LICOM AlphaCAM

CAD/CAM system for Windows™

SIMPLE MILLING
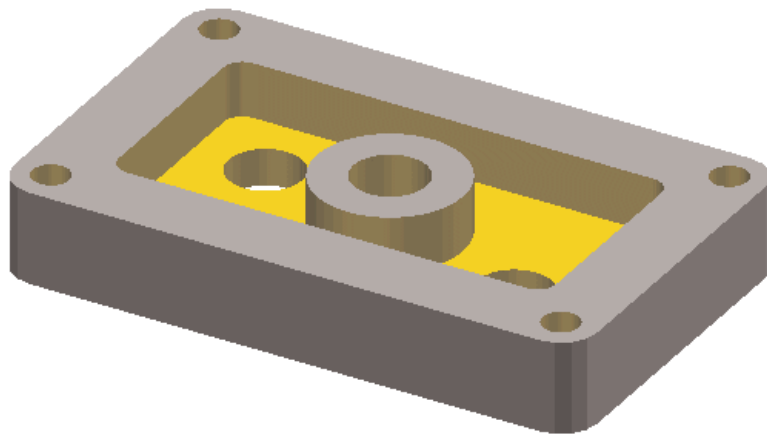### TUTORIAL

Table of Contents

Table of Contents

Conventions When Using The Tutorial

Notes and comments are in *Italics* to separate them from the main text. If there are different ways of performing the same command or option, these are also in *Italics* and can be ignored the first time you work through the tutorial.

The symbol ☞ indicates a new command for you to action.

The symbol ☞ indicates a sub part of command for you to action

The HELP file is very comprehensive. When you have finished the tutorial, please take time to have a look at it. **Select HELP | Contents**.

Introduction

This **AlphaCAM** tutorial has been designed to give you a flavour of how **AlphaCAM** works, how interactive it is, how easy it is to learn and how easy it is to use. We assume that you are familiar with the concepts involved in CNC programming and have a reasonable understanding of your computer and the Windows operating system. **AlphaCAM** has been developed as a true 32-bit Windows 95 or NT application, so if you use other Windows programs you will be familiar with features such as floating button bars, tabbed dialog boxes, etc. If not, you should look at **HELP | Contents | Screen Layout**.

AlphaCAM systems are available for all machining disciplines. Each one is specifically designed for the machine type, but the look and feel of all the systems is standard. This tutorial can be used with both Standard and Advanced **AlphaCAM**, Mill and Router, modules. The tutorial describes the construction and machining of the geometry for the following drawing.

If you have not already done so, start the **'Advanced Mill'** module. This will take you into the graphical portion of the system. Your screen will look similar to the one below.
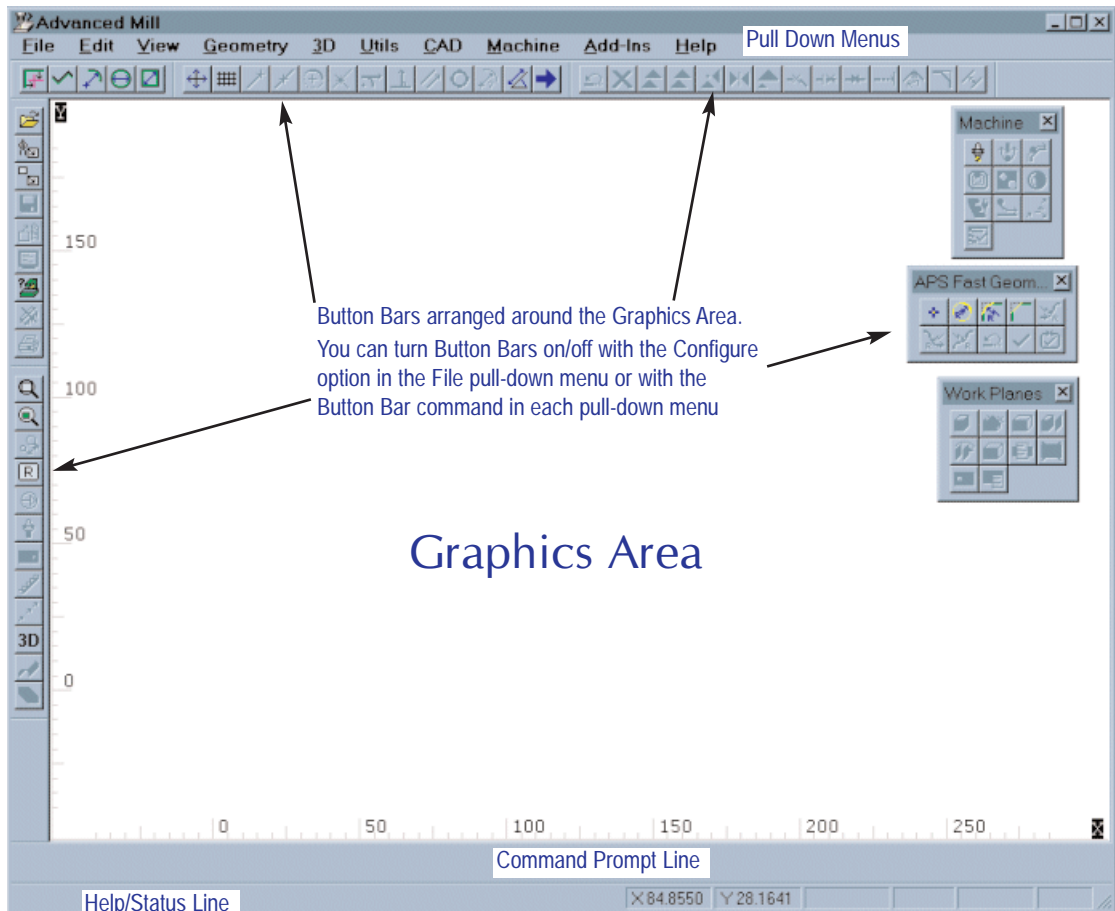
During this tutorial, we tell you where to find commands on the pull-down menus. If there is a button for the command, this will also be shown.

Take the trouble to locate the buttons on your screen. You can speed up your work by avoiding having to pull down menus and side menus to click on commands, when one click on a button will suffice.

To see what command a button performs, place the screen pointer over the button for a couple of seconds, and a prompt will appear beside the pointer.

The Part Geometry

This drawing is the part on which you will work in this tutorial. It was drawn, dimensioned and printed using **AlphaCAM**. By the end of the tutorial, you will have created the NC program for the drawing.
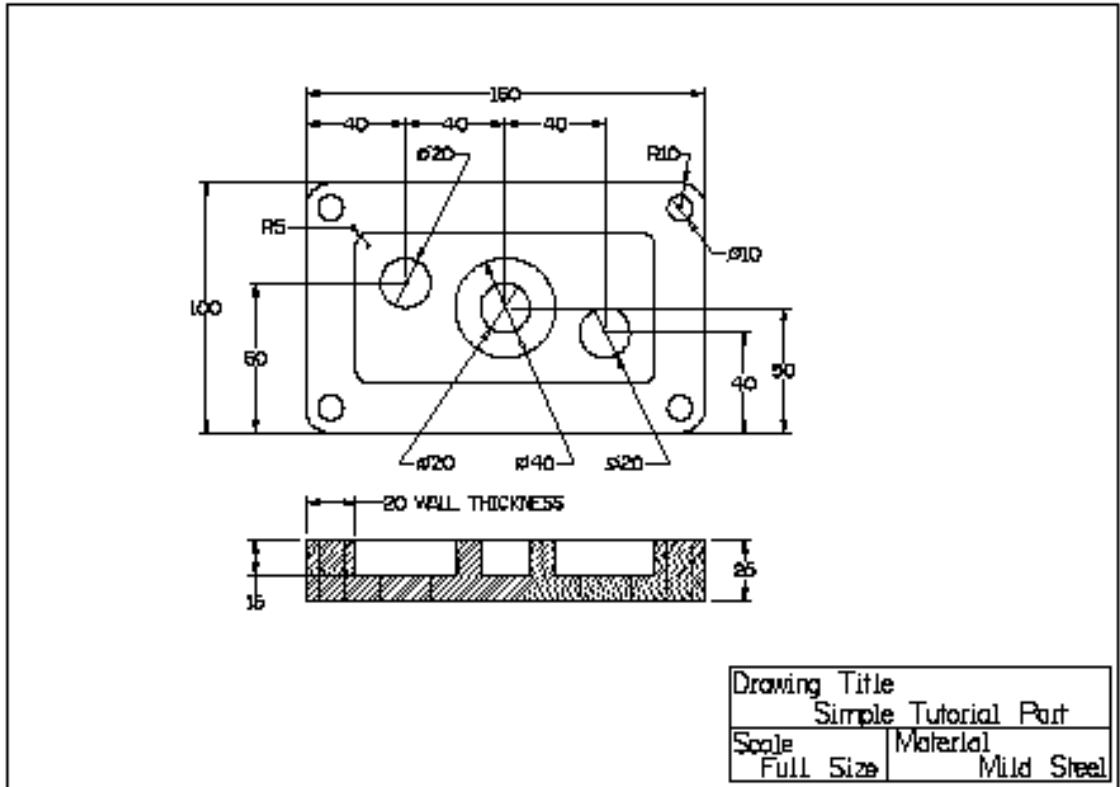
AlphaCAM provides various ways of creating part geometry.

Conventional CAD style geometry, creation, whereby individual geometric features are created and then trimmed in order to create geometric contours. Toolpaths are then applied to the geometric contours, from which the NC program is produced;

'APS Fast Geometry' is a unique way of creating 'bounded' geometry. This can turn some designs into geometric contours much faster than with any conventional CAD system.

With APS Fast Geometry, you do not specify individual lines and arcs. Instead, you say how the tool should move from one element to the next. Each change in direction is called a **Turn**. APS Fast Geometry builds a 'bounded' geometry profile by automatically trimming, blending and filleting as Turn details are entered. This method is very powerful, because it allows you to answer 'Unknown' to questions about poorly specified co-ordinates and once AlphaCAM has enough information it will 'back-calculate' to solve the unknowns.

However, **CAD-style** geometry creation using Line, Arc and Circle commands is sometimes appropriate for simple shapes, and all the conventional CAD commands are included, plus Special Geometries, which automatically produce standard geometric shapes.

The Process Plan

Before creating the geometry to be machined, it is important to decide what geometry is required to control the machining. The geometry necessary to control the machining is defined by the process plan. In our example drawing, the top and outer surface are not to be machined, so it is not necessary to draw the outer rectangle, although the outer profile will be used to define the material size.

OP No.	Operation Description	T No.	Tool Description
1	Drill Holes 4 Holes through at corners 1 Hole in middle 15 deep. 1 hole as pilot for area clearance (to be drilled after Op2 and re-ordered)	1	Drill 10mm
2	Rough Area Clear the inner rectangular pocket	2	Flat 10mm
3	Finish Area Clear the inner rectangular pocket Finish Profile inner rectangle and boss	3	Flat 5mm
4	Draw circles and drill holes as pilot for the area clearance. This operation to be moved and merged into Op1 Drill 2 holes as pilot for 20-diameter through holes	1	Drill 10mm
5	Spiral Area Clear through holes and centre recess. Leave 0.5 stock	2	Flat 10mm
6	Finish Profile through holes and centre recess	4	Flat 12mm +Holder

It will be necessary to create additional geometry during the creation of the machining, in order to drill the pilot hole for the area clearance in the correct position.

From the process plan, it can be seen that the geometries required to control the machining are the inner rectangle and the circles. In this example, the outer rectangle will be created in order to define the material size.

Creating The Geometry

Draw the Outer Rectangle

☞ Select **GEOMETRY** | **Rectangle** ☐ . This command creates a rectangle that is defined about 2 diagonal points.

☞ The command line prompts you to enter the co-ordinates of the first corner of the rectangle. Type **0** ⏎ **0** ⏎

☞ The command line now prompts you to enter the co-ordinates of the second corner of the rectangle. Type **160** ⏎ **100** ⏎

The rectangle is drawn on the screen.

☞ Select **VIEW** | **Zoom All** 🔍 Ctrl+A to make the geometry fit the screen

Draw the Inner Rectangle

The rectangle command could be used to create the inner rectangle, but because of the way the drawing is dimensioned, we will use the offset command.

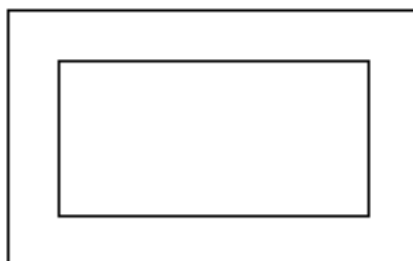☞ Select **EDIT** | **Break Join etc.** | **Offset** ⟋

☞ A dialog box is displayed.
The first section prompts for the **Distance**: type **20**. The second section has 2 or 3 settings, which set **What** is to be offset. Click 👆 on the **Geometry** option. When the dialog is complete, click 👆 on ⟨ OK ⟩.

☞ The command line prompts you to select the geometry that is to be offset. Click 👆 on the rectangle on the screen. The complete rectangle should turn blue. If it does not, click 👆 on the screen and start this section again.

☞ The command line now prompts you to indicate which side the geometry is to be offset. Click 👆 on a position inside the rectangle on the screen.

The inner rectangle is drawn.

Add the Fillet Radii

Geometry can be modified by the addition of a Fillet radius. Fillets can be applied either to individual corners, or to all sharp corners in the contour.

☞ Select **EDIT | Break Join etc. | Fillet** 🔼

☞ A dialog box is displayed. The first section prompts for the **Fillet Radius:** type **10**. The second section has 2 settings, which set which **Corners** are to be filleted. Click 🖰 on the **All** option.
When the dialog is complete, click 🖰 on ⌈ OK ⌉

☞ The command line prompts you to select the geometry that is to be filleted. Click 🖰 on the outer rectangle on the screen. The complete rectangle should turn blue. Click 🖰 on ⌈ Finish (ESC) ⌉ or click 🖰 on a position on screen.

The fillets are added.

☞ Select **EDIT | Break Join etc. | Fillet** 🔼

☞ A dialog box is displayed.
The first section prompts for the **Fillet Radius:** type **5**. The second section has 2 settings, which set which **Corners** are to be filleted.
Click 🖰 on the **All** option. When the dialog is complete, click 🖰 on ⌈ OK ⌉

☞ The command line prompts you to select the geometry that is to be filleted. Click 🖰 on the inner rectangle on the screen.
The complete rectangle should turn blue.
Click 🖰 on ⌈ Finish (ESC) ⌉ or click 🖰 on a position on screen.

The fillets are added
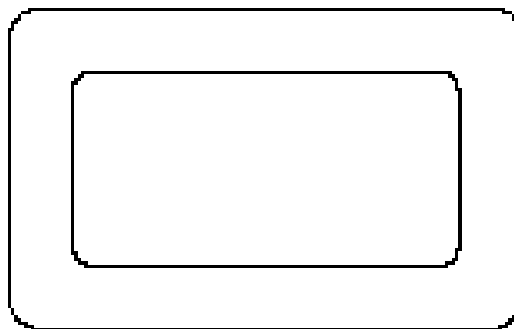
☞ Select **View | Redraw** 🔲 ⌈Ctrl⌉ + ⌈R⌉ to tidy up the display.

Draw The Circles

20 diameter circles

☞ Select **GEOMETRY** | **Circle** | **centre + diameter**

☞ The command line prompts you to enter the circle diameter. Type **20** ↵

☞ The command line now prompts you to enter the co-ordinates of the circle centre.
Type **40** ↵ **60** ↵. The left 20-diameter circle is drawn.

☞ The command line prompts you again to enter the circle diameter.
The default is now 20, so it is only necessary to press ↵

☞ The command line prompts you again to enter the co-ordinates of the circle centre.
Type **40 + 40** ↵ **50** ↵. The middle 20-diameter circle is drawn.

☞ The command line prompts you to enter the circle diameter.
The default is 20, so it is only necessary to press ↵

☞ The command line prompts you again to enter the co-ordinates of the circle centre.
Type **40 + 40 + 40** ↵ **40** ↵. The right 20-diameter circle is drawn.
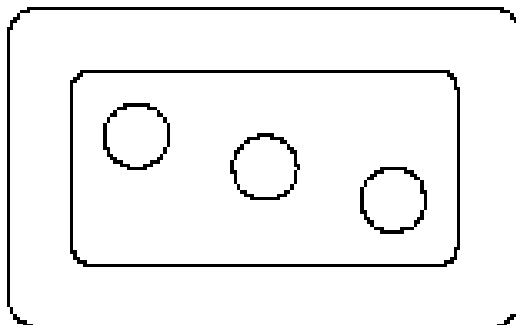
40 diameter circle

☞ The command line prompts you to enter the circle diameter.
The default is 20, so it is only necessary to change this value. Type **40** ⏎

☞ The command line prompts you again to enter the co-ordinates of the circle
centre. This circle is to be positioned with its centre at the same point as
the middle 20-diameter circle. So instead of typing in the co-ordinates, you
can use a snapping function to snap the circle centre to an existing
position.

☞ Select **UTILS | Snaps | centre of** ⊕ F8 . The command line changes
from prompting for the circle centre to prompting you to select the circle to
whose centre you want to snap.

☞ Click 🖰 on the edge of the middle 20-diameter circle. The 40-diameter
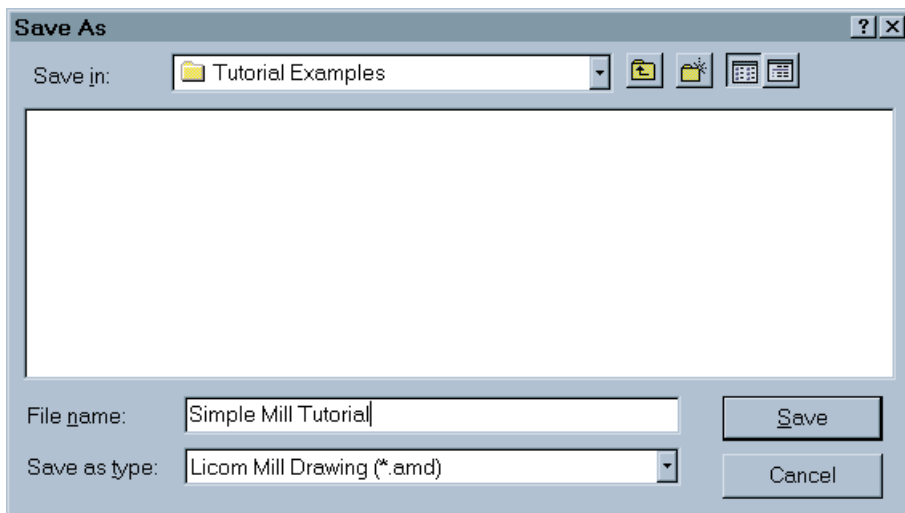circle is drawn.

10 diameter circles

☞ The command line prompts you again to enter the circle diameter. The
default is 40, so it is only necessary to change this value. Type **10** ⏎

☞ Select **UTILS | Snaps | centre of** ⊕ F8 . Click 🖰 on the edge of an outer
fillet radius.

☞ The command line prompts you again to enter the circle diameter. The
default is 10, so it is only necessary to press ⏎

☞ Select **UTILS | Snaps | centre of** ⊕ F8 . Click 🖰 on the edge of an outer
fillet radius.
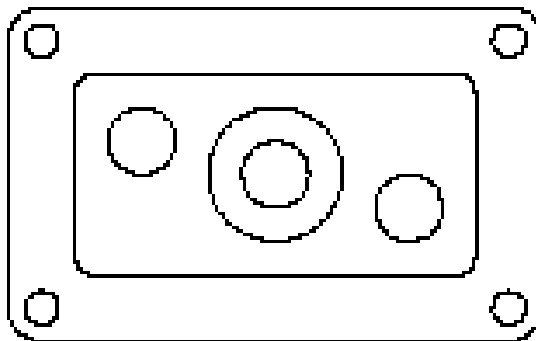
Repeat the previous two sub options twice more.

The geometry is now complete.

☞ Select **FILE | Save As** and the Save As dialog is displayed.
The default **Save In** folder is Licomdir. Set the save in folder to the folder where the work is to be saved. In the **File Name** field enter the name by which the file is to be referenced.

Save As		?☐✕
Save in:	☐ Tutorial Examples	▾ ⬆ ☐ ▦ ▥

File name:	Simple Mill Tutorial	Save
Save as type:	Licom Mill Drawing (*.amd)	▾	Cancel

Preparation For Machining

It is necessary to set the right conditions before the machining commands can be applied to the geometry profiles.

Set The Machine Type

☞ Select **FILE** | **Select Post** . The available post processors are displayed.

☞ Select a suitable post processor.
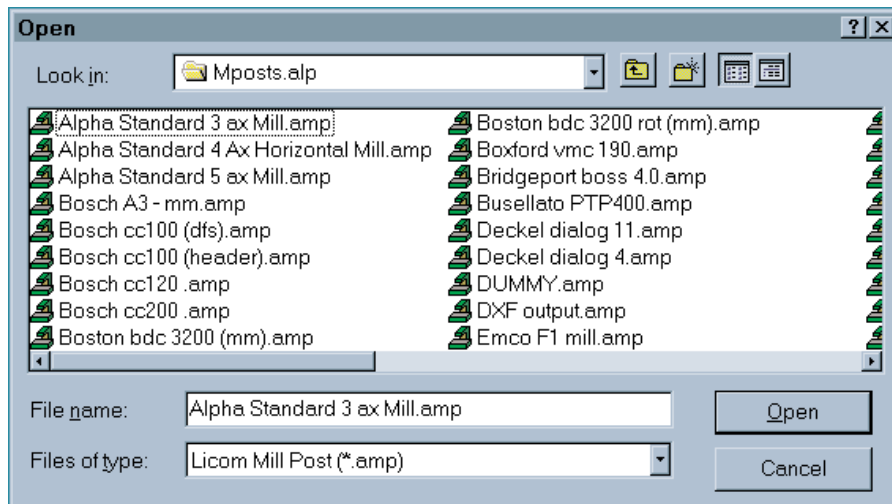
Set The Cutting Conditions

Setting the cutting conditions involves setting the way in which the tools cut the component. The conditions only need to be set for geometries against which the tool is driven. The default tool directions are set as the direction of definition and on centre line.

☞ Select **View** | **Display Options** | **Ghost Tools** Ctrl + G to see the tool directions.
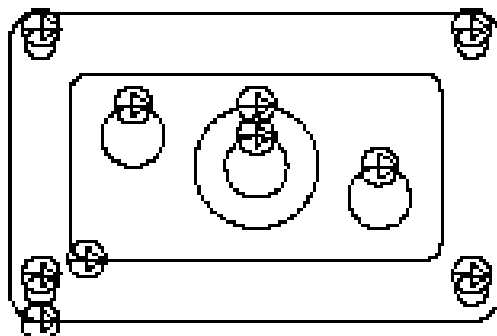
In line with the operations list, it is necessary to configure the following cutting conditions.

The inner rectangle and the 20-diameter circles are to be cut in a CCW direction with the tool positioned on the inside.

The 40-diameter circle is to be cut in a CW direction with the tool positioned on the outside.

In addition to setting the tool directions and side of cutting, it may also be necessary to adjust the point at which the tool starts cutting the profile.

In this case, it is necessary to move the start point on the inner rectangle, as it is not good practice to start cutting on an internal corner.

Tool Directions

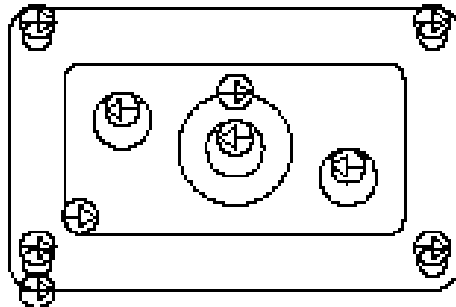☞ Select **Machine** | **Tool Directions** 🔧

A dialog box with two sections is displayed. The left section controls the direction of cutting and the right section controls the side to which the tool is positioned when cutting. The side of cutting is defined whilst looking in the cutting direction.
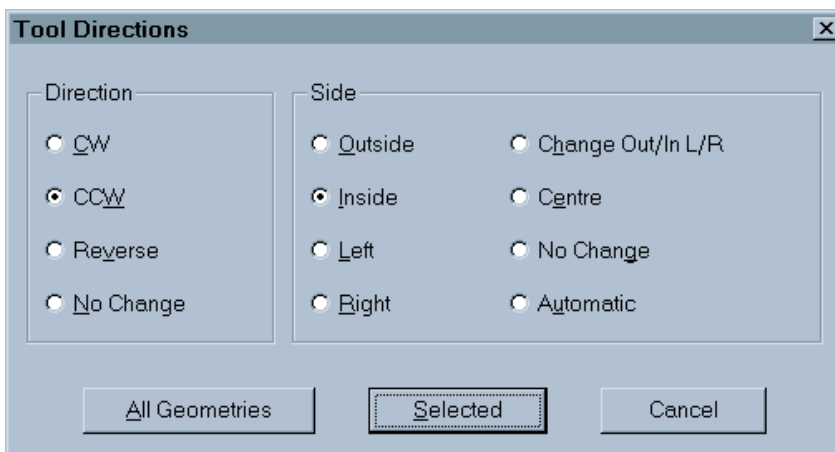
☞ Click 🖱 on the **CCW** option to set the direction and click 🖱 on the **Inside** option. When the selection is correct, click 🖱 on the **Selected** button. The Selected button allows you to pick the geometries to which the tool directions are to be applied.

☞ Click 🖱 on the three 20-diameter circles and the inner rectangle.

👉 Select **View** | **Redraw** 🅁 Ctrl + R to tidy up the display.
The ghost tool display should look like the following diagram.

👉 Select **Machine** | **Tool Directions** again.

👉 Click 🖰 on the **CW** option to set the direction and click 🖰 on the **Outside** option. When the selection is correct, click 🖰 on the **Selected** button.

👉 Click 🖰 on the 40-diameter circle.

👉 Select **View** | **Redraw** 🅁 Ctrl + R to tidy up the display.

Start Point

As it is not good practice to start machining on an internal corner, we will reposition the start point to the middle of the left vertical edge of the inner rectangle.

👉 Select **EDIT** | **Start, Order** | **Start Pt**. 📤 Ctrl + F

👉 Select **UTILS** | **Snaps** | **middle of** ⟋ F7 . Click 🖰 on the left edge of the inner rectangle.

👉 Select **View** | **Redraw** 🅁 Ctrl + R to tidy up the display.
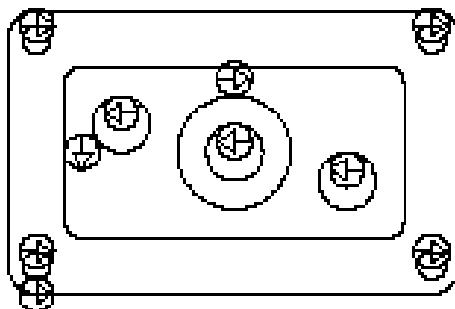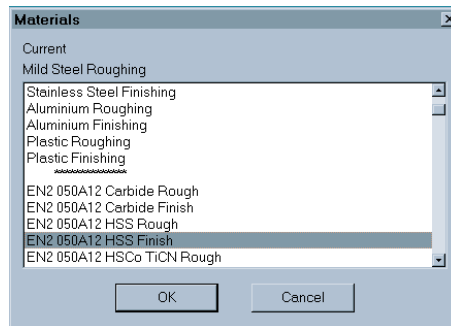The ghost tool display should look like the following diagram.

Tool Selection

Before selecting the tool for the first machining operation, it is necessary to select the cutting speed for the material and cutting operation.
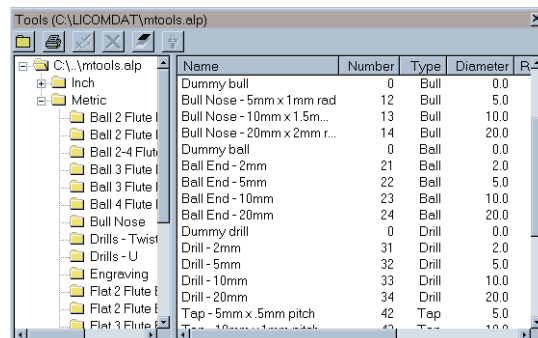
Cutting Speed

☞ Select **Machine** | **Select Material** 📄 . The material dialog is displayed.

☞ Select the **EN2 050A12 HSS Finish** option. To see this option, it is necessary to use the scroll bar on the right to scroll up and down through the options. Click 🖱 on the option to highlight it, then click 🖱 on ⬚ OK ⬚

Select Tool

☞ Select **Machine** | **Select Tool** 🔧 . The tool library dialog is displayed.

☞ Pick the tool name **Drill – 10mm**, then click 🖱 on 🔧 (the select tool button within the tool library dialog). The tool library dialog disappears and a schematic of the tool is displayed.

☞ The command prompt line prompts you to press ⎂Enter⏎⎂ to accept the tool selection or press ⎂Esc⎂ to return to the tool library dialog. Press ⎂⏎⎂.
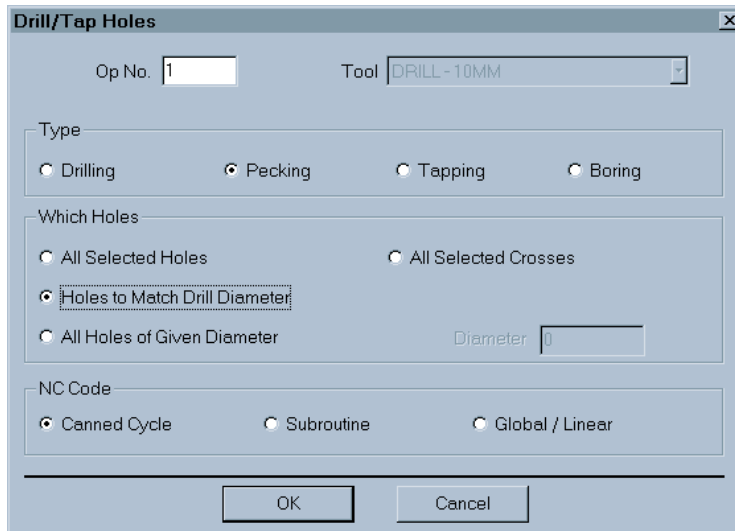
YOU MUST ACCEPT THE TOOL TO SELECT IT!
PICKING ANY OTHER OPTION CAUSES THE TOOL NOT TO BE SELECTED.

The Machining Operations

Operation 1

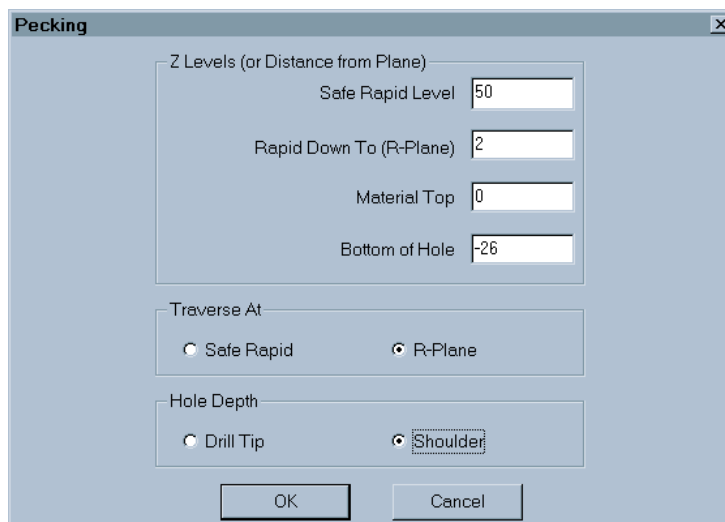#### Drilling

☞ Select **MACHINE** | **Drill/Machine Holes** 🔘 | **Drill Tap Holes** 🔲 and the first dialog box is displayed.

```
Drill/Tap Holes                                              ×

        Op No.  1            Tool  DRILL - 10MM            ▼

   ┌─ Type ──────────────────────────────────────────────┐
   │  ○ Drilling     ● Pecking      ○ Tapping    ○ Boring │
   └─────────────────────────────────────────────────────┘

   ┌─ Which Holes ───────────────────────────────────────┐
   │  ○ All Selected Holes         ○ All Selected Crosses │
   │  ● Holes to Match Drill Diameter                     │
   │  ○ All Holes of Given Diameter      Diameter  0      │
   └─────────────────────────────────────────────────────┘

   ┌─ NC Code ───────────────────────────────────────────┐
   │  ● Canned Cycle    ○ Subroutine    ○ Global / Linear │
   └─────────────────────────────────────────────────────┘

              ┌──────OK──────┐    ┌────Cancel────┐
```
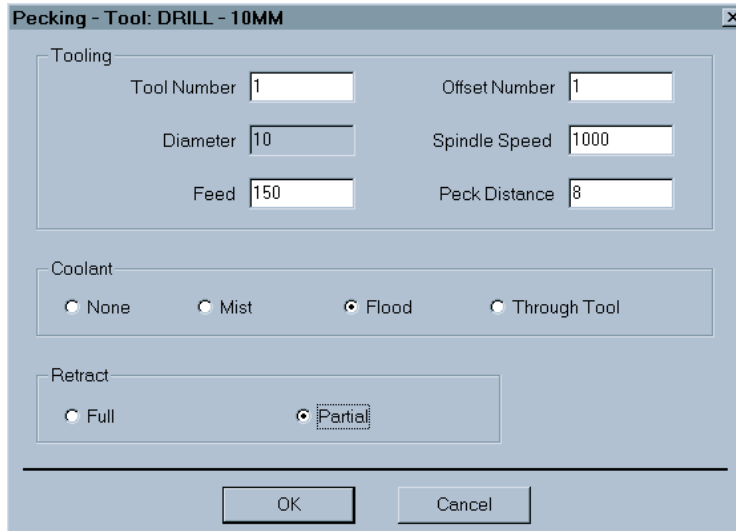
☞ Set the options as shown above. When complete, click 🖑 on ⬛ OK ⬛ and the second dialog box is displayed.

Remember to use the ⬛Tab⬛ key to move to each field in a dialog box. DO NOT PRESS ⬛Enter ↵⬛, as this is the same as clicking 🖑 on ⬛ OK ⬛, which means that all entries are complete.

```
Pecking                                                      ×

   ┌─ Z Levels (or Distance from Plane) ─────────────────┐
   │                  Safe Rapid Level  50               │
   │                                                     │
   │          Rapid Down To (R-Plane)  2                 │
   │                                                     │
   │                    Material Top  0                  │
   │                                                     │
   │                   Bottom of Hole  -26               │
   └─────────────────────────────────────────────────────┘

   ┌─ Traverse At ───────────────────────────────────────┐
   │            ○ Safe Rapid      ● R-Plane              │
   └─────────────────────────────────────────────────────┘

   ┌─ Hole Depth ────────────────────────────────────────┐
   │            ○ Drill Tip       ● Shoulder             │
   └─────────────────────────────────────────────────────┘

              ┌──────OK──────┐    ┌────Cancel────┐
```

☞ Set the options as shown. When complete, click 🖱 on ⬚ OK ⬚ and the third dialog box is displayed.

Pecking - Tool: DRILL - 10MM ✕

Tooling

Tool Number	1	Offset Number	1
Diameter	10	Spindle Speed	1000
Feed	150	Peck Distance	8

Coolant

○ None   ○ Mist   ⊙ Flood   ○ Through Tool

Retract

○ Full   ⊙ Partial

OK   Cancel

☞ Set the options as shown. When complete, click 🖱 on ⬚ OK ⬚

☞ The dialog box disappears and the command line prompts you to select the circles. All the circles may be selected in one go by using a selection box/window.

When all the geometry is blue, click 🖱 on ⬚ Finish (ESC) ⬚
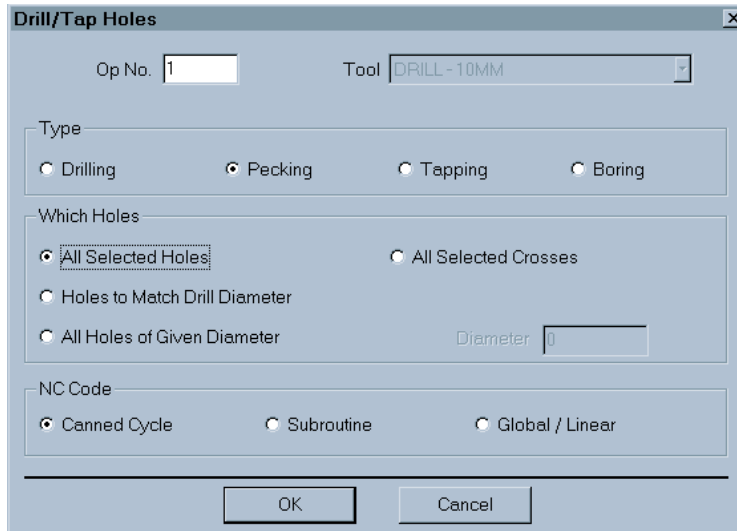
To make a selection using a window, position the cursor to a position above and to the left of the geometry and left click 🖱. When you move the cursor, the system draws a box. Position the cursor so that all the geometry fits inside the box and left click 🖱.

An information box is displayed, informing you that 4 circles have been selected.

☞ Click 🖱 on ⬚ OK ⬚

☞ To repeat the command, you can press [Spacebar] or select

MACHINE | **Drill/Machine Holes** 🔵 | **Drill Tap Holes** 🔵 , and the first
dialog box is displayed.

Drill/Tap Holes

Op No. 1   Tool DRILL - 10MM

Type
○ Drilling   ● Pecking   ○ Tapping   ○ Boring

Which Holes
● All Selected Holes   ○ All Selected Crosses
○ Holes to Match Drill Diameter
○ All Holes of Given Diameter   Diameter 0

NC Code
● Canned Cycle   ○ Subroutine   ○ Global / Linear

OK   Cancel

☞ Set the options as shown above. When complete, click 🖱 on ⬚ OK ⬚ and
the second dialog box is displayed.

Pecking

Z Levels (or Distance from Plane)
Safe Rapid Level 50
Rapid Down To (R-Plane) 2
Material Top 0
Bottom of Hole -14.5

Traverse At
○ Safe Rapid   ● R-Plane

Hole Depth
● Drill Tip   ○ Shoulder

OK   Cancel

☞ Set the options as shown. When complete, click 🖱 on ⬚ OK ⬚ and the third
dialog box is displayed.

Pecking - Tool: DRILL - 10MM

Tooling

Tool Number	1	Offset Number	1
Diameter	10	Spindle Speed	1000
Feed	150	Peck Distance	8

Coolant

○ None   ○ Mist   ⦿ Flood   ○ Through Tool

Retract

○ Full   ⦿ Partial

OK   Cancel

☞ Set the options as shown. When complete, click 🖱 on ⌐ OK ⌐

☞ The dialog box disappears and the command line prompts you to select the circles. Left click 🖱 on the middle circle and it turns blue.
Click 🖱 on ⌐ Finish (ESC) ⌐

An information box is displayed, informing you that the circle selected is a different size from that of the tool.

☞ Click 🖱 on ⌐ OK ⌐

An information box is displayed, informing you that 1 circle has been selected.

☞ Click 🖱 on ⌐ OK ⌐

A cross marks all the holes that have been machined.
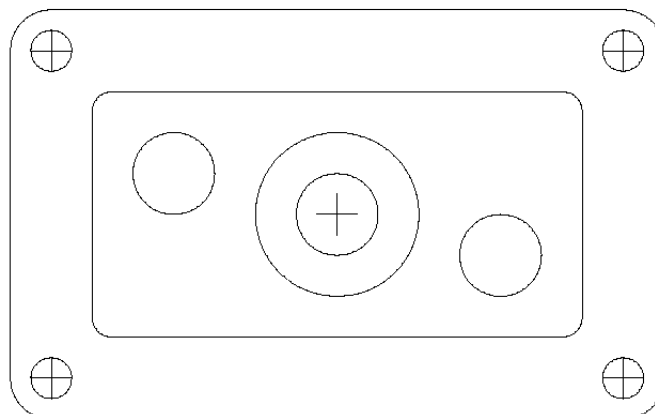
Tool Selection

Operation 2 is to area clear the pocket and this requires a new tool. In this example, we are going to assume that the tool being used for the area clearance is solid carbide. In order for the system to calculate the correct feeds and speed it is necessary to select a new material cutting condition.

Cutting Speed

☞ Select **Machine** | **Select Material** ◢. The material dialog is displayed.

☞ Select the **EN2 050A12 Carbide Rough** option. Click 🖱 on the option to highlight it, then click 🖱 on ⬚ OK ⬚

Select Tool

☞ Select **Machine** | **Select Tool** 🔧 . The tool library dialog is displayed.
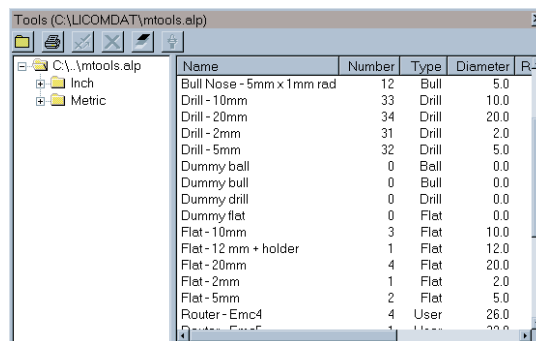
☞ Pick the tool name **Flat – 10mm**, then click 🖱 on 🔧 . The tool library dialog disappears and a schematic of the tool is displayed.

☞ The command prompt line prompts you to press [Enter↵] to accept the tool selection or [Esc] to return to the tool library dialog. Press [↵].

Operation 2

Rough Area Clearance

☞ Select **MACHINE** | **Pocketing** 🔲

The first dialog box shown differs between the Advanced and Standard modules.
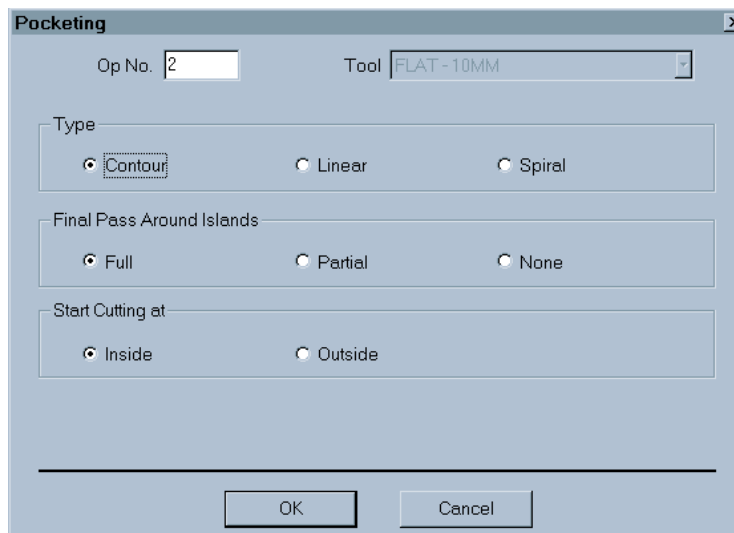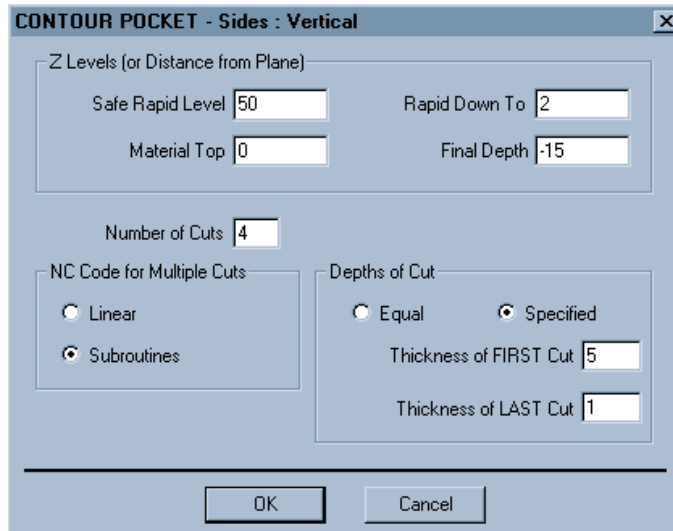
| Advanced | Standard |

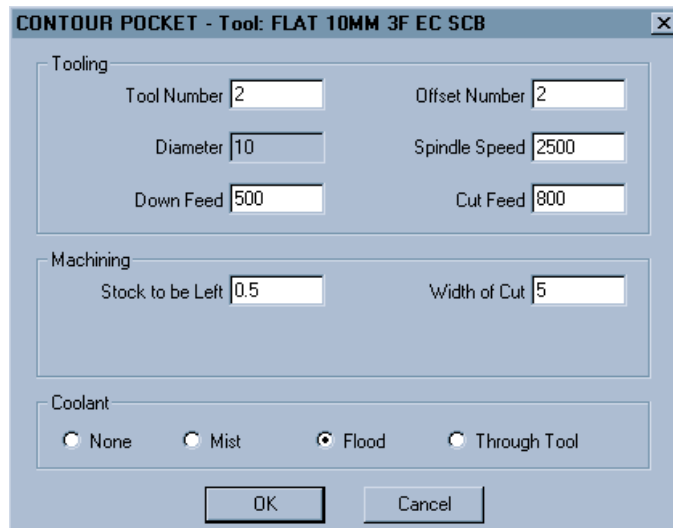☞ Click 🖰 on the Selected button and the second dialog box is displayed.

☞ Set the options as shown. When complete, click 🖰 on OK and the third
   dialog box is displayed.

CONTOUR POCKET - Sides : Vertical

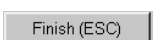Z Levels (or Distance from Plane)

Safe Rapid Level 50   Rapid Down To 2

Material Top 0   Final Depth -15

Number of Cuts 4

NC Code for Multiple Cuts
- ○ Linear
- ● Subroutines

Depths of Cut
- ○ Equal   ● Specified

Thickness of FIRST Cut 5

Thickness of LAST Cut 1

OK   Cancel

☞ Set the options as shown. When complete, click 🖱 on ⌐ OK ⌐ and the fourth dialog box is displayed.

CONTOUR POCKET - Tool: FLAT 10MM 3F EC SCB

Tooling

Tool Number 2   Offset Number 2

Diameter 10   Spindle Speed 2500

Down Feed 500   Cut Feed 800

Machining

Stock to be Left 0.5   Width of Cut 5

Coolant
- ○ None   ○ Mist   ● Flood   ○ Through Tool

OK   Cancel

☞ Set the Tool Number, Offset Number, Stock to be Left and Coolant options as shown. When complete, click 🖱 on ⌐ OK ⌐

☞ The dialog box disappears and the command line prompts you to select the geometries. Left click 🖱 on the middle outer circle and the inner rectangle. They turn blue. Click 🖱 on Finish (ESC) .

The pocketing toolpaths are displayed.
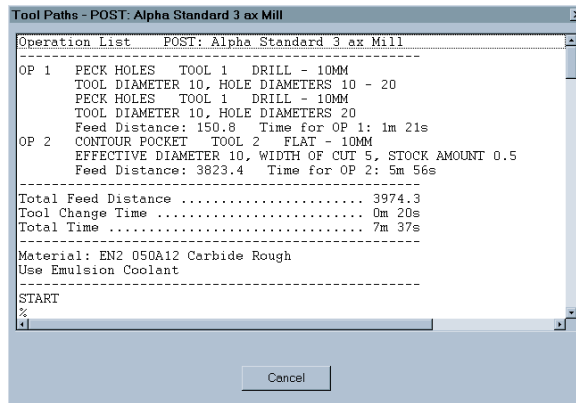
Display Options

At this stage, you may wish to utilise some of the different display options available. The commands can be found in the **VIEW** | **Display Options** menu.

To see the area of material cut by the tools, select 🔲 followed by 🅁. To return the display to normal, select 🔲 followed by 🅁.

To see the tool in an animated display, select 🔩, 🡕 and **3D**. The screen automatically splits into four views and an animated tool is displayed. To see the animation again, select 🅁.

To return the display to normal, select 🔩, 🡕 and **3D**.

To see the NC code that has been generated, select **FILE** | **List NC Code** . The list screen is displayed. Select the **List All** button. The display prompts you to enter a Program Number: enter **1**, then select OK . The NC program is displayed. To return to the graphics display, select the Cancel button.

```
Tool Paths - POST: Alpha Standard 3 ax Mill                        x
Operation List    POST: Alpha Standard 3 ax Mill
----------------------------------------------------
OP 1   PECK HOLES    TOOL 1   DRILL - 10MM
            TOOL DIAMETER 10, HOLE DIAMETERS 10 - 20
            PECK HOLES    TOOL 1   DRILL - 10MM
            TOOL DIAMETER 10, HOLE DIAMETERS 20
            Feed Distance: 150.8   Time for OP 1: 1m 21s
OP 2   CONTOUR POCKET   TOOL 2   FLAT - 10MM
            EFFECTIVE DIAMETER 10, WIDTH OF CUT 5, STOCK AMOUNT 0.5
            Feed Distance: 3823.4   Time for OP 2: 5m 56s
----------------------------------------------------
Total Feed Distance ..................... 3974.3
Tool Change Time ........................ 0m 20s
Total Time .............................. 7m 37s
----------------------------------------------------
Material: EN2 050A12 Carbide Rough
Use Emulsion Coolant
----------------------------------------------------
START
%
```

Cancel

The 10-diameter tool will not have completely machined the pocket, as there are areas the tool can not machine. To machine the rest of the pocket, a smaller tool is required.

The next tool will also be a solid carbide tool and the operation type is still roughing, so it is not necessary to select a different material cutting speed.

Tool Selection

Select Tool

☞ Select **Machine** | **Select Tool** . The tool library dialog is displayed.
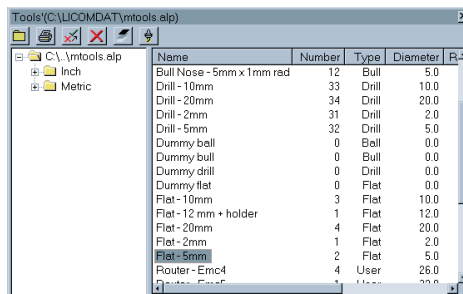
☞ Pick the tool name **Flat – 5mm**, then click on .

The tool library dialog disappears and a schematic of the tool is displayed.

☞ The command prompt line prompts you to press R to accept the tool selection or press Esc to return to the tool library dialog. Press ↵.
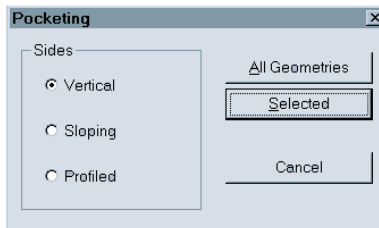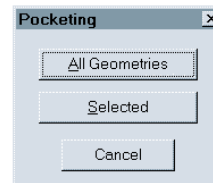
Operation 3

Finish Area Clearance

☞ Select **MACHINE** | **Pocketing** 🔲

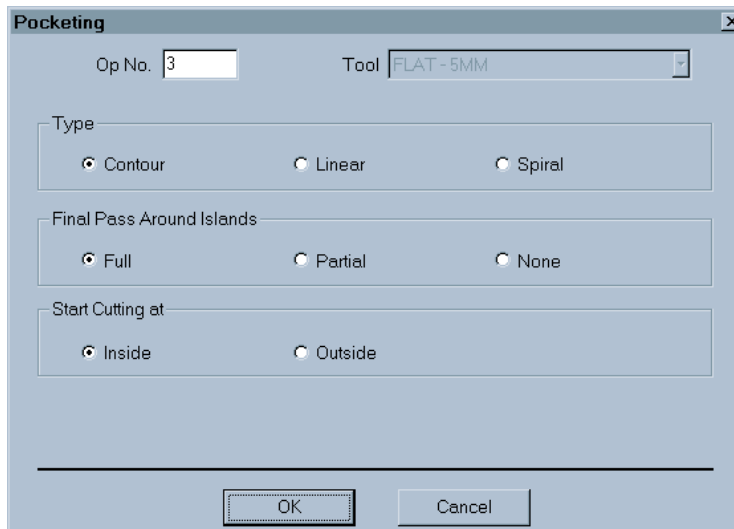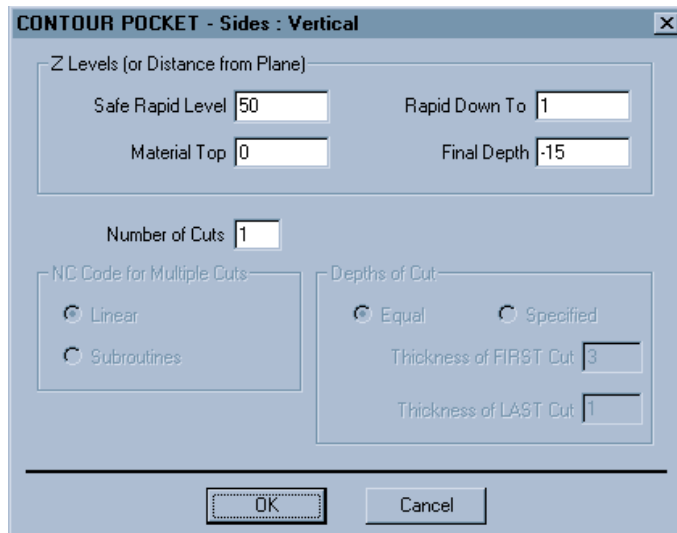The first dialog box shown differs between the Advanced and Standard modules.

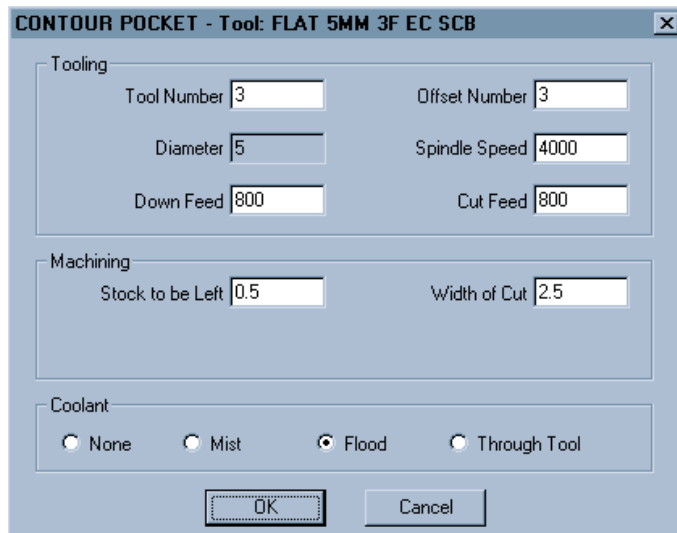**Advanced**   **Standard**

☞ Click 🖱 on the Selected button and the second dialog box is displayed.

☞ Set the options as shown. When complete, click 🖱 on ⌐ OK ¬ and the third dialog box is displayed.

CONTOUR POCKET - Sides : Vertical

Z Levels (or Distance from Plane)

Safe Rapid Level 50   Rapid Down To 1

Material Top 0   Final Depth -15

Number of Cuts 1

NC Code for Multiple Cuts
- Linear
- Subroutines

Depths of Cut
- Equal   - Specified

Thickness of FIRST Cut 3

Thickness of LAST Cut 1

OK   Cancel

☞ Set the options as shown. When complete, click 🖱 on ⌐ OK ⌐ and the fourth dialog box is displayed.

CONTOUR POCKET - Tool: FLAT 5MM 3F EC SCB

Tooling

Tool Number 3   Offset Number 3

Diameter 5   Spindle Speed 4000

Down Feed 800   Cut Feed 800

Machining

Stock to be Left 0.5   Width of Cut 2.5

Coolant
- None   - Mist   ⦿ Flood   - Through Tool

OK   Cancel
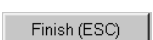
☞ Set the Tool Number, Offset Number, Stock to be Left and Coolant options as shown. When complete, click 🖱 on ⌐ OK ⌐

☞ The dialog box disappears and the command line prompts you to select the geometries. Pick the **Previous** button or left click 🖰 on the middle outer circle and the inner rectangle: they turn blue. Click 🖰 on [Finish (ESC)]
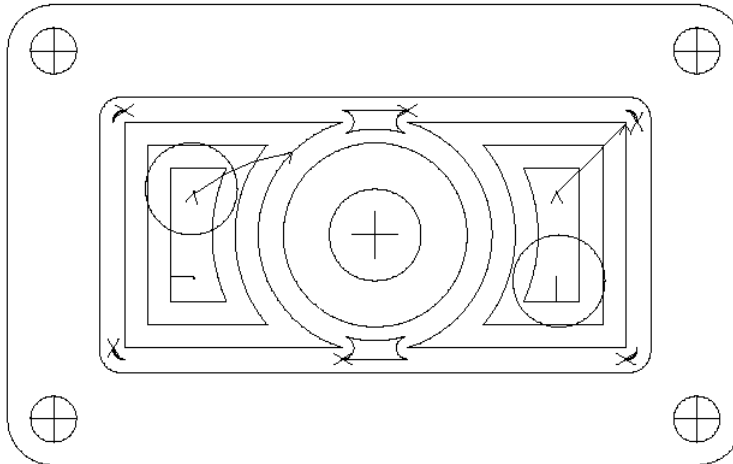
The pocketing toolpaths are displayed.

The 5-diameter tool will only machine the areas
that the 10-diameter tool could not machine.

These additional toolpaths will require a lead in and out toolpath to be added to them.

It is only possible to set the lead in and out to
pocketing toolpaths using the manual option.

Tool Lead In / Out

At this stage, an arc lead in and out can be applied to the 5-diameter toolpaths. This will allow the tool to sink into an area where the material has been removed. By leading out to the lead in position, the tool will NOT retract between cuts.

> *Leading in and out on just an arc is a valid option,*
> *as pocket toolpaths do not use Cutter Comp.*

☞ Select **MACHINE** | **Tool Lead In/Out**

A dialog box is displayed. There are several sections, each setting an aspect of the lead in and/or out to be applied.
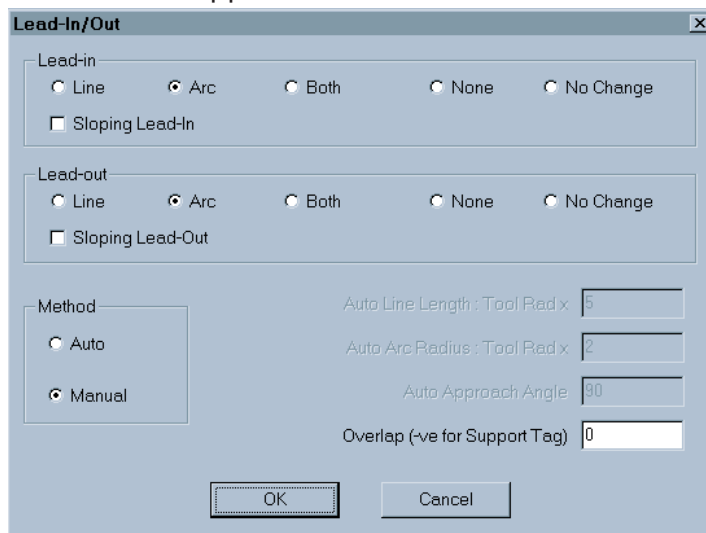
☞ Set the options as shown. When complete, click 🖰 on ⟦ OK ⟧
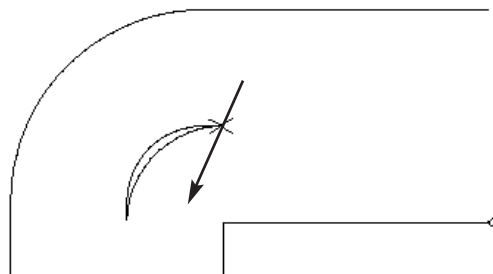
☞ The dialog box disappears and the command line prompts you to select the toolpath. Click 🖰 on a toolpath in the corner.

☞ The command line prompts you to pick the lead in position. Select a suitable position for the lead in position.
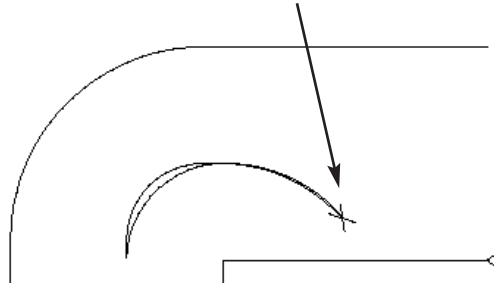
☞ The command line prompts you to pick the lead out position.

Select ⬈ 〔F6〕 and click 🖱 on the end of the lead in arc.

☞ Repeat the last three sub options for each corner toolpath.

☞ To apply the Lead in and out to the middle toolpaths,

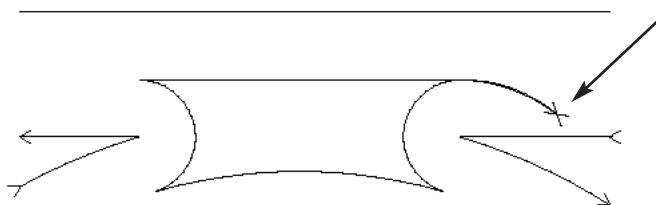firstly select **VIEW | Zoom Window** 🔍 〔Ctrl〕+〔W〕 and draw the window around the top middle toolpath.
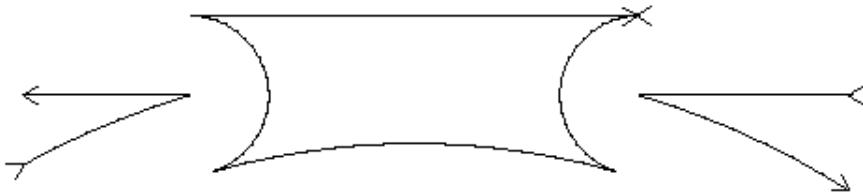
The zoom commands are transparent and can be used at any time without aborting the current command.

☞ The command line prompts you to select the toolpath. Click 🖱 on a toolpath in the middle at the top.

☞ The command line prompts you to pick the lead in position. Position the cursor at a suitable starting point and click 🖱 to pick the position

☞ The command line prompts you to pick the lead out position.

Select ⬈ 〔F6〕 and click 🖱 on the start of the lead in arc.

☞ To apply the Lead in and out to the bottom middle toolpath, firstly select **VIEW** | **Zoom All** 🔍 Ctrl + A to display all the geometries and toolpaths.

☞ Select **VIEW** | **Zoom Window** 🔍 Ctrl + W and draw the window around the bottom middle toolpath.

☞ The command line prompts you to select the toolpath.
Click 🖱 on a toolpath in the middle.

☞ The command line prompts you to pick the lead in position. Position the cursor at a suitable starting point and click 🖱 to pick the position.

☞ The command line prompts you to pick the lead out position.

Select 📈 F6 and click 🖱 on the start of the lead-in arc.
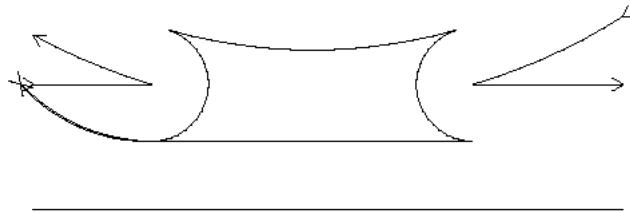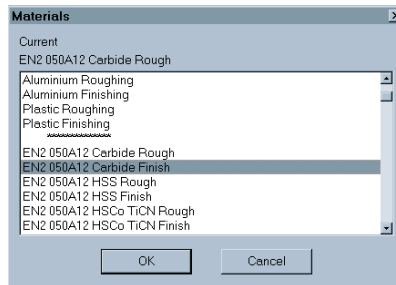
Tool Selection

The next Operation is to finish profile the pocket and boss. This does not require a new tool, as the 5-diameter tool will be used. In order for the system to calculate the correct feeds and speed, it is necessary to select a new material cutting condition.

Cutting Speed

☞ Select **Machine** | **Select Material** ◢. The material dialog is displayed.

☞ Select the **EN2 050A12 Carbide Finish** option. Click 🖰 on the option to highlight it, then click 🖰 on ⬚ OK ⬚

Although the tool has not changed, it is necessary to reselect it, so that the system can recalculate the new speeds and feeds to suit the new conditions. A tool change will not be output.

Select Tool

☞ Select **Machine** | **Select Tool** 🔧. The tool library dialog is displayed.
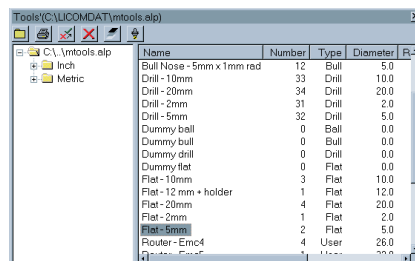
☞ Pick the tool name **Flat – 5mm**, then click 🖰 on 🔧.

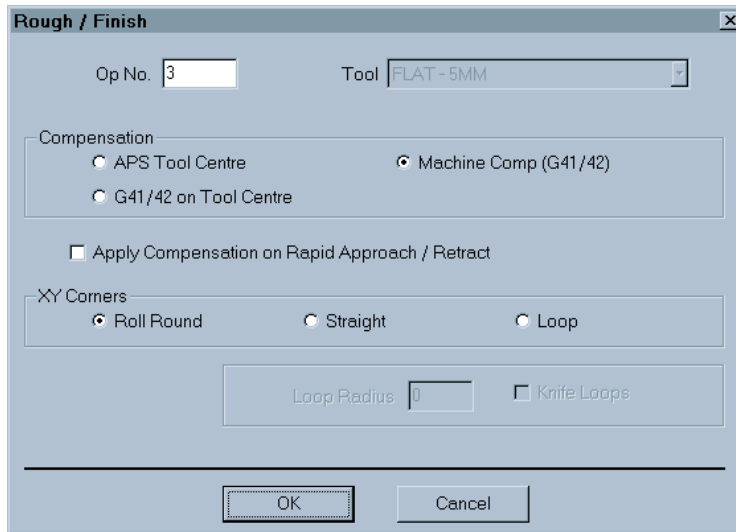The tool library dialog disappears and a schematic of the tool is displayed.

☞ The command prompt line prompts you to press ⬚R⬚ to accept the tool selection or press ⬚Esc⬚ to return to the tool library dialog. Press ⬚↵⬚.
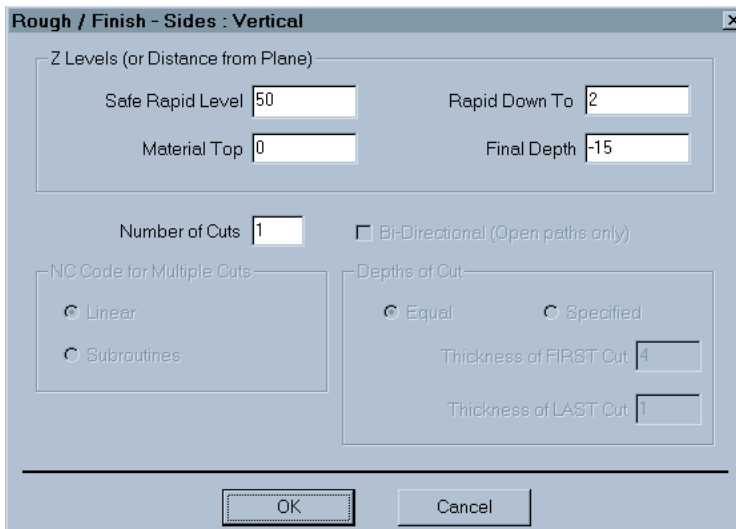
Finish Profile

☞ Select **MACHINE** | **Rough or Finish** 🔏.

The first dialog box shown differs between the Advanced and Standard modules.

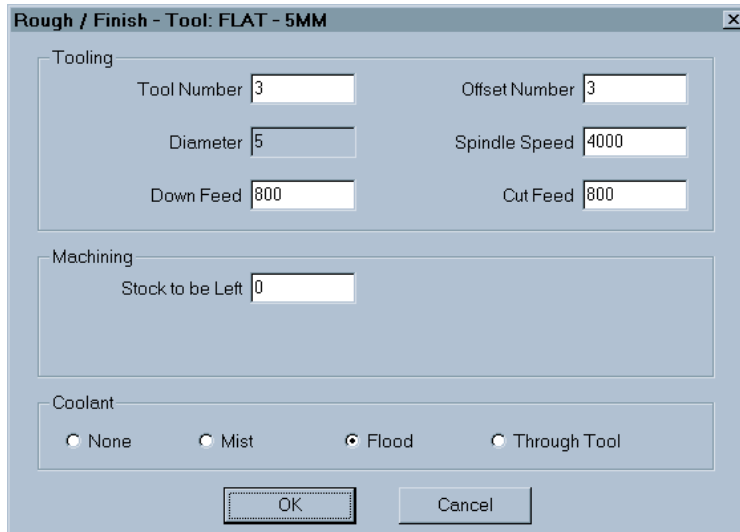☞ Click 🖱 on (vertical and then) the **Selected** button and the second dialog box is displayed.

Rough / Finish

Op No. 3   Tool FLAT - 5MM

Compensation
○ APS Tool Centre   ● Machine Comp (G41/42)
○ G41/42 on Tool Centre

☐ Apply Compensation on Rapid Approach / Retract

XY Corners
● Roll Round   ○ Straight   ○ Loop

Loop Radius 0   ☐ Knife Loops

OK   Cancel

☞ Set the options as shown. When complete, click 🖱 on ⌐ OK ⌐ and the third dialog box is displayed.

Rough / Finish - Sides : Vertical

Z Levels (or Distance from Plane)

Safe Rapid Level 50   Rapid Down To 2

Material Top 0   Final Depth -15

Number of Cuts 1   ☐ Bi-Directional (Open paths only)

NC Code for Multiple Cuts
○ Linear   Depths of Cut
                            ○ Equal   ○ Specified
○ Subroutines
                            Thickness of FIRST Cut 4

                            Thickness of LAST Cut 1

OK   Cancel

☞ Set the options as shown. When complete, click 🖱 on ⌐ OK ⌐ and the fourth dialog box is displayed.

Rough / Finish - Tool: FLAT - 5MM

Tooling

Tool Number	3	Offset Number	3
Diameter	5	Spindle Speed	4000
Down Feed	800	Cut Feed	800

Machining

Stock to be Left 0

Coolant

○ None   ○ Mist   ◉ Flood   ○ Through Tool

OK   Cancel

☞ Set the options as shown. When complete, click 🖱 on ⬚ OK

☞ The dialog box disappears and the command line prompts you to select the geometries. Left click 🖱 on the middle outer circle and the inner rectangle: they turn blue. Click 🖱 on Finish (ESC)
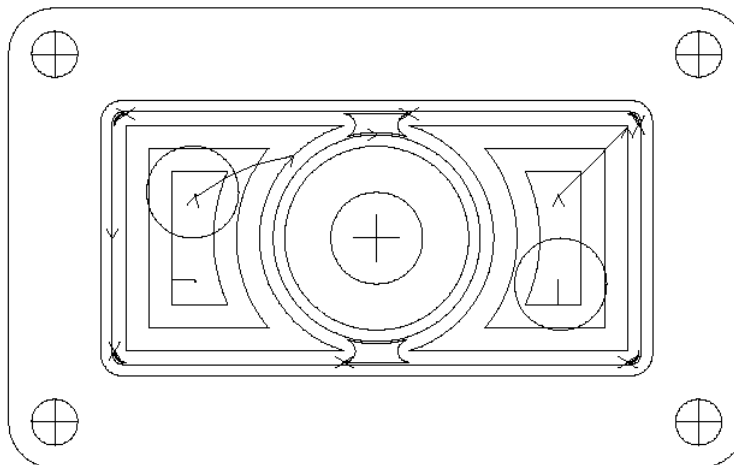
The finishing toolpaths are displayed.

Tool Lead In / Out

The profiling cuts use cutter compensation and will require a lead in and out to be applied.

As the profile toolpaths use Cutter Comp,
arc only is not a valid lead in and out.

☞ Select **MACHINE** | **Tool Lead In/Out**

A dialog box is displayed. There are several sections, each one setting an aspect of the lead in and/or out to be applied.
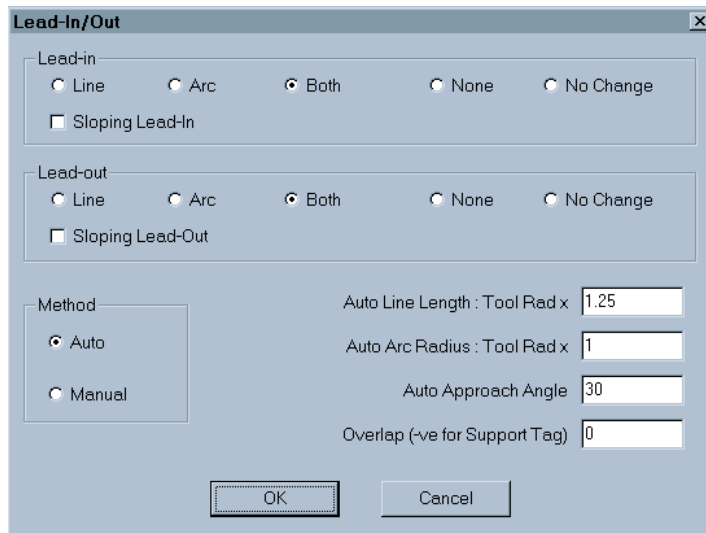
☞ Set the options as shown. When complete, click on ☐ OK

☞ The dialog box disappears and the command line prompts you to select the toolpaths. Click on the profiling toolpaths around the inner pocket and boss.
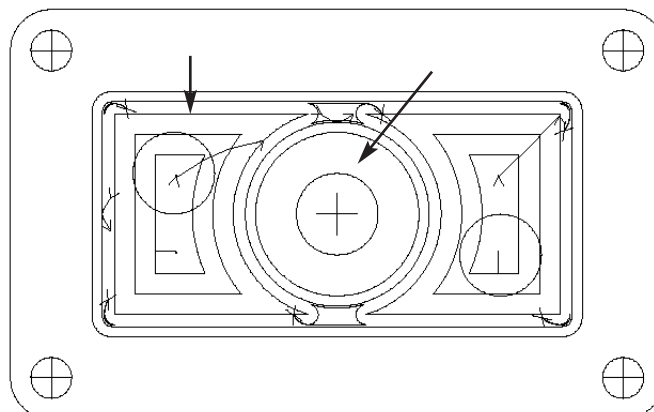
Pilot Holes for Pocketing

Draw the Circles

To drill the pilot holes for the pocketing operation, you need to draw the 10mm diameter circles at a suitable position.

☞ Select **GEOMETRY** | **Circle** | **Centre + Diameter**

☞ For the circle diameter: type **10** ⏎

☞ For the circle centre, use ✎ F7 and pick the rapid line between the start and end of the left side pocketing toolpath.

☞ Repeat the last two sub options for the right side pocketing toolpath.
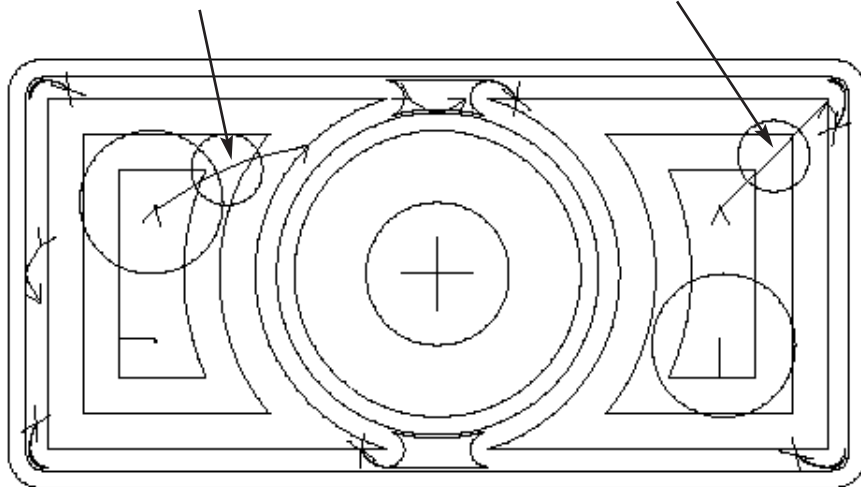
Drill The Pilot Holes

Tool Selection

Operation 4 is to drill both the pilot holes for the pocketing operation and to pilot drill through, for the 20-diameter holes, into the bottom of the pocket.
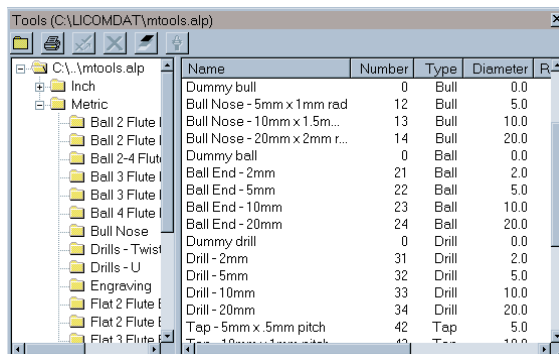
Cutting Speed

☞ Select **Machine** | **Select Material** ◪. The material dialog is displayed.
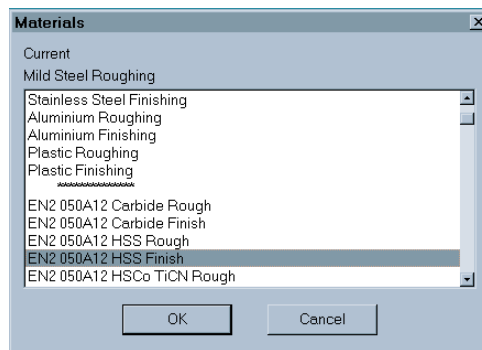
☞ Select the **EN2 050A12 HSS Finish** option. Click 🖱 on the option to highlight it, then click 🖱 on ▭ OK ▭

Select Tool

☞ Select **Machine** | **Select Tool** 🔧. The tool library dialog is displayed.

☞ Pick the tool name **Drill – 10mm**, then click 🖱 on 🔧.
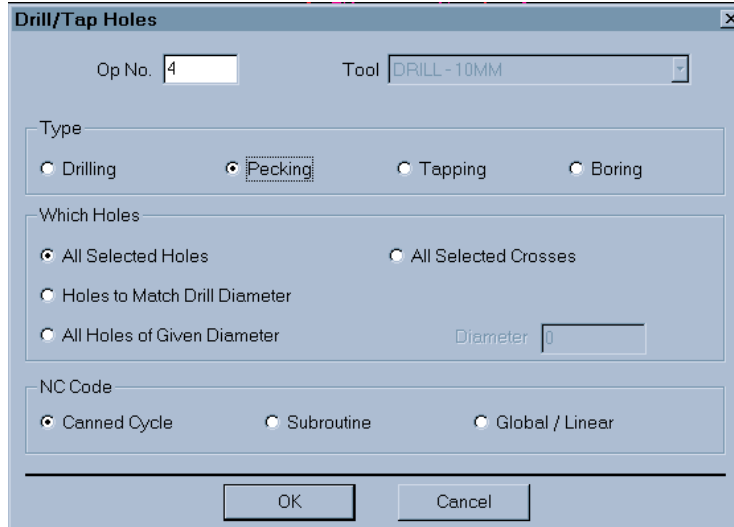
The tool library dialog disappears and a schematic of the tool is displayed.

☞ The command prompt line prompts you to press ⟨R⟩ to accept the tool selection or press ⟨Esc⟩ to return to the tool library dialog. Press ⟨↵⟩.

Operation 4

Drilling

☞ Select **MACHINE** | **Drill/Machine Holes** 🔘 | **Drill Tap Holes** 🔳 and the first dialog box is displayed.

☞ Set the options as shown above. When complete, click 🖑 on ⬚ OK ⬚ and the second dialog box is displayed.

☞ Set the options as shown. When complete, click 🖑 on ⬚ OK ⬚ and the third dialog box is displayed.

Pecking - Tool: DRILL - 10MM

Tooling

Tool Number	1	Offset Number	1
Diameter	10	Spindle Speed	1000
Feed	150	Peck Distance	8

Coolant

○ None   ○ Mist   ◉ Flood   ○ Through Tool

Retract

○ Full   ◉ Partial

[OK]   [Cancel]

☞ Set the options as shown. When complete, click 🖑 on [OK]

☞ The dialog box disappears and the command line prompts you to select the circles.
Left click 🖑 on the two 10-diameter pilot hole circles and they turn blue.
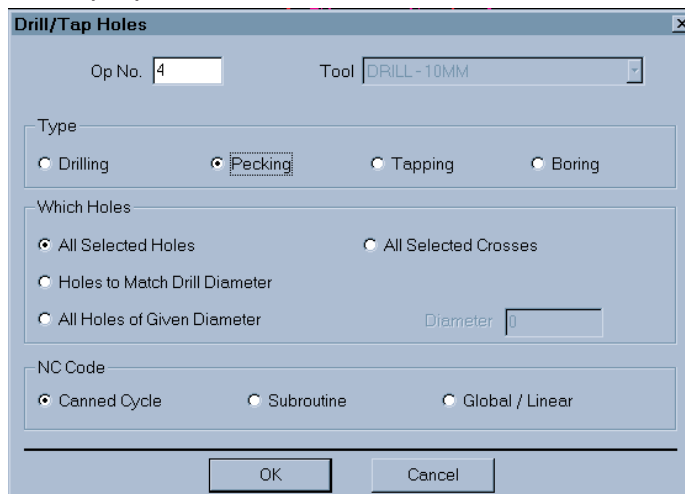Click 🖑 on [Finish (ESC)]

If you select an incorrect item, picking it again de-selects it.

An information box informs you that two circles have been selected.

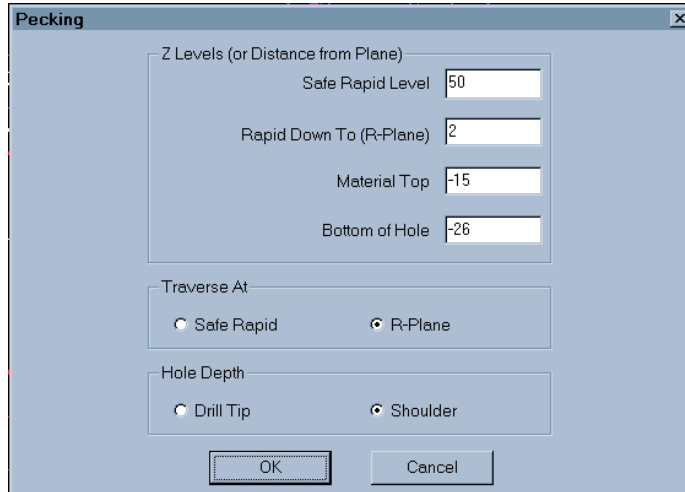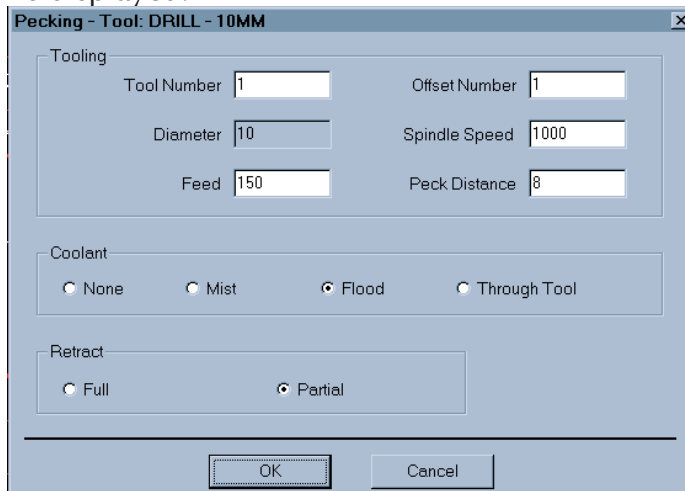☞ Click 🖑 on [OK]

☞ To repeat the command, you can press [Spacebar] or select

MACHINE | **Drill/Machine Holes** 📷 | **Drill Tap Holes** 📷 and the first dialog box is displayed.

Drill/Tap Holes

Op No. 4   Tool [DRILL - 10MM]

Type

○ Drilling   ◉ Pecking   ○ Tapping   ○ Boring

Which Holes

◉ All Selected Holes   ○ All Selected Crosses

○ Holes to Match Drill Diameter

○ All Holes of Given Diameter   Diameter 0

NC Code

◉ Canned Cycle   ○ Subroutine   ○ Global / Linear

[OK]   [Cancel]

☞ Set the options as shown above. When complete, click 🖰 on ⬚ OK ⬚ and the second dialog box is displayed.

Pecking

Z Levels (or Distance from Plane)

Safe Rapid Level	50
Rapid Down To (R-Plane)	2
Material Top	-15
Bottom of Hole	-26

Traverse At
- ○ Safe Rapid   ⊙ R-Plane

Hole Depth
- ○ Drill Tip   ⊙ Shoulder

⬚ OK ⬚   ⬚ Cancel ⬚

☞ Set the options as shown. When complete, click 🖰 on ⬚ OK ⬚ and the third dialog box is displayed.

Pecking - Tool: DRILL - 10MM

Tooling

Tool Number	1	Offset Number	1
Diameter	10	Spindle Speed	1000
Feed	150	Peck Distance	8

Coolant
- ○ None   ○ Mist   ⊙ Flood   ○ Through Tool

Retract
- ○ Full   ⊙ Partial

⬚ OK ⬚   ⬚ Cancel ⬚

☞ Set the options as shown. When complete, click 🖰 on ⬚ OK ⬚

☞ The dialog box disappears and the command line prompts you to select the circles. Left click 🖰 on the two 20-diameter circles in the pocket and they turn blue. Click 🖰 on ⬚ Finish (ESC) ⬚
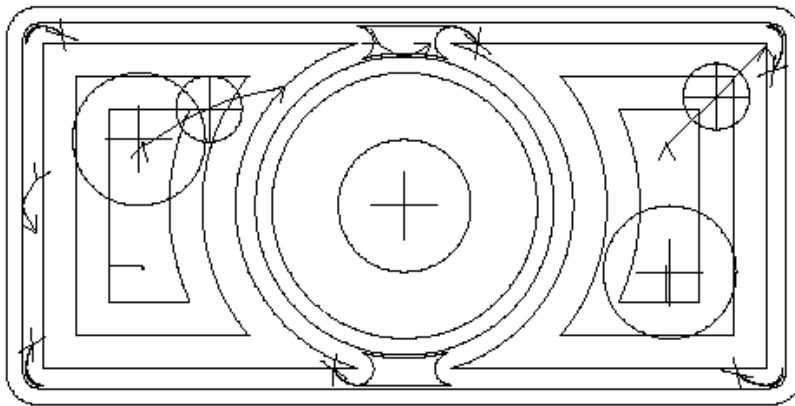
An information box is displayed, informing you that the circle selected is a different size from that of the tool.

☞ Click 🖰 on ⬚ OK ⬚

An information box is displayed, informing you that the 2 circles have been selected.
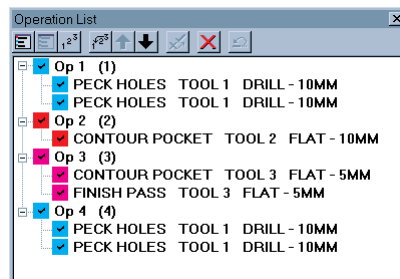
☞ Click 🖱 on [OK]

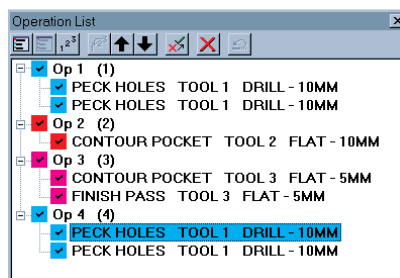A cross marks all the holes that have been machined.
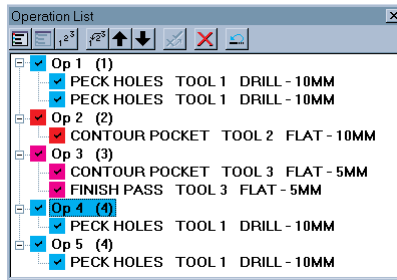

Move the Drilling Operation.

☞ Select **MACHINE | Edit Operations** 🖾 [Ctrl]+[E]. The edit operations window is displayed.


☞ Select the first part of operation 4 by clicking 🖱 on the text. The toolpath cross turns blue.


☞ Click on the bold up arrow ⬆. This causes the operation to split into two separate operations. Select Op 4 again by clicking on the text.

☞ Click on the bold up arrow ⬆. This causes the operation to move up the list. Each click moves the operation up one level. Move the operation to Op 2. Then click 🖰 on the text part of the operation content.
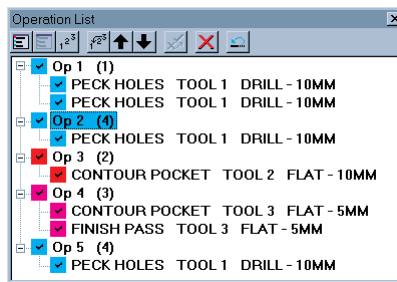
☞ Click on the bold up arrow ⬆.
This causes the operation to be merged into Op1.
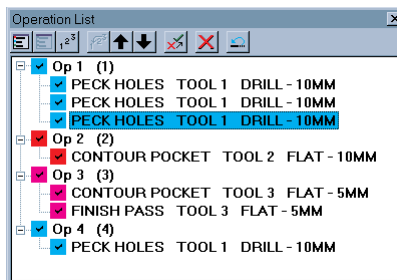
Display Options

At this stage, you may wish to utilise some of the different display options available. The commands can be found in the **VIEW | Display Options** menu.

To see the area of material cut by the tools, select ⬛ followed by 🅡. To return the display to normal, select ⬛ followed by 🅡.

To see the tool in an animated display, select 🔧, ↗ and **3D**. The screen automatically splits into four views and an animated tool is displayed. To see the animation again, select 🅡.
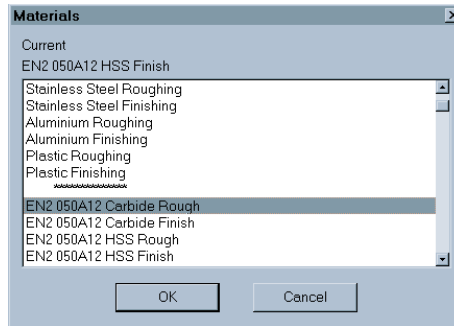
To return the display to normal, select 🔧, ↗ and **3D**.

Tool Selection

Operation 5 is to spiral area clear the 20-diameter pockets and this requires another tool. In this example, we are going to use the 10mm diameter tool that was used to rough out the rectangular pocket.
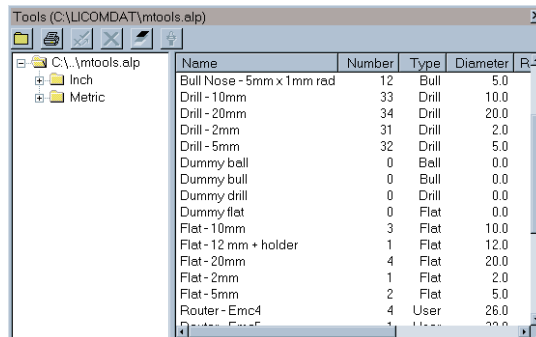
Cutting Speed

☞ Select **Machine** | **Select Material** ◧. The material dialog is displayed.

☞ Select the **EN2 050A12 Carbide Rough** option. Click 🖰 on the option to highlight it, then click 🖰 on ⬚ OK ⬚

Select Tool

☞ Select **Machine** | **Select Tool** ⬚ . The tool library dialog is displayed.

☞ Pick the tool name **Flat – 10mm**, then click 🖰 on ⬚ . The tool library dialog disappears and a schematic of the tool is displayed.

☞ The command prompt line prompts you to press R to accept the tool selection or Esc to return to the tool library dialog. Press ↵.
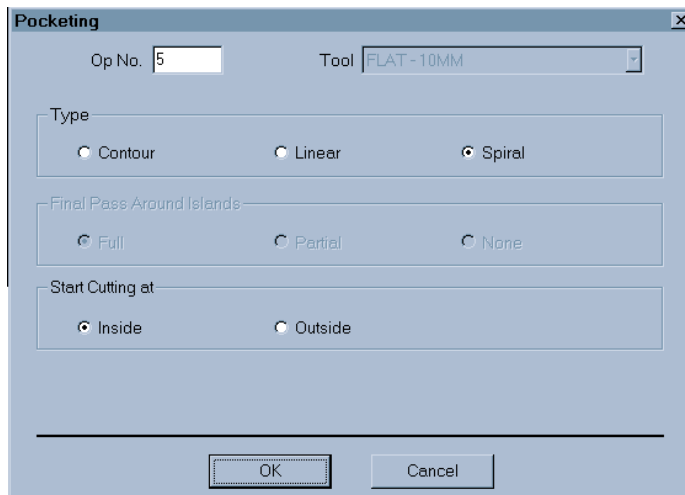
Operation 5

Spiral Area Clearance

☞ Select **MACHINE** | **Pocketing** 🔲.

The first dialog box shown differs between the Advanced and Standard modules.

☞ Click 🖱 on (vertical and then) the **Selected** button and the second dialog box is displayed.
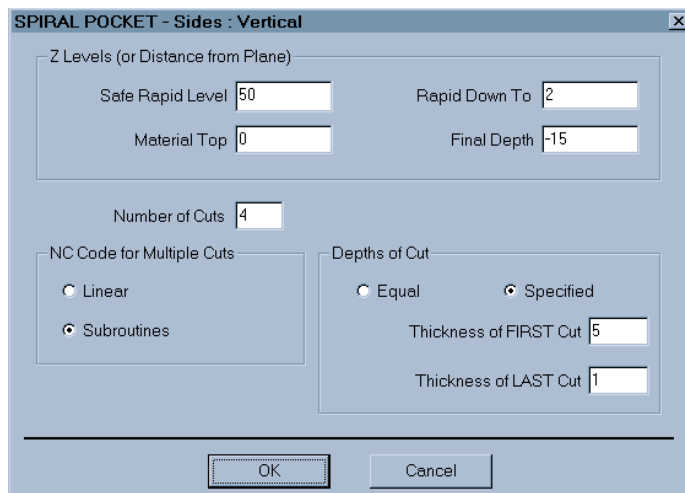
Pocketing

Op No. 5   Tool FLAT - 10MM

Type
- ○ Contour
- ○ Linear
- ● Spiral

Final Pass Around Islands
- ● Full
- ○ Partial
- ○ None

Start Cutting at
- ● Inside
- ○ Outside

OK   Cancel

☞ Set the options as shown. When complete, click 🖱 on ⌐ OK ⌐ and the third dialog box is displayed.

SPIRAL POCKET - Sides : Vertical

Z Levels (or Distance from Plane)
Safe Rapid Level 50   Rapid Down To 2
Material Top 0   Final Depth -15

Number of Cuts 4

NC Code for Multiple Cuts
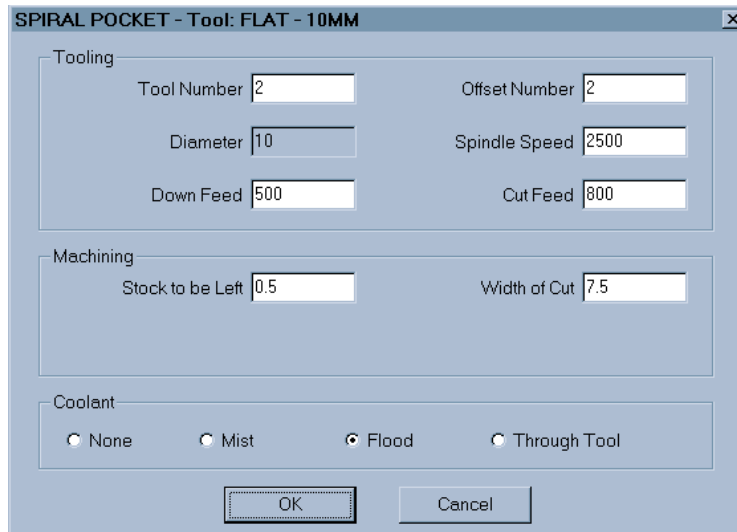- ○ Linear
- ● Subroutines

Depths of Cut
- ○ Equal
- ● Specified

Thickness of FIRST Cut 5

Thickness of LAST Cut 1

OK   Cancel

☞ Set the options as shown. When complete, click 🖱 on ⌐ OK ⌐ and the fourth dialog box is displayed.

SPIRAL POCKET - Tool: FLAT - 10MM

Tooling

Tool Number	2	Offset Number	2
Diameter	10	Spindle Speed	2500
Down Feed	500	Cut Feed	800

Machining

Stock to be Left	0.5	Width of Cut	7.5

Coolant

○ None   ○ Mist   ⦿ Flood   ○ Through Tool

OK   Cancel

☞ Set the options as shown. When complete, click 🖱 on OK

☞ The dialog box disappears and the command line prompts you to select the geometries. Left click 🖱 on the middle 20 diameter circle: it turns blue. Click 🖱 on Finish (ESC)
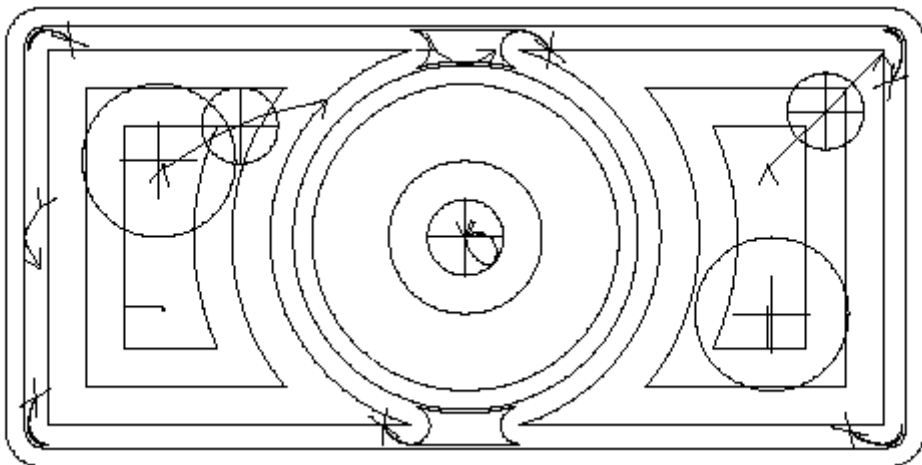
The pocketing toolpaths are displayed.

☞ To repeat the command, you can press ⌐Spacebar⌐ or select

  MACHINE | **Pocketing** 🔲 and the first dialog box is displayed

☞ Click 🖰 on (vertical and then) the **Selected** button and the second dialog box is displayed.

Pocketing

Op No. 5   Tool FLAT - 10MM

Type
  ○ Contour   ○ Linear   ● Spiral

Final Pass Around Islands
  ● Full   ○ Partial   ○ None

Start Cutting at
  ● Inside   ○ Outside

  [OK]   [Cancel]

☞ Set the options as shown. When complete, click 🖰 on [OK] and the third dialog box is displayed.

SPIRAL POCKET - Sides : Vertical

Z Levels (or Distance from Plane)

Safe Rapid Level 50   Rapid Down To -13

Material Top -15   Final Depth -26

Number of Cuts 2

NC Code for Multiple Cuts
  ○ Linear
  ● Subroutines
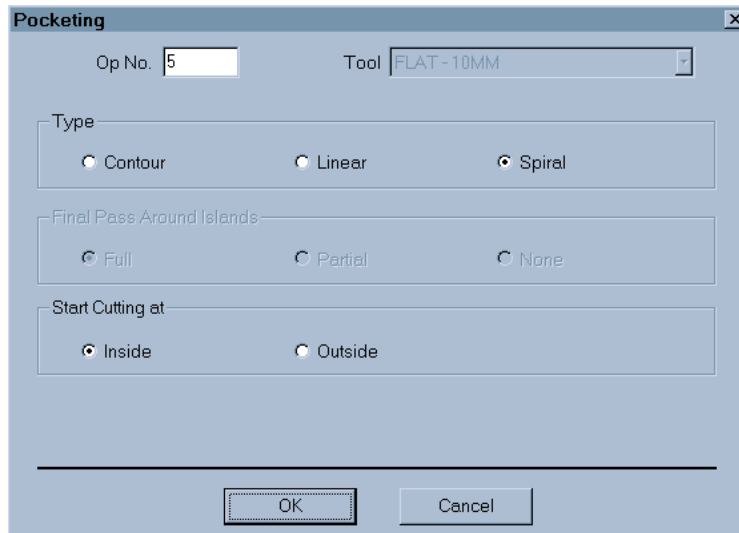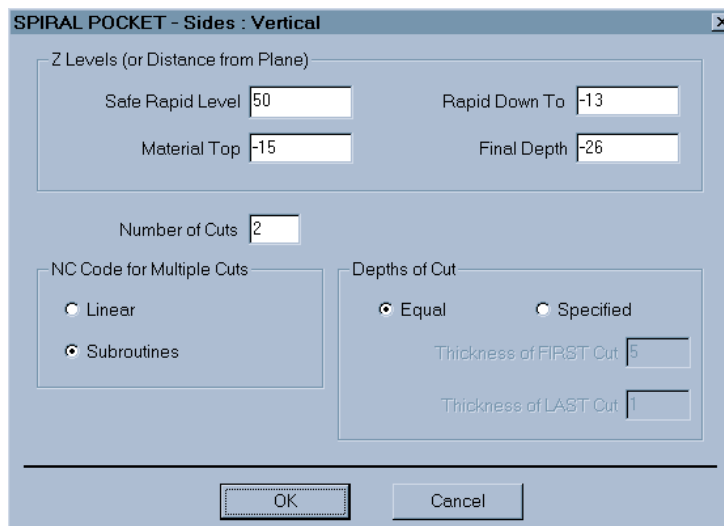
Depths of Cut
  ● Equal   ○ Specified

  Thickness of FIRST Cut 5

  Thickness of LAST Cut 1

  [OK]   [Cancel]

☞ Set the options as shown. When complete, click 🖰 on [OK] and the fourth dialog box is displayed.

SPIRAL POCKET - Tool: FLAT - 10MM

Tooling

Tool Number	2	Offset Number	2
Diameter	10	Spindle Speed	2500
Down Feed	500	Cut Feed	800

Machining

Stock to be Left	0.5	Width of Cut	7.5

Coolant

○ None　　○ Mist　　◉ Flood　　○ Through Tool

[OK]　　[Cancel]
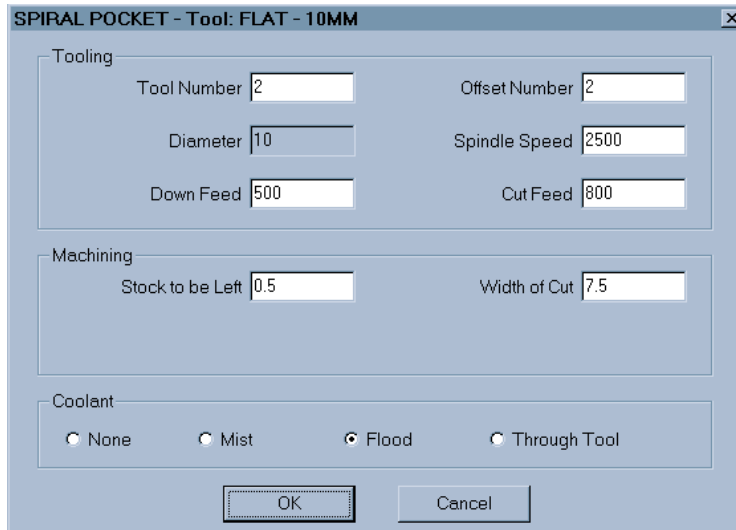
☞ Set the Tool Number, Offset Number, Stock to be Left and Coolant options as shown. When complete, click 🖱 on [OK]

☞ The dialog box disappears and the command line prompts you to select the geometries. Left click 🖱 on the two 20 diameter circles in the pocket and they turn blue. Click 🖱 on [Finish (ESC)]
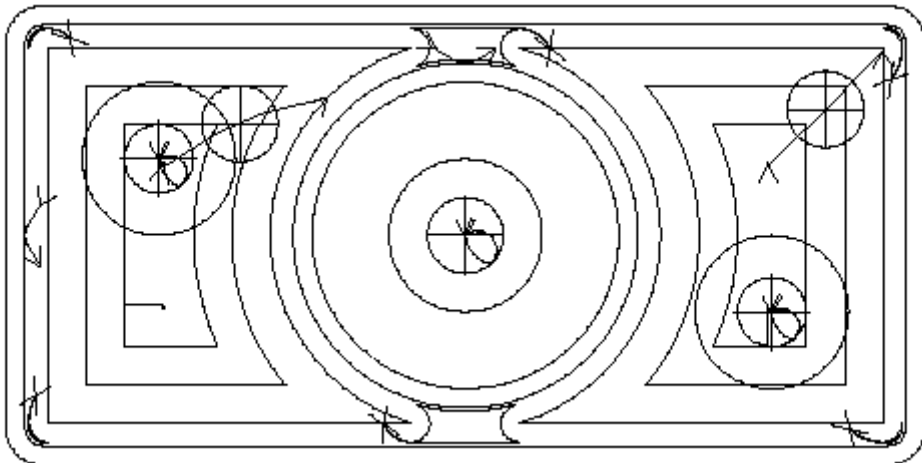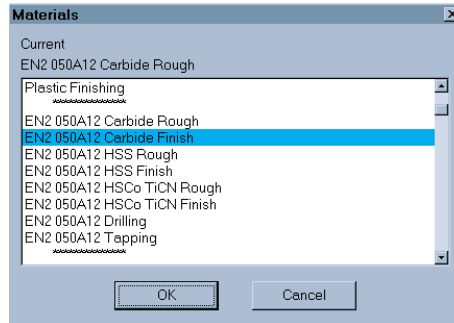

The pocketing toolpaths are displayed.

Tool Selection

Operation 6 is to finish profile the 20-diameter holes and this requires another tool. In this example, we are going to use the 12mm diameter tool.
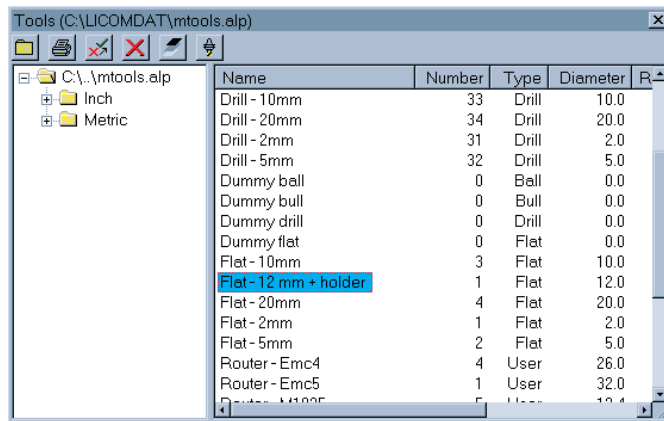
Cutting Speed

☞ Select **Machine** | **Select Material** 🖼. The material dialog is displayed.

☞ Select the **EN2 050A12 Carbide Finish** option. Click 🖱 on the option to highlight it, then click 🖱 on ⬚ OK ⬚

Select Tool

☞ Select **Machine** | **Select Tool** 🔧 . The tool library dialog is displayed.

☞ Pick the tool name **Flat – 12 mm + holder**, then click 🖱 on 🔧. The tool library dialog disappears and a schematic of the tool is displayed.

☞ The command prompt line prompts you to press R to accept the tool selection or Esc to return to the tool library dialog. Press ↵.

Operation 6

Finish Profile

☞ Select **MACHINE** | **Rough or Finish** .

The first dialog box shown differs between the Advanced and Standard modules.

☞ Click 🖰 on (vertical and then) the **Selected** button and the second dialog box is displayed.
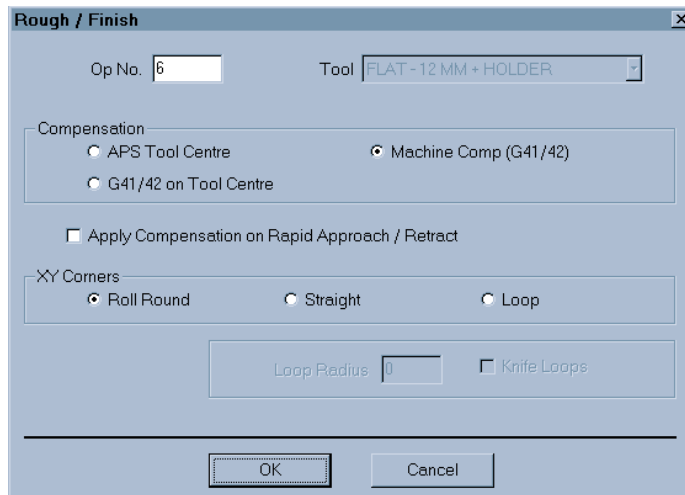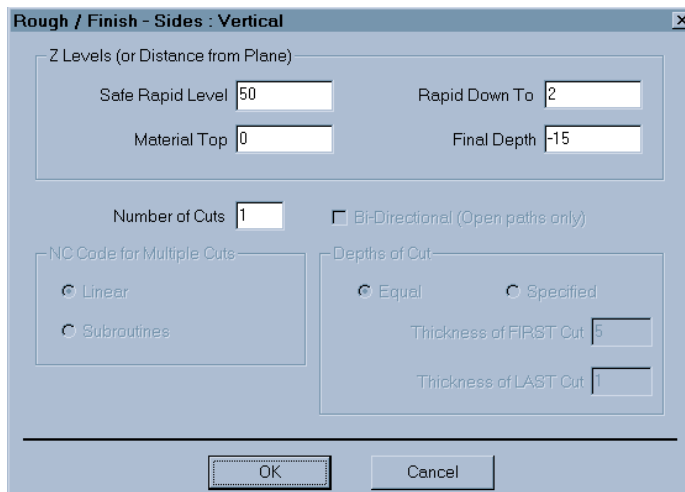
Rough / Finish

Op No. 6　　　　Tool FLAT - 12 MM + HOLDER

Compensation
- ○ APS Tool Centre　　　● Machine Comp (G41/42)
- ○ G41/42 on Tool Centre

☐ Apply Compensation on Rapid Approach / Retract

XY Corners
- ● Roll Round　　　○ Straight　　　○ Loop

Loop Radius 0　　☐ Knife Loops

OK　　Cancel

☞ Set the options as shown. When complete, click 🖰 on ⬜ OK and the third dialog box is displayed.

Rough / Finish - Sides : Vertical

Z Levels (or Distance from Plane)
- Safe Rapid Level 50　　　Rapid Down To 2
- Material Top 0　　　Final Depth -15

Number of Cuts 1　　☐ Bi-Directional (Open paths only)

NC Code for Multiple Cuts
- ○ Linear
- ○ Subroutines

Depths of Cut
- ○ Equal　　　○ Specified
- Thickness of FIRST Cut 5
- Thickness of LAST Cut 1

OK　　Cancel

☞ Set the options as shown. When complete, click 🖰 on ⬜ OK and the fourth dialog box is displayed.

Set the options as shown. When complete, click 🖰 on ⸢ OK ⸥

The dialog box disappears and the command line prompts you to select the geometry. Left click 🖰 on the middle inner 20-diameter circle: it turns blue. Click 🖰 on ⸢ Finish (ESC) ⸥

The pocketing toolpaths are displayed.

To repeat the command, you can press ⸢Spacebar⸥ or

Select MACHINE | Rough or Finish 🔧.

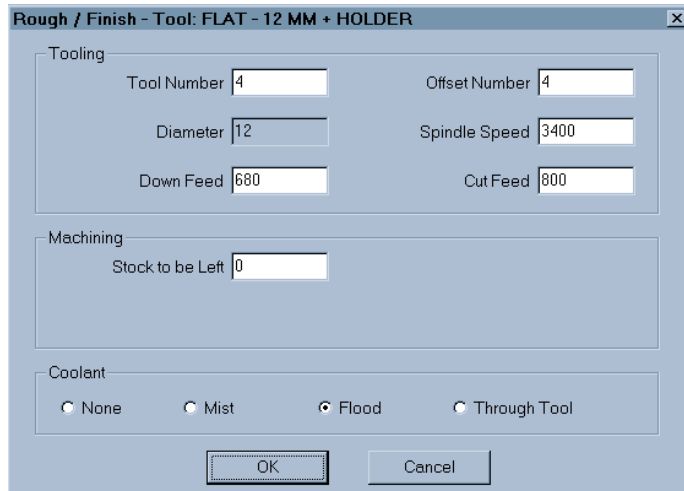The first dialog box shown differs between the Advanced and Standard modules.

Click 🖰 on (vertical and then) the Selected button and the second dialog box is displayed.

Rough / Finish - Sides : Vertical

Z Levels (or Distance from Plane)

Safe Rapid Level	50	Rapid Down To	-13
Material Top	-15	Final Depth	-26

Number of Cuts 1   ☐ Bi-Directional (Open paths only)

NC Code for Multiple Cuts
- ○ Linear
- ○ Subroutines

Depths of Cut
- ○ Equal   ○ Specified

Thickness of FIRST Cut 5

Thickness of LAST Cut 1

OK   Cancel

☞ Set the options as shown. When complete, click 🖱 on ⟦ OK ⟧ and the third dialog box is displayed.

Rough / Finish - Tool: FLAT - 12 MM + HOLDER

Tooling

Tool Number	4	Offset Number	4
Diameter	12	Spindle Speed	3400
Down Feed	680	Cut Feed	800

Machining
Stock to be Left 0

Coolant
- ○ None   ○ Mist   ⦿ Flood   ○ Through Tool

OK   Cancel

☞ Set the options as shown. When complete, click 🖱 on ⟦ OK ⟧ and the fourth dialog box is displayed.
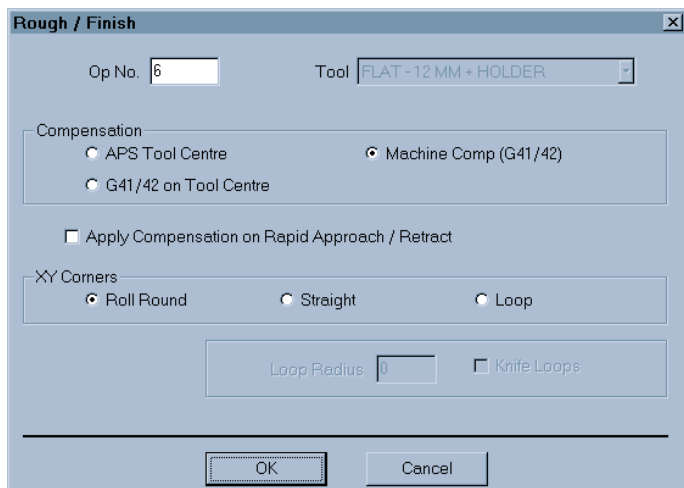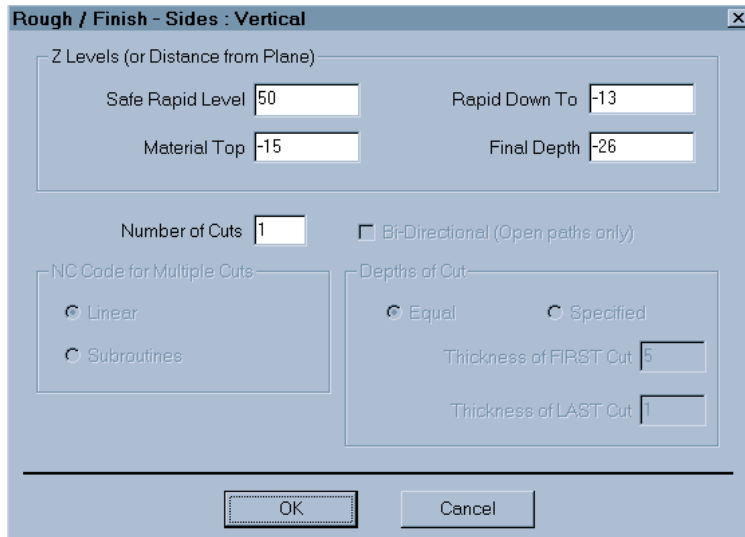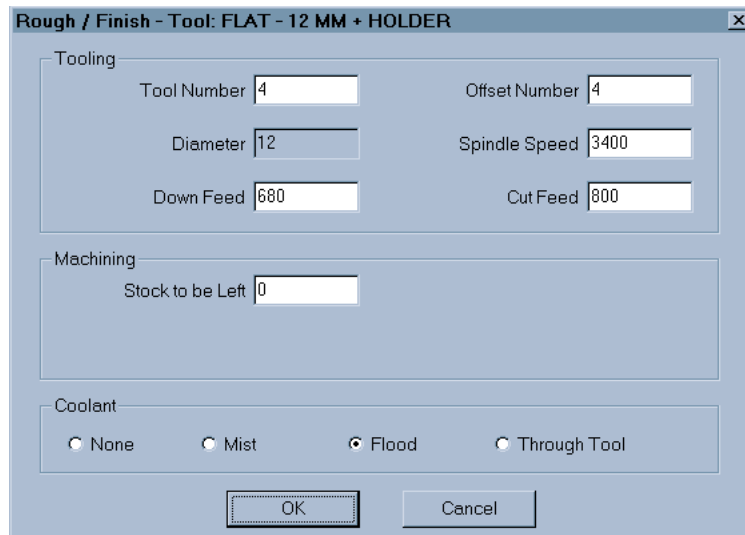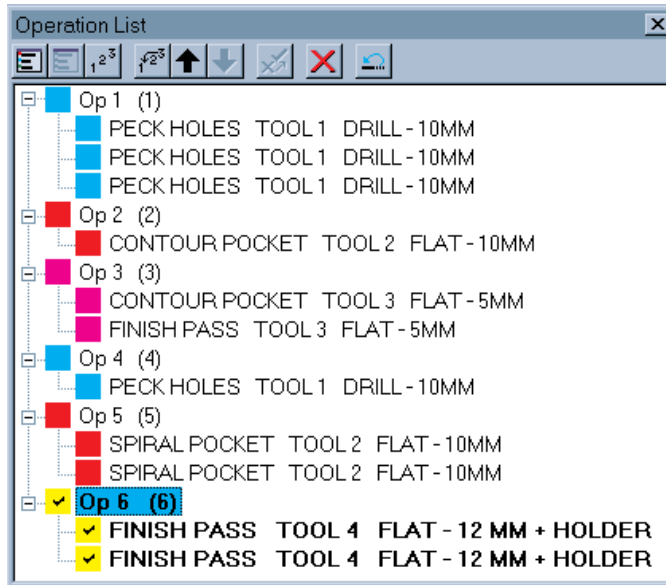
☞ Set the options as shown. When complete, click 🖱 on ⟦ OK ⟧

The dialog box disappears and the command line prompts you to select the geometry. Left click 🖱 on the 20-diameter circles in the pocket: they turn blue. Click 🖱 on ⟦ Finish (ESC) ⟧
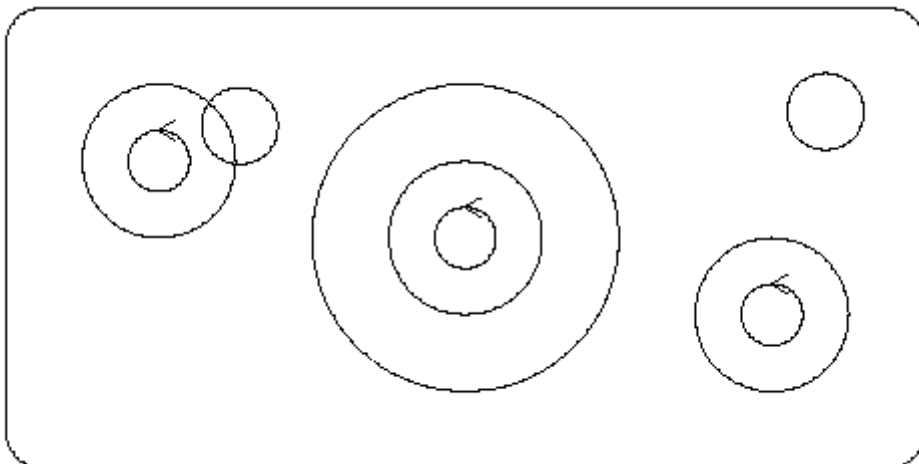
The pocketing toolpaths are displayed.

It is now necessary to apply the lead in and out. In order to simplify the toolpath selection, the previous operations can be turned off from the display.

☞ Select **MACHINE | Edit Operations** 🖼️ ⌈Ctrl⌉ + ⌈E⌉. The edit operations window is displayed.

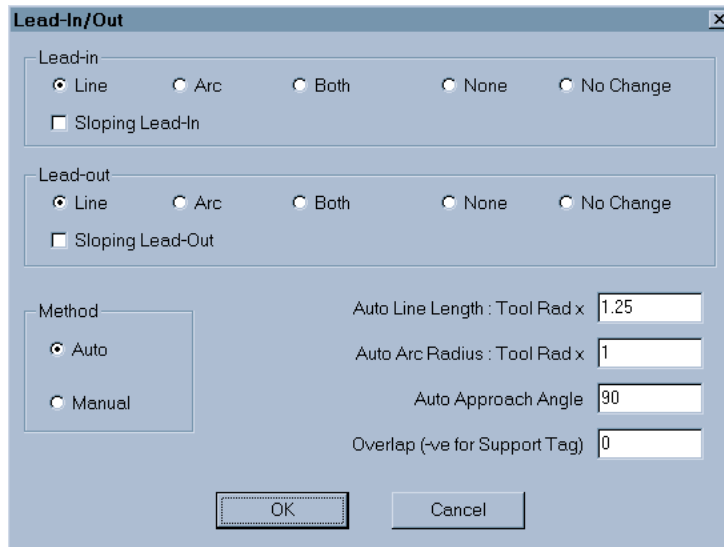☞ Select the **Hide All** 🖼️ button, then reselect **Op 6**

Tool Lead In / Out

The profiling cuts use cutter compensation and will require a lead in and out to be applied.

☞ Select **MACHINE | Tool Lead In/Out** 📇

The dialog box is displayed. There are several sections, each setting an aspect of the lead in and/or out that is applied.

☞ Set the options as shown. When complete, click 🖱 on ☐ OK

☞ The dialog box disappears and the command line prompts you to select the toolpaths. Click 🖱 on the profiling toolpaths inside the 20-diameter circles.
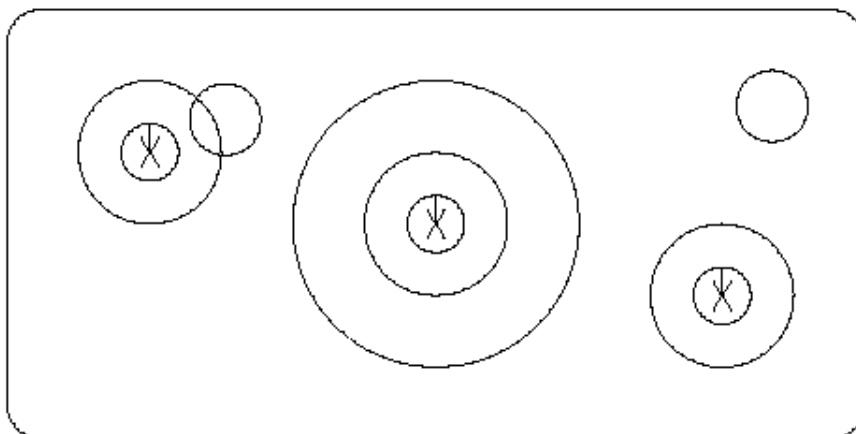
It is now necessary to turn on all the operations and check through the program before saving and outputting the NC-Code.

LICOM AlphaCAM Milling Tutorial

☞ Select **MACHINE** | **Edit Operations** 📧 Ctrl + E. The edit operations window is displayed.
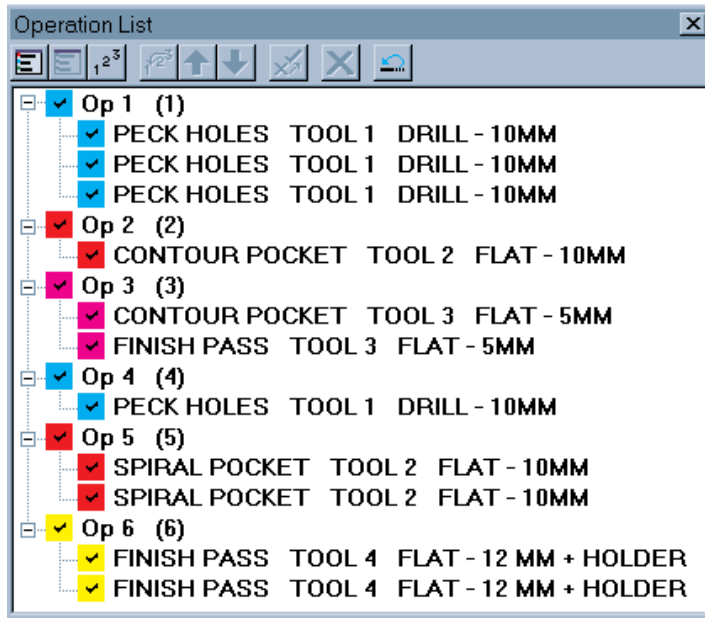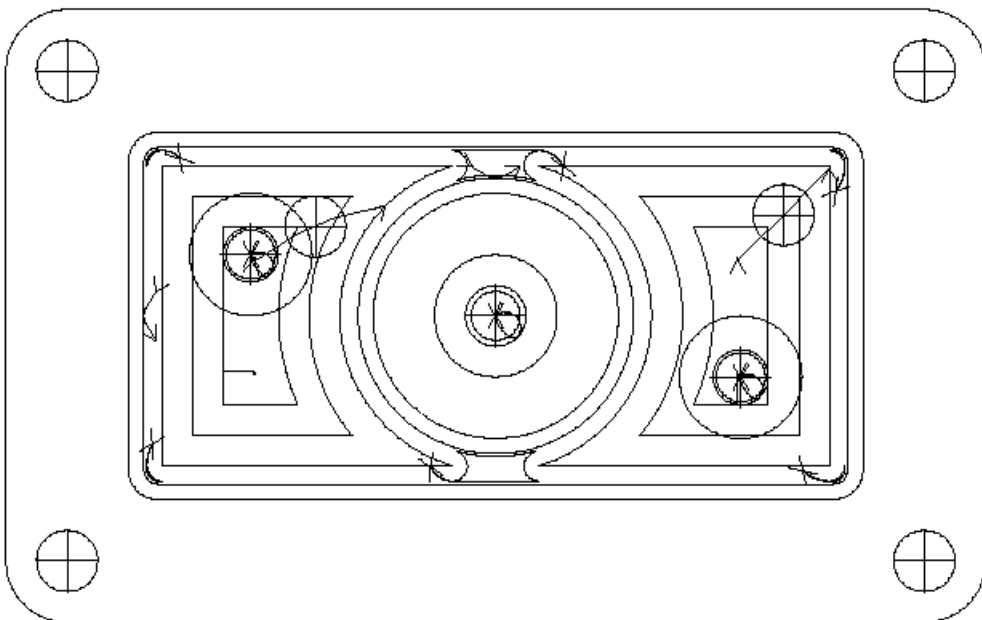
Operation List

- ☑ **Op 1 (1)**
  - ☑ PECK HOLES   TOOL 1   DRILL - 10MM
  - ☑ PECK HOLES   TOOL 1   DRILL - 10MM
  - ☑ PECK HOLES   TOOL 1   DRILL - 10MM
- ☑ **Op 2 (2)**
  - ☑ CONTOUR POCKET   TOOL 2   FLAT - 10MM
- ☑ **Op 3 (3)**
  - ☑ CONTOUR POCKET   TOOL 3   FLAT - 5MM
  - ☑ FINISH PASS   TOOL 3   FLAT - 5MM
- ☑ **Op 4 (4)**
  - ☑ PECK HOLES   TOOL 1   DRILL - 10MM
- ☑ **Op 5 (5)**
  - ☑ SPIRAL POCKET   TOOL 2   FLAT - 10MM
  - ☑ SPIRAL POCKET   TOOL 2   FLAT - 10MM
- ☑ **Op 6 (6)**
  - ☑ FINISH PASS   TOOL 4   FLAT - 12 MM + HOLDER
  - ☑ FINISH PASS   TOOL 4   FLAT - 12 MM + HOLDER

☞ Select the **Show All** 🔲 button.
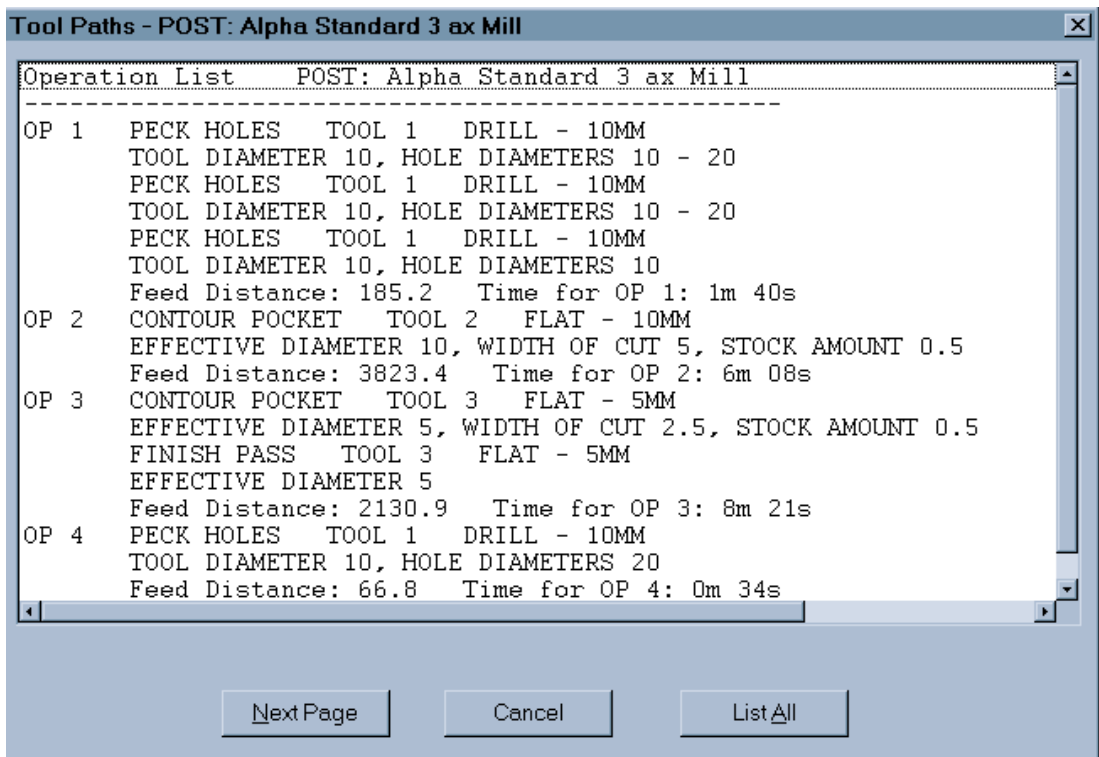
Display Options

At this stage, you may wish to utilise some of the different display options available. The commands can be found in the **VIEW | Display Options** menu.

To see the area of material cut by the tools, select ![icon] followed by ![R]. To return the display to normal, select ![icon] followed by ![R].

To see the tool in an animated display, select ![icon], ![icon] and **3D**. The screen automatically splits into four views and an animated tool is displayed. To see the animation again, select ![R].

To return the display to normal, select ![icon], ![icon] and **3D**.

To see the NC code that has been generated, select **FILE | List NC Code** ![icon]. The list screen is displayed. Select the **List All** button. The display prompts you to enter a Program Number: enter **1**, then select ⌜ OK ⌟. The NC program is displayed. To return to the graphics display, select the ⌜ Cancel ⌟ button.

```
Tool Paths - POST: Alpha Standard 3 ax Mill                          ×

Operation List     POST: Alpha Standard 3 ax Mill
-------------------------------------------------
OP 1    PECK HOLES    TOOL 1    DRILL - 10MM
        TOOL DIAMETER 10, HOLE DIAMETERS 10 - 20
        PECK HOLES    TOOL 1    DRILL - 10MM
        TOOL DIAMETER 10, HOLE DIAMETERS 10 - 20
        PECK HOLES    TOOL 1    DRILL - 10MM
        TOOL DIAMETER 10, HOLE DIAMETERS 10
        Feed Distance: 185.2    Time for OP 1: 1m 40s
OP 2    CONTOUR POCKET    TOOL 2    FLAT - 10MM
        EFFECTIVE DIAMETER 10, WIDTH OF CUT 5, STOCK AMOUNT 0.5
        Feed Distance: 3823.4    Time for OP 2: 6m 08s
OP 3    CONTOUR POCKET    TOOL 3    FLAT - 5MM
        EFFECTIVE DIAMETER 5, WIDTH OF CUT 2.5, STOCK AMOUNT 0.5
        FINISH PASS    TOOL 3    FLAT - 5MM
        EFFECTIVE DIAMETER 5
        Feed Distance: 2130.9    Time for OP 3: 8m 21s
OP 4    PECK HOLES    TOOL 1    DRILL - 10MM
        TOOL DIAMETER 10, HOLE DIAMETERS 20
        Feed Distance: 66.8    Time for OP 4: 0m 34s

        [Next Page]       [Cancel]       [List All]
```

Finally, before you output the NC-Code, the solid display can be used to check the program. Before using the solid simulation, it is preferable to set the material size.
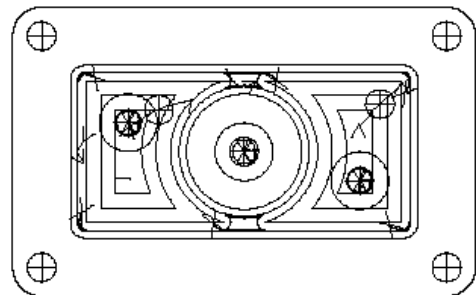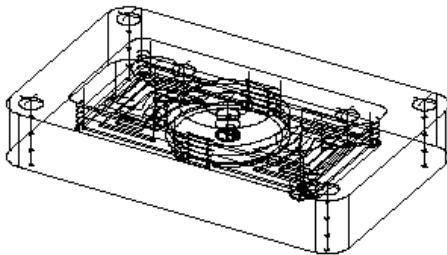
☞ **Select 3D | Set Material Size** ⬦

☞ The command line prompts you to select the geometry, which represents the material. Click 🖱 on the outer rectangle and a dialog box is displayed.

MATERIAL	✕
Material Top: Z	0
Material Bottom: Z	-25
OK	Cancel
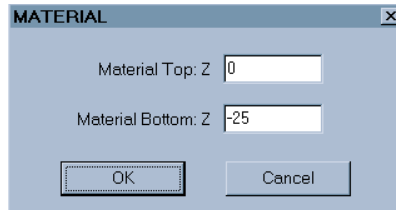
☞ Set the options as shown. When complete, click 🖱 on ⬚ OK ⬚

☞ To see the effect of setting the material, select **VIEW | 3D Views** **3D**

☞ Select **VIEW | 3D Simulation** 📦. to display tool motion removing material from a 'solid block'.

☞ AlphaCAM displays a warning message if you haven't defined a material boundary and offers to define one for you. Click 👆 on ⬚ OK ⬚ to accept and the 3D simulation windows is displayed, or go back to the previous section and define the material size.
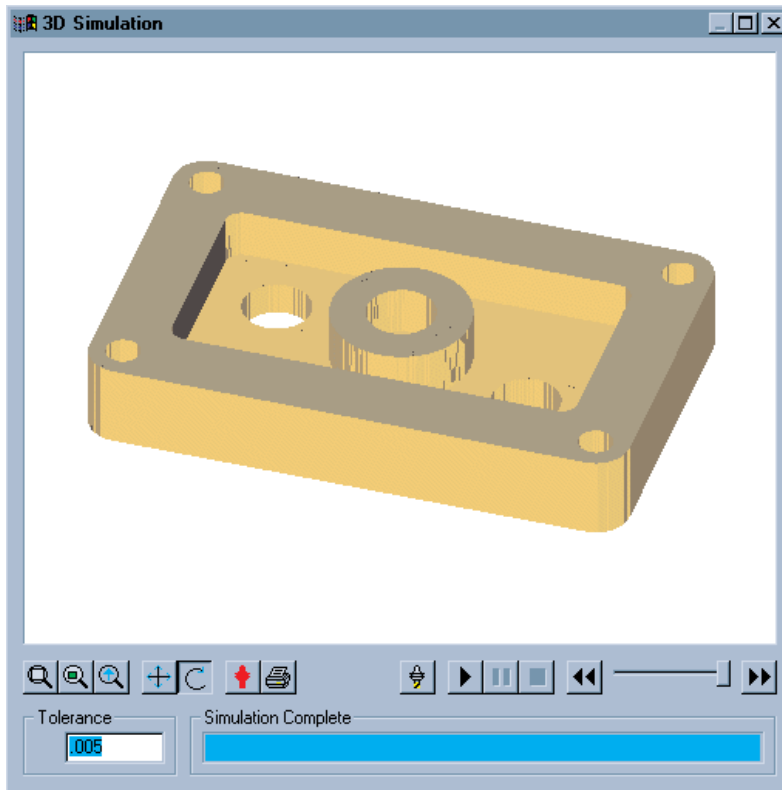
Switching the tool display off 🔧 will speed up the simulation results.

Altering the Tolerance to a smaller value will improve the detail of the display.

NOTE:

> The tolerance is related to the overall size of the machined part. Making the tolerance too small will have a dramatic effect on the speed and may even cause the computer to run out of virtual memory and hang the system.
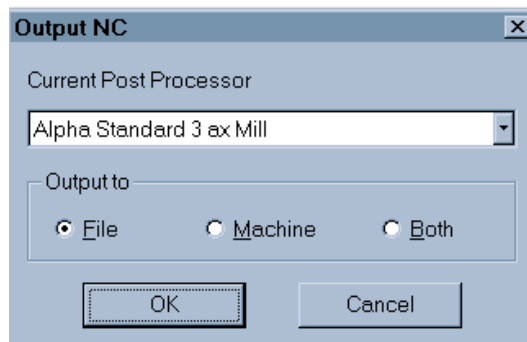> E.G. This part is 160mm x 100mm x 25mm setting the tolerance smaller than 0.005 would cause the system to run very slowly.

When you are ready to conclude the machining, close the 3D simulation by clicking on the ⊠ at the top right corner of the 3D simulation window.

Save and Output.

Finally, it is necessary to save the job and output the NC-code.

☞ Select **FILE | Save As**. The save as dialog box is displayed.

☞ Select a suitable save in drive and directory.
  (see the Windows 95 training notes.)

☞ Enter a suitable filename for this job.

☞ Select **FILE | Output NC** . 🗃 The output dialog is displayed.

Output NC	✕
Current Post Processor	
Alpha Standard 3 ax Mill ▾	
┌ Output to ───────────────────	
⦿ _F_ile ○ _M_achine ○ _B_oth	
[OK] [Cancel]	

☞ Set the options as shown. When complete, click 🖰 on [OK] and the Save
  As dialog box is displayed. The filename will be set to the same as the
  previously used job name. It is possible to save the NC-Code in the same
  folder and using the same name as the job, because **AlphaCAM** uses
  different extensions for drawings (.amd) and NC files (.anc).

☞ Either press [Enter↵] or click 🖰 on [OK] and save the NC-Code.

Manipulation of the NC file and dispatch of the NC file to the machine tool is
done using

AlphaEdit + RS232 Comms   AlphaEDIT
                                    + RS232 ...