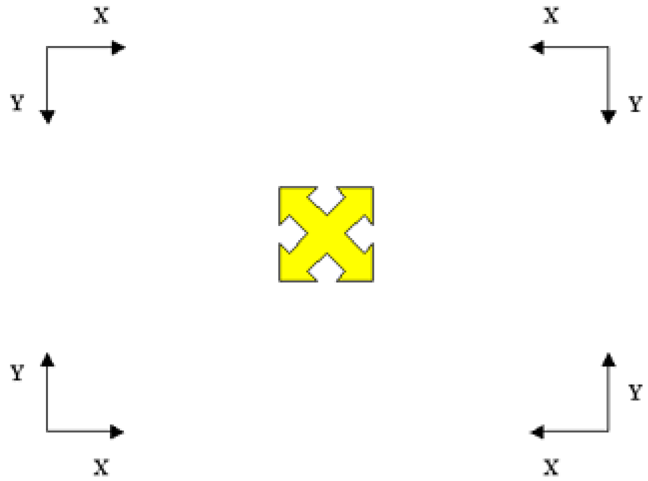
[NONSPEC]SYMX_ : in the passage from one are to another, if the specularity in X of the area changes, the suction cup will be marked as a potential unwanted positioning.



[NONSPEC]SYMY_: in the passage from one are to another, if the specularity in Y of the area changes, the suction cup will be marked as a potential unwanted positioning.



[NONSPEC]SYMXY: in the passage from one are to another, if the specularity in X and in Y of the area changes, the suction cup will be marked as a potential unwanted positioning.



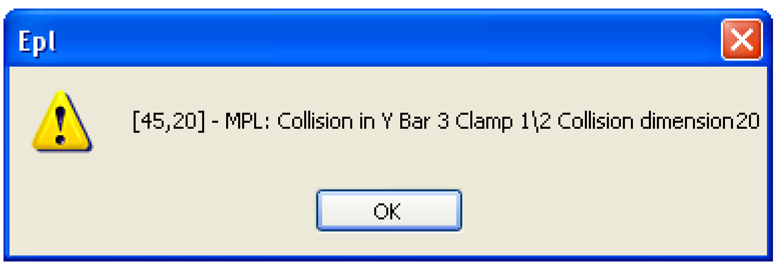
In addition to the individual values it is also possible to configure the combinations of the values that the [NONSPEC] key can assume, by placing the single values followed by a comma “,” (for example
[NONSPEC] SYMX_,SYIMY_) in these cases, the final effect is given by the sum of the single values.

8.7.9 Cycle verification (motorised cross bar and suction cup table)

Cycle verification is a function which controls the positioning inside the motorised table and verifies the presence of errors in the Epl (work table editor) layout before carrying out the Machine Panel program.

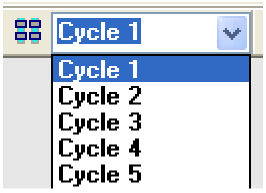
Click on the button with the left mouse button to start the function.

In the case of errors, they are signalled as shown in the following example:



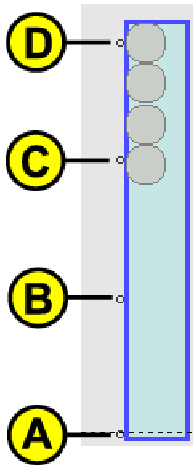
If there are instructions presents, of the PB type, it is possible to have the supports positioned (cross bars, suction cups and clamps) using the cycle verification function, at the same positions given in the PB instruction.

The function may also be carried out for each machining cycle in the piece program (pgm).



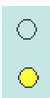
8.7.10 Bottom stops

For each cross bar, up to 4 bottom stops can be configured.



A) Row 1 of stops
B) Row 2 of stops
C) Row 3 of stops
D) Row 4 of stops

Each stop may assume three positions: left (LH), centre (C), right (RH).
All of the configured stops are set, by default, as non active and are graphically displayed on the cross bar at the specified position.



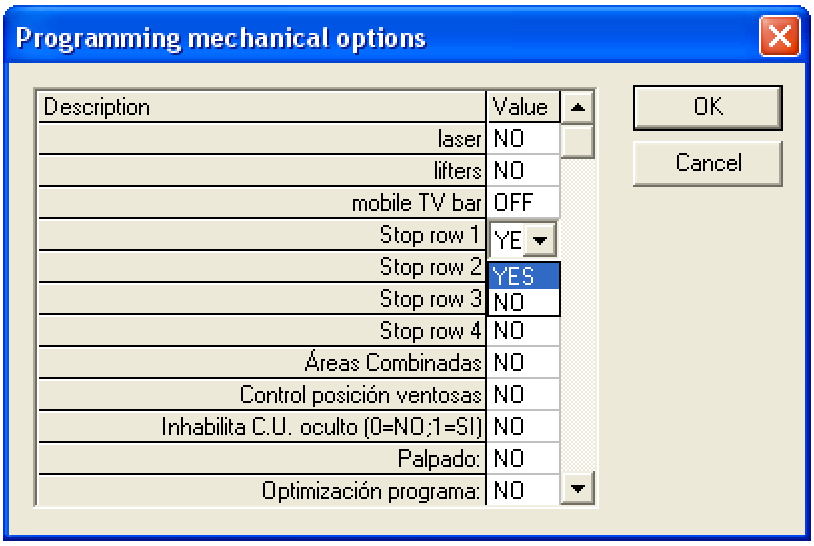
Non active stop
Active stop

All of the active stops are considered in the collision checks.
The non active side stops are considered in the collision checks.

► Activate a row of stops.

1 Click on the **OPTIONS T** button to open the program Header

2 The Option setting window will appear. Double click on the field **ROW OF STOPS..** to enable and select **YES** from the option menu (to enable the row of stops) **NO** (to disable the row of stops).



8.8 Files Description

In the ASCII file format of the worktables editor there is a series of sections defined by [*Section*] … [*END_Section*], each of which represents the programming of a particular object of the crossbeams and suction cups table or of the multipurpose table. Inside each section there is data which characterises the programming of each object: [*DatumName*]   *Value*.

Example of an ASCII file for crossbeams and suction cups table.

[TIPOPIANO]	TV	*Type of work table.*
[OLD_SPECULARE_X]	0	*Specularity value of X upon saving.*
[OLD_SPECULARE_y]	0	*Specularity value of Y upon saving.*
[TRAVERSA]		
[X]	300.000	*Position referred to the OP origin (="panel origin") at the centre of the crossbeam.*
[E]	1	*Activation status of the crossbeam (cf. parameter E of the old VROW). Enablement is possible using the displayers.*
[PAV_NUMTRAV]	1	*Number of the crossbeam from the algorithm of the automatic programming of the suction cups.*
[PAV_ENABLE]	1	*Activation status of the crossbeam during the automatic programming of the suction cups.*
[PAV_STEP]	0.000	*Distance between two suction cups on the same crossbeam during the automatic programming of the suction cups.*
[PAV_MAXNUMVENT]	3	*Maximum number of suction cups which can be inserted on the same crossbeam during the automatic programming of the suction cups.*
[BAC]		
[NAME]	Name	*Name of the load lifting bar (or the lifting bar).*
[POS]	SX	*Position on the crossbeam (SX="LH", DX="RH").*
[END_BAC]		
[VENTOSA]		
[NAME]	Name	*Name of the suction cup.*
[X]	300.000	*Position referred to the OP origin (="panel origin") on the centre X of the base.*
[Y]	134.765	*Position referred to the OP origin on the centre Y of the base.*
[XAPP]	340.000	*Position referred to the OP origin on the centre X of the main rest.*
[YAPP]	104.765	*Position referred to the OP origin on the centre Y of the main rest.*
[XAPP2]	0.000	*Position referred to the OP origin on the centre X of the secondary rest (only if there is a secondary rest present).*
[YAPP2]	0.000	*Position referred to the OP origin on the centre Y of the secondary rest (only if there is a secondary rest present).*
[ALFAAPP]	0.000	*Position in degrees referred to the rotation angle of the main rest, with respect to its centre of rotation.*
[ALFAAPP2]	0.000	*Position in degrees referred to the rotation angle of the secondary rest, with respect to its centre of rotation (only if there is a secondary rest present).*
[ALFABASE]	0.000	*Dimension refers in degrees to the angle of rotation of the base.*
[SIMSUPPORT]	SIMX	*Type of main rest symmetry (SIMX, SIMY, SIMXY, NDEF="not defined").*
[SIMSUPPORT2]	SIMY	*Type of secondary rest symmetry (SIMX, SIMY, SIMXY, NDEF="not defined"; only if there is a secondary rest present).*
[LKEY]	1	*ID (identifier) associated with the support.*
[LKEY2]	2	*ID associated with the secondary rest (only if there is a secondary rest present).*
[END_VENTOSA]		
[MORSETTO]		
[NAME]	Name	*Clamp name.*

[Y]	111	*Position referred to the OP origin (="panel origin") of the centre Y of the base.*
[LATOINSM]	NDEF	*Side on which the clamp is inserted on the cross bar, which can be SX (="LH"), NDEF (not defined or central) or DX (="RH").*
[LKEY]	2	*ID associated with the support.*
[ALFABASE]	0.000	*Dimension refers in degrees to the angle of rotation of the base.*
[SIMSUPPORT]	SIMX	*Type of plate symmetry (SIMX, SIMY, SIMXY, NDEF="not defined").*
[END_MORSETTO]		
[END TRAVERSA]		

[BARRAM]		
[TYPE]	OFF	*Type of movable bar configured (FIXED, ON, OFF, CONTINUA="continuous").*
[X]	300.000	*Position referred to the PO origin (="panel origin") on the centre of the crossbeam.*
[BAP]		
[NAME]	Name	*Name of the piece support stop.*
[TIPO]	LATERALE	*Type of stop on the work table: LATERALE (="lateral") or CENTRALE (="central").*
[OFFSET]	23.5	*Distance referred to the OP origin (="panel origin") of the centreY of the piece support stop.*
[END_BAP]		
[END_BARRAM]		

Example of an ASCII file for a multi-purpose work table.

[TIPOPIANO]		MLTF	*Type of work table.*
[OLD_SPECULARE_X]		0	*Specularity value of X upon saving.*
[OLD_SPECULARE_y]		0	*Specularity value of Y upon saving.*
[VENTOSA]			
	[NAME]	Name	*Name of the suction cup.*
	[X]	300.000	*Position referred to the OP origin("panel origin") of the centre X of the base.*
	[Y]	134.765	*Position referred to the OP origin of the centre Y of the base.*
	[XAPP]	340.000	*Position referred to the OP origin of the centre X of the main rest.*
	[YAPP]	104.765	*Position referred to the OP origin of the centre Y of the main rest.*
	[XAPP2]	0.000	*Position referred to the OP origin on the centre X of the secondary rest (only if there is a secondary rest present).*
	[YAPP2]	0.000	*Position referred to the OP origin on the centre Y of the secondary rest (only if there is a secondary rest present).*
	[ALFAAPP]	0.000	*Position in degrees referred to the rotation angle of the rest, with respect to its centre of rotation.*
	[ALFAAPP2]	0.000	*Position in degrees referred to the rotation angle of the secondary rest, with respect to its centre of rotation (only if there is a secondary rest present).*
	[ALFABASE]	0.000	*Dimension refers in degrees to the angle of rotation of the base.*
	[SIMSUPPORT]	SIMX	*Type of main rest symmetry (SIMX, SIMY, SIMXY, NDEF="not defined").*
	[SIMSUPPORT2]	SIMY	*Type of secondary rest symmetry (SIMX, SIMY, SIMXY, NDEF="not defined"; only if there is a secondary rest present).*
	[LKEY]	1	*ID associated with the support.*
	[LKEY2]	2	*ID associated with the secondary rest (only if there is a secondary rest present).*
[END_VENTOSA]			
[MODULSET]			
	[NAME]	Name	*Name of the modulset.*
	[XM]	111	*Position referred to the OP origin (="panel origin") of the centre X of the male.*
	[YM]	222	*Position referred to the OP origin of the centre Y of the male.*
	[XF]	333	*Position referred to the OP origin of the centre X of the female.*
	[YF]	444	*Position referred to the OP origin of the centre Y of the female.*
	[ALFA]	45	*Position in degrees referred to the rotation angle of the female point with respect to the male point.*
	[LKEY]	2	*ID associated with the support.*
[END_MODULSET]			
[GUARNIZIONE]			
	[POINT]	111,222,3	*111: position of centre X of the point (with respect to OP="panel origin").* *222: position of the centre Y of the point (with respect to OP).* *3: ID associated with the support.*
	[POINT]	333,444,4	*(see above)*
	[POINT]	555,666,5	*(see above)*
	…		
[END_GUARNIZIONE]			
[BAP]			
	[NAME]	Name	*Name of the piece support stop.*
	[TIPO]	LATERALE	*Type of stop on the work table: LATERALE (="lateral") or CENTRALE (="central").*
	[OFFSET]	23.5	*Distance referred to the OP origin (="panel origin") of the centre X or Y of the piece support stop:* *- if the Type is LATERALE/CENTRALE, the offset is the centre X;* *- if the Type is bottom, the offset is the centre Y.*
[END_BAP]			

APPENDIX

APPENDIX A. Menu and Toolbars

Main Window, Programs Editor and Tooling Editor

File Menu

New	Opens a new document.
Open	Opens an existing document.
Close	Closes and open document.
Close all	Closes all open documents.
Save	Saves a document.
Save as	Saves a document with possibility to change name.
Save in pgm format	Saves the document in pgm format.
Save all	Saves all open documents.
Configuring machine	Opens the configuration parameters editor.
Print	Prints a document.
Print preview	Displays a print preview of a document (click on the **CLOSE** button to return to the document).
Set printer	Allows to set a printer for print.
Exit	Exit from Xilog Plus.

Edit Menu

Cancel	Cancels the last change.
Cut	Removes data selected from the document and stores it in Clipboard.
Copy	Copies a selection into Clipboard.
Paste	Inserts the contents of the Clipboard (cut or copied with one of the previous commands) in the insert point.
Delete	Deletes the selected data.
Move	(Graphic editor) Moves selected machining.
Search	(Text and macro editor) Searches a text.
Replace	(Text and macro editor) Searches a text and substitutes it with other specified text.
Refresh	(Graphic editor) Refreshes the graphic representation of a program.
Reinterpret	(Graphic editor) Reinterprets a program.
Property	(Graphic editor) Displays the parameters for selected machining.

View Menu

Toolbar...	Displays/hides the toolbar selected from the list.
Restore default	Restores the original view of the Xilog Plus window.

Free editor	(Text and macro editor) Passes to Free Editor mode.
Guided editor	(Text and macro editor) Passes to Guided Editor mode.
Tools diameter	(Graphic editor) Show/hide tools diameter.
References	(Graphic editor) Show/hide panel reference systems.
Program structure	(Graphic editor) Show/hide program structure.

Tools Menu

Calculator	Displays scientific calculator (see: Appendix M).
Backup	Activates the function for data backup (see: Appendix M).
Restore	Activates the restore function for backup data (see: Appendix M).
Update list of macro	Updates the icons of the instructions toolbar if a new macro is added.
Spindle and tools graphic	(Text and graphic editor) Opens the window for Operating heads graphic display.
Program graphic	(Text and macro editor) Displays the graphic representation of the program.
Parameters editor	(Graphic and mix editor) Opens the Parameters Editor.
Parameters Wizard	(Text and graphic editor) Opens the Parameters Wizard.
Crosspiece and suction cups editor	(Text and graphic editor) Opens the Worktables Editor.
Delete worktable programming	Cancel worktable programming.
Displaying "trace" file	Opens the table of messages set with the TRACE instruction.
Program optimizer	(Text and graphic editor) Activated the programs optimizing function.
List of programs	(Mix) Opens the window for inserting a program into a mix.
Zoom +	(Graphic editor) Enlarges the area selected by the cursor (drag with the zoom rectangle with the mouse over the point to be enlarged and press the **[ENTER]** key).
Zoom –	(Graphic editor) Cancels enlargement zoom.
OK Zoom	(Graphic editor) Confirm zoom.
Esc Zoom	(Graphic editor) Cancel zoom.
Change graphic view	(Graphic editor) Edit type of graphic display.
Options	(Guided text, mix and macro editor). Open the table to set Header parameter T.
Block type	(Guided text, mix and macro editor). Open the table to set Header parameter V.

Options Menu

International settings	Opens the window for setting the language and unit of measure (millimeter or inches) used by Xilog Plus.
Date/Time	Opens the window for setting the date and time.

Text editor	Sets the text editing environment for a program.
Graphic editor	Sets the graphic editing environment for a program.
Macro editor	Sets the editing environment for macros.
Advanced	Opens the window for setting some features of the Xilog Plus interface (background, window, instructions tables).

Windows Menu

Cascade	Overlap open windows
Tile horizontally	Arranges open windows.
Arrange icons	Arranges icons along the bottom part of the window.

Help Menu

Guide topics	Opens the window with on-line Help. Can be activated by pressing key **[F1]**: help topics will appear that are available for the element selected.
About Xilog Plus	Opens the window for information of the Xilog Plus editor (copyright note and version number).

Tool Bar

	Opens a new document.
	Opens an existing document.
	Saves document.
	Removes the selected data from the document and stores in the Clipboard.
	Copies selection onto Clipboard.
	Inserts the contents of the Clipboard (cut or copies using one of the previous commands) into the insert point.
	Cancel last operation.
	Print active document.
	Opens the configuration parameters editor.
	Displays scientific calculator (see: Appendix M).
	Activates data backup function (see: Appendix M).
	Activates restore backup data function (see: Appendix M).
	Opens window with information on the Xilog Plus editor (copyright note and version number).
	Activates search of on-line Help by topic. The mouse pointer turns into an arrow and a question mark: by clicking on the point desired the relative help topic will be displayed. The function can also be activated with a combination of the keys **[SHIFT]** + **[F1]**.

Function Bar

	(Text and Graphic Editor) Opens the window for Operating heads graphic display.

	Loads graphic display window.
	(Graphic and mix Editor) Opens the Parameters Editor.
	(Text and Graphic Editor) Opens the Parameters Wizard.
	(Text and graphic and Twin editor) Opens the work table Editor.
	(Text and graphic and Twin editor) Deletes the work table programming. It the table has not been programmed for the current program, this command is disabled.
	(Mix) Opens window for inserting a program into a mix.
	(Text and macro Editor) Inserts/removes a new line in a program.
	(Guided text, mix and macro editor). Open the table to set Header parameter T.
	(Guided text, mix and macro editor). Open the table to set Header parameter V.

If working with DXF files, the following commands are active:

	Opens window for setting general data.
	Opens window for setting graphic mode.
	Joins disjointed profiles (auto-join).
	Opens window for setting entry into profile.
	Performs inversion on profile.
	Displays order to geometries.

Program data bar

	Opens the Header modification window.
	(Guided text, mix and macro editor). Open the table to set Header parameter T.
	(Guided text, mix and macro editor). Open the table to set Header parameter V.
	(Graphic and mix Editor) Opens the Parameters Editor.
	(Text and Graphic Editor) Opens the Parameters Wizard.

Graphic Tool Bar (Graphic Editor)

	Enlarges the area selected by the cursor (drag with the zoom rectangle with the mouse over the point to be enlarged and press the **[ENTER]** key).
	Cancels enlargement zoom.
	Changes display of graphic face.
	Displays the radius correction for the tool.
	Displays/hides the panel reference system.
	Displays/hides the program structure.

Modal Data Bar (Graphic Editor)

	Activates face 1.
	Activates face 2.
	Activates face 3.
	Activates face 4.
	Activates face 5.
	Toll correction nil.
	Right correction.
	Left correction
	In depth correction
	Incremental disabled.
	Incremental in X.
	Incremental in Y.
	Specular disabled.
	Specular in X.
	Specular in Y
	Original G2/G3 direction.
	Invert G2/G3 direction.

Status Bar

The status bar is displayed in the lower part of the Xilog Plus window.
This area allows the operator to always have machine status under control, since it immediately reflects any changes in machine status and functions, providing the following information:

- contextual help for the operations that are being performed (**area to the left**);
- interface status (**area in the center**);
- *C.N.* status (**area in the center**);
- status of physical keyboard (**area to the right**);

The **area to the left** of the status bar describes the actions performed by the items on the menu and the buttons on the toolbar, when the mouse is over them.

The **area in the center** of the status bar provided the following information:

mm in	Highlights the unit measure (mm = millimeters; in. = inches).
	Graphic Editor active.
	Text Editor active.
	Macro Editor active.

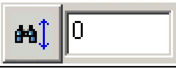
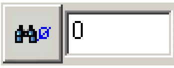
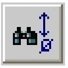
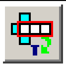
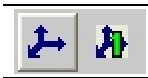
The **area to the right** of the status bar indicates whether the following keys are locked:

MA	The Caps Lock key is locked.

NUM	The Num Lock key is locked.
BS	The Scroll Lock key is locked.

Operating Heads Graphic Display

Tool Bar

	Select single tool.
	Deselect single tool.
	Search by tool length.
	Search by tool diameter.
	Search by lenght and diameter.
	Button for viewing the current tooling file.
	Button for inserting fixed and external tools for the programs editor.
	NC axis unit (for Ergon).

Displaying Graphic

File Menu

Close	Exit displaying graphic.

View Menu

Correcting tool	Enables/disables view of tool correction.
Other type of graphic	Displays projection on sides (only if graphic is relative to one of the single faces).
Side surfaces graphic	Displays single faces.

Zoom Menu

Zoom +.	Enlarges area selected by the cursor (drag the zoom rectangle with the mouse over the point to be enlarged and press the **[ENTER]** key.
Zoom -	Cancel zoom enlargement.
OK zoom	Enable zoom.
Cancel zoom	Cancel zoom.

Options Menu

Refresh	Refreshes graphic representation.
Stop	Blocks any infinite cycles (see: chapter 6.2.4 - Producing and executing a cycle).

Environment for Importing DXF Files

When importing a file in DXF format into Xilog Plus, the **Edit** menu provides some specific commands listed below:

Edit Menu

Auto Join	Joins disjointed profiles (auto-join).
Input/hole data	Opens the window for setting entry into profile.
Inverting profile route	Inverts the direction on the profile.
Reference system	Opens window for setting graphic mode.
General settings	Opens window for setting general data.
Geometry order	Displays the arrangement of geometries.
OK next geometry	Passes to next geometry.
Exit order	Exit display of the arrangement of geometries.

Worktables Editor

File Menu

Open layout...		Opens and existing program file.
Save layout		Stores the table program with the machining program.
Save layout with name …		Stores the table program in a different file from that of the machining program.
Save and close		Stores the table program together with the machining program and closes editor.
Export/Excel sheet		Exports table program in an Excel calculation sheet.
Export/XML Layout		Exports table program in an XML file.
Print...		Prints active table program.
Print preview		Provides a print previews.
Set up printer...		Changes printer settings.
Exit		Closes the editor.

Modify Menu

Undo		Deletes the last action.
Cut		Cuts the selected support.
Copy		Copies selected support.
Paste		Opens the screen to insert the cut or copied support with one of the two previous commands.
Clear		Deletes the selected support.

View Menu

Toolbar...		Shows/hides the toolbar selected from the list.
Restore original Toolbars		Reset original toolbar arrangement.
Show/Hide PGM Origin		Shows/hide the panel reference system.
Show/Hide Machine Origin		Shows/hide the machine reference system.
Show/Hide crosspiece display		Show/hide crosspiece diplays.
Panel to front		Shows the panel in the foreground.
Activate Light Drawing		Shows the work table in simplified graphics.
Full screen		Activates/deactivates the full screen view.

Select Menu

Previous support		Selects previous support.
Next support		Selects next support.

Commands Menu

Move support left	Moves selected support to the left (if the distance has been set in the **X/Y step** box of the supports movement toolbar).
Move support right	Moves selected support to the right (if the distance has been set in the **X/Y step** box of the supports movement toolbar).
Move support up	Moves selected support upwards (if the distance has been set in the **X/Y step** box of the supports movement toolbar).
Move support down	Moves selected support downwards (if the distance has been set in the **X/Y step** box of the supports movement toolbar).
Insert support	Makes it possible to insert a selected support in the list of supports by entering the information using the keyboard.
Rotate support	Activates/deactivates the rotation feature for the selected support.
Rotate support	Rotates the selected support 180°.
Open/close the step separator	Shows the open/closed position of the step separator.

Tools Menu

Anti-collision control	Checks whether collisions may occur.
Suction cups auto. prog.	Automatically positions the suction cups (only for crossbeam tables and suction cups).
Zoom/Real time zoom	Allows the work table image to be enlarged or reduced: - click on the left mouse button: the image is enlarged; - click on the right mouse button: the image is reduced. The zoom is centered on the clicking point.
Zoom/Zoom area	Allows a rectangle to be drawn by dragging the mouse over the drawing: when the mouse is released, the area inside the rectangle is enlarged.
Zoom/Workpiece program zoom	Enlarges the area occupied by the panel.
Zoom/Restore original zoom	Resets the original size of the drawing.
Position seal	Activates/deactivates the feature that inserts the seals (only for the multipurpose table).
Joins adjacent polylines	Joins separate seal lines that area close to each other into one sole profile (only for the multipurpose table).

Menu ?

Help topics	Opens the on-line Help screen.
About EPL…	Displays information on the program, the number of the version, and copyright information.

Toolbar

	Opens and existing program file.

	Stores a table program together with a machining program.
	Stores a table program together with a machining program and closes the editor.
	Shows/hides panel reference system.
	Shows/hides machine reference system.
	Show/hide crosspiece diplays.
	Shops panel in foreground.
	Shows work table in simplified graphics.
	Activates/deactivates full screen display.
	Allows the work table image to be enlarged or reduced: - click on the left mouse button: the image is enlarged; - click on the right mouse button: the image is reduced. The zoom is centered on the clicking point.
	Allows a rectangle to be drawn by dragging the mouse over the drawing: when the mouse is released, the area inside the rectangle is enlarged.
	Enlarges the area occupied by the panel.
	Resets the original size of the drawing.
	Checks for possible collisions.
	Automatically positions suction cups (only for crossbeam tables and suction cups).
	Prints active table program.
	Displays information on the program, the number of the version, and copyright information.
	Enables the search for topics in the on-line Help guide. The cursor turns into an arrow with a question mark: by clicking on the desired point, the relative guide topic will be displayed. The feature may also be activated using the key combination **[SHIFT]**+**[F1]**.
	Shows the open/closed position of the step separator.

Special Toolbar

	Cuts the selected support.
	Copies the selected support.
	Opens the screen to insert support that was cut or copied using one of the previous commands.
	Makes it possible to insert a selected support in the list of supports by entering the information from the keyboard.
	Deletes selected support.
	Activates/deactivates rotation feature for selected support.
	Rotates the selected support 180°.
	Selects previous support.
	Selects next support.
	Activates/deactivates the feature that inserts the seals (only for the multipurpose table).
	Joins separate seal lines that area close to each other into one sole profile (only

	for the multipurpose table).
	Displays the properties screen for selected panel or support.

Status Bar

The **area to the left,** when the moving over them with the mouse, the status toolbar describes the actions of the menu items and buttons on the toolbar.

To the **right** there are boxes that show the coordinates for the point where the cursor is located compared to the panel starting point (OP) and machine starting point (OM).

To the **extreme right** of the toolbar there are three boxes that indicated whether the following keys are blocked.

MA The Caps Lock key is blocked.
NUM The Num Lock key is blocked.
BS The Scroll Lock key is blocked.

APPENDIX B. Variables and Expressions

Variables

In programs, subprograms, and macros it is possible to define up to a maximum f 256 variables (see: instruction L).
Variables are objects used to contain numerical values. These values may be absolute numbers, or the result of an arithmetic or logical expression. The values stores go from 1.7E-308 to 1.7E+308. Each variable has a unique name, that must be selected according to the following rules:

- must not be longer than 15 characters;
- the characters can be letters of the alphabet (lowercase and/or uppercase), numbers and the characters '_' (underscore);
- the first character cannot be a number.

Some variables are reserved and can be used to read the dimensions of the piece, the work area, and represent *pi* (these variables cannot be written):

DX	Dimension in X.
DY	Dimension in Y.
DZ	Dimension in Z.
BX	Workpiece movement in X relative to the stop.
BY	Workpiece movement in Y relative to the stop.
BZ	Workpiece movement in Z relative to the stop.
FLD	Machining area: 1=A, 2=B, 3=C, 4=D 12=AB, 21=BA, .., 41=DA 101=E, 102=F, 103=G, 104=H 112=EF, 121=FE, ..., 141=HE 201=I, 202=J, 203=K, 204=L 212=IJ, 221=JI, ..., 241=LI 301=M, 302=N, 303=O, 304=P 312=MN, 321=NM, ..., 341=PM 900 = area 00, 901 = area 01 910 = area 10, 911 = area 11
PI	Pi-greco.

WARNING

An undefined variable assumes the value of 0 if used in expressions, but assumes "undefined" is used as quota.

Expressions

Expressions are rules for reading the value of the operands and calculating new values by the application of operators.

Operands may be fixed values (numbers) and names of variables. The operators are the conventional operators, plus many others to facilitate the construction of arithmetic and logical expressions. Arithmetic expressions provide a numerical result. Logical expressions give a *True* or *False* logical result. Xilog Plus considers the value 0 (zero) as corresponding to False and each other value as corresponding to True; Xilog Plus generates the value 1 as True. The rules for the composition of the expressions group the operators into priority classes. Unary operators have maximum priority, then in the order, multiplication, addition, comparison and logical.

The table below contains the list of the operators grouped by priority. The lines separate the levels. The maximum priority is at the top.

-	Minus (unary).
+	Plus (unary).
DEF	Parameter/variable definition control (0= not defined,1= defined).
NDEF	Parameter/variable definition control (0= defined, not defined).
ABS	Absolute value.
ACOS	Arc cosine.
ASIN	Arc sine.
ATAN	Arc tangent.
COS	Cosine.
SIN	Sine.
TAN	Tangent.
RD	Rounds a number down, to the nearest integer.
RU	Rounds a number up, to the nearest integer.
EXP	Exponent.
LN	Natural logarithm.
LOG	Base-10 logarithm.
SQR	Square root.
TLRAD	Max. tool radius.
TLURAD	Working tool radius.
TLLEN	Tool length.
TLULEN	Working tool length.
TLROT	Direction of rotation of tool (0=no rotation, 1=+, 2=-).
TLOPPSTX	Tool counter position in X.
TLOPPSTY	Tool counter position in Y.
NXTNUMTOTAL (a,n)	Returns the number of the next tool to **n** (n=1, ...96, 101...196, ..., 801, ..., 896) which belongs to set **a** (a=1...999). If n=0, 100, 200, ..., 800 returns the number of the first tool belonging to set **a**. If the value returned is 0 there are no further tools belonging to set **a**.
_ORGVAL(1)	Returns X origin value.
_ORGVAL(2)	Returns Y origin value.
_ORGVAL(3)	Returns Z origin value.
HEADER(n)	Returns the data specified in the program Header.
HEADER(1)	Returns the value of the DX parameter.
HEADER(2)	Returns the value of the DY parameter.
HEADER(3)	Returns the value of the DZ parameter.
HEADER(4)	Returns the value of the - parameter.
HEADER(5)	Returns the value of the C parameter.

HEADER(6)	Returns the value of the T parameter.
HEADER(7)	Returns the value of the R parameter.
HEADER(8)	Returns the value of the * parameter.
HEADER(9)	Returns the value of the V parameter.
HEADER(10)	Reserved.
HEADER(11)	Returns the value of the BX parameter.
HEADER(12)	Returns the value of the BY parameter.
HEADER(13)	Returns the value of the BZ parameter.
RETV ("NomeDiVariabile", Expression)	May be used in a sub-program or macro to return a value to the sub-program/macro calling. The value consisting of the Expression is copied to the NomeDiVariabile variable belonging to the sub-program/macro calling. NomeDiVariabile may be a standard variable (L), a parameter (PAR), an alias (D). The operator returns 0 if the NomeDiVariabile variable is not defined in the sub-program/macro calling or if there is no sub-program/macro calling; otherwise it is 1. Example: ... L Pos = 1200 L Res = RET("Alfa", Pos*1000) ... If it exists, the Alfa variable of the sub-program/macro calling takes the value 1200000. The RETV operator may be applied only once for each variable, irrespective of the sub-program/macro in which it is applied to the variable. The ambient variables cannot be edited by the RETV operator under any circumstances.
TLSERIES	Tool series (0=none, 1=X, 2=Y).
OPROG	Reserved.
FCKP(1)	Returns the value of the active face (F=1 o 2 o 3 o 4 o 5).
FCKP(2)	Programmed correction C returns (0=no,1,2,3,13,23,31,32) (that can differ from the active one if the first is programmed at half profile).
FCKP(3)	Returns the value of active K incremental (0=no, 1=X, 2=Y, 3=X+Y).
FCKP(4)	Returns the value of active P specular (0=no, 1=X, 2=Y, 3=X+Y, 10=G2G3, 11=X+G2G3, 12=Y+G2G3, 13=X+Y+G2G3).
FMAC	Reserved.
TMAC	Reserved.
ZMAC	Reserved.
TOOL	Obtains data from tooling file specified in current program:
TOOL(n, 1)	Returns the type (0=N.D., 1=P, 2=F, 3=D, 4=T, 5=S) of tool n (n=1,…, 96,101,…,196,…,801,…,896).
TOOL(n, 2)	Returns subtype (0=N.P., 1=L, 2=P, 3=S).
TOOL(n, 3)	Returns the number store.

TOOL(n, 4)	Returns the type of store.
TOOL(n, 5)	Returns positions store.
TOOL(n, 6)	Returns number of opposite tool in X.
TOOL(n, 7)	Returns number of opposite tool in Y.
TOOL(n, 8)	Returns series (0=N.D., 1=axis X, 2=axis Y).
TOOL(n, 9)	Returns radius.
TOOL(n, 10)	Returns working radius.
TOOL(n, 11)	Returns the length/blade radius.
TOOL(n, 12)	Returns working length.
TOOL(n, 13)	Returns direction of rotation (0=N.D., 1=+, 2=-).
TOOL(n, 14)	Returns work direction (0=N.D., 1=+, 2=-).
TOOL(n, 15)	Returns number total.
TOOL(n, 16)	Returns work face (0=all,1=F1,..,5=F5).
TOOL(n, 17)	Returns height countersink.
TOOL(n, 18)	Returns standard rotation speed.
TOOL(n, 19)	Returns maximum rotation speed.
TOOL(n, 20)	Returns standard speed.
TOOL(n, 21)	Returns maximum speed.
TOOL(n, 22)	Returns G0/B speed.
TOOL(n, 23)	Returns offset X.
TOOL(n, 24)	Returns offset Y.
TOOL(n, 25)	Returns offset Z.
TOOL(n, 26)	Returns offset D.
TOOL(n, 27)	Returns offset R.
TOOL(n, 28)	Returns angle A (B for Disk type).
TOOL(n, 29)	Returns overall dimensions.
TOOL(n, 30)	Returns wear coefficient in length.
TOOL(n, 31)	Returns wear coefficient in diameter.
TOOL(n, 32)	Returns maximum wear in length.
TOOL(n, 33)	Returns maximum wear in diameter.
STRCMP(s1,s2)	Compares string s1 with string s2 (only for fixed cycles). Returns: -1 if s1 < s2; 0 if s1=s2; 1 if s1 > s2. s1 and s2 can be immediate strings between double quotes or the parent variable pN.
FIELD(a,1)	Returns the specular value of X of work field a (the values allowed for a are all those that can assume the variable FLD).
FIELD(a,2)	Returns specular value Y of work field a.
FIELD(a,3)	Returns transverse number of work field a.
FIELD(a,4)	Returns first transverse number of work field a.
FIELD(a,5)	Returns X origin value in work field a.
FIELD(a,6)	Returns Y origin value in work field a.
FIELD(a,7)	Returns Z origin value in work field a.
FIELD(a,8)	Returns dimension X in work field a.
FIELD(a,9)	Returns dimension Y in work field a.
FIELD(a,10)	Returns the Y offset of the centre of the stop.
FIELD(a,11)	Returns the X offset of the stops on the multi-function table.
FIELD(a,12)	Returns the Y offset of the stops on the multi-function table.
FIELD(a,13)	Returns the X origin with the mobile stop OFF.
FIELD(a,14)	Returns the width of the area with the mobile stop OFF.

FIELD(a,15)	Step option.
FIELD(a,16)	Joined front and side stops.
FIELD(a,17)	Automatic mobile stop offset management.
FIELD(a,18)	Step separator stroke.
EPL_TV(t,v,d)	Returns information on the crosswise table and suction cups programmed using the work table Editor. 1. If **t** and **v** are at 0: general data is returned. Example: EPL_TV(0,0,1) indicates the position of the continuous centre bar referred to the piece origin. 2. If **t** is between 1 and the index of the last cross bar: a) if **v** is at 0, the data for cross bar t is returned; b) if **v** is between 1 and the index of the last device, the data of the device v of cross bar t is returned.
^	Tools selection (only for macro).
**	Power.
*	Multiplication.
/	Division.
MOD	Remainder from division (module).
+	Sum.
-	Subtraction.
<	Less than.
<=	Less than or equal to.
>	Greater than.
>=	Greater than or equal to.
=	Equal to.
<>	Not equal to.
NOT	Logical Not.

AND	Logical And.
OR	Logical Or.
XOR	Logical Xor.

It is possible to overrule the evaluation order by placing the expression within parenthesis. An expression enclosed by a parenthesis is evaluated independently of the operator before and after it.

Examples:

```
L FOOT = 300
L HEAD = 200
L SHOULDER = 200
L CENTER = 50
L L1 = DY-(SHOULDER*2)-CENTER
L L1 = L1/2
L R1 = CENTER*SIN 45
L R2 = (DY/2)-(SHOULDER+CENTRE+R1)
L L2 = DX-FOOT-HEAD-R2-R1
L L4 = L1-CENTRE
L L3 = L2-L4
```

APPENDIX C. *File Formats*

Formats

CFG

Format for Xilog Plus configuration files.

CNC

Format for ISO programs. ISO programs can also be edited by an external ASCII editor. On some types of machines with NUM® controller, the ISO programs can be managed directly (see: Direct management of ISO programs, in Appendix N: Notes). Programs in CNC format can be imported in PGM format (only if they do not contain jump instructions) in the **Open file** window.

DXF

Format for programs generated by CAD. DXF files can be imported by the Xilog Plus editor in PGM and XXL format.

EPL

Format for Worktables Editor files.

FRZ

Format for mix whose execution has been interrupted in Automatic mix status (see: Machine Panel Manual). These files can be executed but not edited.

MIX

Standard mix format.

PGM

Standard programs format. The programs in PGM format can be exported in XXL format into the **Open file** window; in the same window the unit of measure for a PGM file can be converted from millimeters to inches and vice versa.

TLG

Format for tooling files.

TWN

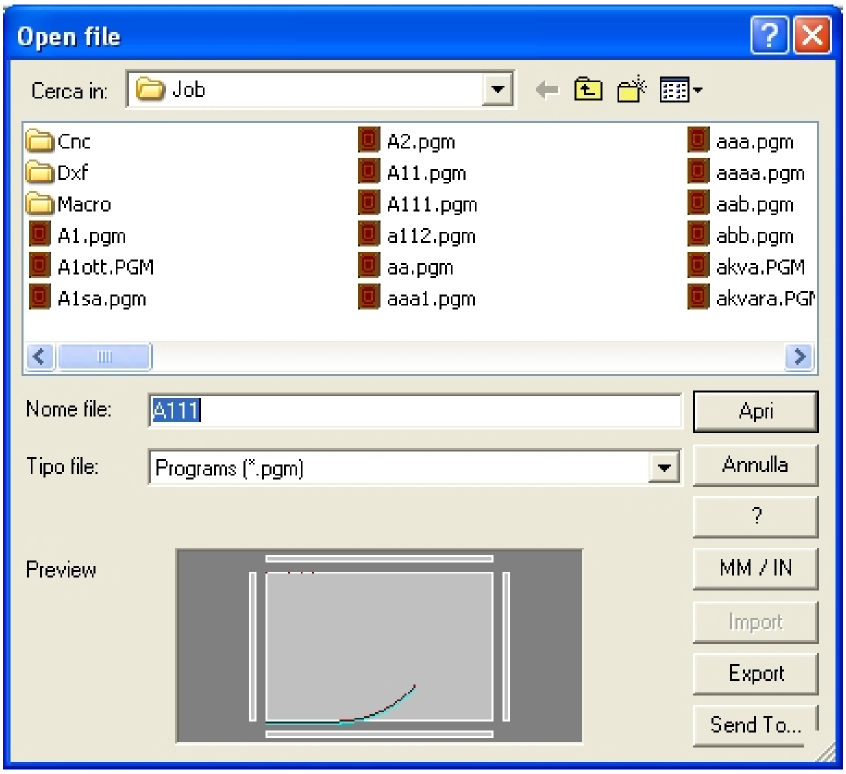
Format for Multiple programs Editor files.

XXL

ASCII programs format. Programs saved in this format can be edited with any text editor. The files in XXL format can be saved in PGM format.

Open File and Conversion Window

The **Open file** window of Xilog Plus in different from the standard windows because it contains some buttons with specific functions.



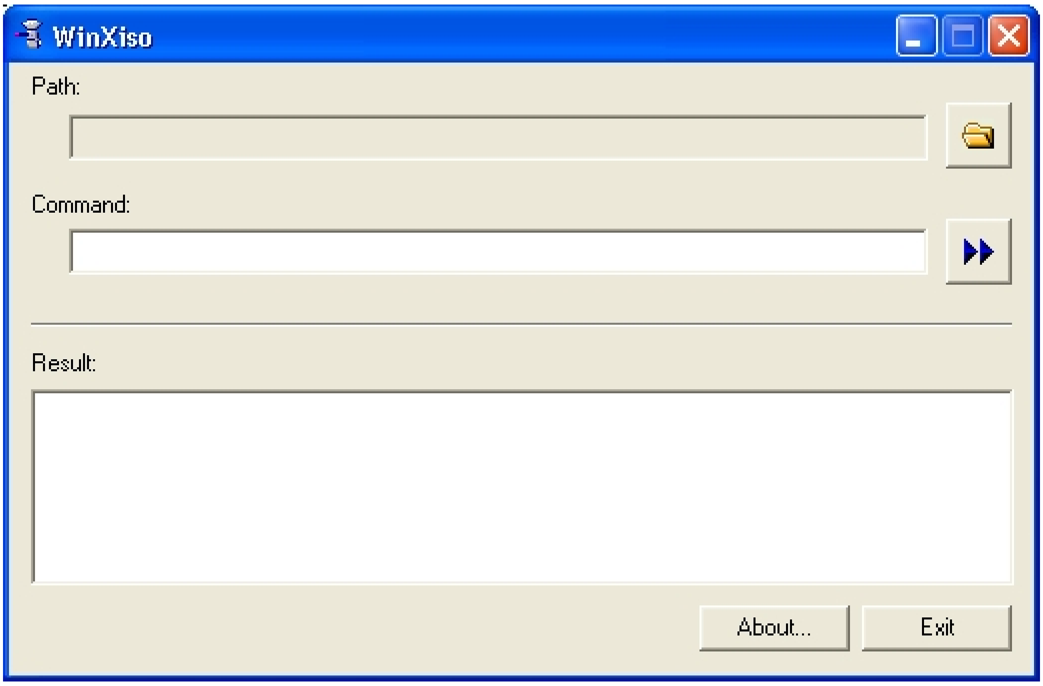
- ?: display on-line Help for Xilog Plus editor;
- **MM/IN**: converts unit of measure for PGM file from millimeters to inches and vice versa (activated only by selecting PROGRAMS from the file type box). The conversion does not effect numbers that are followed by a hatch sign (#), inserted in the numeric fields of any instructions.
- **IMPORT**: converts a CNC file into a PGM file (activated only by selecting iso programS in the file type box).
- **EXPORT**: converts a PGM file into a XXL file (activated only by selecting ProgramS from the file type box).
- **SEND TO**: makes it possible to copy the selected program into another position in the computer or onto external storage support (e.g. a floppy disk).

WinXiso

With the WinXiso program it is possible transform a program in ASCII format (with XXL extension) into a program in PGM format, and vice versa.
WinXiso can be used directly or activated by an external program, for example a Cad or Cam. The external program can activate WinXiso through API features, such as WinExec and ShellExecute.

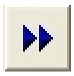
► To start WinXiso directly, click on the **START/ALL PROGRAMS/SCM GROUP/WINXISO** menu.

The program will show the following screen:



By clicking on the following button

it is possible to select the program in XXL format to be compiled or the macro (in PGM format) to be broken down. One the file is selected, just click on the following button



to activate compilation or breakdown.

Some parameters are available for setting program functions. The parameters are codes preceded by the symbol "-" and can be inserted after the name of the program in the **COMMAND** box.

-o <filename>	Converts a program in ASCII format (with XXL extension) into a program in PGM format and assigns a <filename> (mandatory).
-x <filename>	Converts a program in PGM format into a program in ASCII format (with XXL extension) This option excludes options -o and -l.
-i <filename>	Produces a file in ASCII format (with .INF extension) with the results of the translation: - [LINES]=<number of instructions converted> - [ERRORS]=<number of errors found>
-l <filename>	Produces a program list in ASCII format (with .LST extension).
-s	Prevents windows from opening on the screen.
-v	Together with the -x option, compiles/breaks down the EPL section of a program. To breakdown, select the .PGM file: a .XXL file and a

	separate .EPL file will be created. To compile, select a .XXL file and insert the name of the .EPL file after the -v option: a .PGM file containing the EPL section will be created.
-t	In conjunction with the -1 option, adds the table of symbols to the list of programs.
-f <bitmap>	Adds the specified bitmap image to the program.

- WinXiso always tries to translate an ASCII file into PGM, unless the -x option is specified.
- <filename> can be any file name composed of a maximum of 8 characters plus three characters for the extension, preceded by the path if necessary. For option -o the file name is mandatory; for the other options it is optional (if not specified, the selected file name is used again with a different extension).

Example of WinXiso recalled from an external application:

```
...
WinExec("c:\\tools\\WinXiso.exe ..\\test\\sample.xxl -s -i",SW_SHOW);
...
```

When the control reaches this point the translation is terminated; therefore, the ..\test\sample.inf. file can be controlled.

APPENDIX D. Programming the Exhaust Hood

Pneumatic Hood

a) Hood programmed during machining (parameter E):

E=-1 Hood high.
E=0 Hood high.
E=1 Hood low in position 1.
E=2 Hood low in position 2.
E=3 Hood low in position 3.
E=4 Hood low in position 4.

The maximum number of positions corresponds to the number of position (in sequence) other than zero programmed in the "Hood" section of the configuration file axis.cfg and depends on the type of machine.

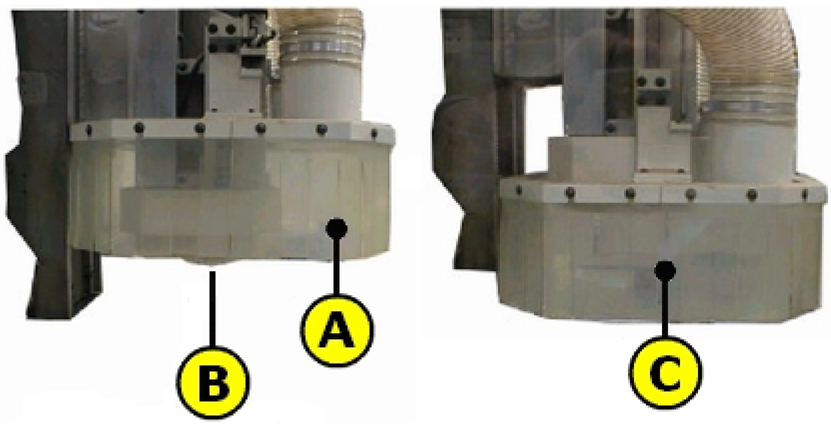
b) Hood not programmed during machining
The best order is automatically calculated based on current machining, together with the characteristics of the tool used and the hood position programmed in the "Hood" section of the configuration file axis.cfg.

Power-driven Hood

The position programmed in parameter E corresponds to the actual (in mm or inches) descent position of the hood. **Caution:** in this case no anti-collision controls are carried out on the position of the hood.
If E is not programmed, the hood is automatically positioned based on current machining and on the characteristics of the tool used.

RD260 – Prisma Basic head external hood

The Record 260 Prisma "Basic Head" is fitted not only with the standard hood (motor-driven and height-programmable), but also a second external hood with pneumatic positioning (ON - OFF).



A) External hood (OFF).

B) Internal hood with programmable height positioning.
C) External hood (ON).

Programme the position of this hood by directly adjusting the value of parameter E.

E=0	Both hoods are switched off.
E=desired height	The external hood is switched off; the standard hood is positioned at the desired height.
E=not programmed	The external hood is switched off; the standard hood is positioned at a height calculated according to the dimensions of the tool in the spindle.
E=-1	The external hood is lowered (ON); the standard hood is switched off.
E=-2	The external hood is lowered (ON); the standard hood is positioned at a height calculated according to the dimensions of the tool in the spindle.

APPENDIX E. Programming Reference Stops

The reference stops are programmed through the T field of the H instruction (TV ON/OFF bar)
By provision, the pause position (OFF) for all reference stops is always that of unblocked; usually (always by provision) the bars are all in OFF position. Any controls to be performed by the PLC are always carried out before the bars are positioned on start. Activating manual reference stops is equivalent to controlling the position.
Activating the physical reference stop and controlling the pause (OFF) position for other stops, by the PLC, depends on the configuration of the machine and of the program area being executed.

Machine without central zero (2 work areas: A,B; E,F;...) and with two reference stops

If the reference area is A or E or I or M, then the controlled TV bar is the one on the left of the machine (when standing in front of the machine); if the reference area is B or F or J or N, then the controlled TV bar is the one on the right of the machine (always when standing in front of the machine).

In the case of **single areas**, any TV bar other than the one controlled is not taken into consideration.

Examples:

Area	*left TV*	*right TV*
A	Active	-
B	-	Active

In the case of **double areas** (including the entire machine surface), the controlled TV bar is always the one relative to the reference area (that corresponds to the first letter in the list of areas). The other TV bar is placed in unblocked position: if physically this is not so, the PLC signals the situation with an error message before the program is run.

Examples:

Area	*left TV*	*right TV*
AB	Active	Control OFF
BA	Control OFF	Active

Machine with central zero (4 work areas: A, B, C, D; E, F, G, H;...) and with four reference stops

With standard areas

For reference areas A or E or I or M and D or H or L or P (which do not have a central reference), the previous paragraph pertains with D or H or L or P in place of B or F or J or N.

In particular, the condition of the TV1 bar corresponds to the condition of the TV bar mentioned above. If the area is single, programming of the TV2 bar is ignored.
For reference areas B or F or J or N and C or F or J or N (which share a central reference) the TV2 bar is always active and the condition of the TV1 bar, if the area is single, is ignored.

Examples:

Area	*left TV1*	*left TV2*	*right TV2*	*right TV1*
A	Active	-	-	-
B	-	Active	-	-
C	-	-	Active	-
D	-	-	-	Active

If the area is double, the TV bar controlled is always the one relative to the reference area (TV1 or TV2, depending on the name of the area). The other TV bar (TV1 or TV 2, opposite to the first one) always assumes an unblocked position. Otherwise the PLC will show an error.

Examples:

Area	*left TV1*	*left TV2*	*right TV2*	*right TV1*
AB	Active	Control OFF	-	-
BA	Control OFF	Active	-	-
CD	-	-	Active	Control OFF
DC	-	-	Control OFF	Active

If the area is a multiple area (including the entire machine) the previous paragraph on double areas pertains but with D or H or L or P in place of B or F or J or N, and the TV2 bars are considered in unblocked position (even though in this case, when calculating the movement of the bars, only the limit runs for these, that are programmed in the configuration file supports.cfg, will be considered).

Examples:

Area	*left TV1*	*left TV2*	*right TV2*	*right TV1*
AD	Active	Control OFF	Control OFF	Control OFF
DA	Control OFF	Control OFF	Control OFF	Active

With virtual areas (AB, CD, BA, DC, AD, DA; EF, GH, FE, HG, EH, HE;...)

The same as described for standard areas, with the difference that the areas are always in pairs.

APPENDIX F. Ergon Machines

This appendix to the manual covers the use of Xilog Plus installed on Ergon machines. Given the particular configuration of this type of boring – routing machine, new configuration parameters had to be introduced, as well as new instructions and new programming rules. For a clearer understanding of the content of this document, refer to the diagram below, which represents an Ergon machine with four independent Z axes (or units), each fitted with an electrospindle as the main head and an additional (secondary) head, which can be of various types (e.g.: boring head, Universal, blade, etc.).

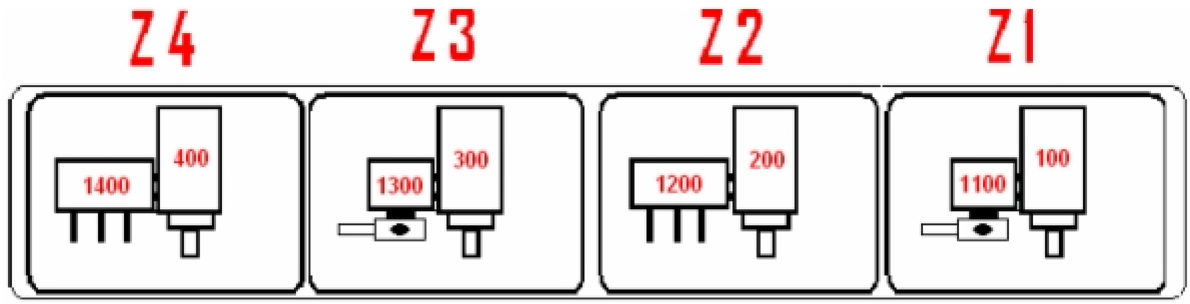


Fig. 1

Tooling

The tooling of an Ergon is similar to standard tooling, except that more groups of parameters are displayed for the fixed and external tools, equal to the number of independent Z axes (units) on the machine.

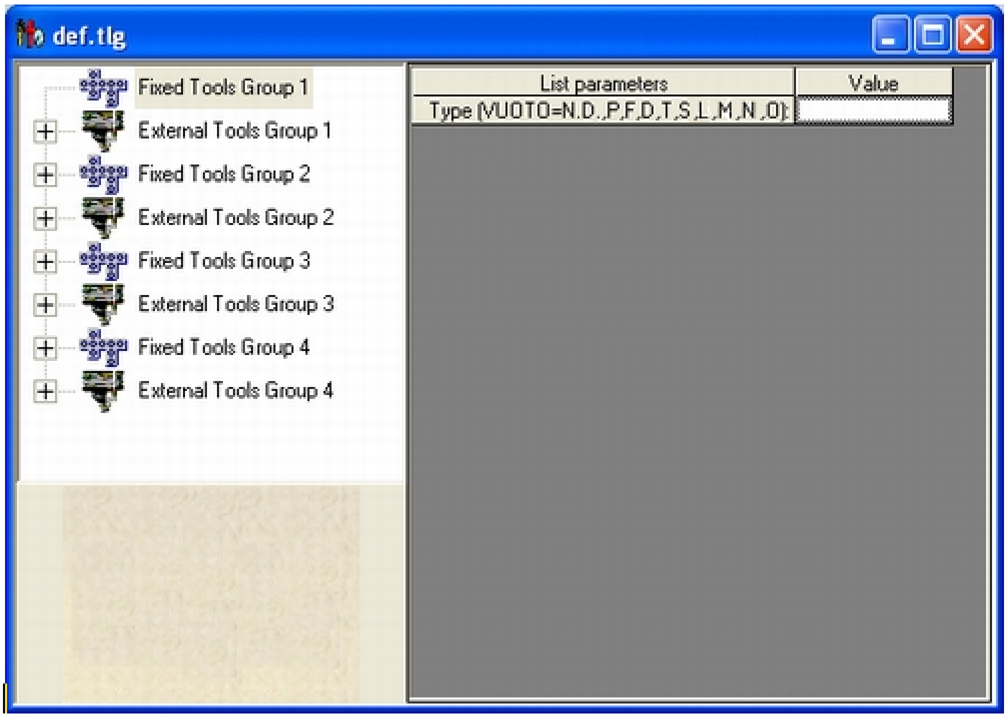


Fig. 2

Expanding the branch relative to each unit displays 96 different programmable tools. The tools in unit 1 are characterised by the fact that all of their parameters can be modified, whether for fixed tools (i.e.: boring heads) or external tools (spindle heads). Notice in the figure below the example relative to the external tools for unit 1:

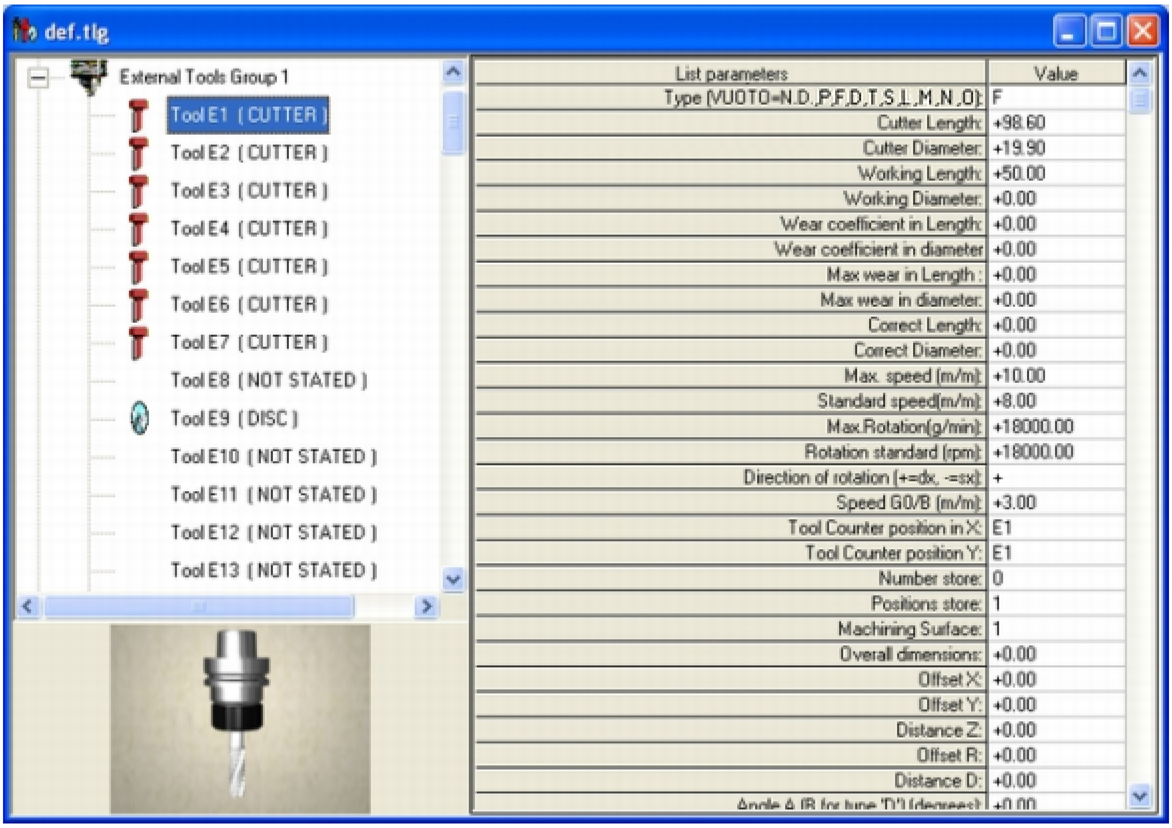


Fig. 3

In contrast, for the tools of units other than unit 1, you can only change some of the parameters, that is to say, those which change for each tool according to the Z unit to which they belong.

The figure below shows the example of external tool 1 for unit 2, where you can see how only some of the parameters can be changed, whilst all of the others (shown in grey) are inherited from those programmed for tool 1 in unit 1. Therefore, to configure a new tool, you must first edit its parameters relative to unit 1 (even if for said unit the tool in question may not be physically present) then modify the parameters, not common to all of the units, for the units in which the tool is physically present.

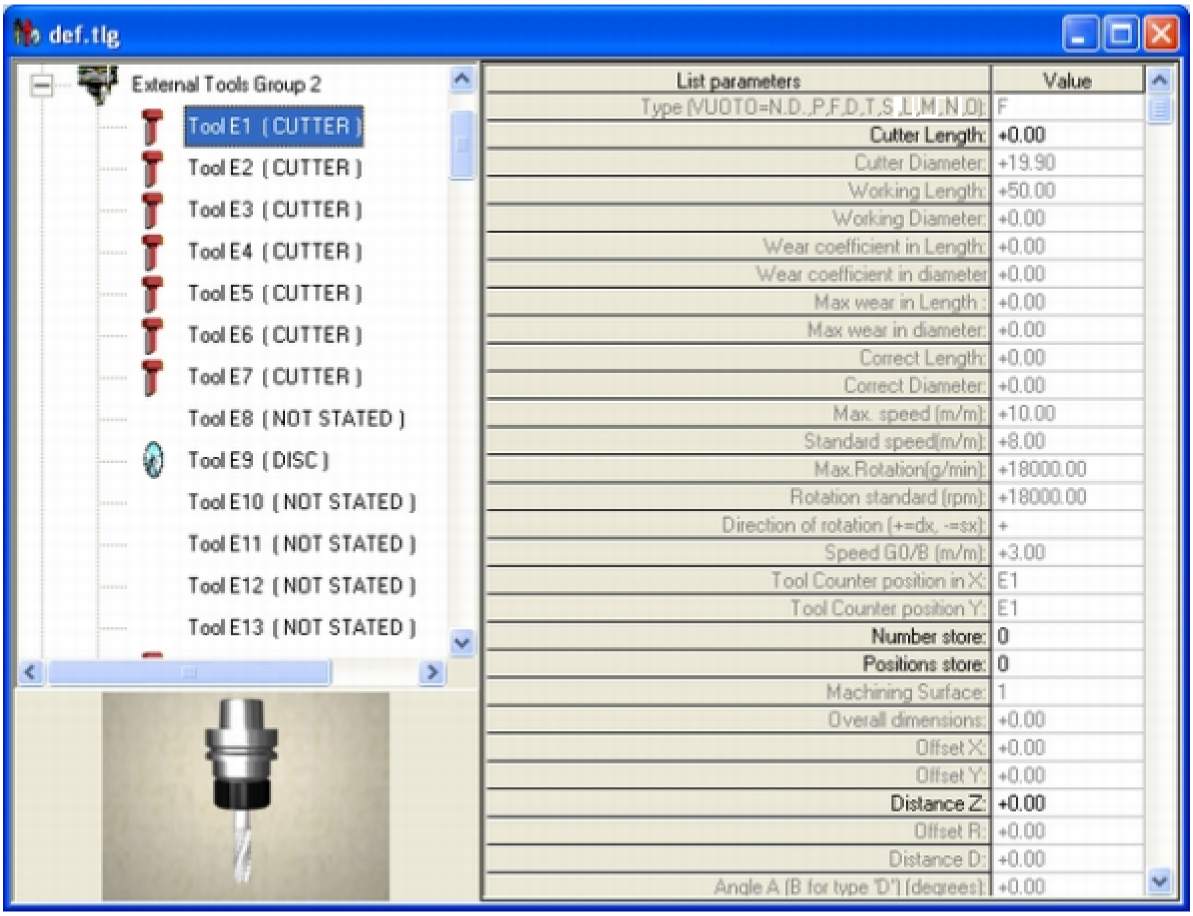


Fig. 4

For example, to add external tool 5 to the tooling, select position 5 in unit 1 and modify all of its parameters, starting with the type of tool.
When unit 1 tool parametrisation is complete, change the specific parameters in the other units. If the tool is not present in a given unit, you do not need to complete its parameters, unless it is unit 1, for which tool parameter programming is mandatory.
To save the tools associated with the various Z units, the system generates as many files as there are machine Z units. If, for example, the tooling is “def” and the machine has 4 Z units, the system generates and manages the following 4 files:

def.tlg	“def” tooling for tools (fixed and external) in unit 1
def.tl2	“def” tooling for tools (fixed and external) in unit 2
def.tl3	“def” tooling for tools (fixed and external) in unit 3
def.tl4	“def” tooling for tools (fixed and external) in unit 4

The same applies for the configuration of the boring head spindles (in the configuration file spindles.cfg). If the machine has 4 Z units, the system generates and manages the following 4 files:

spindles.tlg	Parametrisation of boring spindles on unit 1
spindles.tl2	Parametrisation of boring spindles on unit 2
spindles.tl3	Parametrisation of boring spindles on unit 3
spindles.tl4	Parametrisation of boring spindles on unit 4

This means that the correct and complete backup of machine toolings and configuration always requires that all of these files are saved (not just the configuration files def.tlg or spindles.cfg).

Programming

The programming rules at the basis of a PGM program are the same as those of any other SCM machine managed by the Xilog Plus system. However, given the nature of the Ergon machine, a mechanism is required which allows you to state, for each machining operation to be performed in the program, not only the tool to be used, but also which heads must operate together, if simultaneous machining of two or more workpieces must be carried out, synchronising two or more operating heads. In the example in this document, machining may be required with the main electrospindles of Z units 1 and 2 or with the boring heads of Z units 2 and 4, etc. To identify these combinations, simply specify, in the tool field (T) of any PGM operating instruction (e.g.: XB, XBR, XG0…) a list of tools, each representing the head which must perform the machining and the tool to be used on it.
To facilitate mnemonic association of the head number to be programmed with number of the Z unit to which the head belongs, the configuration of the heads (pheads section) is performed in such a way that the tool number to be used in the operating instructions follows this rule:

- from **100** to **900** indicate the machine mains heads;
- from **1100** to **1900** indicate the machine secondary heads (first additional head);
- from **2100** to **2900** indicate the machine third level heads (second additional head).

In this way, the tool number hundreds value represents the Z unit to be identified, whilst the thousands value represents the type of head to be used (0: main – 1: first additional – 2: second additional).
Except in particular cases which may arise with special machines, this convention will be considered the standard.

Example program

Assume, for example, that you want to simultaneously machine two workpieces, separated from one another by a distance equal to that between Z units 1 and 3 (this distance assumed to be the same as that between Z units 2 and 4), the machining being routing with tool 5 of the main electrospindles of Z units 1 and 3 and routing with the spindles 7 and 8 of the auxiliary boring heads of Z units 2 and 4. Also assume that you want to machine on table Y, with reference to the origin of the workpiece on the bottom left (area AB). The arrangement of the two workpieces will be that shown in Fig. 5:

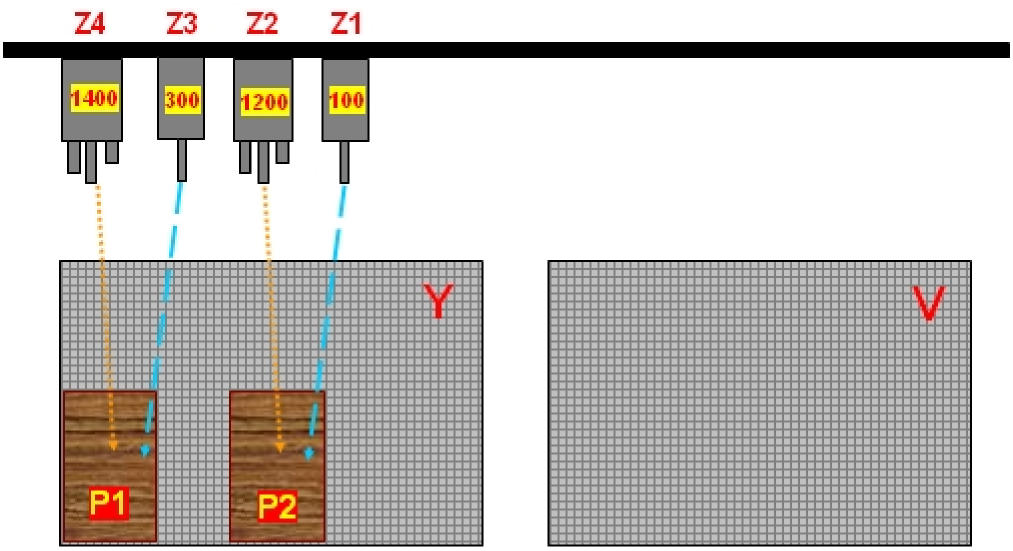


Fig. 5

As is graphically indicated by the arrows, the machining will involve the two pairs of heads 100/300 (electrospindles) and 1200/1400 (boring heads).
With reference to Fig. 1 and 5, simply write the following instructions in the program:

H DX=… DY=… DZ=… BX=… BY=… BZ=… /"Def"
XG0 X=… Y=… Z=… **T=305 105**
XG1 X=… Y=… Z=…
XB X=… Y=… Z=… **T=1407 1408 1207 1208**

An analysis of the two lists of tools entered for instructions XG0 and XB indicates the following:

- T=305 105. The system understands that synchronised use of heads 100 and 300 is required, which, in the current example, identify the main spindle of Z units 1 and 3 respectively, and that tool 5 must be fitted for both, its features described in the group of external tools in the def.tlg tooling (selected in the program header); finally, that all of the X and Y dimensions indicated for this machining relate to head 300, since the first tool in the list is 305.

- T=1407 1408 1207 1208. The system understands that synchronised use of heads 1200 and 1400 is required, which in the current example identify the auxiliary (secondary) boring head of Z units 2 and 4 respectively. For both, spindles 7 and 8 whose features are described in the fixed tools group of the def.tlg tooling (selected in the program header) are extracted and, finally, all of the X and Y dimensions indicated for this machining relate to head 1400, since the first tool in the list is 1407.

It is important to emphasise that for operating head electrospindles you can specify various tool numbers, whilst for boring heads you must specify the same spindles for the heads programmed to work in a synchronised fashion. So, the instruction XG0 X=…Y=…Z=…**T=305 121** is allowed, but the instruction XB X=…Y=…Z=…**T=1407 1408 1207 1215** is not allowed.

Further examination of machine behaviour relative to X/Y/Z movement of the synchronised heads is now useful, relating to the programming carried out.

Movement on the Z axis

Take, for example, the machining with the two electrospindles, required by instruction XG0 X=…Y=…Z=…T=305 105. Assume that tool 5 is a fluted cutter and the tool mounted on head 100 is 80 mm long, whilst the tool on head 300 is 80.5 mm long. In this case, before performing the machining on the workpiece, the head 300 will perform a vertical movement (Z3 axis) measuring 0.5 mm upwards, so that the "tips" of the two tools, on heads 100 and 300, are perfectly aligned (two or more heads are always synchronised by keeping the first belonging to the Z unit with the lowest number stationary and moving the other heads vertically on their Z axes). After this operation, all of the instructions in the program which require movements on the Z axis will involve simultaneous movement of heads 100 and 300.
The same applies for the boring heads 1200 and 1400, except that at present for boring heads the vertical alignment (in Z) is not performed by the PLC. In general, when working with heads other than electrospindles (boring heads, Universal heads, etc.) the PLC performs the synchronisation, but not the alignment in Z, according to the different length of the tools fitted.

Movements on the X and Y axes

Following synchronisation and alignment of the heads on the Z axis, movement on the X and Y axes is performed, taking the first head in the list of tools as the reference head. Therefore, for machining XG0 X=…Y=…Z=…T=305 105, all of the X/Y dimensions will be calculated by the system with reference to head 305. Obviously, if the two workpieces on the work table are aligned on the Y axis and their distance on the X axis is equal to the centre-to-centre distance between the heads 100 and 300, the machining performed by head 300 on workpiece P1 and by head 100 on workpiece P2 (see Fig. 5) will be absolutely identical.
The same applies to machining with the boring heads. Therefore, for machining XB X=…Y=…Z=…T=1407 1408 1207 1208, after synchronisation of heads 1200 and 1400 (no alignment – see above), all movements on the X/Y axis will be performed with reference to spindle 7 of head 1400, since the first tool in the list is 1407. Again, the result is two absolutely identical machining operations, performed by head 1400 on workpiece P1 and head 1200 on workpiece P2.

SET TWIN instruction

To facilitate programming, above all for machining carried out with boring heads, you can use the SET instruction on the TWIN parameter to indicate which Z units must work in a synchronised fashion. For example, the program:

H DX=… DY=… DZ=… BX=… BY=… BZ=… /"Def"
XG0 X=… Y=… Z=… **T=305 105**
XG1 X=… Y=… Z=…
XB X=… Y=… Z=… **T=1407 1408 1207 1208**

would become the following:

H DX=… DY=… DZ=… BX=… BY=… BZ=… /"Def"

SET TWIN=13
XG0 X=… Y=… Z=… T=305
XG1 X=… Y=… Z=…
SET TWIN=24
XB X=… Y=… Z=… T=1407 1408

In practice, use of the TWIN does not require you to specify in field T the list of tools for all of the heads which must work in a synchronised fashion. You can simply indicate the tool (electrospindle) or spindles (boring head) for the “master” head, that is to say, the head relative to which the program dimensions are processed.
To better illustrate the usefulness of SET TWIN, assume that you want to perform machining with the boring heads of units 2 and 4 and you want to use spindles 1, 2, 3, 4, 5, 6 and 7. In the first programming mode presented, the program is as follows:

H DX=… DY=… DZ=… BX=… BY=… BZ=… /“Def“
XB… **T=1401 1402 1403 1404 1405 1406 1407 1201 1202 1203 1204 1205 1206 1207**

But using SET TWIN, the program would be as shown below:

H DX=… DY=… DZ=… BX=… BY=… BZ=… /“Def“
SET TWIN=24
XB… **T=1401 1402 1403 1404 1405 1406 1407**

The usefulness of SET TWIN increases the greater the number of heads to be synchronised and spindles (in the case of boring heads) to be used.

SET TAU Instruction

The SET TAU instruction makes it possible to manage the feeler mounted on the electric spindle heads. With Ergon machines, the use of this instruction follows special rules.

SET TAU = 0
If inserted in the program following at least one machining operation which requested the use of the feeler, it informs the system that from the next machining operation it will not be requested anymore, but only in physical terms. This means that the feeler ring will be disconnected and the dynamic operators will be placed on "stand-by", but will continue to remain enabled.

SET TAU = -1
If inserted in the program following at least one machining operation which requested the use of the feeler, it informs the system that from the following machining operation it will no longer be requested either in physica or dynamic terms, for the correctly synchronised heads (or for just the current master head if ther is no other head synchronised with it). This means that the feller ring will be disconnected from the heads thus selected and the dynamic operators will be "unloaded" (subprogram H9051) from the NC execution cycle.

SET TAU = -2
If inserted in the program following at least one machining operation which requested the use of the feeler, it informs the system that from the following machining operation it will no

longer be requested either in physica or dynamic terms, for all of the heads on the machine with the feeler inserted, whether synchronised, master or "disabled". This means that the feeler ring will be disconnected from the heads, thus selected and the dyanmic operators will be "unloaded" (subprogram H9051) from the NC execution cycle.

SET TAU = -3
Used at the beginning of the program (after the Header) in the same way as SET TAU =-1 or, by necessity, in the middle of the program (here SET TAU=-1 can no longer be used for the same purpose) so that when the tool is already installed on the head which must work, for safety, it is explicitly requested that the command be generated which mechanically excludes the feeler ring, since the previous program may have ended the machining operation with the latter inserted, while at the beginning of the new program the head must work without the feeler ring.
Remember that this is only valid for the first machining operation (with relative tool change) associated to each individual head, since it represents the transition from the current machine state (unknown to the translator) to that at the start of the program. If one of these SET TAU values (-1 or -3) is not used, everything functions as before and no codes are generated inherent to the feeler which, therefore, will remain in the last assumed position (connected or disconnected), unless a tool change is requested.

Example

H ...	
SET TAU = -3	Physical exclusion of feeler requested if CUT is absent(M31 M185). In this case -1 may also be used.
XG0 ... T=101	Machining of T101 without feeler.
SET TAU = 1	Feeler requested to work in next machining operation.
XG0 ... T=101	Machining of T101 with feeler.
SET TAU = -3	Physical exclusion of feeler requested if CUT is absent (M32 M185). In this case -1 **cannot** be used.
XG0 ... T=201	Machining with T201 without feeler.
SET TAU = 1	Feeler requested to work in the next machining operation.
XG0 ... T=201	Machining of T201 with feeler.

If, in the place of the second "SET TAU = -3" a "SET TAU =-1" were to be used, the system would produce a physical exclusion of the feelers for the 200 head, but it would also unload the dynamic operators, since the latter had been loaded by the previous machining operation marked by "SET TAU = 1".

Macro

To facilitate the programming of some Ergon machine functions, some macros were prepared for use at the PGM program editing stage. These macros can be displayed by pressing the push-button for macros associated with machines with parallel heads.

- Selecting piece block sub areas – XSUBAREA.
- Selecting stop files for piece block sub areas – XBATTON.
- Cleaning tables with brushes – XCLEAN.

- Positioning motorised guides on table – XGUIDEM.
- Set distance in X between heads for twin machining – XINTAX.
- Horizontal guide dimension programming – XGUIDEH.
- Delta between actual tool length and corrected by feeler – XDELTAPALPATORE.

See: cap. 5.4.2 - User Macros for Ergon Machines.

Optimisers

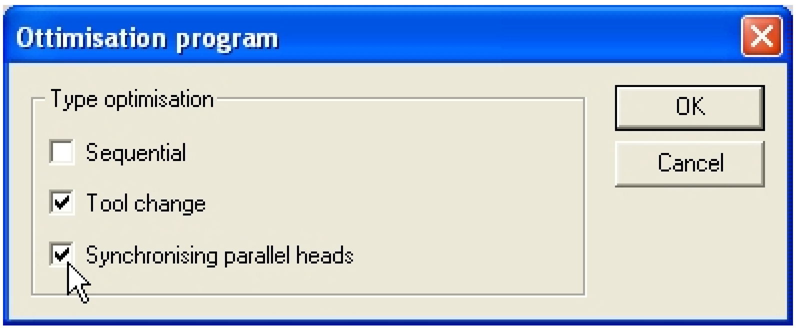
The Xilog Plus editor makes two optimisation algorithms available:

- Tool change-over optimiser (algorithm SO; see: cap. 7 – Optimization program).
- Boring optimiser (see: BO instruction)

Use of such algorithms also applies on Ergon machines following some indications.

Tool change-over optimisation (SO algorithm)

The algorithm follows the rules already defined and explained in the manual for a common PGM program, with the addition of a specific option, made available by the relative dialog box:



Selecting **SYNCHRONISING PARALLEL HEADS** optimisation automatically selects the **TOOL CHANGE** check-box. With this combination of check-boxes selected, the optimisation algorithm not only attempts to optimise the number of tool change-over operations on the heads, but also minimises the number of successive synchronisation and desynchronisation operations. This latter operation is to the detriment of the former, that is to say, limiting the desynchronisation operations does not normally give the lowest number of tool change-overs. However, considering the amount of time required for head synchronisation in the machine, on average better performance is achieved, in terms of reduced operating times.
The SO algorithm requires that the PGM programs to which it is applied do not contain SET TWIN instructions to indicate the synchronised combinations. For each operating instruction in the language, the list of tools must be specified, as already indicated.

Boring optimisation (BO algorithm)

This algorithm follows the rules already defined and explained in the manual for a common PGM program. In addition, for Ergon, operating with two or more synchronised boring heads involves mandatory use of the SET TWIN instruction to indicate the combination of heads to be synchronised.

Example:

H DX=1000 DY=1000 DZ=50 BX=0 BY=0 BZ=0 /“Def”
SET TWIN = 24
BO X=100 Y=100 Z=-5 D=8 N=“P“ R=2 x=32 V=10

In this example the boring optimisation algorithm generates a set of boring instructions which will be executed with the two boring heads 1200 and 1400 synchronised with one another.
In contrast, to work with a single boring head (e.g.: 1200), for the TWIN value, simply specify the number of the Z unit to which the head belongs:

Example:

H DX=1000 DY=1000 DZ=50 BX=0 BY=0 BZ=0 /“Def”
SET TWIN = 2
BO X=100 Y=100 Z=-5 D=8 N=“P“ R=2 x=32 V=10

If SET TWIN is not present, the algorithm works by default on the boring head configured in the configuration file pheads.cfg and belonging to Z unit 1. If they do not exist, an error is generated.

APPENDIX G. Universal Unit (on RD220) with Rotation Axis Parallel to Axis Y of the Machine

Unit Configuration

- **xilog3.cfg**

Set the "Universal" parameter = 0.

- **pheads.cfg** (head 4)

Set the "Actuator" parameter = 5.

Measure the distance along X between the main spindle center of rotation and the axis of rotation of the unit spindle in the horizontal position, as illustrated in Fig. 1 and set the "Set up 0 X" parameter using said value, observing the sign.

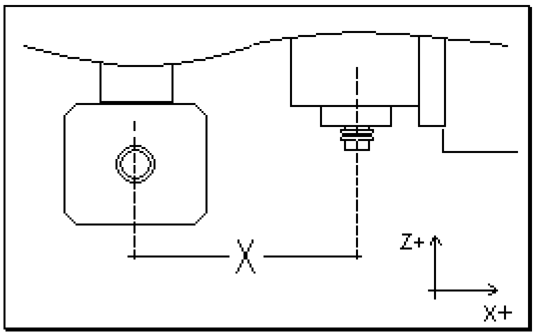


Fig. 1

Measure the distance along Y between the main spindle center of rotation and the axis of rotation of the unit spindle in the vertical position, as illustrated in Fig. 2 and set the "Set up 0 Y" parameter using said value, observing the sign.

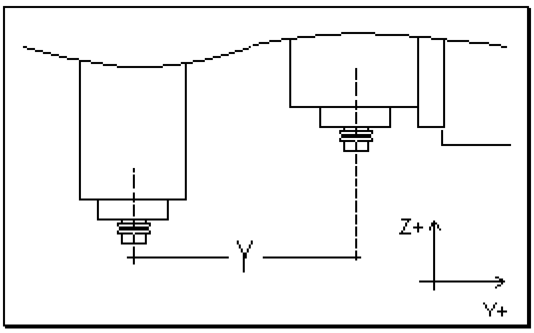


Fig. 2

Measure the distance along Z between the reference point for tool measurement of the main spindle and the axis of rotation of the unit spindle in the lowered-horizontal position, as illustrated in Fig. 3 and set the "Set up 0 Z" parameter using said value, observing the sign.

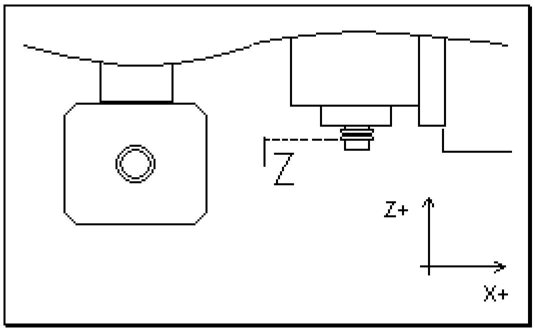


Fig. 3

Set the “Limit switch+ (positive) of Vector” parameter = 0.

- **nci.cfg**

Edit the file with a common text editor and add the following section if not already present:

```
{Tilting-Axis Lock Cutting Unit}
{Start Section}
$H04_XVECTOR
B
$
$H04_XFCVECTOR
43
$
$H04_UP
M81
$
$H04_DOWN
M71
$
$H04_ROTCW
M65M3S%ld
M21
$
$H04_ROTCCW
M65M3S%ld
M21
$
$H04_NOROT
M50
$
{End Section}
```

Creating Unit Tooling

Edit any tool between **E1** and **E96** which is a CUTTER or BIT.
Set the tool length as the distance between the unit reference plane and the tip of the cutter to be controlled (see Fig. 4).

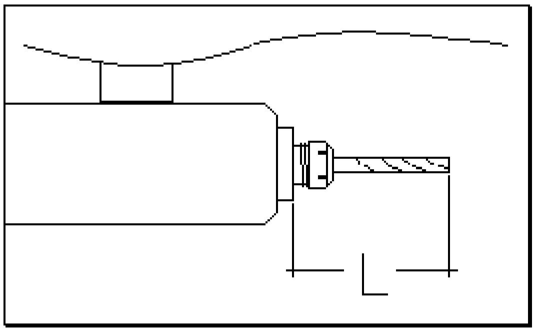


Fig. 4

Set the “Tool Diameter”, “Working Length”, “Working Diameter”, “Forward Speed” (maximum and standard), “Rotation Speed” (maximum and standard) and “Speed G0/B” parameters.
Set the “Overall dimensions” parameter as the distance between the tool axis of rotation and the maximum dimension Z of the unit in the horizontal position (Fig.5).
Set the “Distance Z” parameter = 0.
Set the “Offset R” parameter with the angle on the XY surface that takes on the tool with the unit positioned on 0° rotation.
Set the “Distance D” parameter as the distance between the unit center of rotation and the reference plane for measurement of the unit tool length (Fig. 5).

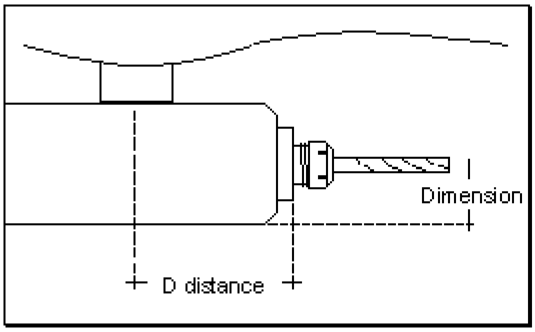


Fig. 5

Set the “Angle A” parameter with the angle of the tool to the vertical axis when the unit is at 0° rotation.

Programming the Unit

To program the unit in question, use the standard programming procedures with the Xilog Plus interface, remembering to enter the unit tilting instruction as follows:

;Positioning unit **B** tool at 110° to the vertical plane.
XAXROT R=110 N=”B”

;Angled boring instruction with boring angle of 270° on the XY plane with tool **E1** of head **2**.
BR X=800 Y=50 Z=-10 A=270 Q=1 T=201
The angle programmed in XAXROT must always be positive.

To change the angle of machining relative to the vertical plane, enter the **XAXROT** instruction again and repeat the machining. This is not necessary if the angle does not have to change.
You cannot change the unit angle of rotation once the profile is started, since the XAXROT instruction envisages unit positioning only before machining starts and not unit interpolation during machining

Example:

XAXROT R=90 N="B" --- CORRECT
G0R X=0 Y=0 Z=-10 A=90 H=20 N="profile" T=201
G1R X=100

XAXROT R=110 N="B" -- CORRECT
G0R X=0 Y=0 Z=0 A=90 H=20 N="profile" T=201
G1R X=100 Z=-10
XAXROT R=90 N="B" ----------------ERRONEOUS LINE
G1R X=0 Z=-20
G1R X=100

You can machine all the faces of the panel (using the unit), correct the tool radius or center the tool, using the normal B, XG0, XL2p instructions, etc., specifying the face required. You can also machine compatible angled planes using the XPL X... Y... Z.. instructions etc...
In these cases, you do not have to position the tilting axis with the XAXROT command since the Xilog3 compiler will use the angle compatible with the work plane selected:

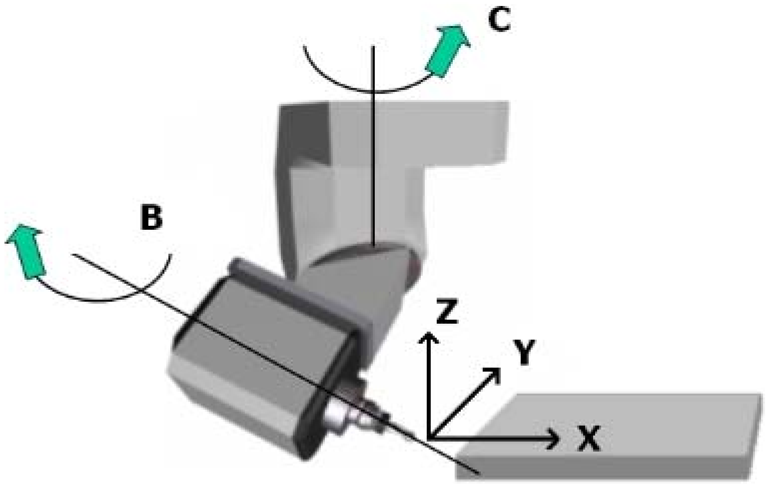
;Enabling of a plane parallel to face 4 of the panel.
XPL X=0 Y=10 Z=0 Q=0 R=90
XG0 X=100 Y=20 Z=-10 T=201 N="profile" C=2
XL2P X=300

To repeat the profile just created by changing the work plane, you can use the **XGREP** instruction, specifying the name of the profile:

;Enabling of a plane parallel to face 4 of the panel.
XPL X=0 Y=0 Z=0 Q=0 R=80
;Repetition of the profile on an angled plane with tool 202. 202.
XGREP N="profile" T=202

APPENDIX H. Prisma Head

Prisma is a head consisting of two rotary axes: C (Vector) and B (Tilting). In addition to normal machining on panels, Prisma allows you to perform 3D machining.



Tooling

Configure the tools with the normal criteria based on their appearance after loading in the spindle. To define the direction of machining, angle the blade on face 1 and towards the operator. Set the machining face to 0 for routers and 1 for blades.

Standard machining

Programme standard machining with instructions (X)B, (X)BR, (X)G0, (X)G0R.

3D machining

Programme 3D machining with instructions (X)G03D and (X)G13D.
During particularly broken up 3D machining (with numerous very small (x)G13D sections) unwanted pauses in machining may occur, due to the fact that Xilog Plus sends the ISO blocks to the NC too slowly for the NC to execute them. There are two methods for increasing the speed of ISO block transfer to the NC:

- Insert the “%ISOBLOCKSIZE=xx” ISO instruction at the start of the program, where xx must be a number between 5 and 20. The instruction validity range is limited to the program in which it is inserted.
- Insert block %%Bxx in the key GEN_INIT of the configuration file nci.cfg, where xx must be a number between 5 and 20. The instruction validity range extends to all programs, even those which do not contain 3D machining.

Use both instructions (but in particular the latter) with caution and only if really necessary, since they slow down all Xilog Plus activities not linked to transfer of the ISO blocks to the NC. Never exceed the value xx=10.
See also: instruction SET JERK3D.

Example of 3D programming

```
...
G03D X=85.148 Y=100 H=20.544 Q=90.054 R=59.036 T=101 D=30 V=1000 S=9000
G13D X=87.675 Y=91.2 H=20.544 Q=89.964 R=59.031 V=4000
G13D X=87.666 Y=82.709 H=20.544 Q=89.935 R=59.047
G13D X=87.676 Y=74.292 H=20.543 Q=90.324 R=59.066
G13D X=87.746 Y=65.234 H=20.569 Q=89.433 R=58.87
G13D X=87.4 Y=57.109 H=20.569 Q=85.129 R=58.764
G13D X=86.491 Y=50.424 H=20.569 Q=78.984 R=58.852
...
```

Running the ISO program directly

The Xilog Plus machine panel may take considerable time to convert lengthy programs (i.e. programs that involve 3D type processing) into ISO files and transmit them to the Number Control to be run..
With these programs we therefore suggest that you generate the ISO file on the PC used for programming so you can then move it to the Machine Panel and run it directly.

To carry out this operation you must:

- install the Machine Panel on the PC used for programming.
- verify that the machine programmed in the Xilog Plus editor is equipped and configured in the same way as the machine.

► To generate a ISO file (with a .CNC extension) for use on the Machine Panel:

1. Start the Machine Panel.
2. Import the PGM program file using the **FILE/CARRIED OUT AUTOMATICALLY** menu (see: Machine Panel manual).
3. Open the ISO file generated by the Machine Panel with a text editor (the Machine Panel editor or an external editor). The ISO file is in the directory APC (.../SCM Group/Xilog Plus/APC), inside the directories A, B, C or D, according to the area in which the program is loaded. The ISO file is named <name>.inn, where nn is a number from 00 to 99.
4. Delete the "%" character in the last line and insert the "M2" string in its place.
5. Save the file by modifying the .CNC extension (for example, the file **example.i00** will be renamed **example.cnc**).
6. The .CNC file which has been generated is then imported into the Machine Panel on the machine using the **FILE/OPEN CNC FILE** menu.

Notes

The Prisma head is subject to the normal position presence and not outside software end of stroke controls. Any errors due to movements generated from the Numeric Control are not indicated by Xilog Plus.

You can switch off all controls carried out by Xilog Plus in Editor mode and in Automatic mode by setting the “Check axis limit switches” field to zero in the “GEN 2” section of the configuration file axis.cfg.

The Prisma head always turns at a safety dimension above the workpiece. This dimension is used to position the head for the first machining operation, for movements between one machining operation and the next, and at the end of the final machining operation to move the head to the home position.
The safety dimension is variable, since it is linked to the head machining positions and to the tool and head dimensions.
Without the safety dimension above the workpiece, machining is impossible.
For 3D machining with the head's smallest tool, we recommend that you set the tool workpiece lead in and lead out angles to 0°.

Calculate the "Safe cylinder diameter" field in the configuration file pheads.cfg so that, when you turn axes B and C with the Prisma head fully raised, there are no collisions.

In MDI the M6Txxx tool change-over commands are sent to the NC using file %8197 and are executed using M197. Therefore, you must set the association between the program and the M code.

APPENDIX I. Programming Duomatic Clamps

Selection of hold-down using the DUOMATIC clamps must be made at the start of the program before the work piece hold-down request (M28/M39/M58).
It is selected using the usual E30xxx parameters:

E30001 (Selects the type of work piece hold-down to be used for zone 1).
E30002 (Selects the type of work piece hold-down to be used for zone 2).
(Also see: Diagnostics manual and M Codes provided with the machine)

In particular: **E30001 =11 : DAB (Duomatic zone AB)**
E30002 =11 : DCD (Duomatic zone CD)

1) If the **V** parameter is correctly programmed inside the Header of the Xilog Plus PGM program this will allow values for **E30001** and **E30002** to be set automatically for the empty zone (A, B, C, D etc…) selected:

V=5x (Selects the DUOMATIC clamps)

Example:

(H DX=… DY=… DZ=… -A C=0 T=0 R=99 … V=5x)
E30001=11
…
M28

Example:

(H DX=… DY=… DZ=… -B C=0 T=0 R=99 … V=5x)
E30001=11
…
M38

At the start of the program, before the work piece hold-down request (M28/M38/M58), you must also select which DUOMATIC clamps are to be used to start work [front (operator side in Y direction), rear (side opposite operator in Y direction) or both].

E30059 (Selects the front/rear/both CLAMPS DUOMATIC zone 1)
E30060 (Selects the front/rear/both CLAMPS DUOMATIC zone 2)

E30059	= 1: Front clamps zone 1	**E30060**	= 1: Front clamps zone 2
	= 2: Rear clamps zone 1		= 2: Rear clamps zone 2
	= 3: Both clamps zone 1		= 3: Both clamps zone 2

2) If the **V** parameter is correctly programmed, these values will be automatically set with regards to the zones concerned.

V=50 (no effect)
V=51 (Selects front DUOMATIC clamps)

V=52 (Selects rear DUOMATIC clamps)
V=53 (Selects front and rear DUOMATIC clamps)

Example:

(H DX=… DY=… DZ=… -CD C=0 T=0 R=99 … V=51)
E30002=11
E30060=1
…
M58

Example:

(H DX=… DY=… DZ=… -AB C=0 T=0 R=99 … V=53)
E30001=11
E30059=3
…
M58

At any point of the program, you can then exchange (during machining) the front and rear clamps using the code:
M193 (exchange front and rear clamps)
(Also see: Diagnostics manual and M Codes provided with the machine)

3) Xilog Plus contains an appropriate macro instruction capable of performing this operation directly from the program (XISO instruction).

DUOMATIC G=0 (exchange front and rear clamps)

Example:

(H DX=… DY=… DZ=… -AB C=0 T=0 R=99 … V=53)
E30001=11
E30059=3
…
M58
…
(DUOMATIC G=0)
M193

At any point of the program, you can decide to release the front or rear clamps if the **V=53** option (both) has been selected.

E30059 (Selects the front/rear CLAMPS DUOMATIC zone 1)
E30060 (Selects the front/rear CLAMPS DUOMATIC zone 2)
M194 (overrides the clamps selected)

E30059 = 4 : Front clamps
= 5 : Rear clamps

E30060 = 4 : Front clamps
= 5 : Rear clamps

N.B. Once the front clamps have been released, you can no longer release the rear clamps and vice versa or decide to re-apply hold-down with both clamps.

4) The instruction referred to above can be used to carry out this operation directly from the program (XISO instruction):

DUOMATIC **G=1** (overrides front clamps)
DUOMATIC **G=2** (overrides rear clamps)

Example:

(H DX=… DY=… DZ=… -AB C=0 T=0 R=99 … V=53)
E30001=11
E30059=3
…
M58
…
(DUOMATIC G=1)
E30059=4
M194

APPENDIX J. Management of Customer Equipment

Before switching the tooling on or off, you must ensure that the starting condition on all zones of the work table is the home position. You can carry out pendulum machining, but you cannot manage the tooling together with other locking systems.

With Xilog Plus:

To activate equipment management enter V = 40 in the V field of the Header.

In ISO:

a) 2 zone management

E30001=1 Zone A
E30002=1 Zone B

M191 = Tooling ON
M190 = Tooling OFF

These codes must be followed by the normal vacuum request codes.

b) 4 zone management

E30001=1 Zone A
E30001=2 Zone B
E30001=3 Zone AB or BA
E30002=1 Zone C
E30002=2 Zone D
E30002=3 Zone CD or DC

M191 = Tooling ON
M190 = Tooling OFF

These codes must be followed by the normal vacuum request codes.

APPENDIX K. Configuration of supports and table elements of worktables Editor

File libsupp.cfg

The supports and table elements which can be used in the worktables Editor can be configured with the configuration file **MACHINE/SUPPORTS LIBRARIES EPL (LIBSUPP)**. The corresponding file is called libsupp.cfg and is in text format, which means that it can be freely edited even using an editor outside Xilog Plus.
The file contains a single section for each type of support or table element, in which the individual supports or table elements are defined separately (except the seal for multi-function tables, of which there is only one).

Setting the number of supports/table elements available

The libsupp.cfg configuration file installed with Xilog Plus already contains all of the parameters needed to configure supports and table elements. However, the supports and table elements are not immediately visible in the worktables Editor, since you need to set the number, according to the number of pieces actually available on the machine. Enter this data with the [MAXNUMTYPE] parameter, if available for that type of support/table element.

Configuration parameters

If the line for a parameter of a support or table element is missing, Xilog Plus considers its value to be 0 (or null, if text must be entered).

Suction cups

The suction cups are contained between the following markers:

[INIT_LIBVT]	Start of section.
...	
[END_LIBVT]	End of section.

Definition of a suction cup

[INIT_VT]	Start of definition.
[GEOMBASE]...	Base geometry. Values: C (circular), R (rectangular).
[WBASE]...	Base width (if base is rectangular).
[LBASE]...	Base length (if base is rectangular).
[RBASE]...	Base radius (if base is circular).
[WBASEAGG]...	Width of rectangle connecting base to multi-function table.
[LBASEAGG]...	Length of rectangle connecting base to multi-function table.
[GEOMAPP]...	Geometry of rest. Values: C (circular); R (rectangular); G (generic).
[WAPP]...	Width of rest (if rest if rectangular).

[LAPP]...	Length of rest (if rest is rectangular).
[RAPP]...	Radius of rest (if rest is circular).
[HAPP]...	Height of rest.
[EN_ROTAZ]...	Enabling of rest rotation relative to base. Values: 0 (not enabled), 1 (enabled).
[OFFX_OVR]...	Offset in X of centre of rotation of rest relative to centre of base.
[OFFY_OVR]...	Offset in Y of centre of rotation of rest relative to centre of base.
[OFFX_OVAPP]...	Offset in X of centre of rest relative to centre of rotation of rest.
[OFFY_OVAPP]...	Offset in Y of centre of rest relative to centre of rotation of rest.
[HTOT]...	Height of suction cup (base+rest).
[ALFAROT]...	Rest initial angle of rotation (parameter not currently usable).
[FILEBMP]...	Path/FileName of bitmap image to be displayed in the suction cup properties window.
[MAXNUMTYPE]...	Number of suction cups available.
[OFFRLASER]...	Angle of asymmetrical laser cross for laser 0°-360° with rest positioned in its default configuration.
[OFFX_MIR]...	Offset in X between centre of base and suction cup sight.
[OFFY_MIR]...	Offset in Y between centre of base and suction cup sight.
[GEOMAPP2]...	Geometry of secondary rest. Values: C (circular), R (rectangular).
[WAPP2]...	(Suction cups with two rests). Width of secondary rest (if rest is rectangular).
[LAPP2]...	(Suction cups with two rests). Length of secondary rest (if rest is rectangular).
[RAPP2]...	(Suction cups with two rests). Radius of secondary rest (if rest is circular).
[HAPP2]...	(Suction cups with two rests). Height of secondary rest.
[EN_ROTAZ2]...	(Suction cups with two rests). Enabling of secondary rest rotation relative to base. Values: 0 (not enabled); 1 (enabled).
[OFFX_OVR2]...	(Suction cups with two rests). Offset in X of secondary rest centre of rotation relative to centre of base.
[OFFY_OVR2]...	(Suction cups with two rests). Offset in Y of secondary rest centre of rotation relative to centre of base.
[OFFX_OVAPP2]...	(Suction cups with two rests). Offset in X of centre of secondary rest relative to secondary rest centre of rotation.
[OFFY_OVAPP2]...	(Suction cups with two rests). Offset in Y of centre of secondary rest relative to secondary rest centre of

	rotation.
[ALFAROT2]...	(Suction cups with two rests). Secondary rest initial angle of rotation (parameter currently not usable).
[OFFRLASER2]...	(Suction cups with two rests). Angle of asymmetrical laser cross for laser 0°-360° with secondary rest positioned in its default configuration.
[DELTAAPP]...	(Suction cups with two rests). Parameter only valid if main rest and secondary rest have the same point of rotation: indicates the minimum angle between the two rests.
[SIMAPP]...	(Suction cups with generic rest). Rest symmetry relative to its own centre. Values: NDEF (no symmetry), SIMX (symmetry in X), SIMY (symmetry in Y), SIMXY (symmetry in X and Y).
[G0] X=...,Y=...	(Suction cups with generic rest). See below.
[G1] X=...,Y=...	(Suction cups with generic rest). See below.
[G2] X=...,Y=...,I=...,J=...,R=...	(Suction cups with generic rest). See below.
[G3] X=...,Y=...,I=...,J=...,R=...	(Suction cups with generic rest). See below.
[NOME]...	Name of suction cup (must be unique in the file).
[NONSPEC]...	Defines the areas in which the suction cup is not specularised. Values: SYMX_ (not specularisable on areas with Symmetry in X), SYMY_ (not specularisable on areas with Symmetry in Y), SYMXY (not specularisable on areas with Symmetry in XY). These values may also be present simultaneously, separated by a comma ([NONSPEC]:SYMX_, SYMY_ ,SYMXY).
[END_VT]	End of definition.

Configuration of a generic rest for a suction cup

If the suction cup has a generic rest (e.g.: triangular with rounded vertices), set some specific parameters, as indicated in the list of parameters for suction cup definition.
In particular, you can use the following instructions to define the rest profile:

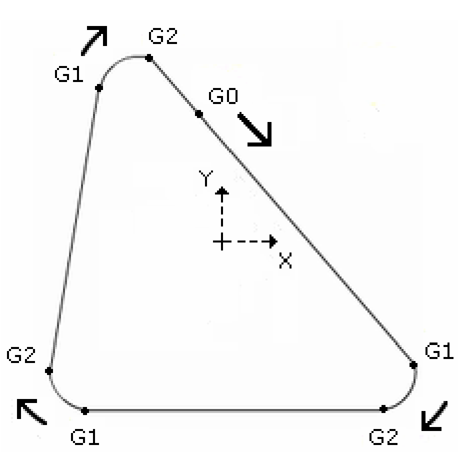
[G0] X=...,Y=...	Rest profile starting point. X = X co-ordinate; Y = Y co-ordinate.
[G1] X=...,Y=...	Straight line segment. X = end point X co-ordinate; Y = end point Y co-ordinate.
[G2] X=...,Y=...,I=...,J=...,R=...	Clockwise arc. X = end point X co-ordinate; Y = end point Y co-ordinate;

I = X co-ordinate of centre of arc;
J = Y co-ordinate of centre of arc;
R = radius of arc.

[G3] X=...,Y=...,I=...,J=...,R=... Anti-clockwise arc.

X = end point X co-ordinate;
Y = end point Y co-ordinate;
I = X co-ordinate of centre of arc;
J = Y co-ordinate of centre of arc;
R = radius of arc.

Example:



[GEOMAPP] G
[SIMAPP] NDEF
[OFFX_OVR] 0
[OFFY_OVR] 0
[OFFX_OVAPP] 0
[OFFY_OVAPP] 0
[EN_ROTAZ] 1
[HAPP] 63.5
[G0] X=-25,Y=75
[G1] X=100,Y=-75
[G2] X=75,Y=-100,I=75,J=-75,R=25
[G1] X=-75,Y=-100
[G2] X=-100,Y=-75,I=-75,J=-75,R=25
[G1] X=-75,Y=75
[G2] X=-50,Y=100,I=-50,J=75,R=25

To use these instructions some conditions must be satisfied:

- there must be only one G0;
- the rest profile must be defined by at least three points;
- all instructions used (G0, G1, G2, G3) must be entered in such a way as to define a closed perimeter consisting of a sequence of points continuing on from one another.

The suction cup in the example may also be entered using instruction G3. In that case, the points must be entered following the anti-clockwise direction from the origin (G0).

Clamps

The clamps are contained between the following markers:

[INIT_LIBCLAMP] Start of section.
...
[END_LIBCLAMP] End of section.

Definition of a clamp

[INIT_CLAMP]	Start of definition.
[TYPEMORS]...	It defines the type of clamp. Values: G (generic) S (standard).
[GEOMBASE]...	Base geometry. Values: C (circular), R (rectangular).
[WBASE]...	Base width (if base is rectangular).
[LBASE]...	Base length (if base is rectangular).
[RBASE]...	Base radius (if base is circular).
[RSTELO]...	Stand radius.
[RPIATT]...	Plate radius.
[OFFX_OMORS]...	Offset in X of centre of stand relative to centre of base.
[OFFY_OMORS]...	Offset in Y of centre of stand relative to centre of base.
[SIMMORS]...	(Clamps with generic type bodies). Type of Symmetry of the body with respect to its own centre. Values: NDEF (no symmetry), SYMX (symmetry in X), SYMY (symmetry in Y, SYMXY (symmetry in X and Y).
[HTOT]...	Clamp height (base+stand+plate).
[HUP]...	Stand+plate height.
[INSERONBASE]...	Confirms the possibility of inserting the clamp on the base (only for motorised tables). Values: 0 (the support cannot be inserted on the motorised bases) 1 (the support can be inserted on the motorised bases).
[POS_MORS]...	Position for clamp insertion on crosspiece. Values: 0 (central), 1 (on left-hand side), 2 (on right-hand side).
[FILEBMP]...	Path/FileName of bitmap image to be displayed in the clamp properties window.
[OFFX_MIR]...	Offset in X between centre of base and suction cup sight.
[OFFY_MIR]...	Offset in Y between centre of base and suction cup sight.
[MAXNUMTYPE]...	Number of clamps available.
[INIT_CORPO]...	Start definition of the body shape.
[G0] X=...,Y=...	(Clamps with generic type body). See below.
[G1] X=...,Y=...	(Clamps with generic type body). See below.
[G2] X=...,Y=...,I=...,J=...,R=...	(Clamps with generic type body). See below.
[G3] X=...,Y=...,I=...,J=...,R=...	(Clamps with generic type body). See below.
[END_CORPO]...	End definition of the body shape.
[NOME]...	Clamp name (must be unique in the file).
[END_CLAMP]	End of definition.

Configuration of a generic type body for clamp

If the clamp is equipped with a generic type body (for example, triangular with rounded corners), several specific parameters must be set, as indicated in the list of parameters for the definition of a clamp.
In particular, to define the profile of the body, the following instructions are available:

[G0] X=...,Y=...	Starting point of the body profile. X = X co-ordinate;

Y = Y co-ordinate.

[G1] X=...,Y=... Straight line segment.

X = end point X co-ordinate;
Y = end point Y co-ordinate.

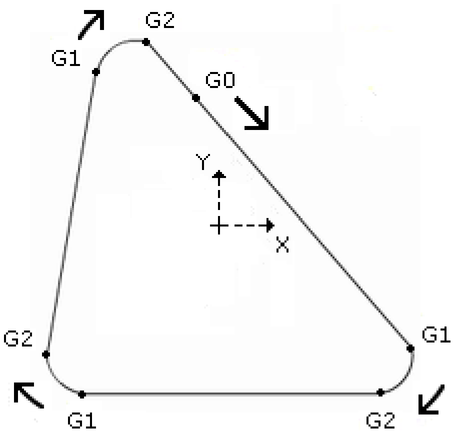
[G2] X=...,Y=...,I=...,J=...,R=... Clockwise arc.

X = end point X co-ordinate;
Y = end point Y co-ordinate;
I = X co-ordinate of centre of arc;
J = Y co-ordinate of centre of arc;
R = radius of arc.

[G3] X=...,Y=...,I=...,J=...,R=... Anti-clockwise arc.

X = end point X co-ordinate;
Y = end point Y co-ordinate;
I = X co-ordinate of centre of arc;
J = Y co-ordinate of centre of arc;
R = radius of arc.

Example:



[INIT_CLAMP]
[TYPEMORS] G
[GEOMBASE] R
[WBASE] 145
[LBASE] 145
[OFFX_OMORS] -32.7
[OFFY_OMORS] 0
[INIT_CORPO]
[G0] X=-25,Y=75
[G1] X=100,Y=-75
[G2] X=75,Y=-100,I=75,J=-75,R=25
[G1] X=-75,Y=-100
[G2] X=-100,Y=-75,I=-75,J=-75,R=25
[G1] X=-75,Y=75
[G2] X=-50,Y=100,I=-50,J=75,R=25
[END_CORPO]

In order to use these instructions, several conditions must be respected:

- The profile of the body must be defined by at least three points;
- All of the instructions used (G0, G1, G2, G3) must be inserted so as to define a closed perimeter made up of succesive points, continuous with respect to each other.

The clamp in the example can be also designed using the G3 instruction. This this case, the points must be inserted by counter-clockwise starting from the origin (G0).

Modulset

The modulsets are contained between the following markers:

[INIT_LIBMSET]	Start of section.
...	
[END_LIBMSET]	End of section.

Definition of a modulset

[INIT_MSET]	Start of definition.
[LINS]...	Length of insertion side.
[LSEC]...	Length of secondary side.
[DIST]...	Distance between sides.
[FILEBMP]...	Path/FileName of bitmap image to be displayed in modulset properties window.
[MAXNUMTYPE]...	Number of modulsets available.
[NOME]...	Modulset name (must be unique in the file).
[END_MSET]	End of definition.

Seal

There is only one seal, and it is described in the following section:

[INIT_LIBGU]	Start of section.
[W]...	Seal width.
[FILEBMP]...	Path/FileName of bitmap image to be displayed in seal properties window.
[END_LIBGU]	End of section.

Loading Aid Bars

The loading aid bars are contained between the following markers:

[INIT_LIBBAC]	Start of section.
...	
[END_LIBBAC]	End of section.

Definition of a loading aid bar

[INIT_BAC]	Start of definition.
[W]...	Bar width.
[OFFX]...	Offset in X of bar relative to centre of crossbeam.
[FILEBMP]...	Path/FileName of bitmap image to be displayed in bar properties window.
[MAXNUMTYPE]...	Number of bars available.
[NOME]...	Bar name (must be unique in the file).
[END_BAC]	End of definition.

Movable Bars

The movable bars are contained between the following markers:

[INIT_LIBBM]	Start of section.
...	
[END_LIBBM]	End of section.

Definition of a movable bar

[...]	Start of definition. Values: INIT_BLSX (left side bar), INIT_BCSX (left central bar), INIT_BCDX (right central bar), INIT_BLDX (right side bar).
[W]...	Bar width.
[OFFX]...	Offset in X between centre of bar and point of supporting stops in contact with workpiece.
[FILEBMP]...	Path/FileName of bitmap image to be displayed in bar properties window.
[EN_BM]...	Bar enabling. Values: 0 (assent), 1 (present).
[ZONE]	Bar position. Values: 1 (left side bar), 2 (left central bar), 3 (right central bar), 4 (right side bar).
[OFFXQUOTA]...	Offset in X of table dimension between machine origin and 0 on metric rod applied to base.
[NOME]...	Bar name (must be unique in the file).
[...]	End of definition. Values: INIT_BLSX (left side bar), INIT_BCSX (left central bar), INIT_BCDX (right central bar), INIT_BLDX (right side bar).

Supporting Stops

The supporting stops are contained between the following markers:

[INIT_LIBBAP]	Start of section.
...	
[END_LIBBAP]	End of section.

Definition of a supporting stop

[INIT_BAP]	Start of definition.
[R]...	Stop radius.
[INIT_CORPO]...	Start definition of the body shape.
[G0] X=...,Y=...	(Stop with generic type body). See below.
[G1] X=...,Y=...	(Stop with generic type body). See below.
[G2] X=...,Y=...,I=...,J=...,R=...	(Stop with generic type body). See below.
[G3] X=...,Y=...,I=...,J=...,R=...	(Stop with generic type body). See below.
[END_CORPO]...	End definition of the body shape.
[FILEBMP]...	Path/FileName of bitmap image to be displayed in the stop properties window.
[MAXNUMTYPE]...	Number of stops available.

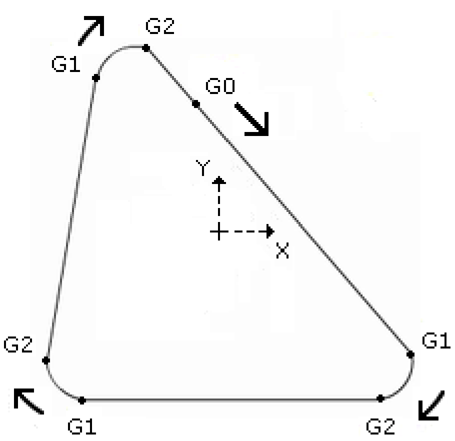
[NOME]... Stop name (must be unique in the file).
[END_BAP] End of definition.

Configuration of a generic type body for stop

If the stop is equipped with a generic type body (for example, triangular with rounded corners), several specific parameters must be set, as indicated in the list of parameters for the definition of a stop.
In particular, to define the profile of the body, the following instructions are available:

[G0] X=...,Y=...	Starting point of the body profile. X = X co-ordinate; Y = Y co-ordinate.
[G1] X=...,Y=...	Straight line segment. X = end point X co-ordinate; Y = end point Y co-ordinate.
[G2] X=...,Y=...,I=...,J=...,R=...	Clockwise arc. X = end point X co-ordinate; Y = end point Y co-ordinate; I = X co-ordinate of centre of arc; J = Y co-ordinate of centre of arc; R = radius of arc.
[G3] X=...,Y=...,I=...,J=...,R=...	Anti-clockwise arc. X = end point X co-ordinate; Y = end point Y co-ordinate; I = X co-ordinate of centre of arc; J = Y co-ordinate of centre of arc; R = radius of arc.

Example:



[INIT_BAP]
[R] 12.5
[INIT_CORPO]
[G0] X=-25,Y=75
[G1] X=100,Y=-75
[G2] X=75,Y=-100,I=75,J=-75,R=25
[G1] X=-75,Y=-100
[G2] X=-100,Y=-75,I=-75,J=-75,R=25
[G1] X=-75,Y=75
[G2] X=-50,Y=100,I=-50,J=75,R=25
[END_CORPO]

In order to use these instructions, several conditions must be respected:

- The profile of the body must be defined by at least three points;
- All of the instructions used (G0, G1, G2, G3) must be inserted so as to define a closed perimeter made up of succesive points, continuous with respect to each other.

The stop in the example can be also designed using the G3 instruction. This this case, the points must be inserted by counter-clockwise starting from the origin (G0).

APPENDIX L. Blade Cuts Management

Tool Configuration

Auxiliary Units

The auxiliary units (Blade unit, Lock slot unit) can be fitted with pneumatic rotation or continuous rotation.
Pneumatic rotation makes it possible to rotate the blade at 90 degree intervals (based on the tool characteristics). In tooling-up, the pneumatic rotation tools are identified by:

- even tool numbers, in non rotated position (parameter "Offset R" = 0);
- odd tool numbers, in rotated position (parameter "Offset R" = -90).

Continuous rotation is used to continuously rotate the blade.

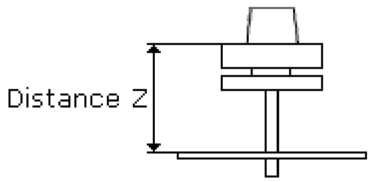
Vector Axis

The Vector axis, which can be mounted both on the main spindle and on the auxiliary units, is used to rotate the blade during machining on table XY. For slanted cuts with Vector axis, the IC parameter of the (X)G0R instruction must always be set on 3: this way, Xilog Plus will align the blade with the cutting direction.

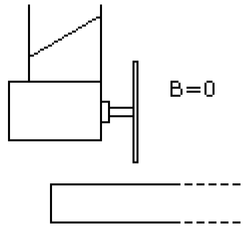
Prisma Head

The Prisma head is used to move the blade in any direction in a three dimensional space during machining. The blade tool mounted on a Prisma head requires a particular configuration:

- parameter "A" must be set on 0;
- parameter "Offset R" must be set on 0;
- parameter "Distance Z" represents the entire distance between the cone base and the internal face of the blade (it is equal to the sum of the "Distance Z" and the "Distance D" parameters of a blade on an angular head).



The Prisma head automatically positions the spindle in a horizontal position, with the orthogonal blade on the work table. The rotation angle of the blade is not programmed during tooling-up, instead it is programmed directly in the G0R/XG0R instruction with the parameter B (the use of this parameter is reserved to the Prisma head).



Note: if B=0, the lead in side programmed with parameter IC/I of instructions G0R/XG0R is not observed.

Programming

Orthogonal Cuts on Face 1

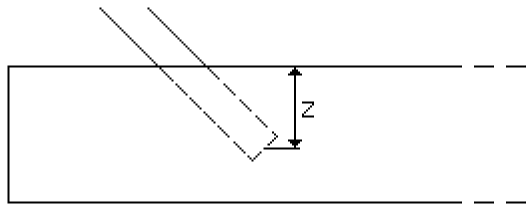
Orthogonal cuts on face 1 may be programmed with the instructions G0 and G1, or XG0 and XL2P. In addition:

- in the case of cuts for the entire length of the piece, the instructions (X)GIN and (X)GOUT can be used to ensure a constant cut depth;
- in the case of hollow cuts, the cut depth can be corrected at the beginning and ending point by setting the instruction C with a value of 3 (depth correction).

Slanted Cuts

Slanted cuts can be programmed with the instructions (X)G0R and (X)G1R. In slanted cuts:

- the instructions (X)GIN/(X)GOUT cannot be used;
- ool correction cannot be used; the cut co-ordinates are always referred to the centre point of the blade cutting edge.



Note. On face 1, the slanted cuts can also be made using the (X)PL instruction. In this case, the tool correction can also be used.

Machining Direction

The machining direction of a blade can be:

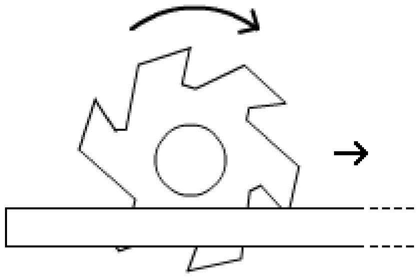
- pro-feed direction: the blade advances by pushing its teeth downward;
- counter-feed direction: the blade advances by pushing its teeth upward.

The machining direction may be set in the tooling-up of a tool with the “Machining direction” parameter (with values: +/-), but remember that the real machining direction also depends on other factors:

- the direction in which the blade is mounted on the spindle;

- the rotation direction of the spindle.

Mechanical conditions permitting, it is advisable to ensure that once the blade is mounted on the spindle, it rotates in front of the operator so that the clockwise movement is in the feed direction.

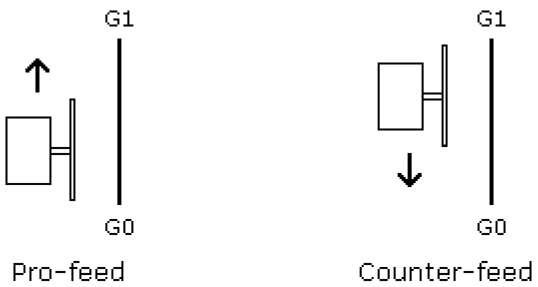


In this case, to set the machining direction with the "Machining direction" parameter, use:

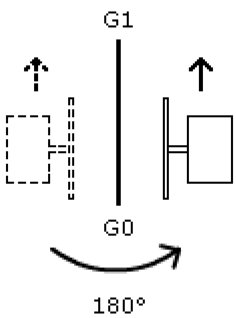
- pro-feed: +;
- counter-feed: -.

During machining operations with the blade, Xilog Plus manages tools using the following principles:

- the gear box is positioned to the left of the cutting direction;
- the counter-feed machining direction is obtained by inverting the machining path direction.



Exception: **orthogonal cuts made with Vector axis and Prisma head**. In these cases, the blade is rotated 180°, without inverting the machining path direction.



To gain time during machining operations which require alternating the machining direction pro/counter-feed, the same tool can be configured as two distinct, but equal, tools (except for the different machining direction); however, each of the two tools is assigned the same aggregate number (parameter "Number total"). This way Xilog Plus will reduce to a minimum the pauses between the machining operations.

APPENDIX M. Accessory Functions

Calculator

If in any given environment the cursor is located on an editable field that can receive only numerical data, the calculator display shows the value present in that field. Press **[Enter]** to paste the value in the display of the calculator to the editable field where the cursor is located.
Apart from using the calculator with the mouse, the following keyboard functions are available (where X is the value displayed on the screen and Y is the value in the calculator memory).

[S]	Sine of X.
[I]	Arcsine of X.
[C]	Cosine of X.
[O]	Arcsine of X.
[T]	Tangent of X.
[N]	Arctangent of X.
[P]	X = constant pi value.
[R]	Square root of X.
[L]	Cancel Memory: Y = 0.

Backup and Restore Functions

With the backup function is it possible to copy all files that are contained in a main directory of the Xilog Plus editor onto a disk or tape (or on the network). The backup function includes all the files contained in all the subdirectories of the main directories, except in the case of configuration files.
The user may choose whether to perform a selective Backup (for example, of the programs only) or a complete Backup. If the floppy disk unit is selected, it is not possible to make a backup copy of the files which exceed the capacity of the floppy disk. This is because the files are copied onto the selected disk unit without being compressed. When the space available on the disk runs out, the user is prompted to insert a new disk in the backup unit.
With the restore function it is possible to restore onto a main disk any files previously saved during a backup session. If the floppy disk unit is selected, since the disks are not automatically catalogued, the user must take care to insert all the disks containing the backup copy, one at a time (regardless of order) when carrying out the restore procedure. When the restore procedure has been completed on each disk, the system prompts the user to insert a new disk (if the file is not yet complete).

APPENDIX N. Notes

PGM Programs Management

Editing

You can edit and graphically view programs with up to approximately 32700 lines.

Execution

You can run all editable programs which produce a maximum of around 65500 operating instructions. The operative instructions are: H, REF, F, BR, B, N, TA, G0, G1, G2, G3, SET, MSG, ISO, VT, VROW, ENDVT, G0R, G1R, G2R, G3R.
If necessary, the limit of around 65500 operating instructions may be extended; however, modification is not recommended since system performance may be compromised.

Direct Management of ISO Programs

Some machines fitted with NUM® control also have direct control of ISO programs. An ISO program is a file with the extension CNC which can be freely edited and graphically displayed; this function is demanded by Numeric Control and therefore only operative when Numeric Control is connected. The names of the CNC files must be chosen according to a NUM® convention: therefore each name must be made up of no more than 5 numeric figures followed by a 0 (zero).

Editing

It is possible to edit and graphically view programs that measure up to approximately 65000 characters; for programs of a larger size it is necessary to use an editor outside Xilog Plus.
Programs below the size limit can only be graphically viewed if the Numeric Control memory has enough space to hold them. Moreover, any sub-programs must be transferred to the Numeric Control memory in advance.

Execution

The limit of around 65000 characters does not apply when running programs. However, the following considerations apply:

- If the program to be run contains jumps, you must set the “Slotting in CNC” parameter of the configuration file gendata.cfg to 0. In such cases, the program will only run if the Numeric Control memory has enough space to hold it. Moreover, any sub-programs must be transferred to the Numeric Control memory in advance.
- If the program to be run does not contain jumps, set the “Slotting in CNC” parameter of the configuration file gendata.cfg to 1. In such cases, the program is transferred to the Numeric Control in blocks. Therefore, the only limit on its size is the space available on the hard disk of the Personal Computer. However, any sub-programs must be transferred to the Numeric Control memory in advance.

M02

All M02 blocks in CNC programs must be preceded by an M201 block; if the M201 block is missing the program continues to run correcting, but when it terminates the color of the work area will remain green and the counter for the pieces produced will not be reset; under these condition press key **[F10]** (Reset) to inform the Xilog Plus Machine Panel that the program has terminated. If you are working with an old CNC program, edit it and manually write the required M201 code.

M20

The presence of code M20 instead of M2 (in pendulum machining) means that machining does not have to end in pass mode, therefore, gaining time during the passage from one machining area (e.g.: area A) to another (e.g.: Area B). If using an old CNC program, edit it and manually enter the desired M20 code.

Work Onset

On some machines the head which must perform the initial machining on the panel descends before or during movement. The dimension Z to which the head moves is defined **exclusively** according to the panel Z dimension (value of program title field DZ). Therefore, the size of any panels present on the other machining areas is not considered. If there is any danger of collision with the other panels, before the program initial machining insert one or more N or XN instructions, which control the movement in safety. The same procedure applies for subsequent machining, if the heads must move over other panels.

Transfer above Panel

The movements of the heads as they approach the starting point of the subsequent machining are optimized according to the tools/assemblies and the panel Z dimension. If there is no safety dimension to which the head can move above the panel with the tools/assemblies mounted, a “No safety dimension above work piece” error message is issued.

Unrecoverable cases

In the following cases the “No safety dimension above work piece” error cannot be recovered:

1. If movement is possible with the pneumatic head raised:

- BR or XBR with tool configured on face 0
- TA or XTA on face 1
- B or XB on inclined worktable

2. If movement is not even possible with the pneumatic head raised:

- BR or XBR with tool configured on face 0
- TA or XTA
- B or XB on inclined worktable

The other cases may be recovered using the SET DONTCARE instruction.

NO EDIT Selector

On the control panel of the bore-milling machine there is a key type selector with which to enable/disable the possibility of editing and permanently saving the data contained inside a program, or the software configuration, for tooling files and ISO files. With this function disabled, it is still possible to access the above Editors, but no changes made to any open documents can be saved (access for viewing purposes only).