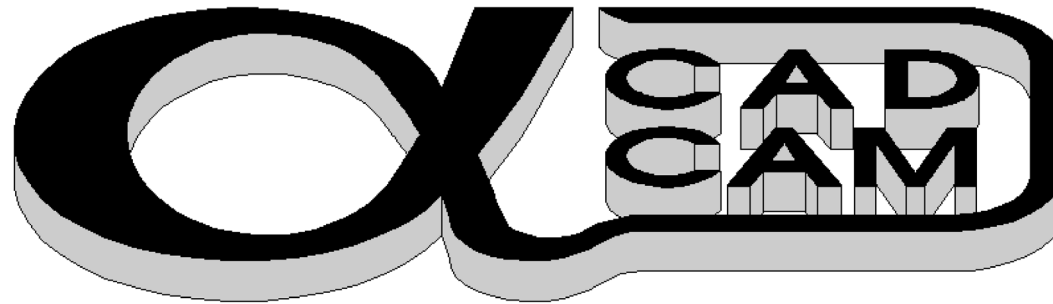


Licom Systems Ltd.,



Training Supplement

AlphaCAM / Xilog Interface

Revision A




Licom Systems Ltd.







Table of Contents

Revision A

NC Program Procedure AlphaCAM	1
NC Program Procedure Xilog Individual Parts	2
NC Program Procedure Xilog Nested Parts	3
AlphaCAM & XilogPlus.	4
AlphaCAM / XilogPlus Dialog	4
Select Tool	4
Rough or Finish	5
Pocketing	6
Drilling / Boring	7
Xilog Special Options.	8
Interpolation with a Vector Axis	8
Cut With Saw	8
Work Planes	9
Engraving	9
Definitions and Configuration.	10
Tooling Definitions.	10
Machine Configuration file	10
Hood Settings	10
Post Processor Configuration	11
Unit of measurement:	11
Profile/boring lead-in speed:	11
Generating arc data ...	11
Moving axes at the end of a program	11
XilogPlus Engraving Tool Definition.	12

NC Program Procedure AlphaCAM

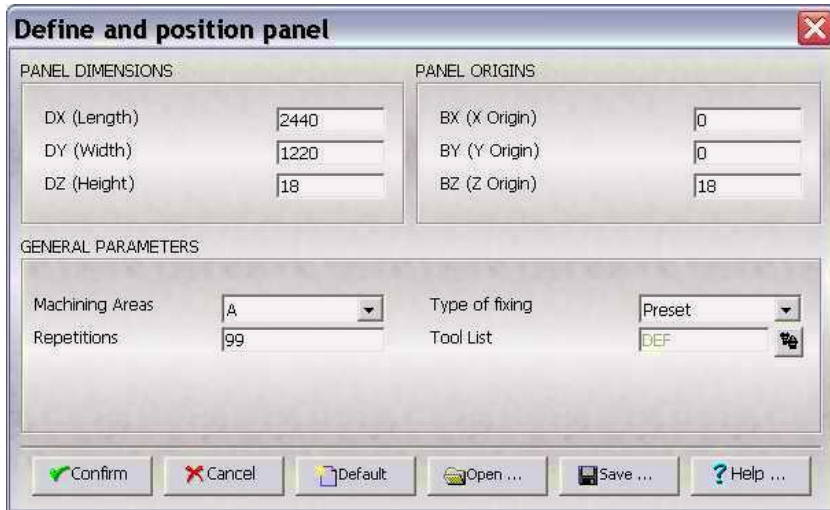
- 1) Create the Process Planning
- 2) Select the relevant Post Processor 
- 3) Create/Import the required geometry.
- 4) Ensure Geometry is in Continuous Profile
- 5) Set Tool Directions  (Direction, Side and Start Point)
- 6) Select Tool 
- 7) Select Machining Option :



	Rough or Finish	for	Profiling Cuts
	Pocketing	for	Area Clearance
	3D Engrave	for	Engraving Profile cuts.
	Drilling	for	Single tool drilling
	Manual ToolPath	for	ToolPath Point to Point creation.
	Multi Drilling	for	Drilling using a Multi Drill Head.
- 8) OutPut Nc 





NC Program Procedure Xilog Individual Parts

Bold Items are selected using the Xilog Menu



- 1) Create the Process Planning
- 2) Select the relevant Post Processor 
- 3) **From the Xilog Menu Set the Panel Details and the Tool List**
- 4) Create/Import the required geometry.
**If the geometry is required on the edge of the panel.
Use the Select XilogPlane menu option**
- 5) Ensure Geometry is in Continuous Profile
- 6) Set Tool Directions  (Direction, Side and Start Point)
- 7) **Select Tool** “When working with a Xilog controls it is Very Important to select the tools from the XilogPlus menu and NOT the AlphaCAM Machine menu.”
- 8) Select Machining Option :

Rough or Finish	for	Profiling Cuts
Pocketing	for	Area Clearance
 3D Engrave	for	Engraving Profile cuts.
Boring	for	Drilling operations
Interpolation With Vector Axis	for	Cutting round the edge of the part
Cut with Blade	for	Cutting with Saw
- 9) OutPut Nc 





Manual ToolPath for ToolPath Point to Point creation.



NC Program Procedure Xilog Nested Parts

Bold Items are selected using the Xilog Menu

Parts with side machining cannot be nested. Parts **MUST NOT** be created on a **Panel**

- 1) Create the Process Planning
- 2) Select the relevant Post Processor 
- 3) Create/Import the required geometry. Ensure Geometry is in Continuous Profile
- 4) Set Tool Directions  (Direction, Side and Start Point)
- 5) **Select Tool** “When working with a Xilog controls it is Very Important to select the tools from the XilogPlus menu and NOT the AlphaCAM Machine menu.”
- 6) Select Machining Option :

Rough or Finish for **Profiling Cuts**

Pocketing for **Area Clearance**



3D Engrave for Engraving Profile cuts.

Boring for **Drilling operations**

Cut with Blade for **Cutting with Saw**



Manual ToolPath for ToolPath Point to Point creation.

- 7) Save each Part Item “DO NOT SAVE PARTS With a MATERIAL DEFINED”

Repeat Steps 1 to 8 for each part to be nested.

- 8) Create Sheet(s)

- 9) Nest Components  (From Drawing or Nest List)




- 10) OutPut Nesting.

AlphaCAM & XilogPlus.

When using a XilogPlus controlled machine tool certain options are MANDATORY.

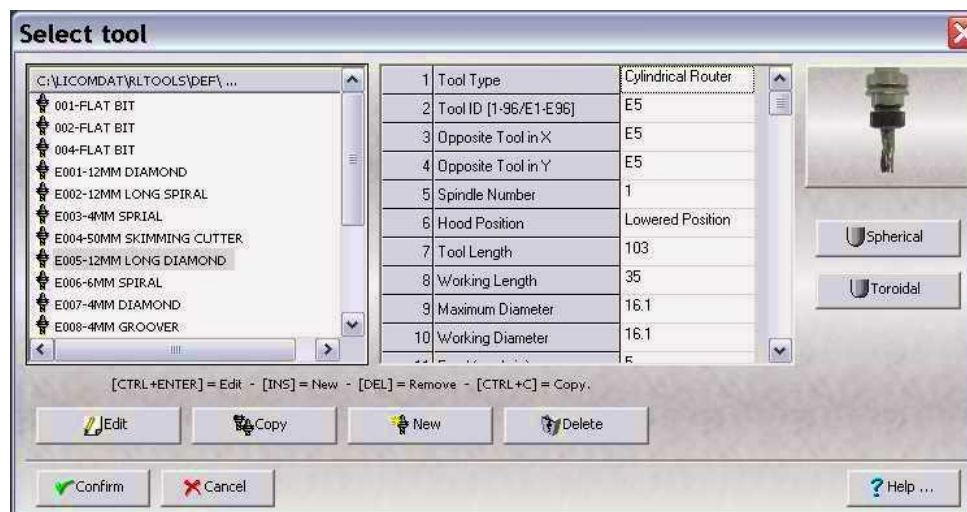
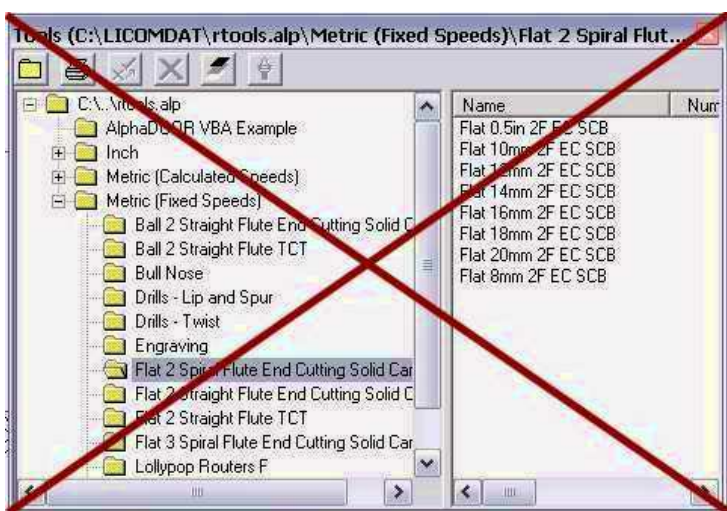
- 1) Tools should be defined at the Machine and imported into AlphaCAM.
- 2) Tools MUST only be selected using the XilogPlus menu.
- 3) Individual Parts MUST be created on a Panel.
The panel should be created using the XilogPlus Menu.
- 4) Parts for NESTING Must Not be created on a panel.
- 5) Machining Options can be applied using
either the XilogPlus Menu or the AlphaCAM Machine menu.

AlphaCAM / XilogPlus Dialog

To Change or Enter values in the XilogPlus dialog options you have to  click in the field to activate it. When a field is activated the associated graphic view is changed to give a pictorial representation of the option. Once the value is entered or selected you have to press  or  to confirm the entry.

Select Tool

When working with a XilogPlus control tools MUST be selected via the XilogPlus menu.



Rough or Finish

Rough / Finish

Sides:

Vertical

Sloping

Profiled

All Geometries

Selected

Partial

Cancel

Rough / Finish

Op No. 1 Tool: FLAT 10MM 2F EC SCB

Change Tool...

Compensation:

APS Tool Centre Machine Comp (G41/G42)

G41/G42 on Tool Centre

Apply Compensation on Rapid Approach / Retract

XY Corners:

Roll Round Straight Loop

Loop Radius: 0 Knife Loops

OK Cancel

Rough / Finish - Sides : Vertical

Z Levels (or Distance from Plane):

Safe Rapid Level: 55 Rapid Down To: 6

Material Top: 0 Final Depth: -10

Number of Cuts: 2 Bi-Directional (Open paths only)

NC Code for Multiple Cuts:

Linear Subroutines

Depths of Cut:

Equal Specified

Thickness of FIRST Cut: 0

Thickness of LAST Cut: 0

OK Cancel

Rough / Finish - Tool: FLAT 10MM 2F EC...

Tooling:

Tool Number: 1 Offset Number: 1

Diameter: 10 Spindle Speed: 18000

Down Feed: 0.75 Cut Feed: 3500

Machining:

Stock to be Left: 0

Coolant:

None Mist Flood Through Tool

OK Cancel

Define Tool path - Roughing/Finishing.

1	Compensation	NO
2	Distance from finished (mm)	0
3	Pass around edges	Constant radius
4	Radius of pass around edges (mm)	0
5	"D" return upstroke Z dimension	20
6	Machining Z dimension	0
7	Number of passes	1
8	Type of passes	Equal
9	Bidirectional	No
10	Height of first cut (mm)	0
11	Height of last cut (mm)	0
12	Tool speed of rotation (rpm)	15000
13	Feed speed (mm/min)	0
14	Tool machining speed (mm/min)	5
15	Lead-in into profile mode	None
16	Lead-out of profile mode	None
17	Line length: Tool Radius x	
18	Arc Radius: Tool Radius x	
19	Approach angle (°)	
20	Overlap (mm)	
21	Sloping lead-in	
22	Sloping lead-out	
23	Hood Position	Lowered Position
24	Enable machining	Yes

Tool: 103 - 4MM SPRIAL, L=80.55, D=4.5, Max. V=8, Max. S=18000

Tool: 103 - 4MM SPRIAL, L=80.55, D=4.5, Max. V=8, Max. S=18000

Tool Geometries Tool Path Open ... Save ...

Confirm Cancel Help ...

To Change or Enter values in the XilogPlus dialog options you have to click in the field to activate it. When a field is activated the associated graphic view is changed to give a pictorial representation of the option. Once the value is entered or selected you have to press **Enter** or **Tab** to confirm the entry.

Pocketing

Pocketing

Sides:

- Vertical
- Sloping
- Profiled
- Multi-Level

All Geometries Selected

Op No. 1 Tool: FLAT - 10MM

Type:

- Contour
- Linear
- Spiral

Final Pass Around Islands:

- Full
- Partial
- None

Start Cutting at:

- Inside
- Outside

Set Start Point(s)

CONTOUR POCKET - Sides : Vertical

Z Levels (or Distance from Plane)

Safe Rapid Level 50 Rapid Down To 5

Material Top 0 Final Depth -10

Number of Cuts 3

NC Code for Multiple Cuts:

- Linear
- Subroutines

Cutting Order:

- by Zone
- by Level

Depths of Cut:

- Equal
- Specified

Thickness of FIRST Cut 3

Thickness of LAST Cut 1

CONTOUR POCKET - Tool: FLAT - 10MM

Tooling:

Tool Number 3 Offset Number 3

Diameter 10 Spindle Speed 23900

Down Feed 4800 Cut Feed 9600

Machining:

Stock to be Left 0 Width of Cut 5

Coolant:

- None
- Mist
- Flood
- Through Tool

Define Tool path - Pocketing.

1	Type of pocketing	1-Concentric
2	Final pass around islands	Complete
3	Start cut	Centre
4	Distance from finished (mm)	0
5	"D" return upstroke Z dimension	20
6	Machining Z dimension	0
7	Number of passes	1
8	Type of passes	Equal
9	Height of first cut (mm)	0
10	Height of last cut (mm)	0
11	Tool speed of rotation (rpm)	15000
12	Feed speed (mm/min)	0
13	Tool machining speed (mm/min)	5
14	Cut width (mm)	2.25
15	Hood Position	Lowered Position
16	Enable machining	Yes

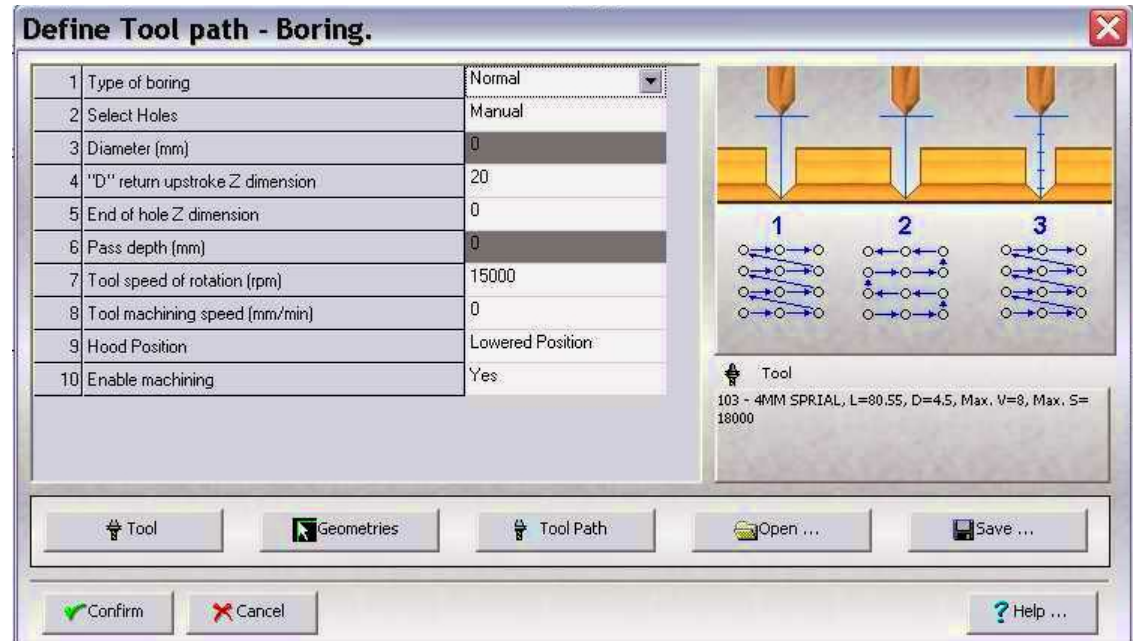
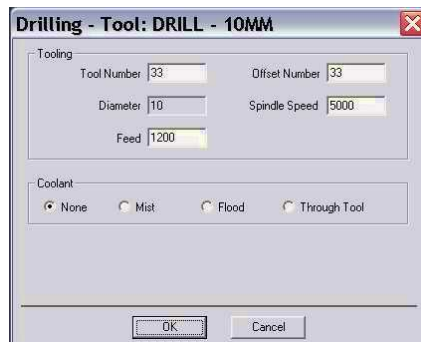
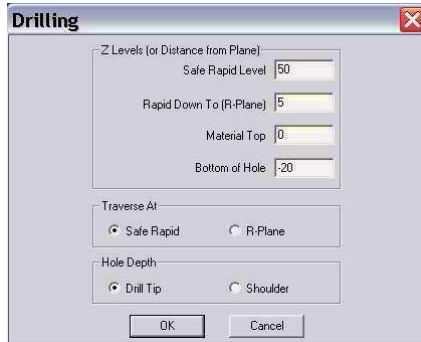
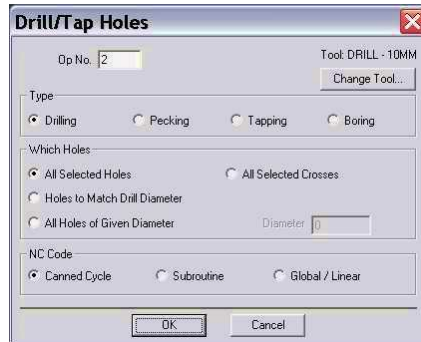
Tool: 103 - 4MM SPRIAL, L=80.55, D=4.5, Max. V=8, Max. S= 18000

Buttons: Tool, Geometries, Tool Path, Open ..., Save ...

Buttons: Confirm, Cancel, Help ...

To Change or Enter values in the XilogPlus dialog options you have to click in the field to activate it. When a field is activated the associated graphic view is changed to give a pictorial representation of the option. Once the value is entered or selected you have to press **Enter** or **Tab** to confirm the entry.

Drilling / Boring

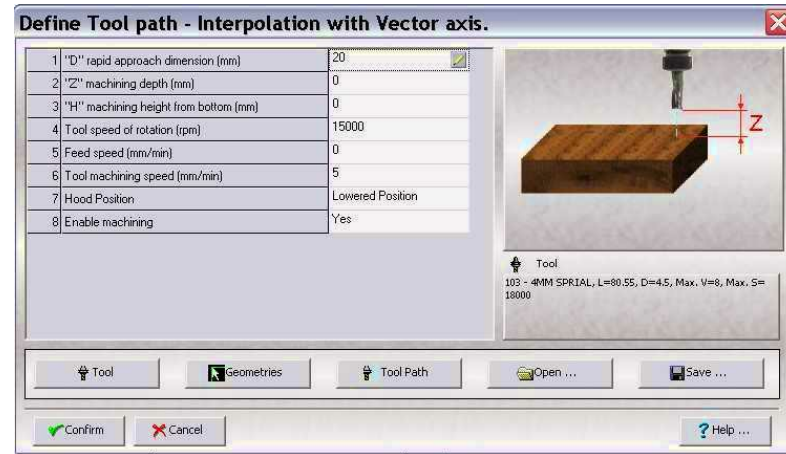


To Change or Enter values in the XilogPlus dialog options you have to click in the field to activate it. When a field is activated the associated graphic view is changed to give a pictorial representation of the option. Once the value is entered or selected you have to press **Enter** or **Tab** to confirm the entry.

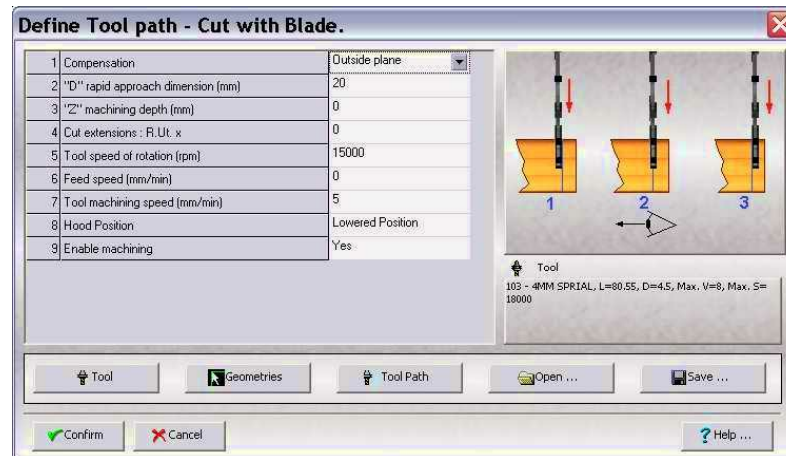
Xilog Special Options.

There are two specialist options on the xilog menu which is achieved using the Rough or Finish option from the AlphaCAM Machine menu.

Interpolation with a Vector Axis

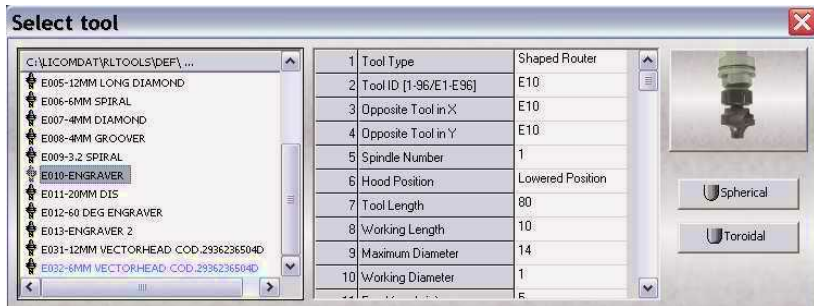
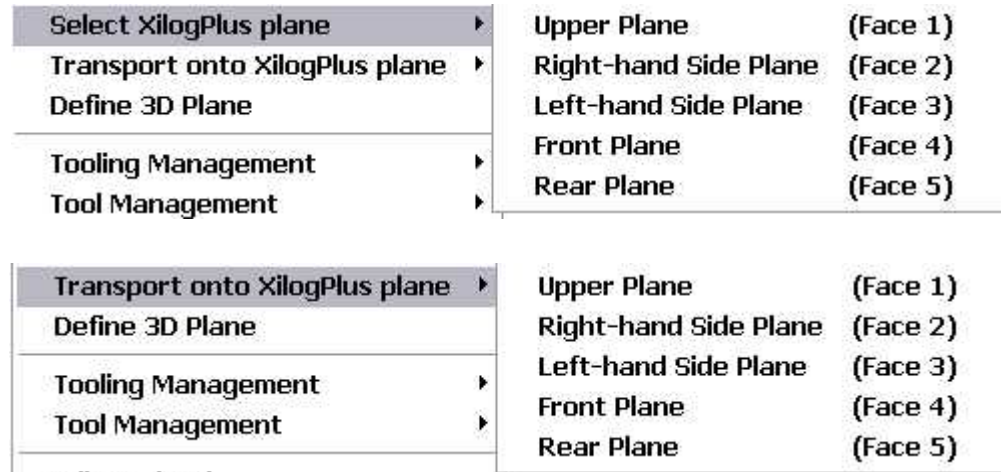


Cut With Saw



Work Planes

When creating geometry on work planes ie panel edges, or manipulation of geometries from one face to another the **Select XilogPlus Plane** or **Transfer onto XilogPlus Plane** options should be used.



Engraving

Engraving is done using the **3D Engraving** option from the AlphaCAM Machine menu. You should however select an Engraving Tool using the XilogPlus menu. The tool **MUST** be of type **SHAPED TOOL**.



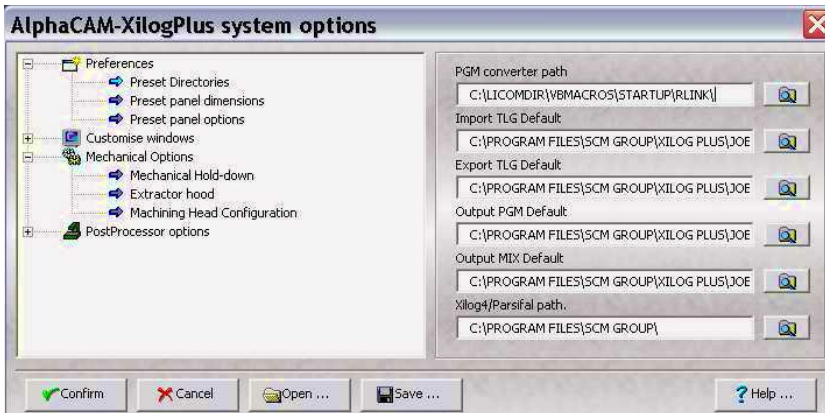
Definitions and Configuration.

Tooling Definitions.

When using the XilogPlus machine tool controller interface. All tooling definitions should be carried out at the machine and the tool def information then imported into alphaCAM.

Although there is an option to create a new tool for XilogPlus on the AlphaCAM computer it is preferable to create new tools on the Machine as this ensures compatibility of information.

The exception to this is User Defined (Shaped) Tools, these have to be defined in AlphaCAM and exported to the Tlg file on the machine. If the Tlg file is not exported, when a new tool is added at the machine and the Tlg file is imported into AlphaCAM the engraving tool is overwritten and the geometry is lost.



Machine Configuration file

When off line programming, the XilogPlus CFG folder on the machine computer must be copied to the XilogPlus folder on the AlphaCAM computer.

The tooling.TLG and the *.PGM files are usually stored in the preset directory:-

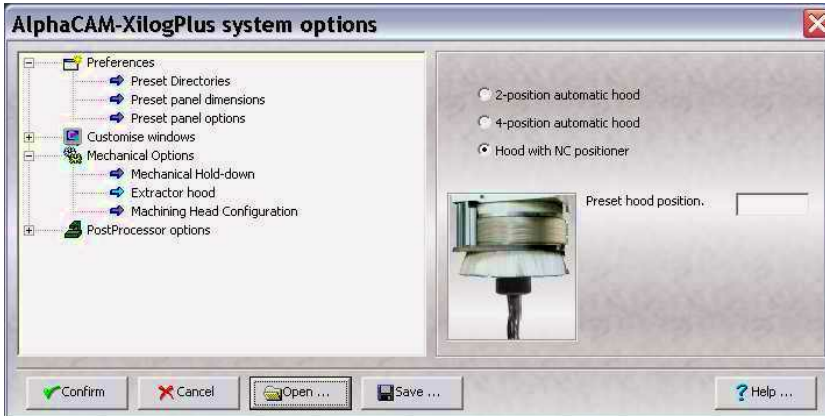
Network Path to Machine\Program Files\SCMGroup\XilogPlus\Job

Some setups require the .PGM file to the local Drive and transferred to the machine later.

C:\Program Files\SCMGroup\XilogPlus\Job

Hood Settings

To activate Automatic Hood there must be "E=" in the program. This is set in the **XilogPlus | Options | Mechanical Options** and the **Extractor Hood** is set to **Hood with NC Positions** with a blank value (NOT 0)



Post Processor Configuration

There are several settings that affect the program output. These are altered according to your requirements and machine settings.

Unit of measurement:

Sets the unit of measurement for tool data during tooling importation and tooling exportation.

Profile/boring lead-in speed:

Sets the default feed speed for tool lead-in into the workpiece or for execution of boring, which will be proposed during tool path parameter definition.

Generating arc data ...

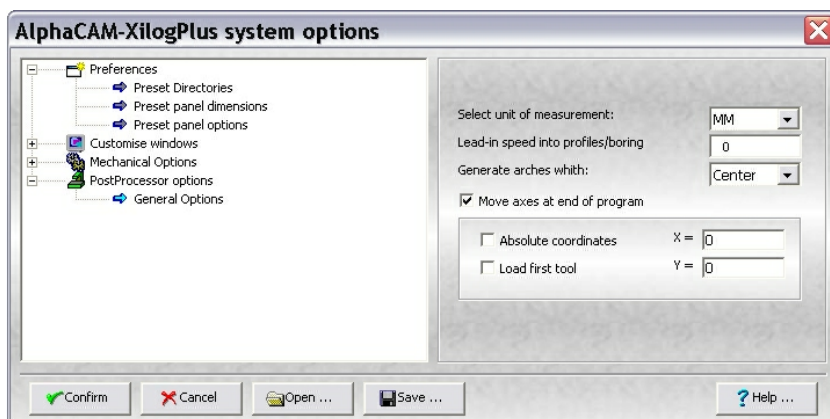
Sets the Post-process arc generation mode. Arcs can be generated by programming the I and J centre coordinates or by programming their radius. The Xiso language instructions that enable these two programming modes are XA2p and XAR2. When a workpiece is finished the arc generation mode that has been selected is no longer significant but the arc generation radius data (XAR2) makes the Xiso program easier to read and edit manually..

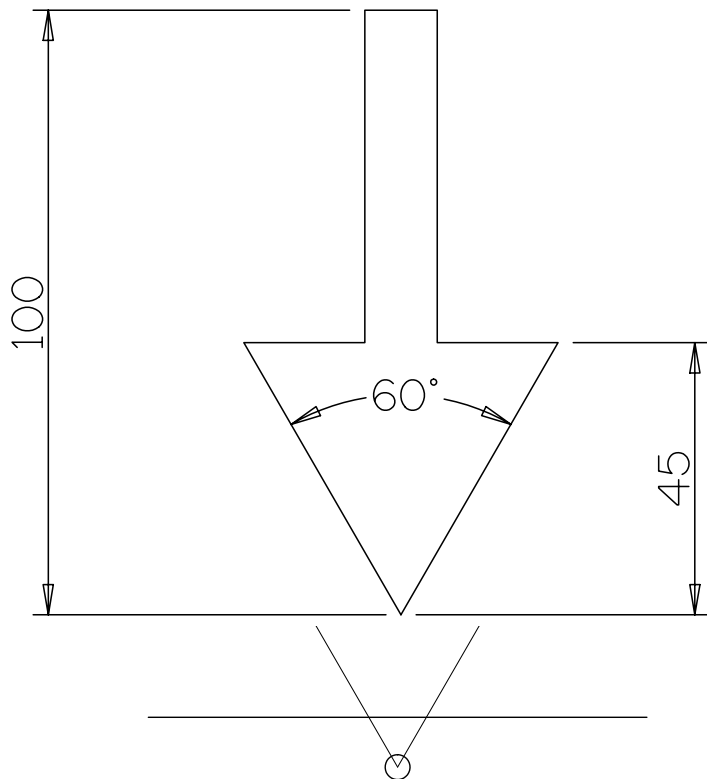
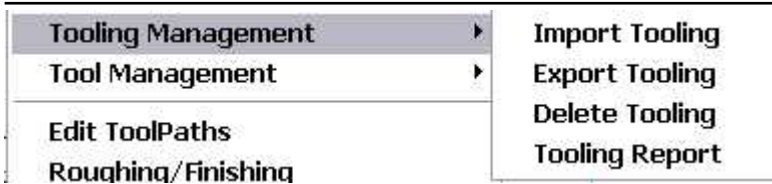
Moving axes at the end of a program

This option enables the Xiso XN instruction to be inserted automatically at the end of each Post-P generated program. This instruction is responsible for moving the main X,Y and Z machine axes to a given position by specifying the coordinates in relation to the machine origin without taking into account the specular areas (Absolute coordinates) or the coordinates in relation to the piece origin, taking into account the specular areas.




It also enables the first tool used in the program to be fitted onto the main spindle by activating the tool changeover while the machine is moving into the programmed position and the operator is unloading the finished piece and loading on a new one.

In this way when the workpiece program starts up, the machine will already have fitted the tool needed for the first machining stage.





XilogPlus Engraving Tool Definition.

- 1) Define the Engraving tool on the machine and import the tlg file into AlphaCAM.
- 2) Draw the geometry for the engraving tool to match that of the actual tool on the machine. Similar to the picture opposite.
- 3) In order to utilise this drawing to define an engraving tool via XilogPlus you must add a line to define the cutting diameter and a circle to define the controlling position of the tool. As shown.
- 4) Using **XilogPlus | Tool Management | Select Tool** highlight the engraving tool in the list.
- 5) Select the  Button to edit the tool then select Shaped Tool from the list.
- 6) Select the  Button you will then be prompted to
Select the Tool Geometry
Select the Line defining the Controlling diameter
Select the Circle defining the Controlling point
- 7) Select the  Button to save the changes.
- 8) Using **XilogPlus | Tooling Management | Export Tooling** export the tool file from AlphaCAM to the .Tlg file.
- 9) Copy the .Tlg file to the Machine tool. (If required)



Licom Systems Ltd. Training Supplement
AlphaCAM / Xilog Interface
