LICOM AlphaCAM

CAD/CAM system for Windows™

SIMPLE LATHE

TUTORIAL

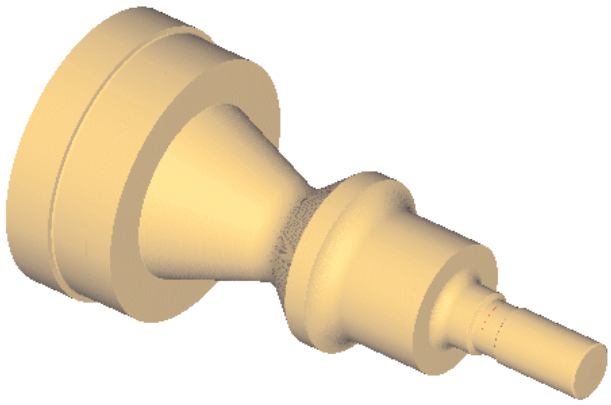


Table of Contents

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

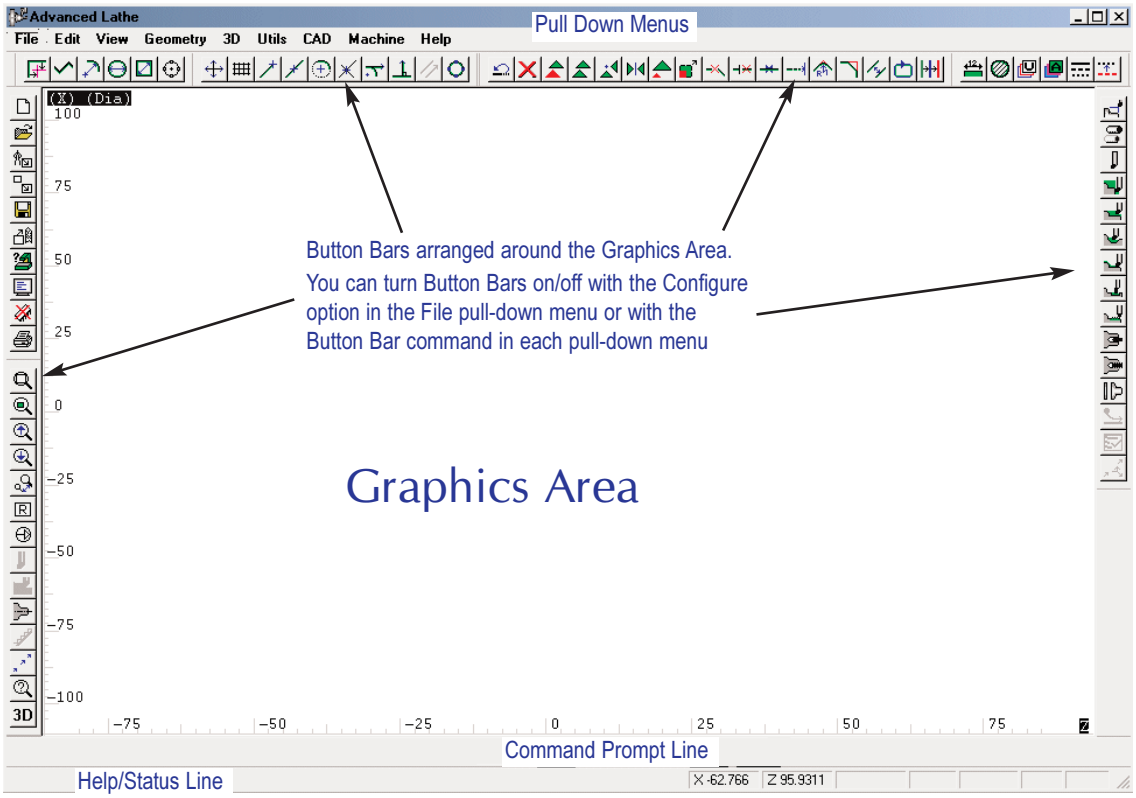
The HELP file is very comprehensive. When you have finished the tutorial, please take time to have a look at it. **Select HELP** | **Contents**.

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, 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.



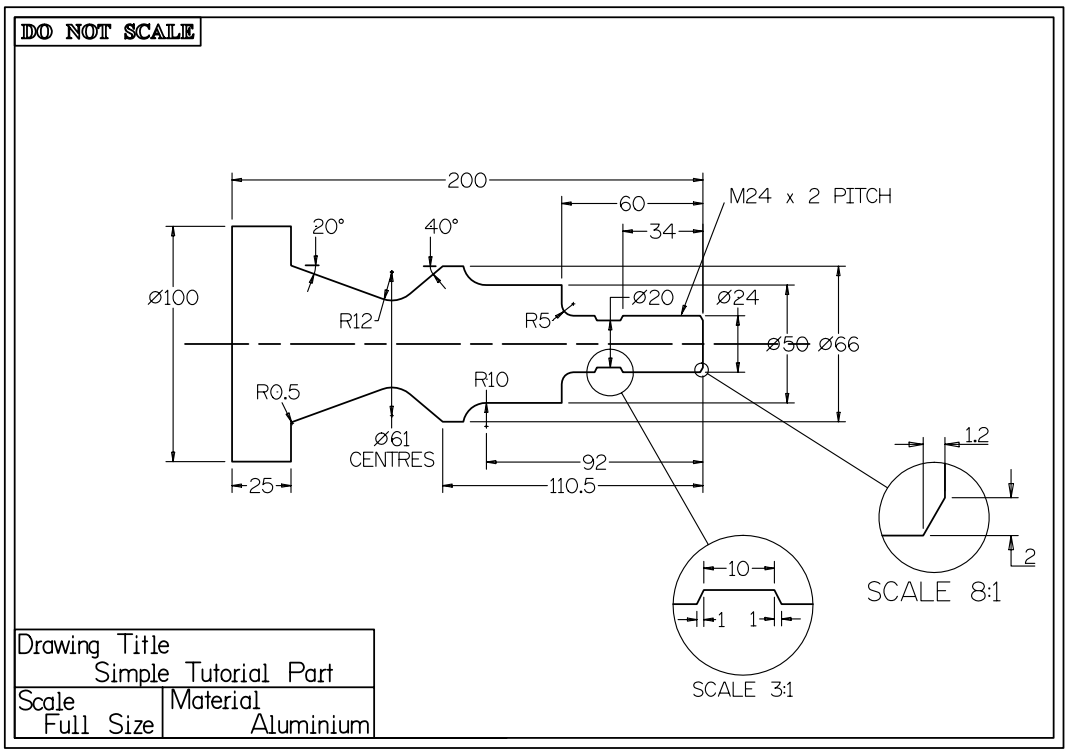
During this tutorial, we tell you where to find commands on the pull-down menus. If there is a button for the command, this will also be shown.

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.

☞ Select **GEOMETRY** | **APS Fast Geometry** | **Point**. 

☞ 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 🖱 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.

☞ Select **GEOMETRY** | **APS Fast Geometry** | **Line to Line Chamfer**. 

☞ The command line prompts for the first chamfer distance.
Type **2** ↵

☞ The command line prompts for the second chamfer distance.
Type **1.2** ↵

☞ The command line prompts for the co-ordinates of the point where the lines that are to be chamfered intersect.Type **24** ↵ ↵

Pressing ↵ 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.

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

☞ Select **GEOMETRY** | **APS Fast Geometry** | **Point**. 

☞ The command line prompts for the co-ordinates of the point.
Type ↵ **–34** ↵

Pressing ↵ 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.

☞ 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.

☞ 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.

☞ 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.

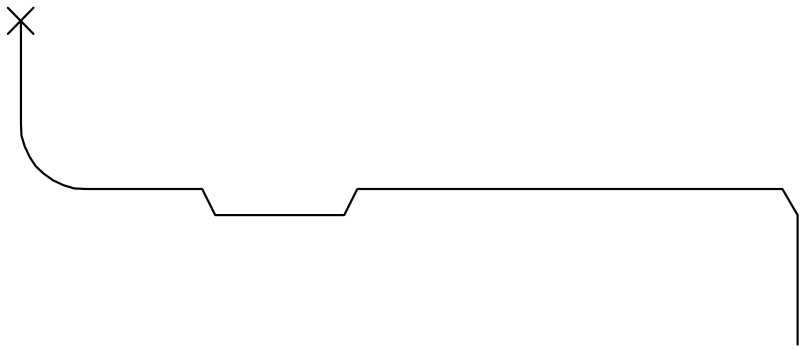
A phantom line is now drawn to this position on the screen, as the radius can not be drawn until the next turn is entered.

☞ Select **GEOMETRY** | **APS Fast Geometry** | **Point.** 

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

*Pressing ↵ 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.



☞ Select **GEOMETRY** | **APS Fast Geometry** | **Known Arc**.

☞ The command line prompts for the radius value of the Arc.
Type **10** ↵.

The first dialog box is displayed

☞ 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**.

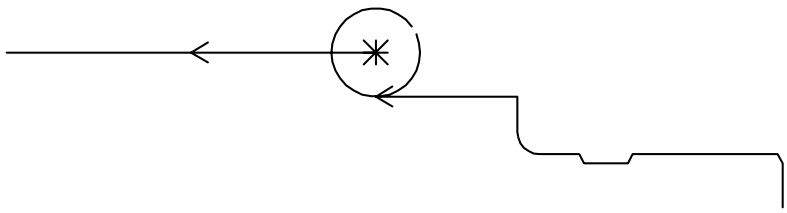
☞ The command line prompts for the radius value of the Blend.
Type **0** ↵.

The first dialog box is displayed

☞ 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 on OK
A second dialog box is displayed

☞ This prompts for the direction of the Line.
Select the 180 direction, then click 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.



☞ Select **GEOMETRY** | **APS Fast Geometry** | **Point.**

☞ 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.

☞ 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

☞ 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.

☞ Select **GEOMETRY** | **APS Fast Geometry** | **Line to Line Blend**. 

☞ 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 **into** 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.

☞ 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** ↵.

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** ↵ ↵.

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.

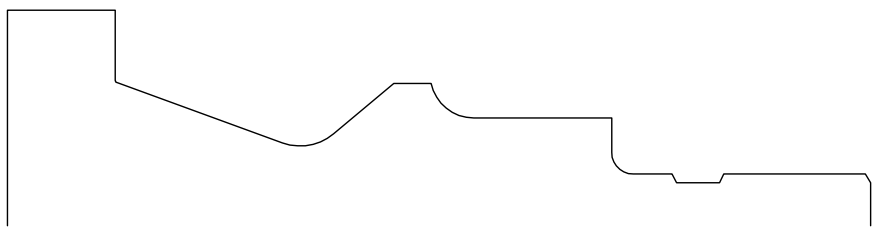
It is only necessary to draw half the component.

☞ Select **GEOMETRY** | **APS Fast Geometry** | **Finish.**

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

☞ 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

☞ 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** ↵.

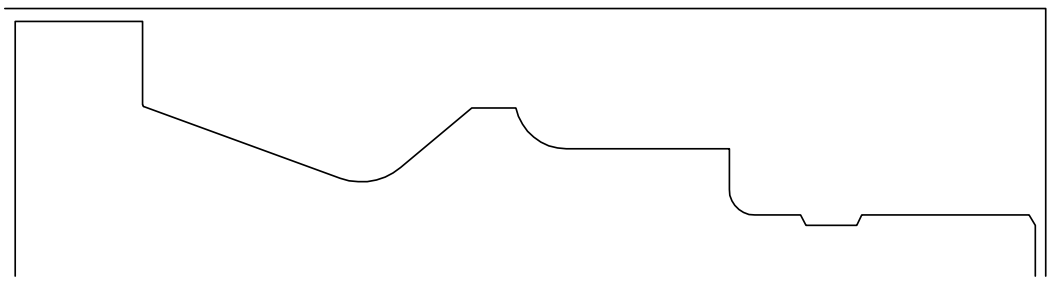
It is only necessary to draw half the billet.

☞ 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.

☞ Select **VIEW** | **Display Options** | **Ghost Tool** Ctrl + G

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 Finish (ESC) 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.

☞ Select **MACHINE** | **Select Material** 

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 [OK]

☞ Select **MACHINE** | **Set Tool Change Pos** 

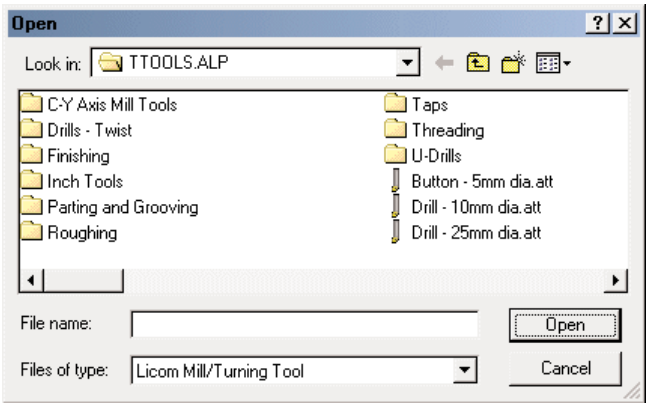
This is the position to which the tool is sent in order to index the turret. This position can be set once for the program or set for each tool prior to tool selection.

☞ 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 [Tab] 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**.



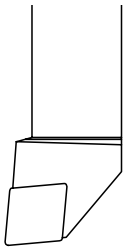
When selecting tools it will be necessary to return to the top level of Ttools, using the up one level button, then select the required folder in order to select the tool specified for the operation.

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.

☞ 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.

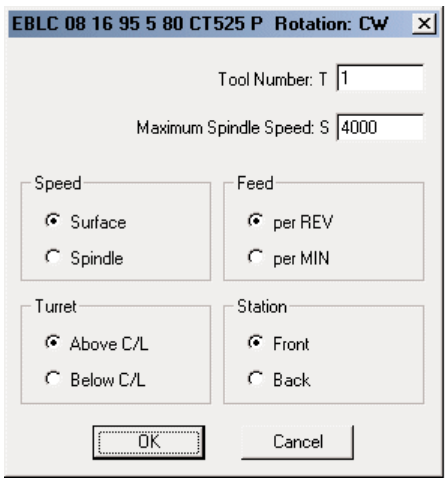


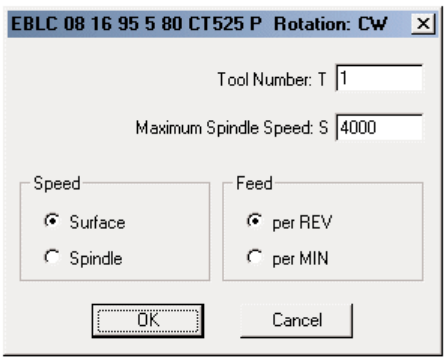
The command line prompts you to press Enter to accept and select the tool or Esc to abort the tool selection and return to the tool list.

☞ 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.





☞ Set the options as shown then select OK.

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

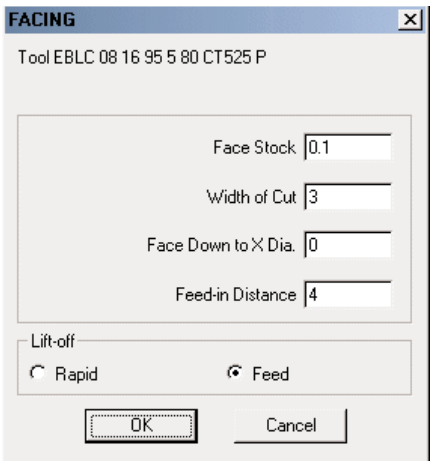
Operation 1 Facing

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

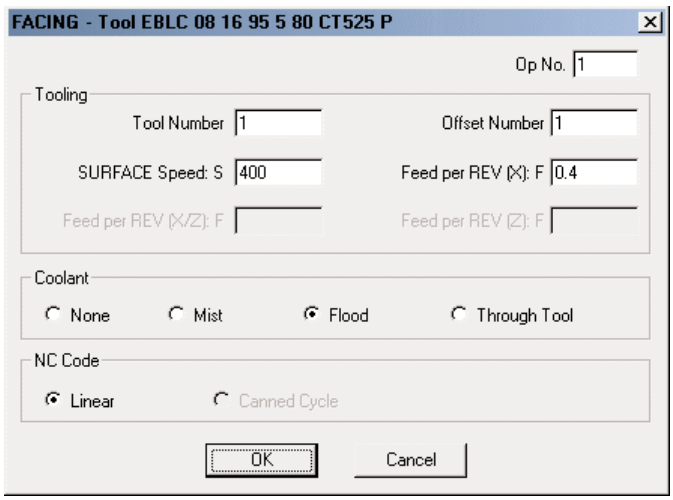
☞ Select **MACHINE** | **Conventional Turning** | **Facing**

☞ 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.

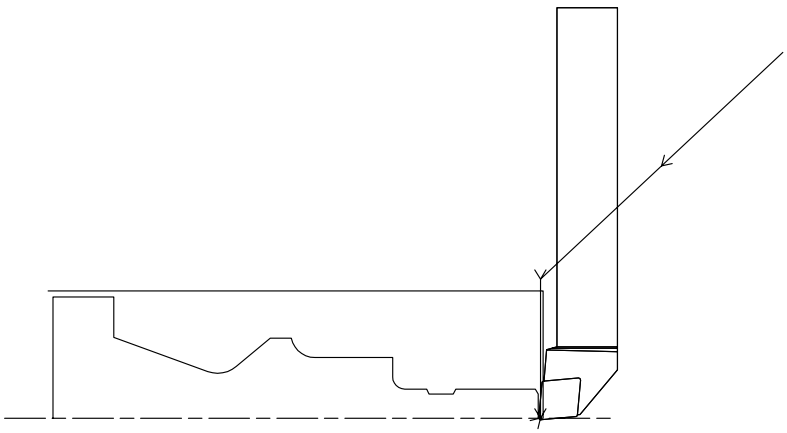


☞ Set the options as shown, then click on OK

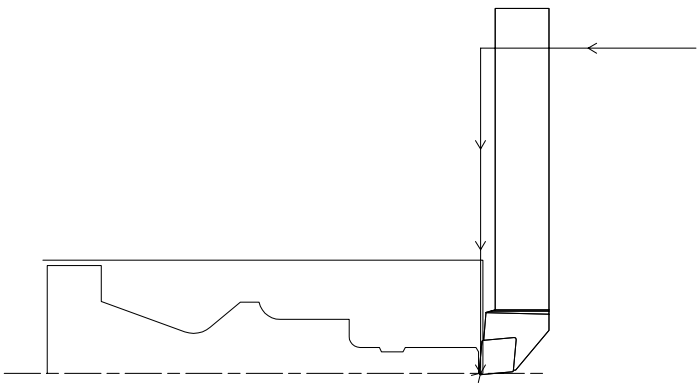


☞ Set the options as shown, then click on OK

☞ 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.
Clicking the 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** R Ctrl + R or **Zoom All** is used.

In order for the rapids to be displayed, the draw rapids option must be set active.

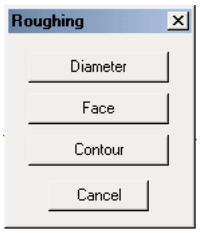
☞ Select **VIEW** | **Display Options** | **Draw Rapids**

followed by **VIEW** | **Redraw** R Ctrl + R

Operation 2 Rough Turn the Diameter

☞ Select **MACHINE** | **Conventional Turning** | **Roughing**

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



☞ Select **Diameter**

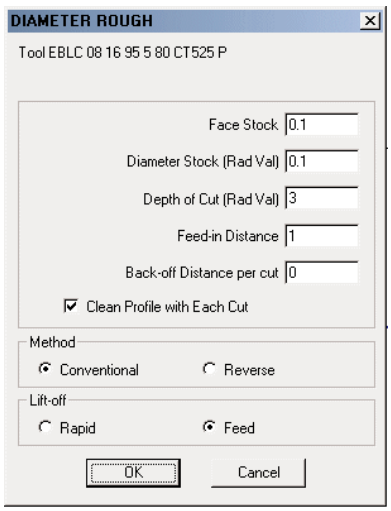
☞ The command line prompts you to select the billet geometry.
Click on the billet outline.

☞ The command line prompts for the Start point on the part geometry.
Using F6, click on the top of the front face.

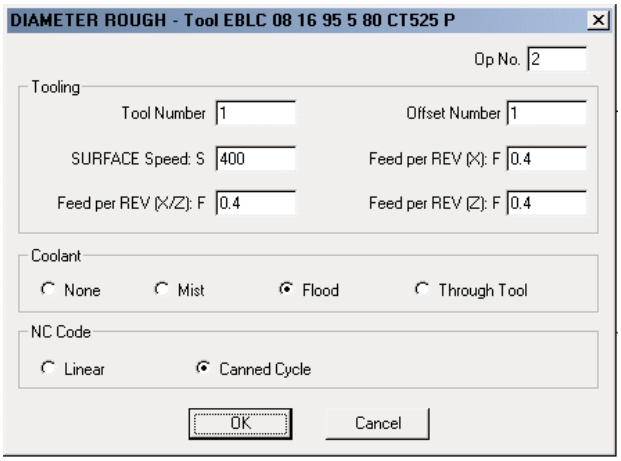
☞ The command line prompts for the End point on the geometry.
Using F6, click on the left end of the OD at the back.

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.



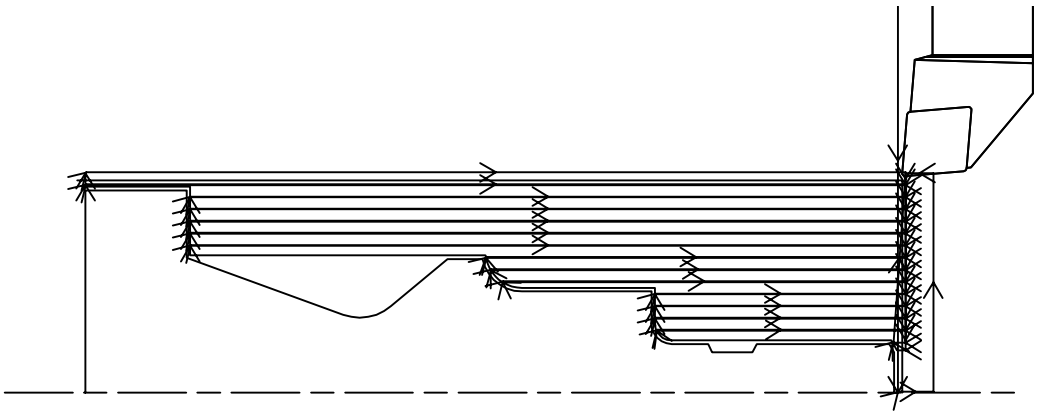
☞ Set the options as shown, then click on OK



☞ 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.

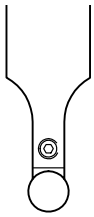
☞ Select to hide all the operations, then press Esc or click 🖱 on ☒

The current toolpaths are no longer displayed.

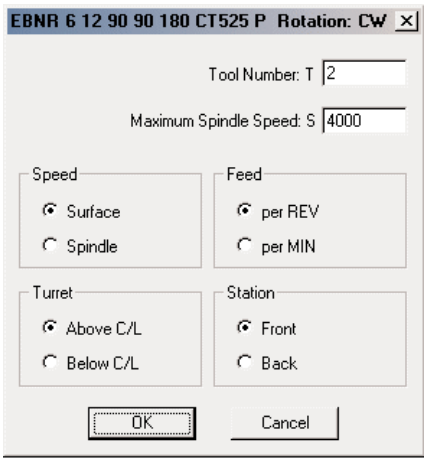
☞ Select **MACHINE** | **Select Tool** . AlphaCAM displays the tool library, listing all the folders and tools currently defined in the Ttools library folder.

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

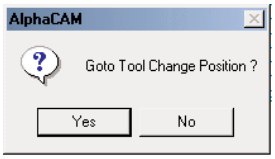
☞ Select **EBNR 6 12 90 90 180 CT525 P** tool and then click on **Open**. The tool is selected from the library and displayed on your screen, as shown below, attached to the cursor.



☞ Press Enter to accept and select the tool



☞ 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

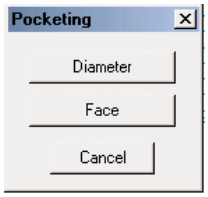
☞ Set the rapid motion to get the tool to the tool change position.

☞ Select **MACHINE** | **Conventional Turning** | **Pocketing** .

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.



☞ Select **Diameter**

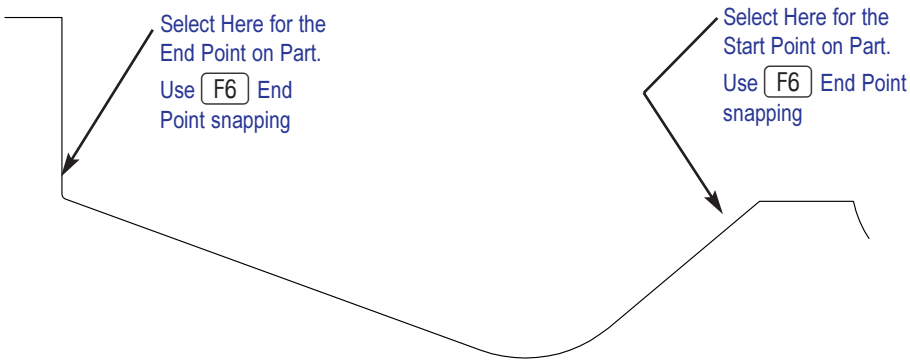
A warning is displayed when the tool clearance angle is not set. If a warning is issued, select OK .

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

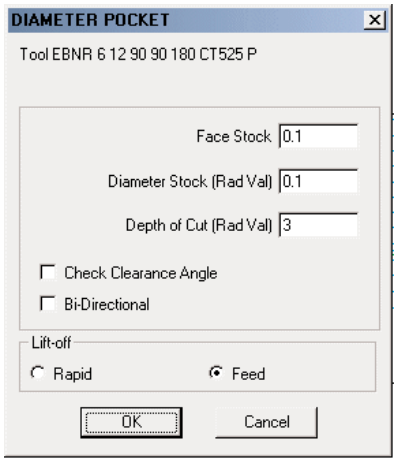
Using F6, click on the top front corner of the recess.

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

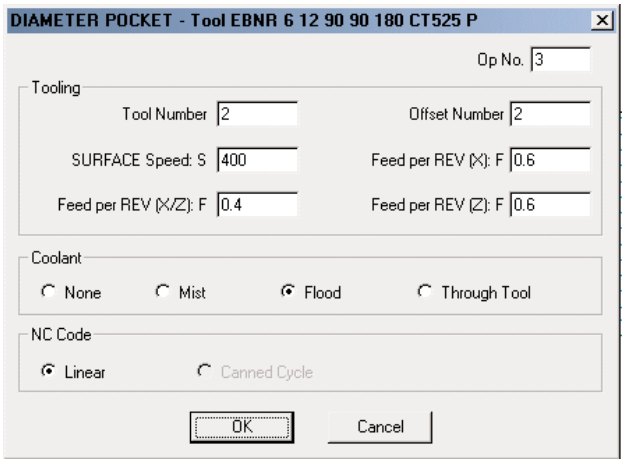
Using F6, click on the bottom end of the face at the back of the recess.



The selected geometry turns blue.



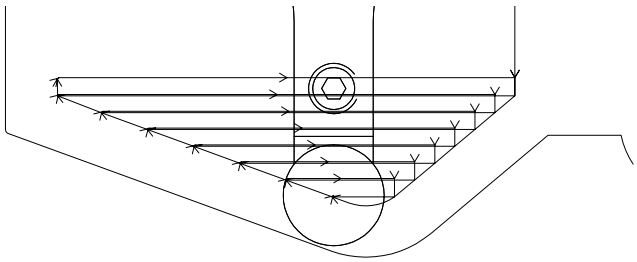
☞ Set the options as shown, then click on OK



☞ Set the options as shown, then click on OK

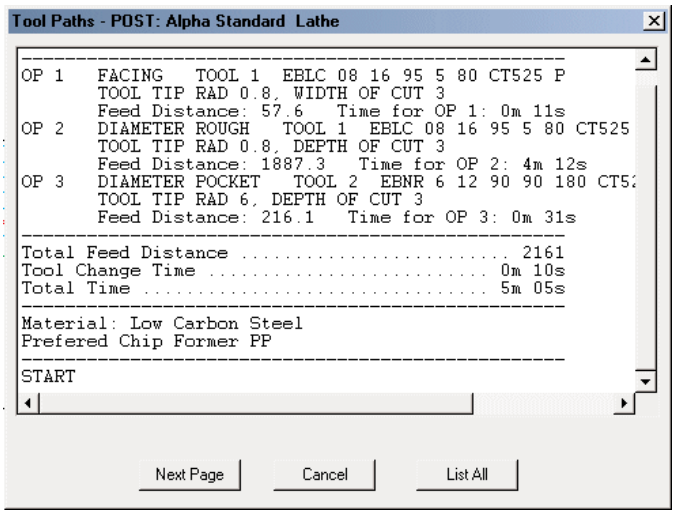
The operation number is automatically increased when the tool is changed.

☞ 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.

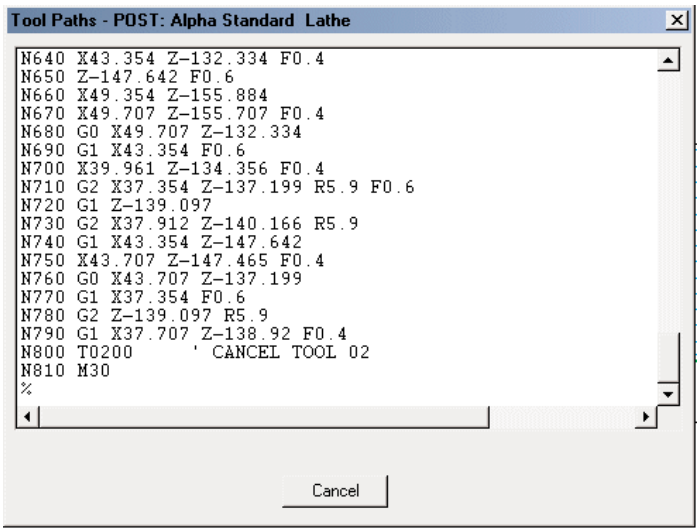


You can view the CNC program at any time. Select **FILE** | **List NC Code** .

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.



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



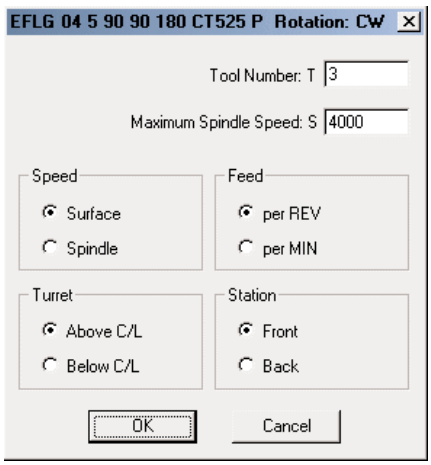
You will see remarks in the code - e.g. '(OP1 FACING ...) and 'SELECT TOOL 01. These are for your information, they are not sent to the machine tool controller.

Operation 4 Grooving

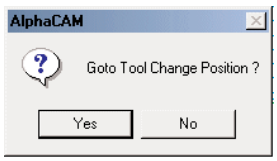
☞ Select **MACHINE** | **Select Tool** . AlphaCAM displays the tool library, listing all the folders and tools currently defined in the Ttools library folder.

The Roughing folder should be active. Select the up one level button and then select the **Parting and Grooving** folder and then click on **Open**. 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 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



☞ Set the options as shown, then click on OK

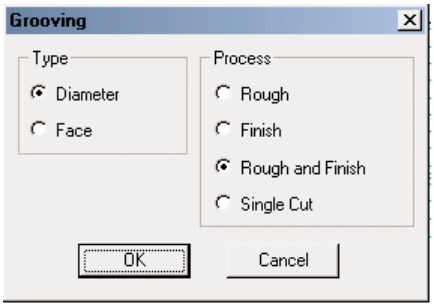
☞ AlphaCAM recognises that the tool is not at the tool change and a dialog box asks if it is necessary to **Go To Tool Change**. Select Yes

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

☞ 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.*



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

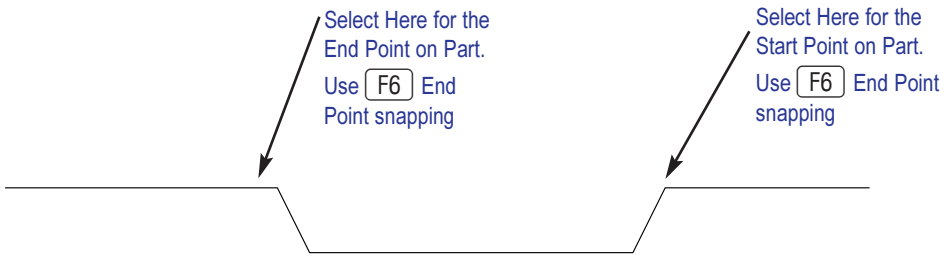
☞ Set the options as shown, then click on OK

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

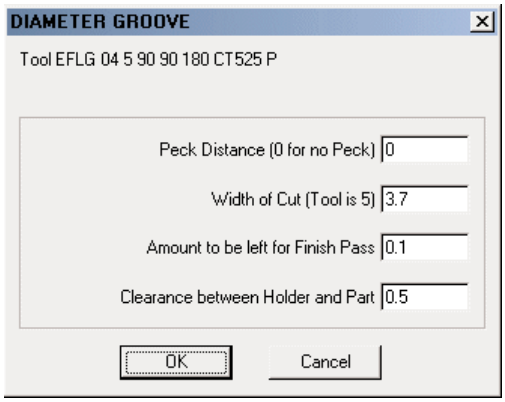
Using F6. Click on the top front corner of the groove.

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

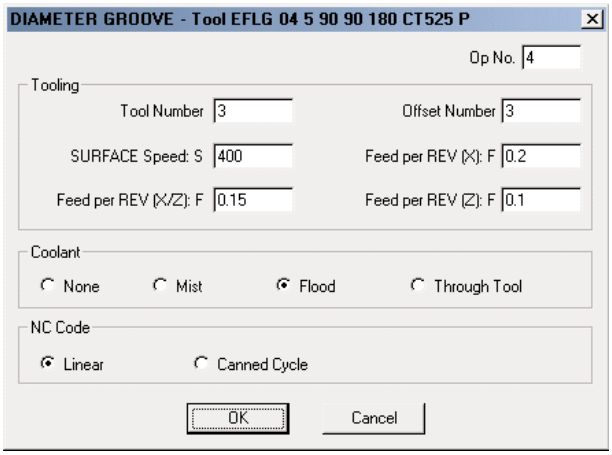
Using F6. Click on the top back corner of the groove.



The selected geometry turns blue.



☞ Set the options as shown, then click on OK

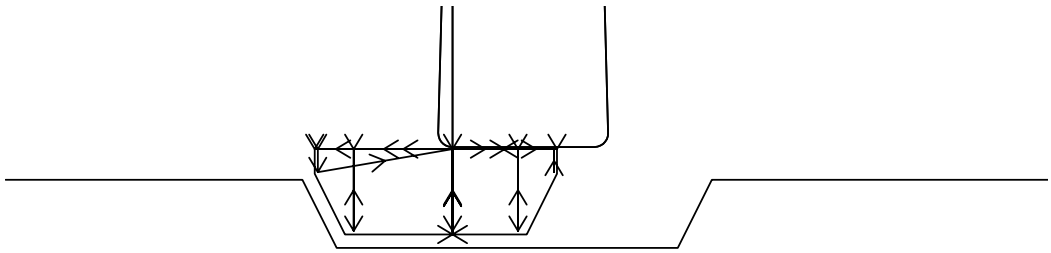


☞ Set the options as shown, then click on OK

☞ 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** ↵

☞ 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

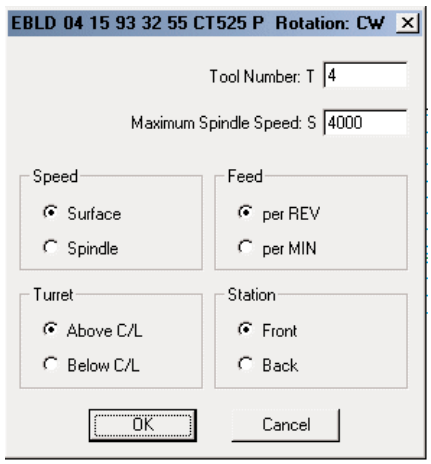
☞ Select **MACHINE** | **Select Tool** . AlphaCAM displays the tool library, listing all the folders and tools currently defined in the Ttools library folder.

The Parting and Grooving folder should be active.

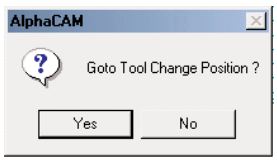
Select the up one level button and then select the **Finishing** folder and then click on **Open**. The tools located in the Finishing folder will be displayed.

☞ Select **EBLD 04 15 93 32 55 CT525 P** tool and then click 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



☞ 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.

☞ Select **MACHINE** | **Conventional turning** | **Finishing** command.

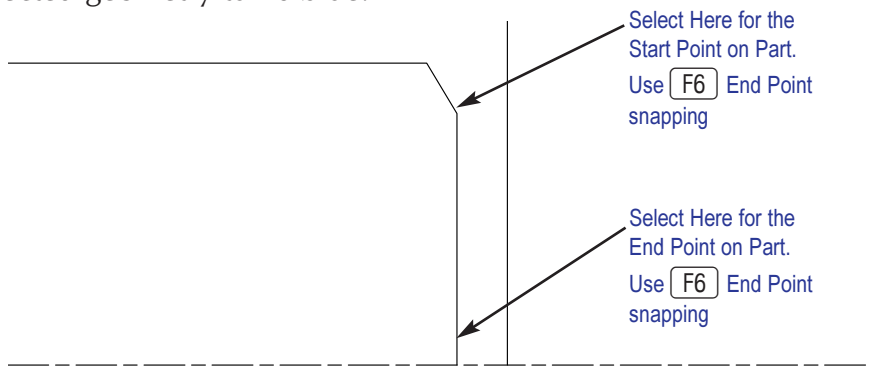
☞ The command line prompts for the Start point on the part geometry.

Using F6, click on the top of the front face.

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

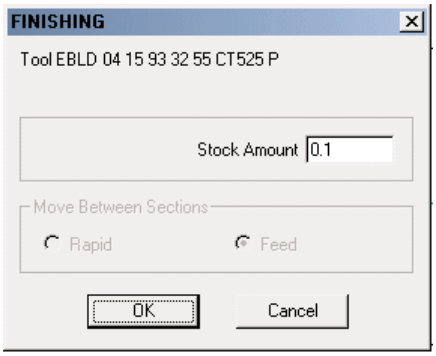
Using F6, click on the bottom of the front face.

The selected geometry turns blue.

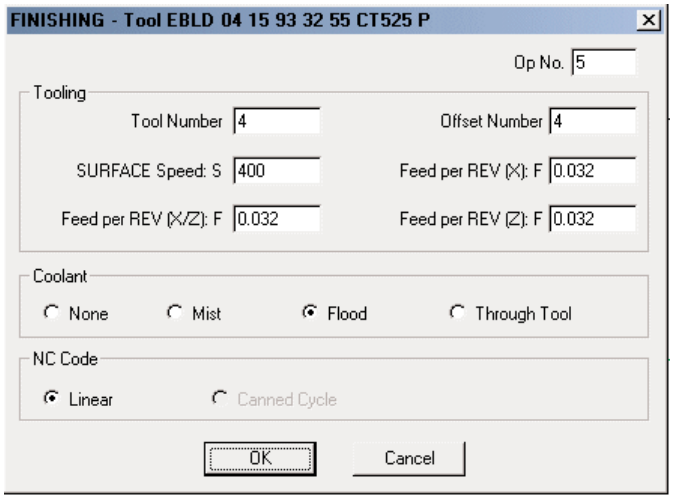


☞ The command line prompts

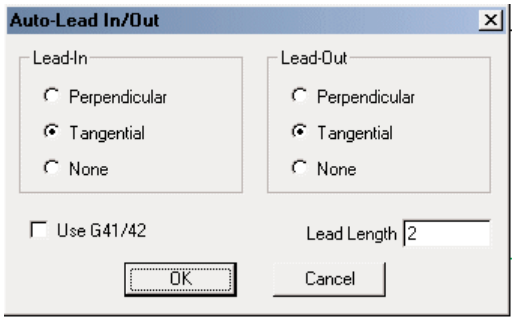
As there are no more sections to machine in this operation, press Esc



☞ Set the Stock Amount as shown, then click on OK

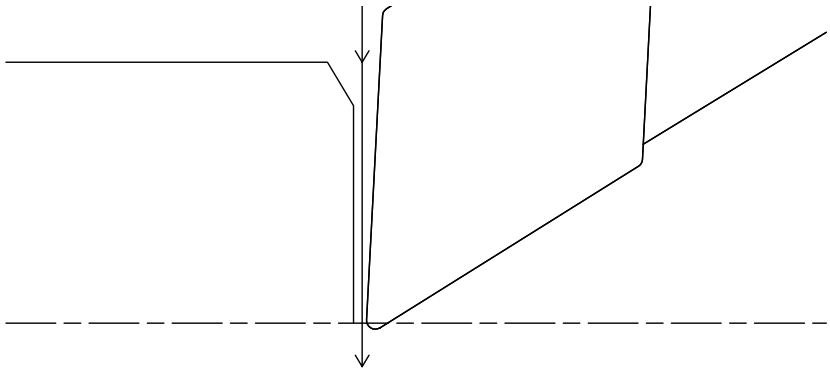


☞ Set the options as shown, then click on OK



☞ Set the options as shown, then click on 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.



Operation 6 Finish Profile

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

☞ Select **MACHINE** | **Conventional turning** | **Finishing** command.

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

Using F6, click on the top of the front face.

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

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

The selected geometry turns blue.

☞ The command line prompts

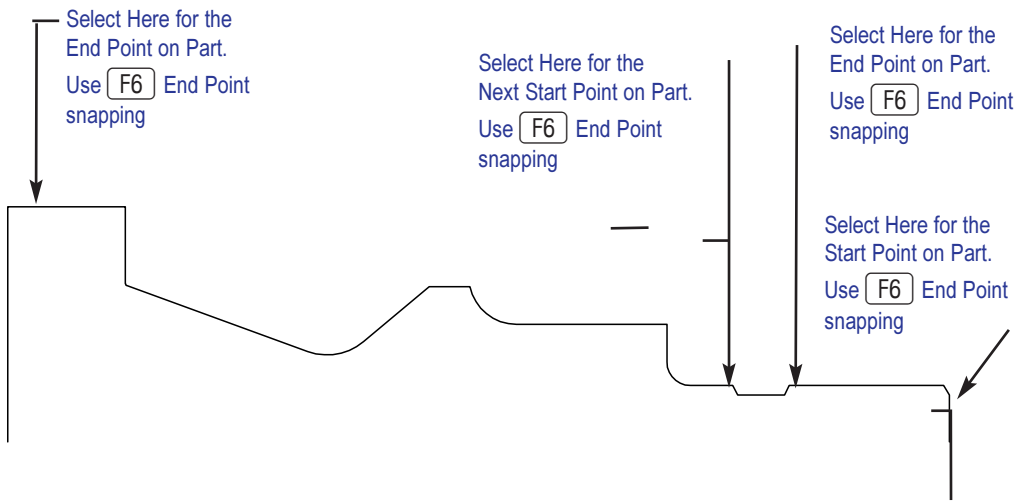
Pick NEXT Start Point on Part <ESC>=FINISHED X 0 Z 0 OK

Using F6, click on the back top corner of the groove.

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

Using F6, click on the back end of OD.

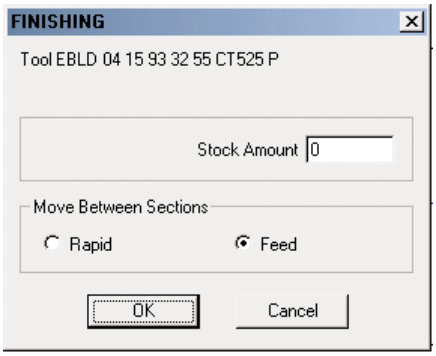
The selected geometry turns blue.



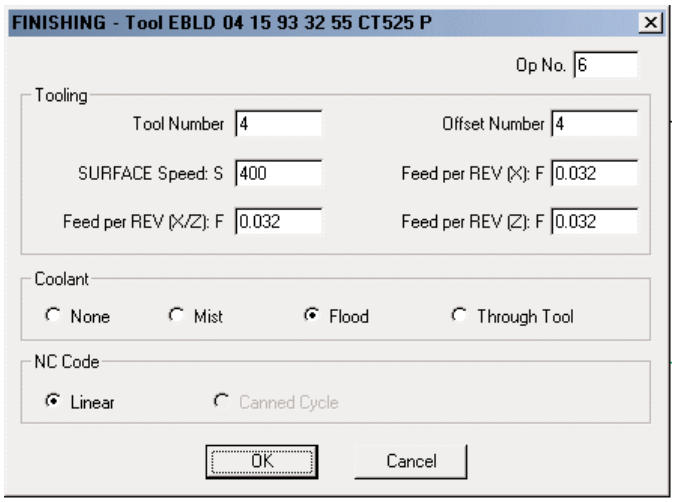
☞ The command line prompts

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

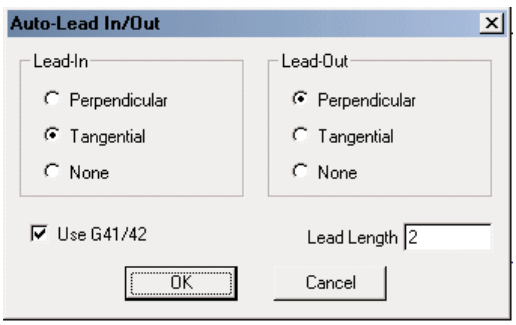
As there are no more sections to machine in this operation, press Esc



☞ Set the Stock Amount as shown, then click on OK

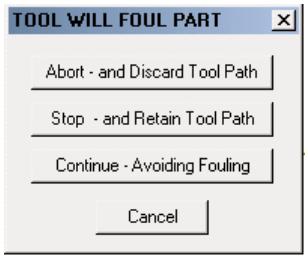


☞ Set the options as shown, then click on OK



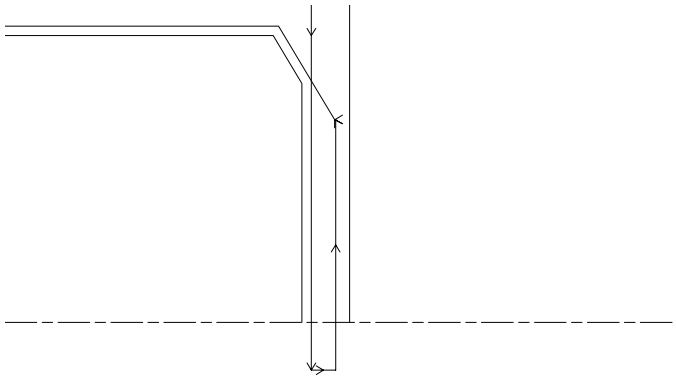
☞ Set the options as shown, then click on OK

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



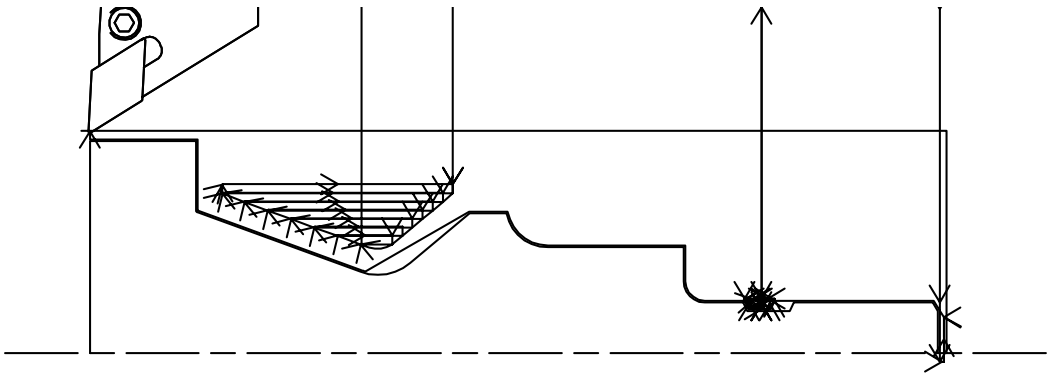
☞ Select the Continue - Avoiding Fouling option.

☞ 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.



It may be necessary to use the VIEW | Zoom Window to enlarge the view to make the selections easier.

Selecting VIEW | Zoom All displays all the part and toolpaths.



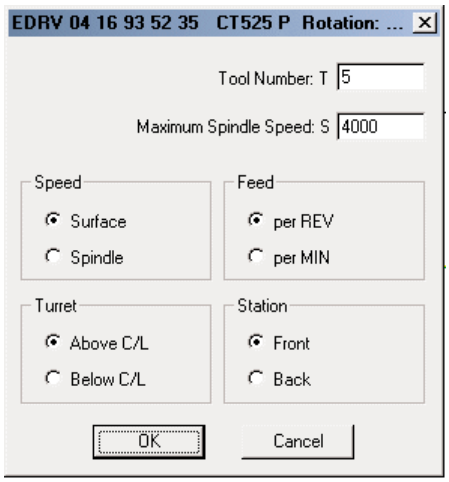
Operation 7 Finish Profile Back Turning.

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

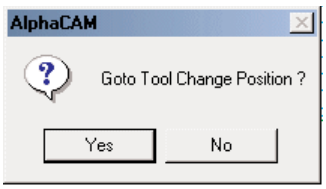
The Finishing folder should be active. If not select the up one level button and then select the **Finishing** folder and then click on **Open**. The tools located in the Finishing folder will be displayed.

☞ Select **EDRV 04 16 93 52 35 CT525 P** tool and then click 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



☞ 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.

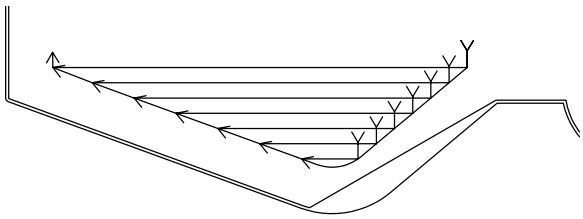
☞ Select **MACHINE** | **Conventional turning** | **Finishing** command.

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

Using F6, click on the left side of the radius in the bottom of the recess.

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

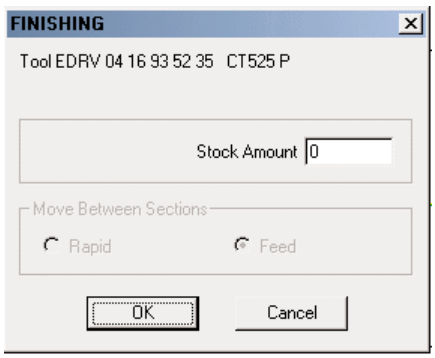
Using F6, click on the right top edged of the recess.



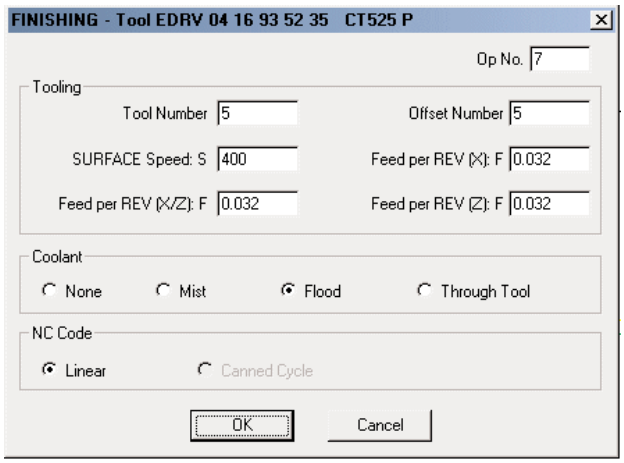
The selected geometry turns blue.

☞ The command line prompts

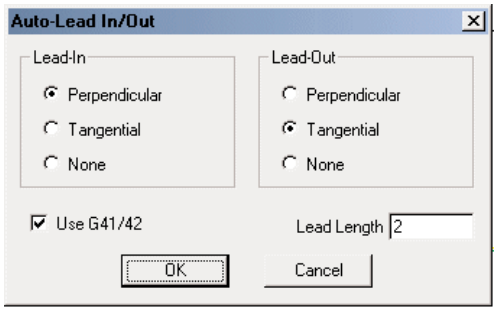
As there are no more sections to machine in this operation, press Esc



☞ Set the Stock Amount as shown, then click on OK

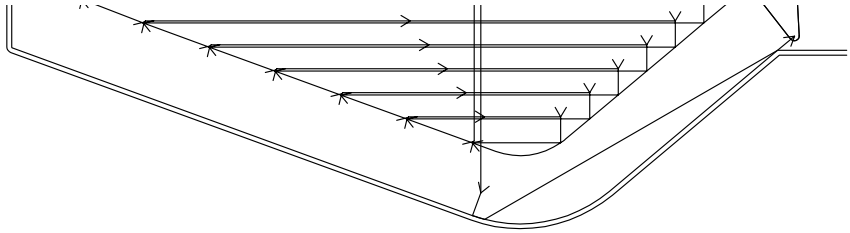


☞ Set the options as shown, then click on OK



☞ Set the options as shown, then click on 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.



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.

☞ Select to hide all the operations, then press Esc or click on ☒

The current toolpaths are no longer displayed.

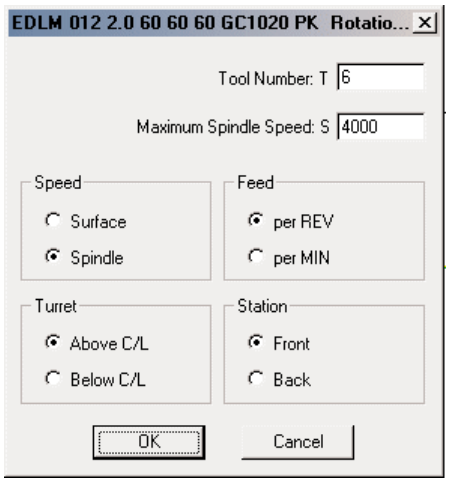
Operation 8 Threading

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

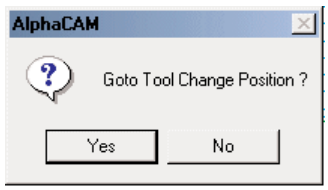
The Finishing folder should be active. Select the up one level button and then select the **Threading** folder and then click on **Open**. The tools located in the Threading folder will be displayed.

☞ Select **EDLM 012 2.0 60 60 60 GC1020 PK** tool and then click 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



☞ 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.

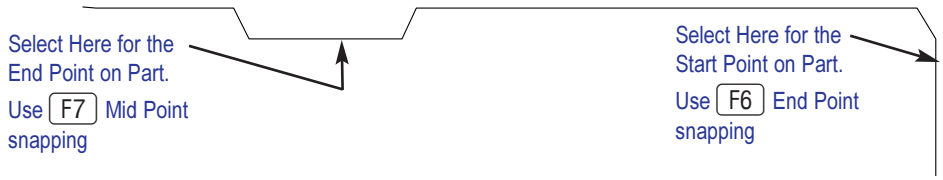
☞ Select **MACHINE** | **Conventional turning** | **Threading** command.

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

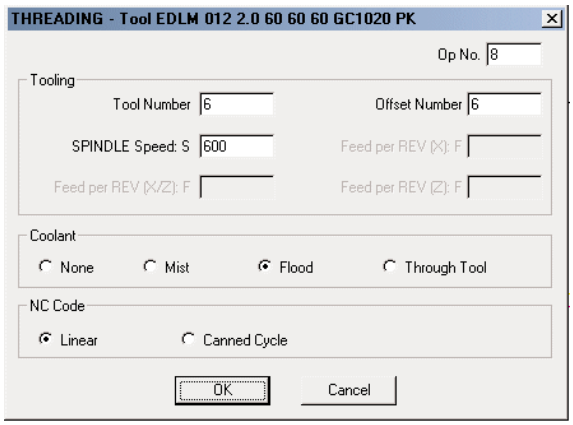
Using F6, click on the top of the front face.

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

Using F7, click on the line at the bottom of the groove.



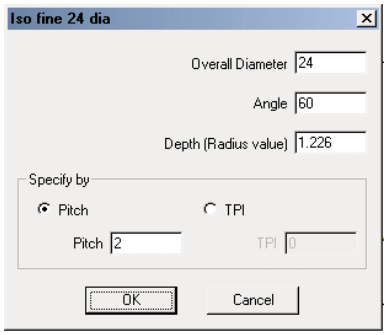
The selected geometry turns blue.



☞ 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.

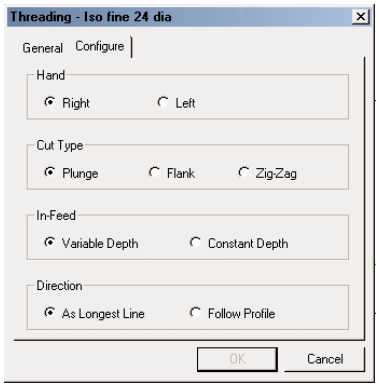
☞ Select the **ISO FINE 24 DIA**.



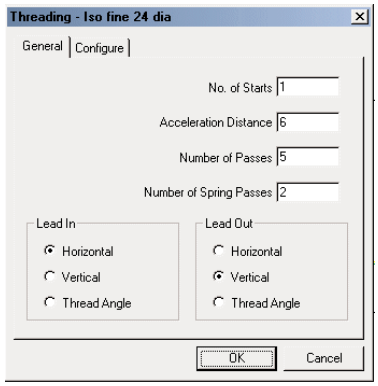
☞ A dialog box displays the pre-defined thread parameters. Select OK

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.

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

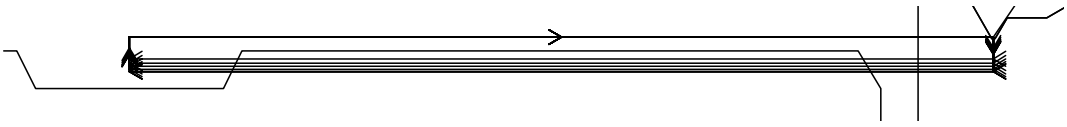


☞ On the General page, set the following options, if not already set.



☞ 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.

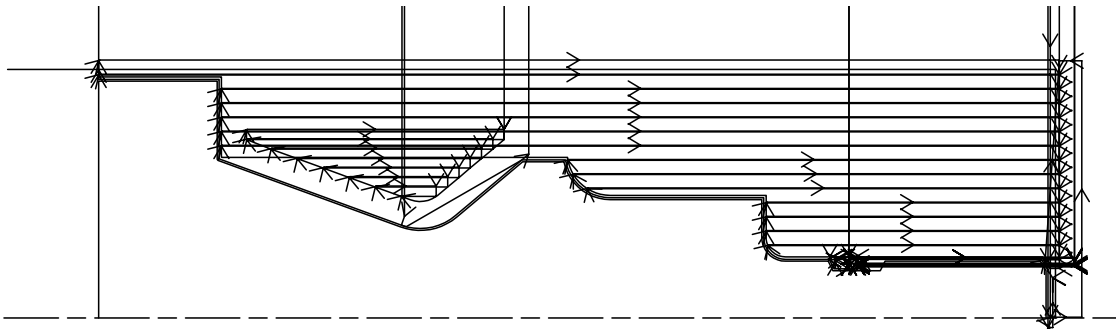
As the job is now complete, we can check the toolpaths using the display options to check for collisions and incorrect rapid motion.

Firstly ensure that all the toolpaths are displayed.

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

☞ Select in the dialog box to show all operations, then press Esc or click on ☒ All toolpaths are displayed.

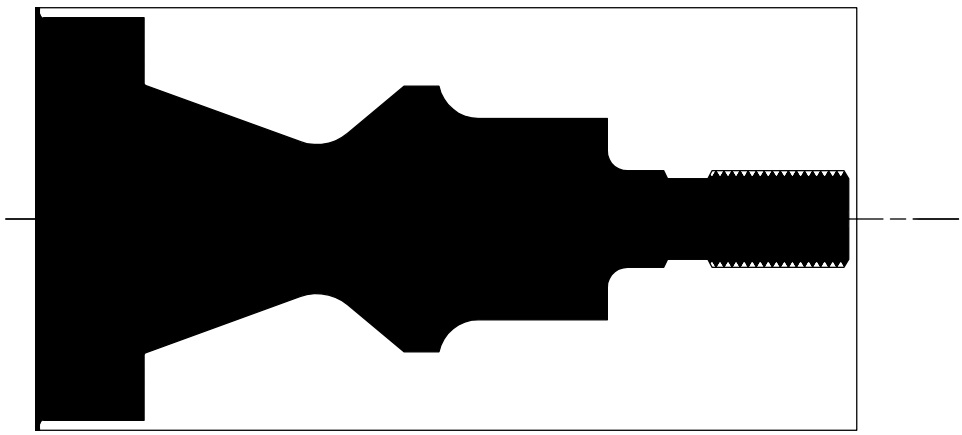
Use the zoom commands to set the display so that the component, toolpaths and rapid motions can be seen clearly.



If any of the rapid motion paths are not correct, they may be altered using the **MACHINE | *[Edit Machining]*| Adjust Rapids** command.

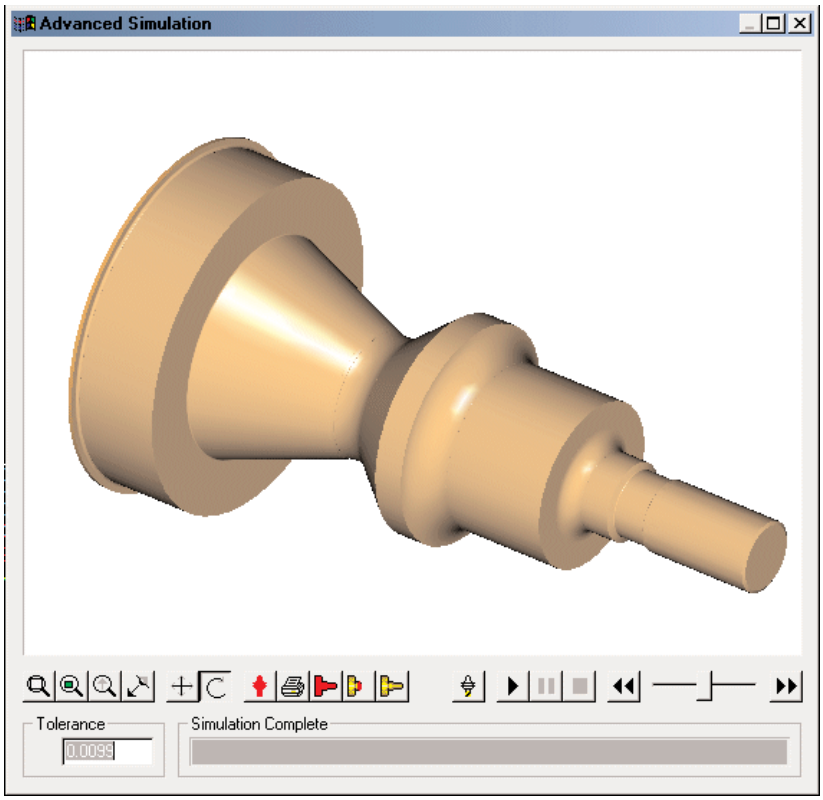
☞ Select the following buttons from the display button bar. Show Tools Show Material Cut Show Both Sides

☞ Select the redraw button . 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** to display tool motion removing material from a 'solid billet'



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.

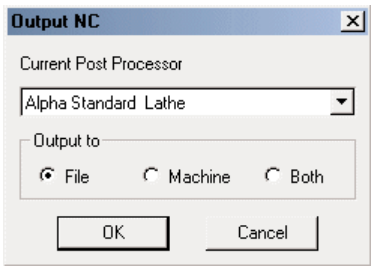
AlphaCAM will display the toolpaths removing material from a 'solid' billet. This will give you extra confidence that the toolpaths you have created are correct.

When you are ready to conclude the machining, close the 3D simulation by clicking on the ☒ 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.
- Select **FILE** | **Output NC** . The output dialog is displayed.



- 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