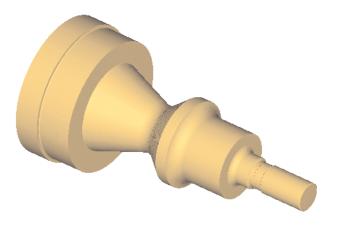
LICOM AlphaCAM

CAD/CAM system for Windows[™]

SIMPLE LATHE





Conventions When Using The Tutorial	3
Introduction	3
The Part Geometry	5
The Process Plan	7
Create the Part Profile	8
Create the billet profile	14
Preparing Geometry for Machining	
Machining Setup	
Operation 1 Facing	19
Operation 2 Rough Turn the Diameter	21
Operation 4 Grooving	27
Operation 5 Finish Face	30
Operation 6 Finish Profile	33
Operation 7 Finish Profile Back Turning.	
Operation 8 Threading	39
Save and Output.	44



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 *if* 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

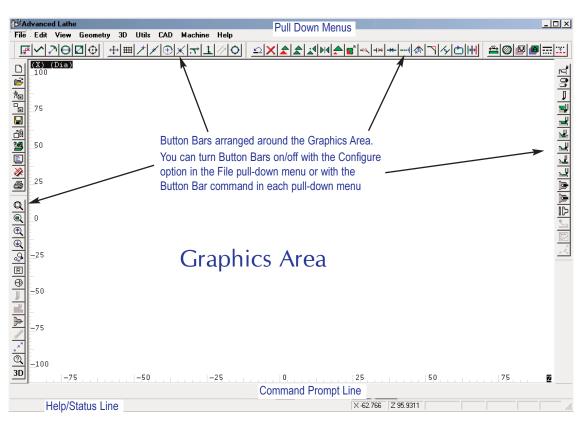
This AlphaCAM tutorial has been designed to give you a flavour of how AlphaCAM works, how interactive it is, how easy it is to learn and how easy it is to use. We assume that you are familiar with the concepts involved in CNC programming and have a reasonable understanding of your computer and the Windows 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, Lathe 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 Lathe'** module. This will take you into the graphical portion of the system. Your screen will look similar to the one following.

LICOM AlphaCAM Lathe Tutorial





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.

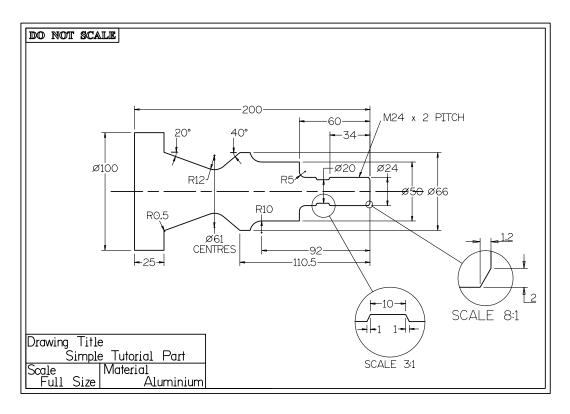
Take the trouble to locate the buttons on your screen. You can speed up your work by avoiding having to pull down menus and side menus to click on commands, when one click on a button will suffice.

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, and information will be displayed in the Help/Status Line.



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.

In the Lathe module it is only necessary to create half the profile. It is normal to define the top half, as this is the positive side.



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 some cases, it may be necessary to create other geometries as well the part geometry in order to control the machining toolpaths.

OP No	Operation Description	T No.	Tool Description
1	Rough Face to remove the excess material from the front of the billet	1	EBLC 08 16 95 5 80 CT525 P
2	Rough Diameter to remove the excess material from the diameters	1	EBLC 08 16 95 5 80 CT525 P
3	Pocket to remove the excess material from re-entrant features.	2	EBNR 6 12 90 90 180 CT525 P
4	Groove	3	EFLG 04 5 90 90 180 CT525 P
5	Finish Face	4	EBLD 04 15 93 32 55 CT525 P
6	Finish Profile diameters	4	EBLD 04 15 93 32 55 CT525 P
7	Finish Profile (Back Turning)	5	EDRV 04 16 93 52 35 CT525 P
8	Thread ISO 24 diameter fine.	6	EDLM 012 2.0 60 60 60 GC1020 PK



Create the Part Profile

In this example, we will use APS Fast Geometry to create the part profile.



F Select GEOMETRY | APS Fast Geometry | Point.

F The command line prompts for the co-ordinates of the point. Type **0 - 0 -**

This position is selected on the screen and a phantom line connects it to the cursor.

Pressing *Esc* or *P* once aborts any repeating commands.

Pressing [Esc] or [] during APS Fast Geometry when there is not a command active displays the APS Fast Geometry Menu.



- F The command line prompts for the first chamfer distance. Type 2
- F The command line prompts for the second chamfer distance. Type **1.2**
- F The command line prompts for the co-ordinates of the point where the lines that are to be chamfered intersect. Type **24**

Pressing \frown for the Z value (without entering a value) accepts the Z value from the last command.

This position is selected on the screen and a phantom line connects it to the 0,0 position.

F Select **VIEW** | **Zoom All** . The geometry fills the drawing screen.

F Select GEOMETRY | APS Fast Geometry | Point.

F The command line prompts for the co-ordinates of the point. Type → -34 →

> Pressing \frown for the X value accepts the default X value 24 from the last command

The geometry is now drawn to this position on the screen and a phantom line connects it to the cursor.

The next turn is also a **Point**.

Since the command repeats, it is not necessary to reselect it.

I → The command line prompts for the co-ordinates of the next point. Type 20 → -35 →.

The geometry is now drawn to this position on the screen and a phantom line connects it to the cursor.

The next turn is also a **Point**.

Since the command repeats, it is not necessary to reselect it.

F The command line prompts for the co-ordinates of the next point.

Type **← −45 ← .**

Pressing 🖅 for the X value accepts the default X value 20 from the last command.

The geometry is now drawn to this position on the screen and a phantom line connects it to the cursor.

The next turn is also a **Point**.

The command line prompts for the co-ordinates of the next point. Type **24** -46 -

The geometry is now drawn to this position on the screen and a phantom line connects it to the cursor.







F Select GEOMETRY | APS Fast Geometry | Line to Line Blend.

- The command line prompts for the Blend radius value. Type 5 -
- The command line prompts for the co-ordinates of the point where the lines that are to be blended intersect.

Type **←** –60 **←**

Pressing 🚽 for the X value accepts the default X value 24 from the last command.

A phantom line is now drawn to this position on the screen, as the radius can not be drawn until the next turn is entered.



The command line prompts for the co-ordinates of the point. Type **50**

Pressing I for the Z value accepts the default Z value -60 from the last command.

The blend radius and the connecting lines are now drawn. A phantom line connects the current position to the cursor.





F Select GEOMETRY | APS Fast Geometry | Known Arc.

F The command line prompts for the radius value of the Arc. Type **10 ↓**.

The first dialog box is displayed

 \mathbb{G} This prompts for the direction of rotation of the Arc. Select

The command line prompts for the co-ordinates of the circle centre. Type 70 -92 - 70 Diameter is calculated 10 radius times 2 plus 50

A phantom circle is now drawn at this position on the screen. The arc can not be drawn until the next turn is entered.



🕼 Select GEOMETRY | APS Fast Geometry | Arc to Line.

F The command line prompts for the radius value of the Blend.

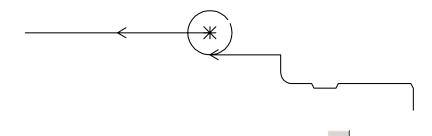
The first dialog box is displayed

 \mathbf{F} This prompts for the direction of rotation and included angle of the blend. The **CW** direction and **<180** angle is offered as default Select the **CCW** and click and **OK**

A second dialog box is displayed

First prompts for the direction of the Line. Select the 180 direction, then click in on OK or Type **180** -

A line is drawn from the centre of the arc in phantom style, and will not be drawn solid until the next turn is entered, which fixes the position of the line.





F The command line prompts for the co-ordinates of the point. Type 66 🖵 –110.5 🗲

The previous half-known Arc to Line turn is now calculated and drawn to the point. A phantom line connects it to the cursor.



For Select GEOMETRY | APS Fast Geometry | Known Arc.

The command line prompts for the radius value of the Arc. Type **12** -

The first dialog box is displayed

 \mathbb{G} This prompts for the direction of rotation of the Arc. Select

The command line prompts for the co-ordinates of the circle centre. Type **61** - F1

Pressing *F1* in place of the Z value means that the Z value is unknown.

A direction dialog box is displayed

This prompts for the direction into the arc. Type **180 + 40** -

A phantom circle is now drawn at this position on the screen. The arc can not be drawn until the next turn is entered.



The command line prompts for the Blend radius value. Type **0.5**

The command line prompts for the co-ordinates of the point where the lines that are to be blended intersect.

Type [F1] **−175**

Pressing *F1* in place of the X value means that the X value is unknown.

A direction dialog box is displayed

This prompts for the direction <u>into</u> the intersection. Type **180–20** -

A phantom line is now drawn to this position on the screen.

🕼 Select VIEW | Zoom All 🔍 . The geometry fills the drawing screen.



For Select GEOMETRY | APS Fast Geometry | Point.

The command line prompts for the co-ordinates of the point. Type **100** - (-)

Pressing 🖅 for the Z value accepts the default Z value -175 from the last command.

The blend radius and the connecting lines are now drawn. A phantom line connects the current position to the cursor.

The command line prompts for the co-ordinates of the next point. Type -200 -200 -200

Pressing 🖅 for the X value accepts the default X value 100 from the last command.

The geometry is now drawn to this position on the screen and a phantom line connects it to the cursor.

The command line prompts for the co-ordinates of the next point. Type **0** - J.

Pressing 🖅 for the Z value accepts the default Z value –200 from the last command.

The geometry is now drawn to this position on the screen and a phantom line connects it to the cursor.

It is only necessary to draw half the component.

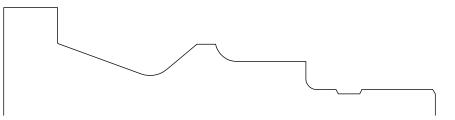
🕼 Select GEOMETRY | APS Fast Geometry | Finish.

The geometry changes

from yellow (current geometry) to green (finished geometry).

F Select VIEW | Zoom All 🔍 . The geometry fills the drawing screen.

This completes the part geometry.



Now we need to create geometry representing the billet material.



Create the billet profile

- For Select GEOMETRY | APS Fast Geometry | Point.
- The command line prompts for the co-ordinates of the point. Type **0** - **2** -
- The command line prompts for the co-ordinates of the next point. Type **105** -

The command line prompts for the co-ordinates of the next point. Type -202 -202

It is only necessary to draw half the billet.

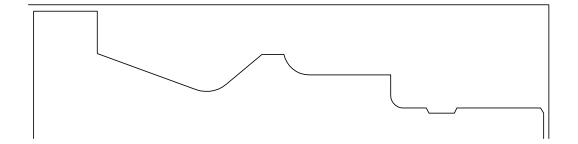
For Select GEOMETRY | APS Fast Geometry | Finish.

The geometry changes

from yellow (current geometry) to green (finished geometry).

This completes the billet geometry.

The billet geometry does not have to be rectangular it can be any shape (as in the case of a casting or forging).





Preparing Geometry for Machining

If the geometry has been created using CAD geometry commands, it is possible that the geometry profile is not in a continuous form. To ascertain whether this is the case, use the Ghost Tools option, which displays a ghost tool on the start point of each geometry profile.



In this case, there should be two ghost tools: one for the part profile and one for the billet profile.

If there are more than two ghost tools, it is necessary to use the Join command to join the geometries into a continuous profile.

Select **EDIT** | **Break join etc.** | **Join** then select the geometries to be joined. When they are all blue, pick the button.

To turn off the ghost tool display, select the ghost tool command again.



Machining Setup

🕼 Select FILE | Select Post

AlphaCAM displays the available post processors. Select a suitable post processor. The post processor determines the format of the output NC code for your machine.





AlphaCAM displays the dialog box with all the materials currently specified in the material library. Feeds and speeds are determined automatically based on the material and tools used.

Select ALUMINIUM IC8025 and click 🖱 on [0K

🕼 Select MACHINE | Set Tool Change Pos 🖻

This is the position to which the tool is sent in order to index the turret. This position can be set once for the program or set for each tool prior to tool selection.

F The command line prompts for the co-ordinates of the next point. Type **300** - **100** .

> As part of the toolpath creation, AlphaCAM displays dialog boxes to obtain all the information required to carry out the operation. Enter all the data as shown in the diagrams. Remember to press Tabth to move to the next entry field. Do NOT press [Enter+] until all entries are complete.

> > In dialog boxes, [Enter] = Finished.



When Selecting tools it will be necessary to navigate the tooling directory structure. The default turning tool library is split up into different folders. Each folder containing different catagories of turning tool. The select tool open file dialog box below shows the default directory structure below **Licomdat/Ttools.alp**.

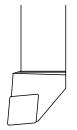
Open	<u>? ×</u>
Look in: 🔄 TTOOLS.ALP	▼ ← 🗈 📸
C-Y Axis Mill Tools	🗋 Taps
Drills - Twist	🚞 Threading
📄 Finishing	🚞 U-Drills
🗋 Inch Tools	📕 Button - 5mm dia.att
Parting and Grooving 🔋 🗍 Drill - 10mm dia.att	
🔁 Roughing	📕 Drill - 25mm dia.att
	<u> </u>
File name:	Open
Files of type: Licom Mill/Turning Tool	Cancel

When selecting tools it will be necessary to return to the top level of Ttools, using the up one level button, then select the required folder in order to select the tool specified for the operation.

The "Turning Tool Library.pdf" file containing information about the structure of the Licom turning tool naming convension and detail pictures of all the default tools can be found in the Licomdir/Ttools folder.



- F Select MACHINE | Select Tool 🧾 . AlphaCAM displays the tool library, listing all the folders and tools currently defined in the Ttools library folder.
- 🕼 Select the **Roughing** folder and then click 🖱 on **Open**. The tools located in the Roughing folder will be displayed.
- 🕼 Select EBLC 08 16 95 5 80 CT525 P tool and then click 🖱 on Open. The tool is selected from the library and displayed on your screen, as shown below, attached to the cursor.



The command line prompts you to press [Enter -] to accept and select the tool or Esc to abort the tool selection and return to the tool list.

Press Enter + to accept the tool

It is important to press either [Enter] or [Esc]. Selecting anything else results in the tool **NOT** being selected.

The select tool dialog is displayed.

BLC 08 16 95 5 80	CT525 P Rotation: CW 🛛 🗙	1		
	Tool Number: T		EBLC 08 16 95 5 80 C	T525 P Rotati
Maximu	m Spindle Speed: S 4000			Tool Number: T
Speed	Feed		Maximum) Spindle Speed: S
 Surface Spindle 	Image: Oper REV Image: Oper MIN		Speed © Surface	Feed
Turret	Station • Front		C Spindle	C per MIN
C Below C/L	C Back			Cancel
(OK	Cancel			

Set the options as shown then select OK

The Turret and Station options are displayed only when using the Advanced module.

×



Operation 1 Facing

The facing option is for area clearance facing, which removes the excess from the face of the billet with multiple cuts.



- The command line prompts you to select the billet geometry. Click 🖱 on the billet outline.
- The command line prompts you to select the part geometry. Click 🖱 on the part outline.

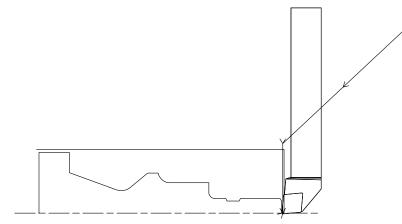
Tool EBLC 08 16 95	5 80 CT525 P
	Face Stock 0.1
	Width of Cut 3
Fa	ce Down to X Dia. 0
	Feed-in Distance 4
Lift-off	
C Rapid	Feed

Set the options as shown, then click on OK

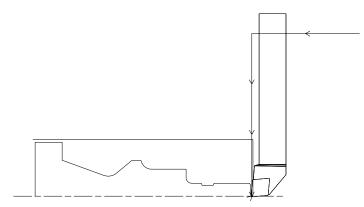
FACING - Tool E	BLC 08 16 95 5	5 80 CT525 P	
			Op No. 1
Tooling	ol Number 1		Offset Number 1
SURFACE	Speed: S 400		Feed per REV (X): F 0.4
Feed per RE	V (X/Z): F		Feed per REV (Z): F
Coolant			
C None	C Mist	Flood	C Through Tool
NC Code			
C Linear	C Canne	ed Cycle	
	OK.	C	Cancel
the options as show	wn, then o	click on	ОК



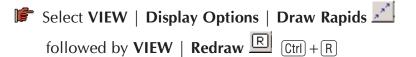
The command line prompts you to indicate the path of the rapid motion from the tool change position to the start of the cut. The dash line on screen shows the rapid motion. Clicking the T RMB will always generate a straight linear rapid motion.



If Ortho mode F3 is active the rapid motion is restricted to single axis motion This is the mode recommended through out this tutorial as not all lathes perform true simultaneous 2 axis linear motion.



Rapids disappear if the **VIEW** | **Redraw** \square \square \square \square **C**trl + \square or **Zoom All** \square is used. In order for the rapids to be displayed, the draw rapids option must be set active.





Operation 2 Rough Turn the Diameter

F	Select MACHINE	Conventional Turning	Roughing 📑

The first dialog box allows for the direction of roughing cuts.

Roug	yhing	×
	Diameter	
	Face	
	Contour	
	Cancel]

Select Diameter

- The command line prompts you to select the billet geometry. Click 🖱 on the billet outline.
- The command line prompts for the Start point on the part geometry.

Using \checkmark [F6], click \bigcirc on the top of the front face.

F The command line prompts for the End point on the geometry.

Using \checkmark [F6], click [] on the left end of the OD at the back.

The selected geometry turns blue.

The command line prompts for a point to indicate on which side of the part geometry the cutting is to take place. Position the cursor above the part geometry in the middle and click 🖱 the left mouse button.

DIAMETER ROUGH	
Tool EBLC 08 16 95 5 80	CT525 P
	Face Stock 0.1
Diamete	er Stock (Rad Val) 0.1
Dept	h of Cut (Rad Val) 3
	Feed-in Distance 1
Back-of	if Distance per cut 0
🔽 Clean Profile w	vith Each Cut
Method	
Conventional	C Reverse
Lift-off	
C Rapid	Feed
OK.] Cancel

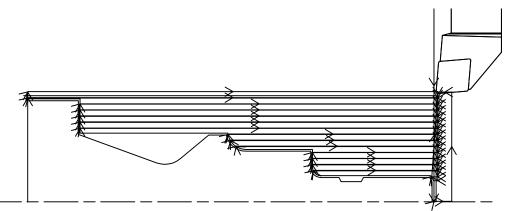
Set the options as shown, then click on OK



DIAMETER ROUGH - Tool EBLC 08 16 95 5 80 CT525 P 🛛 🗙				
			Op No. 2	
Tooling	ool Number 1		Offset Number 1	
SURFAC	E Speed: S 400	_	Feed per REV (X): F 0.4	
Feed per F	EV (X/Z): F 0.4	_	Feed per REV (Z): F	
Coolant				
C None	C Mist	Flood	C Through Tool	
NC Code				
C Linear	🖲 Cann	ed Cycle		
Cancel				

- Set the options as shown, then click on OK
- The command line prompts for the starting position of the canned cycle. Type **107** - **2** -
- The command line prompts you to indicate the path of the rapid motion from the tool's last position to the start of the roughing cycle. The dash line on screen shows the rapid motion, click 🖱 to set the rapid motions path.

Click the 🖱 RMB if a straight linear rapid motion is required.



The screen at this stage is covered with toolpaths. To make it easier to define the next operations, the current toolpaths can be turned off

Select MACHINE | Edit Operations 🖾 . A dialog box showing the current operations is displayed.

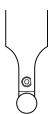
F Select **S** to hide all the operations, then press **Es** or click **T** on **X** The current toolpaths are no longer displayed.



Select MACHINE | Select Tool . AlphaCAM displays the tool library, listing all the folders and tools currently defined in the Ttools library folder.

The Roughing folder should be active. If not select the up one level button and then select the **Roughing** folder and then click \bigcirc on **Open**. The tools located in the Roughing folder will be displayed.

Select **EBNR 6 12 90 90 180 CT525 P** tool and then click ^(*) on <u>**Open</u></u>. The tool is selected from the library and displayed on your screen, as shown below, attached to the cursor.</u>**



 \mathbb{F} Press Enter \rightarrow to accept and select the tool

	EBNR 6 12 90 90 180 CT525 P Rotation: CW 🗴			
	Tool Number: T			
	Maximum Sp	bindle Speed: S 4000		
	Speed	Feed		
	Surface	per REV		
	C Spindle	C per MIN		
	Turret	Station		
	Above C/L	Front		
	C Below C/L	C Back		
	(OK)	Cancel		
Set the options as sho	wn, then click	on OK		
	AlphaCAM	×		
	😲 Goto To	ol Change Position ?		
	Yes	No		

Image: AlphaCAM recognises that the tool is not at the tool change and a dialogbox asks if it is necessary to Go To Tool Change.Select Yes

Set the rapid motion to get the tool to the tool change position.



F Select MACHINE | Conventional Turning | Pocketing 1.

Where tools have been defined with more than one programming point the system asks you to select which programming point is to be used for this operation. The F2 key is used to move through the defined programming point pressing OK activates the tool with the with the displayed programming point. Each time a tool with more than one programming point is used the system asks the user to select the required programming point.

Select OK to accept the default programming point.

The first dialog box allows you to identify whether the pocket is on a face or a diameter.

Pocketing	×
Diameter	
Face	
Cancel	

Select Diameter

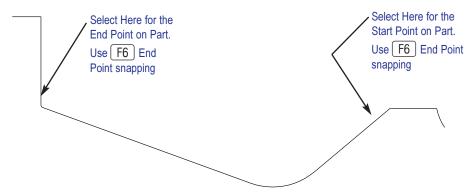
A warning is displayed when the tool clearance angle is not set. If a warning is issued, select OK.

F The command line prompts for the Start point on the part geometry.

Using \swarrow [F6], click \bigcirc on the top front corner of the recess.

F The command line prompts for the End point on the part geometry.

Using ______ F6, click 🖱 on the bottom end of the face at the back of the recess.



The selected geometry turns blue.



IAMETER POCKE	
	Face Stock 0.1
Diamete	er Stock (Rad Val) 0.1
Dept	th of Cut (Rad Val) 3
🖵 Check Clearar	nce Angle
F Bi-Directional	
Lift-off	
C Rapid	Feed
OK	Cancel

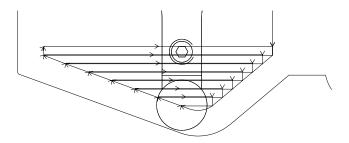
Set the options as shown, then click on OK

AMETER PO	CKET - Tool EBN	IR 6 12 90 90	180 CT525 P
			0p No. 3
Tooling	ool Number 2		Offset Number 2
SURFAC	E Speed: S 400	F	Feed per REV (X): F 0.6
Feed per R	IEV (X/Z): F 0.4	F	Feed per REV (Z): F 0.6
Coolant			
C None	C Mist	Flood	C Through Tool
NC Code			
C Linear	C Canne	ed Cycle	
	OK.	Ca	ncel

Set the options as shown, then click on OK

The operation number is automatically increased when the tool is changed.

The command line prompts you to indicate the path of the rapid motion from the tool change position to the start of the cut. The dash line on screen shows the rapid motion, set the rapid motions path required.





You can view the CNC program at any time. Select FILE | List NC Code \blacksquare .

When operations have been hidden when listing the NC Code, a dialog is displayed allowing the user to select whether to display the code for just the visible toolpaths or all toolpaths. Enter any CNC code data as requested (program number, etc.) and AlphaCAM shows the CNC program. Notice that AlphaCAM has also calculated the cycle time for this part so far.

OP 1	FACING TOOL 1 EBLC 08 16 95 5 80 CT525 P TOOL TIP RAD 0.8, WIDTH OF CUT 3
OP 2 OP 3	Feed Distance: 57.6 Time for OP 1: 0m 11s DIAMETER ROUGH TOOL 1 EBLC 08 16 95 5 80 CT525 TOOL TIP RAD 0.8, DEPTH OF CUT 3 Feed Distance: 1887.3 Time for OP 2: 4m 12s DIAMETER POCKET TOOL 2 EENR 6 12 90 90 180 CT5; TOOL TIP RAD 6, DEPTH OF CUT 3 Feed Distance: 216.1 Time for OP 3: 0m 31s
Tool	l Feed Distance
	rial: Low Carbon Steel ered Chip Former PP
STAR	Γ
	•

Select LIST ALL and AlphaCAM generates the CNC program. Use your scroll bars to view the code.

ool Paths - POST: Alpha Standard Lathe	×
N640 X43.354 Z-132.334 F0.4	
N650 Z-147.642 F0.6 N660 X49.354 Z-155.884	
N670 X49.707 Z-155.707 F0.4	
N680 G0 X49.707 Z-132.334	
N690 G1 X43.354 F0.6	
N700 X39.961 Z-134.356 F0.4 N710 G2 X37.354 Z-137.199 R5.9 F0.6	
N710 G2 X37.354 Z-137.199 R5.9 P0.6	
N730 G2 X37.912 Z-140.166 R5.9	
N740 G1 X43.354 Z-147.642	
N750 X43.707 Z-147.465 F0.4	
N760 G0 X43.707 Z-137.199 N770 G1 X37.354 F0.6	
N780 G2 Z-139.097 R5.9	
N790 G1 X37.707 Z-138.92 F0.4	1
N800 T0200 ' CANCEL TOOL 02	
N810 M30 %	
/*	لے
•	<u> </u>
Cancel	

You will see remarks in the code - e.g. '(OP1 FACING ...) and 'SELECT TOOL 01. These are for your information, they are not sent to the machine tool controller.



Operation 4 Grooving

Select MACHINE | Select Tool I. AlphaCAM displays the tool library, listing all the folders and tools currently defined in the Ttools library folder.

The Roughing folder should be active. Select the up one level button and then select the **Parting and Grooving** folder and then click \bigcirc on <u>Open</u>. The tools located in the Parting and Grooving folder will be displayed.

Select **EFLG 04 5 90 90 180 CT525 P** tool and then click T on <u>Open</u>. The tool is selected from the library and displayed on your screen, attached to the pointer.

EFLG 04 5 90 90 180 CT	1525 P Rotation: CW 🙁
	Tool Number: T 3 pindle Speed: S 4000
Speed © Surface © Spindle	Feed
Turret	Station
[0K]	Cancel

Press Enter to accept and select the tool

AlphaCAM	×
Goto Too	ol Change Position ?
Yes	No

🕼 Set the options as shown, then click on 🛛 🔿

AlphaCAM recognises that the tool is not at the tool change and a dialog box asks if it is necessary to **Go To Tool Change**. Select <u>Yes</u>

Set the rapid motion to get the tool to the tool change position.



F Select MACHINE | Conventional Turning | Grooving 🖳 .

Where tools have been defined with more than one programming point the system asks you to select which programming point is to be used for this operation. The F2 key is used to move through the defined programming point pressing OK activates the tool with the with the displayed programming point. Each time a tool with more than one programming point is used the system asks the user to select the required programming point.

Select OK to accept the default programming point.

Grooving		x
Туре	Process	
Oiameter	C Rough	
C Face	C Finish	
	Rough and Finish	
	C Single Cut	
(OK)	Cancel	

Dialog box 1 requests the type of groove and the process to be actioned.

 $\mathbf{\vec{b}}$ Set the options as shown, then click on \mathbf{OK}

F The command line prompts for the Start point on the part geometry.

Using \checkmark [F6]. Click \bigcirc on the top front corner of the groove.

F The command line prompts for the End point on the part geometry.

Using \checkmark F6. Click \bigcirc on the top back corner of the groove.

A Select Here for the End Point on Part. Use F6 End Point snapping	Select Here for the Start Point on Part. Use F6 End Point snapping

The selected geometry turns blue.



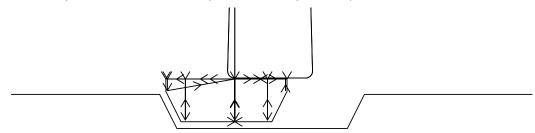
DIAMETER GROOVE	×			
Tool EFLG 04 5 90 90 180 CT5	i25 P			
Peck Distance (0 for no Peck)				
Widt	th of Cut (Tool is 5) 3.7			
Amount to be left for Finish Pass 0.1				
Clearance between Holder and Part 0.5				
[OK]	Cancel			
s shown, then clic	k on OK			
METER GROOVE - Tool EFLG 04				
ooling	Op No. 4			
Tool Number 3	Offset Number 3			
SURFACE Speed: S 400	Feed per REV (X): F			
Feed per REV (X/Z): F 0.15	Feed per REV (Z): F			
oolant				
C None C Mist 🧿	Flood C Through Tool			
C Code				
Canned Cyc	le			
(OK)	Cancel			
	Tool EFLG 04 5 90 90 180 CTS Peck Distar Wid Amount to be Clearance betwe Clearance betwe Clearance betwe Clearance betwe Clearance betwe S shown, then clic METER GROOVE - Tool EFLG 04 SURFACE Speed: S 400 Feed per REV (X/Z): F 0.15 Solant None C Mist C C Code C Linear C Canned Cyc			

🕼 Set the options as shown, then click on 🛛 OK

F The command line prompts for the safe approach position for the groove.

The safe approach position is used to indicate the point from which the grooving cycle starts, and should be with the centre of the tool approximately in the middle, above the groove. Type 25 - 42 - 42

The command line prompts you to indicate the path of the rapid motion from the tool change position to the start of the cut. The dash line on screen shows the rapid motion set the rapid motions path required.





Operation 5 Finish Face

Select MACHINE | Select Tool I. AlphaCAM displays the tool library, listing all the folders and tools currently defined in the Ttools library folder.

The Parting and Grooving folder should be active.

Select the up one level button and then select the **Finishing** folder and then click on **Open**. The tools located in the Finishing folder will be displayed.

- Select **EBLD 04 15 93 32 55 CT525 P** tool and then click T on <u>Open</u>. The tool is selected from the library and displayed on your screen, attached to the pointer.
- Press Enter + to accept and select the tool

	EBLD 04 15 93 32 55 CT	525 P Rotation: CW 🙁
		Fool Number: T
	Maximum Sp	bindle Speed: S 4000
	Speed	Feed
	Surface	📀 per REV
	C Spindle	C per MIN
	Turret	Station
	Above C/L	Front
	C Below C/L	C Back
	(OK)	Cancel
🕼 Set the options as sh	own, then click	c on OK
	AlphaCAM	
	? Goto To	ol Change Resition 2

AlphaCAM recognises that the tool is not at the tool change and a dialog box asks if it is necessary to **Go To Tool Change**. Select <u>Yes</u>

Yes

No

Set the rapid motion to get the tool to the tool change position.



For Select MACHINE | Conventional turning | Finishing Revealed to Command.

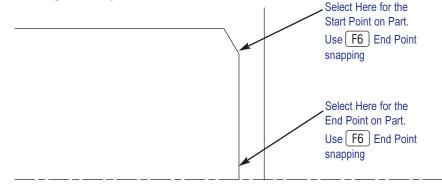
F The command line prompts for the Start point on the part geometry.

Using 🖊 🕞, click 🖱 on the top of the front face.

F The command line prompts for the End point on the part geometry.

Using \checkmark [F6], click \square on the bottom of the front face.

The selected geometry turns blue.



F The command line prompts

Pick NEXT Start Point on Part <esc>=FINISHED</esc>	× 0	z 0	[OK]
--	-----	-----	--------

As there are no more sections to machine in this operation, press Esc

FINISHING	×
Tool EBLD 04 15 93 32 55 CT525 P	
Stock Amount 0.1	
Move Between Sections	
C Rapid 💽 Feed	
Cancel	

Set the Stock Amount as shown, then click on OK



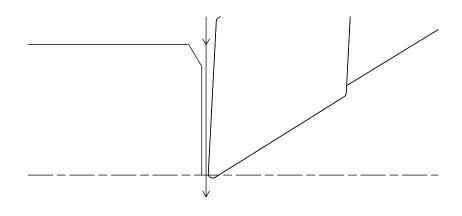
		iΡ		
		Op No. 5		
ool Number 4		Offset Number 4	_	
E Speed: S 400	F	eed per REV (X): F 0.032		
EV (X/Z): F 0.03	32 F	eed per REV (Z): F 0.032	_	
C Mist	Flood	C Through Tool		
C Canned Cycle				
	EV (X/Z): F 0.00	E Speed: S 400 F EV (X/Z): F 0.032 F C Mist C Flood	Dol Number 4 Offset Number 4 E Speed: S 400 Feed per REV (X): F 0.032 EV (X/Z): F 0.032 Feed per REV (Z): F 0.032 C Mist C Flood C Through Tool	

🕼 Set the options as shown, then click on 🛛 🛛 🕅

Auto-Lead In/Out	×
Lead-In	Lead-Out
C Perpendicular	C Perpendicular
Tangential	Tangential
C None	C None
☐ Use G41/42	Lead Length 2
<u> </u>	Cancel

Set the options as shown, then click on OK

F The command line prompts you to indicate the path of the rapid motion from the tool change position to the start of the cut. The dash line on screen shows the rapid motion set the rapid motions path required.





Operation 6 Finish Profile

This operation is performed using the same tool as the previous operation.

Finishing Select MACHINE | Conventional turning | Finishing 🖄 command.

F The command line prompts for the Start point on the part geometry.

Using \checkmark [F6], click \bigcirc on the top of the front face.

F The command line prompts for the End point on the part geometry.

Using \checkmark F6, click \bigcirc on the top front corner of the groove.

The selected geometry turns blue.

🕼 The command line prompts

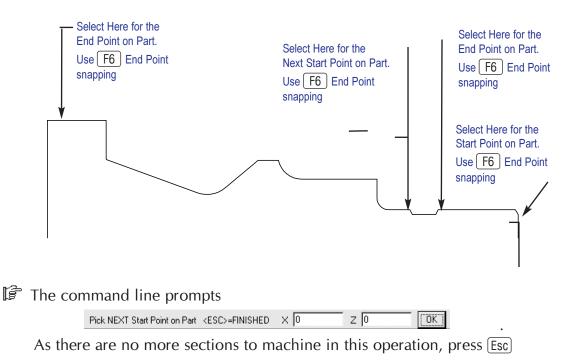
Pick NEXT Start Point on Part <ESC>=FINISHED X 0 Z 0

Using \checkmark [F6], click \bigcirc on the back top corner of the groove.

F The command line prompts for the End point on the part geometry.

Using \angle [F6], click \bigcirc on the back end of OD.

The selected geometry turns blue.





		FINISHING			×	
		Tool EBLD 04 15 93	3 32 55 CT 5	525 P		
			Stock	Amount 0		
		⊂ Move Between Se ⊂ Rapid		Feed		
_		OK.		Cancel		
🕼 Set the Stocl					ОК	
	FINISHING -	Tool EBLD 04 15 9	93 32 55 (CT525 P		X
	. .				Op No. 6	
	Tooling	Tool Number 4		Offset N	lumber 4	-
	SURF	ACE Speed: S 400		Feed per RE\	/ (X): F 0.032	_
	Feed pe	REV (X/Z): F 0.032		Feed per RE\	/ (Z): F 0.032	
	Coolant					
	C None	C Mist	Floo	d CTh	rough Tool	
	NC Code					
	C Linear	C Canne	ed Cycle			
		OK.		Cancel		

 $\mathbf{\hat{F}}$ Set the options as shown, then click on \mathbf{OK}

Auto-Lead In/Out	x
Lead-In	Lead-Out
C Perpendicular	Perpendicular
Tangential	C Tangential
C None	C None
☑ Use G41/42	Lead Length 2
(OK)	Cancel

🕼 Set the options as shown, then click on 🛛 OK



AlphaCAM detects that the tool will foul the part or the pocket and displays a dialog box.

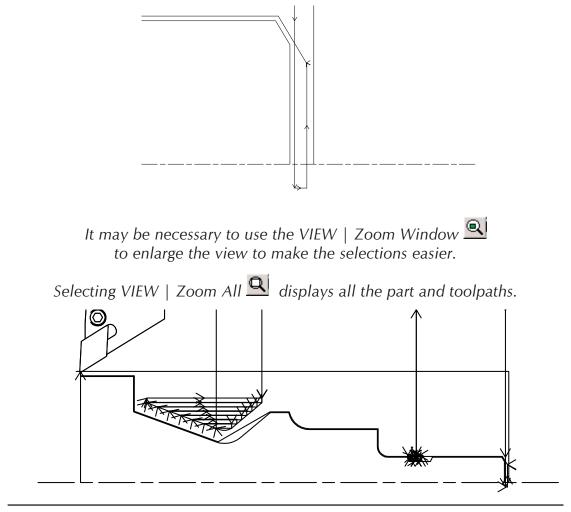


Select the

Continue - Avoiding Fouling

option.

The command line prompts you to indicate the path of the rapid motion from the tool change position to the start of the cut. The dash line on screen shows the rapid motion set the rapid motions path required.





Operation 7 Finish Profile Back Turning.

Select MACHINE | Select Tool . AlphaCAM displays the tool library, listing all the folders and tools currently defined in the Ttools library folder.

The Finishing folder should be active. If not select the up one level button and then select the **Finishing** folder and then click \bigcirc on **Open**. The tools located in the Finishing folder will be displayed.

- Select **EDRV 04 16 93 52 35 CT525 P** tool and then click ⁽¹⁾ on <u>**Open**</u>. The tool is selected from the library and displayed on your screen, attached to the pointer.
- Press Enter + to accept and select the tool

EDRV 04 16 93 52 35	CT525 P Rotation: 🗙			
Tool Number: T 5				
Maximum Spindle Speed: S 4000				
Speed	Feed			
Surface	Per REV			
C Spindle	C per MIN			
Turret	Station			
Above C/L	 Front 			
C Below C/L	C Back			
(OK)	Cancel			

🕼 Set the options as shown, then click on 🗌 OK



AlphaCAM recognises that the tool is not at the tool change and a dialog box asks if it is necessary to **Go To Tool Change**. Select Yes

Set the rapid motion to get the tool to the tool change position.



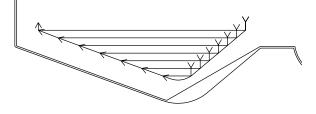
For Select MACHINE | Conventional turning | Finishing we command.

F The command line prompts for the Start point on the part geometry.

Using 156, click on the left side of the radius in the bottom of the recess.

F The command line prompts for the End point on the part geometry.

Using \swarrow [F6], click [] on the right top edged of the recess.



The selected geometry turns blue.

F The command line prompts

Pick NEXT Start Point on Part $\langle ESC \rangle = FINISHED \times 0 $	z jo	OK
---	------	----

As there are no more sections to machine in this operation, press Esc

FINISHING	×
Tool EDRV 04 16 93 52 3	35 CT525 P
	Stock Amount 0
Move Between Section	8
C Rapid	🕫 Feed
(OK)	Cancel

🕼 Set the Stock Amount as shown, then click on 🛛 🛛 🕅

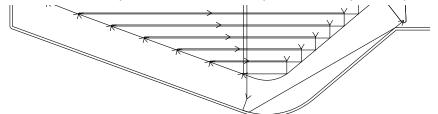


	FINISHING - Tool EDRV 04 16 93 52	2 35 CT525 P ×
		Op No. 7
	Tooling	
	Tool Number 5	Offset Number 5
	SURFACE Speed: S 400	Feed per REV (X): F 0.032
	Feed per REV (X/Z): F 0.032	Feed per REV (Z): F 0.032
	Coolant	
	C None C Mist 🧿	Flood C Through Tool
	NC Code	
	← Linear ← C Canned Cyc	le
		Cancel
Set the option	s as shown, then clic	k on OK
	Auto-Lead In/Out	×
	Lead-In	Lead-Out
	Perpendicular	C Perpendicular
	C Tangential	Tangential
	C None	
	🔽 Use G41/42	Lead Length 2
	(OK)	Cancel

 $\mathbf{\hat{f}}$ Set the options as shown, then click on \mathbf{OK}

Ê

The command line prompts you to indicate the path of the rapid motion from the tool change position to the start of the cut. The dash line on screen shows the rapid motion set the rapid motions path required.



The screen at this stage is covered with toolpaths. To make it easier to define the next operations, the current toolpaths can be turned off

Select MACHINE | Edit Operations 🖾 . A dialog box showing the current operations is displayed.

if Select \blacksquare to hide all the operations, then press Esc or click \square on \boxtimes

The current toolpaths are no longer displayed.



Operation 8 Threading

Select MACHINE | Select Tool . AlphaCAM displays the tool library, listing all the folders and tools currently defined in the Ttools library folder.

The Finishing folder should be active. Select the up one level button and then select the **Threading** folder and then click \bigcirc on **Open**. The tools located in the Threading folder will be displayed.

- Select **EDLM 012 2.0 60 60 GC1020 PK** tool and then click T on **Open**. The tool is selected from the library and displayed on your screen, attached to the pointer.
- Press Enter to accept and select the tool

EDLM 012 2.0 60 60 60 GC1020 PK Rotatio ×				
Tool Number: T 6 Maximum Spindle Speed: S 4000				
Speed C Surface C Spindle	Feed per REV C per MIN			
Turret	Station Front Back			
(K)	Cancel			

Set the options as shown, then click on OK

AlphaCA	4		×
?	Goto Tool C	hange Pos	ition ?
	Yes	No	

- AlphaCAM recognises that the tool is not at the tool change and a dialog box asks if it is necessary to **Go To Tool Change**. Select Yes
- Set the rapid motion to get the tool to the tool change position.



F	Select MACHINE Conventional turning Threa	ading 🚽 command.			
ţ,	The command line prompts for the Start point on Using \checkmark [F6], click [] on the top of the front fac				
ţ,	The command line prompts for the End point on the part geometry. Using \swarrow [F7], click \bigcirc on the line at the bottom of the groove.				
	Select Here for the End Point on Part. Use F7 Mid Point snapping	Select Here for the Start Point on Part. Use F6 End Point snapping			
The	selected geometry turns blue.	I			

THREADING - Tool EDLM 012 2.0 60 60 60 GC1020 PK × Op No. 8 Tooling Tool Number 6 Offset Number 6 Feed per REV (X): F SPINDLE Speed: S 600 Feed per REV (X/Z): F Feed per REV (Z): F Coolant C None Flood C Through Tool C Mist NC Code C Linear C Canned Cycle ÖK Cancel

- Set the options as shown, then click on OK
- Dialog box 2 Lists all the pre-defined thread definitions. Type ***24*** to display only the files with '24' in their name.
- Select the **ISO FINE 24 DIA**.

Iso fine 24 dia		x
	Overall Diameter 24	_
	Angle 60	
	Depth (Radius value) 1.226	_
Specify by		
Pitch	C TPI	
Pitch 2	TPI 0	_
OK	Cancel	

A dialog box displays the pre-defined thread parameters. Select OK



The threading dialog box is displayed. This dialog box has two pages: one is to configure the thread, the other is the General page. To display the different pages, select the tabs at the top of each page.

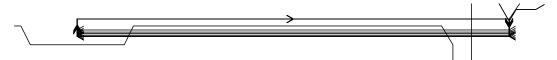
I On the Configure page, select the following options, if not already set.

hreading - Iso fine 24	dia 🕹		×
General Configure			
Hand			
Right	C Left		
Cut Type			
Plunge	C Flank	C Zig-Zaj	
In-Feed			
Variable Depth	С	Constant Depth	
Direction			
As Longest Line	С	Follow Profile	
		OK	Cancel

I On the General page, set the following options, if not already set.

Threading - Iso fine 24 dia	×		
General Configure			
No. of Starts 1 Acceleration Distance 6 Number of Passes 5			
Number of Spring Passes 2			
Lead In	Lead Out		
 Horizontal 	C Horizontal		
C Vertical	Vertical		
C Thread Angle	C Thread Angle		
·			
	OK Cancel		

- \overrightarrow{I} When all settings are correct select OK.
- The command line prompts you to indicate the path of the rapid motion from the tool change position to the start of the cut. The dash line on screen shows the rapid motion set the rapid motions path required.



This is the last operation, so it is necessary to return the tool to the tool change position. Select MACHINE | Go To Tool Change Pos And set

the rapid motion the tool should take back to the tool change position.

As the job is now complete, we can check the toolpaths using the display options to check for collisions and incorrect rapid motion.



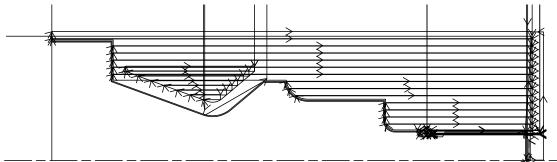
Firstly ensure that all the toolpaths are displayed.

Select MACHINE | Edit Operations 🖾 . A dialog box is displayed showing the current operations.

Select I in the dialog box to show all operations,

then press \fbox{Esc} or click $\fbox{}$ on $\boxtimes~$ All toolpaths are displayed.

Use the zoom commands to set the display so that the component, toolpaths and rapid motions can be seen clearly.

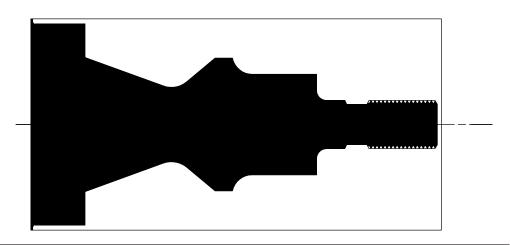


If any of the rapid motion paths are not correct, they may be altered using the

MACHINE | [Edit Machining] | Adjust Rapids 🗾 command.

Select the following buttons from the display button bar. D Show Tools

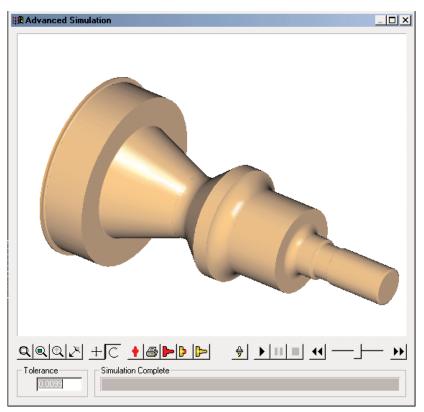
Select the redraw button D. This causes the toolpath animation to start showing a representation of the material being cut. The toolpath animation can be sped up and slowed down using the PgUp and PgDn keys.





A 3D solid model can be displayed if required.

Select VIEW | 3D Simulation Select VIEW | 3D Simulation Select view of the select view



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 it to run out of virtual memory and hang the system. E.G. This part is 100mm Diameter x 200mm Long setting the tolerance smaller than 0.1 would cause the system to run very slowly.

AlphaCAM will display the toolpaths removing material from a 'solid' billet. This will give you extra confidence that the toolpaths you have created are correct.

When you are ready to conclude the machining, close the 3D simulation by clicking on the \boxtimes 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.
- Enter a suitable filename for this job.

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

Output NC		×
Current Post Pro	cessor	
Alpha Standard	Lathe	•
Output to		
🕶 File	C Machine	C Both
OK		ancel

- Set the options as shown. When complete, click on OK and the Save As dialog box is displayed. The options are set the same as the job details. It is possible to save the NC-Code in the same folder, using the same name as the job, because AlphaCAM uses different extensions for drawings (.atd) and NC files (.anc).
- Either press Enter or click on OK to 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