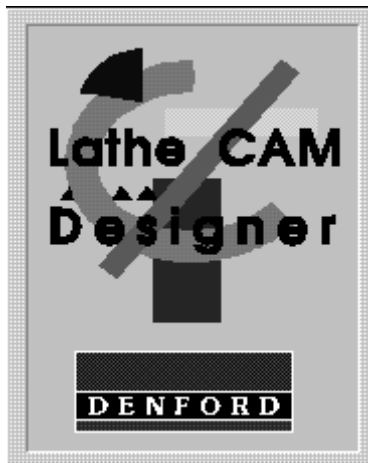




**DENFORD**

Total Commitment to Education and Training WorldWide.

# LatheCAM Designer User Manual.



---

Denford Limited reserves the right to alter any specifications and documentation without prior notice. No part of this manual or its accompanying documents may be reproduced or transmitted in any form or by any means, electronic or mechanical, for any purpose, without the express written permission of Denford Limited.

All brands and products are trademarks or registered trademarks of their respective companies.

Copyright Denford Limited - Version 2.2 - 2000. All rights reserved.

---

---

## About this Manual.

---

Disclaimer	We take great pride in the accuracy of information given in this manual, but due to nature of software developments, be aware that software specifications and features of this product can change without notice. No liability can be accepted by Denford Limited for loss, damage or injury caused by any errors in, or omissions from, the information supplied in this manual.
Screenshots	Please note that any screenshots are used for explanation purposes only.
Language	This manual is written using European English.
Contact	Any comments regarding this manual should be referred to the following e-mail address: <a href="mailto:customer_services@denford.co.uk">customer_services@denford.co.uk</a>
Updates	Any updates to this manual will be posted in the 'Downloads' section of the Denford website: <a href="http://www.denford.co.uk">http://www.denford.co.uk</a>

# Contents.

---

Conventions used in this Manual .....	4
Introduction .....	5
Minimum System Requirements .....	6
Security Devices .....	7
Installation Procedure - Windows 3.xx .....	8
Installation Procedure - Windows 95/98/NT .....	9
Quick Start .....	10
Material Window .....	13
Design Window .....	15
Element Buttons .....	16
Data Panel .....	18
Element Positioning Nudge Buttons .....	19
File Menu .....	20
Elements and Information Menus .....	22
Create G-code Menu .....	23
Tool Types .....	27
Program Structure .....	28
Advanced Program Setup .....	29
Network Installation of LatheCAM Designer .....	32
How To... .....	34
Troubleshooting .....	36
Technical Support .....	37
Glossary .....	38
Index .....	39

---

# Conventions used in this Manual.

---

Mouse Usage	When asked to left click on a menu tile or object, click the LEFT mouse button ONCE. When asked to right click on a menu tile or object, click the RIGHT mouse button ONCE. When asked to double click on an object, click the LEFT mouse button TWICE. When reference to either a left mouse button or right mouse button click command is omitted, always perform one click with the left mouse button.
<u>Underlined text</u>	This is used to show key words. The full definition of any terms are given in the Jargon Buster helpboxes. Similar helpboxes are also used to display any Important Notes or Tips to help you use the program.
"Quotation Marks"	Quotation marks are used to specify any software menu, title and window selections, e.g. click the "File" menu would mean click the left mouse button once, when the cursor is positioned over the File menu label. When a sequence of menu commands are requested, the menu and option names are separated by a vertical line, for example - Click "File   Open" would mean open the File menu, then click on the Open option.
<b>Bold Text</b>	Bold Text is used to show any characters, or text, that must be entered, e.g. type <b>file1</b> would mean type the word file1 into the appropriate text entry box.
[Square Brackets]	Square brackets are used to show any on-screen software button selections, e.g. Click the [OK] button would mean click the left button of the mouse once, when the cursor is directly pointing over the button labeled OK.
<b>[Bold Square Brackets]</b>	Bold square brackets containing text show individual keys to press on your qwerty keyboard, e.g. press <b>[Enter]</b> would mean press the Enter key. If a number of keys must be pressed in sequence they are shown with plus signs outside any square brackets, e.g. press <b>[Alt] + [Enter]</b> would mean press the Alt key first followed by the Enter key second. If a number of keys must be pressed simultaneously they are shown with plus signs inside any square brackets, e.g. press <b>[Alt + Enter]</b> would mean press both the Alt key and Enter key together, at the same time.

---

# Introduction.

---

Congratulations on your purchase of LatheCAM Designer software. LatheCAM Designer has been specially written to lead you through your first experiences of CAD/CAM, into the exciting stages of CNC manufacture!!

LatheCAM Designer is a CNC design system which allows users to build up a turned component on screen, using a graphical environment. A file, in G-Code format, can then be produced automatically.

The work area is specified at the start of the program. This represents the billet size available in the lathe being used. No block can be placed or moved outside the work area. This acts as a safety feature when machining is taking place.

The design is created by placing different styles of blocks together within a limited area. Each block, after being placed, can be manipulated to determine its exact size and shape. LatheCAM Designer is co-ordinate based, allowing extremely accurate movements of the blocks by either keyboard arrow keys or a mouse. Unrequired blocks may be simply deleted at any time.

Any design drawn can be drilled and parted-off . Designs can be saved and loaded at any time, so a library of useful parts can soon be built up.

LatheCAM designer has been developed to be as simple to use as possible. It will allow your CNC turning machine to be used as a 'everyday tool' rather than one reserved for special occasions.

**Jargon Buster** - □ X

CAD/CAM refers to Computer Aided Design and Manufacture, using a computer to help you plan your design from drawing into production.

CNC refers to Computer Numerical Control, the automatic system used to control a machine tool.

A G-Code is a series of letters and numbers that make up the language used by CNC machinery.

A billet is the piece of material you place in the lathe.

Parted-off means to cut the finished work, from the remaining material with a special cutter.

# Minimum System Requirements.

---

The following hardware is required to run LatheCAM Designer.

System Requirements:

- IBM and 100% compatibles.
  - Minimum processor: 486 DX2/66.  
Recommended processor: Pentium.
  - 8MB RAM with Win 3.xx.  
16MB RAM with Windows 95/98/NT.
  - Microsoft 100% compatible mouse.
  - A VGA graphics card is required.
  - A parallel port is required for the security dongle on the full software version.
-

# Security Devices.

---

Your LatheCAM Designer disks will be preformatted, ready for use with one of the following security devices:

## 1) Flash Software

This version does not require a security key. When first starting LatheCAM Designer, a screen will appear displaying your user information and licence details.

A single seat licence can be installed to only one computer station at the registered establishment.

A multiple seat licence can be installed to the specified number of computer stations at the registered establishment.

## 2) Keyed Software

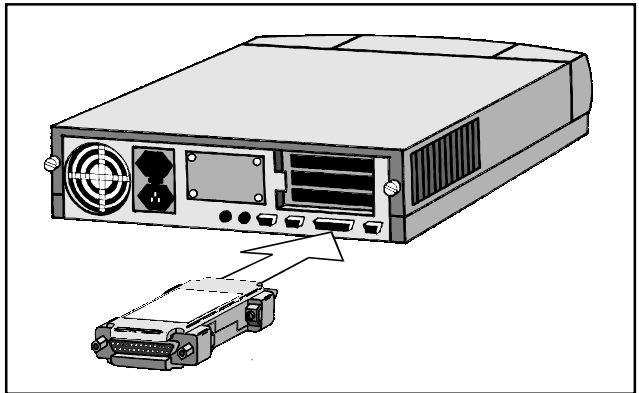
This version uses a security key, called a dongle.

When using the Keyed version of the software, LatheCAM Designer will not run without the Denford Security Dongle.

Fit the dongle onto the parallel port of your computer.

The parallel port is usually positioned on the back panel of your computer - a long, thin 25 pin male connector plug. Note that your parallel port may be labelled as the printer port.

The dongle has a pass-through feature, allowing data to be sent to other external devices, such as printers, scanners or zip drives, when the LatheCAM software is not being used. Simply plug the parallel port cable, supplied with your external device, directly into the back of the dongle.



# Installation Procedure - Windows 3.xx.

---

Follow these instructions to install the Denford LatheCAM Designer software onto a computer running Windows 3.xx. Whilst running Windows on your computer:

- 1) Insert the LatheCAM Designer floppy disk into your floppy disk drive, usually labelled drive 'a:'.
- 2) From 'program manager' select 'run' from the 'file' menu.
- 3) A dialogue box will appear. Type in the command line box "a:install" and press **[Enter]**.
- 4) This will run the Denford install program. Pressing **[Enter]** will install LatheCAM Designer onto your hard disk. It will also create a group window and place the 'Lathe CAM Designer' icon in this group window.
- 5) Before using LatheCAM Designer for the first time, ensure that keyed versions of the software have the dongle fitted to the parallel port of your computer (see page 7). If you did not receive a dongle, you will be using a flash version of the software (see page 7).



The LatheCAM Designer icon looks like this.

---

Jargon Buster - □ X

A dialogue box is a window that opens to let you enter text into a program.  
The command line is where any text entries are entered in a dialogue box.  
An icon is a small picture, displayed in Windows. When you double click the icon, the program associated with that icon will run.  
A program group, or group window is where a selection of icons can be stored together in the Windows environment.

Note - □ X

It is recommended that you allow the Denford installation program to create its own directories and set up its default values. If you find these inconvenient, then feel free to alter them.

Note - □ X

LatheCAM Designer software ships in two different formats: Win 3.xx and Win 95/98/NT. Do not attempt to install the Win 95/98/NT version of LatheCAM Designer onto a Win 3.xx computer !!



# Installation Procedure - Windows 95/98/NT.

---

Follow these instructions to install the Denford LatheCAM Designer software onto a computer running Windows 95/98/NT. Whilst running Windows on your computer:

- 1) Insert the LatheCAM Designer floppy disk into your floppy disk drive.
- 3) Double-click the left mouse button on the 'My Computer' icon. In the 'My Computer' window find your floppy disk drive icon, usually labelled 'a:' and double-click the left mouse button on this icon.
- 4) The contents of the floppy disk will be displayed in a new window. Double-click the left mouse button on the file named 'Setup.exe' to start the Denford software installation program.

Follow the on-screen instructions to install LatheCAM Designer.

- 5) Before using LatheCAM Designer for the first time, ensure that keyed versions of the software have the dongle fitted to the parallel port of your computer (see page 7). If you did not receive a dongle, you will be using a flash version of the software (see page 7).



The LatheCAM Designer icon looks like this.

---

**Note** - □ X

It is recommended that you allow the Denford installation program to create its own directories and set up its default values. If you find these inconvenient, then feel free to alter them.

**Note** - □ X

LatheCAM Designer software ships in two different formats: Win 3.xx and Win 95/98/NT. Do not attempt to install the Win 3.xx version of LatheCAM Designer onto a Win 95/98/NT computer !!

# Quick Start.

---

This section will enable you to become familiar with the features of LatheCAM Designer, without referring to the reference sections.

1) Start LatheCAM Designer by double clicking on the LatheCAM Designer icon, or by clicking 'Startbar|Programs|Denford|LatheCAM' in Windows 95/98/NT.

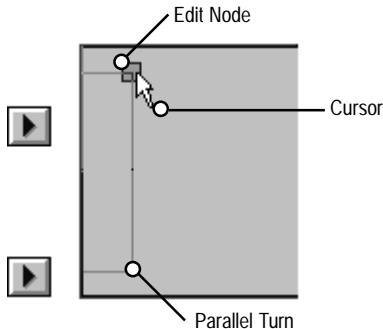
---

2) The materials window allows you to select the material type, the CNC machine tool type, the units of measurement and the billet size, both the diameter and length. For now we will leave these as default values, so click on the [OK] button.

---

3) You will now be presented with the design window. The main grey area of the window represents the side of the work billet, with the red and white dashed line marking the centre of the billet. The rectangle at the far left of this work area is the first element, a parallel turn.

The purple square at the end of the element is called the edit node (a purple edit node identifies the selected element). You can alter the size of this first selected element by moving the cursor over the edit node, then pressing and holding the left mouse key and dragging the shape out to a new position.



## Jargon Buster

An element is a predesigned shape that can be used in your design. There are six element shapes that can be used.

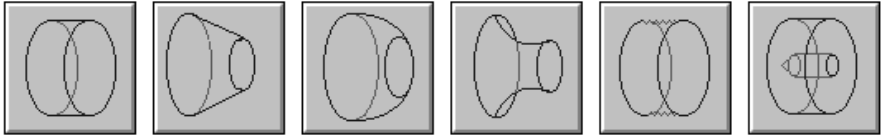
A parallel turn is when material is removed and the finished section is the same diameter along its length.

An edit node is like a handle, you must click on with the cursor, which will allow you to change the shape.

# Quick Start.

---

4) New types of element can be added by clicking on the element buttons above the work area. Each new element will be placed to the right of the highlighted element.



Parallel

Tapered

Convex

Concave

Thread

Drill

## Element Buttons.

---

5) Elements can be edited at any time. Simply highlight the required element by clicking on its edit node once. The information about the highlighted element is displayed in the lower window, called the data panel. The icon in the data panel shows you the type of element selected, the other data is numerical and gives the exact location and size of the element.

6) Pressing **[Delete]** will remove any highlighted element. To delete the whole drawing, simply click 'File|New' from the menu bar.



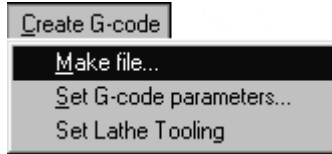
# Quick Start.

---

7) When you have finished designing your shape, save your LatheCAM Designer file by clicking 'File|Save As...' from the menubar. LatheCAM files are saved with the file extension '.lcd'.

---

8) The G-code, required by your CNC machine tool, is made by clicking 'Create G-code|Make file...' from the menubar. G-code files are saved with the file extension '.fml'.



9) The final step is to run the G-code file on your machine tool, to manufacture your part.

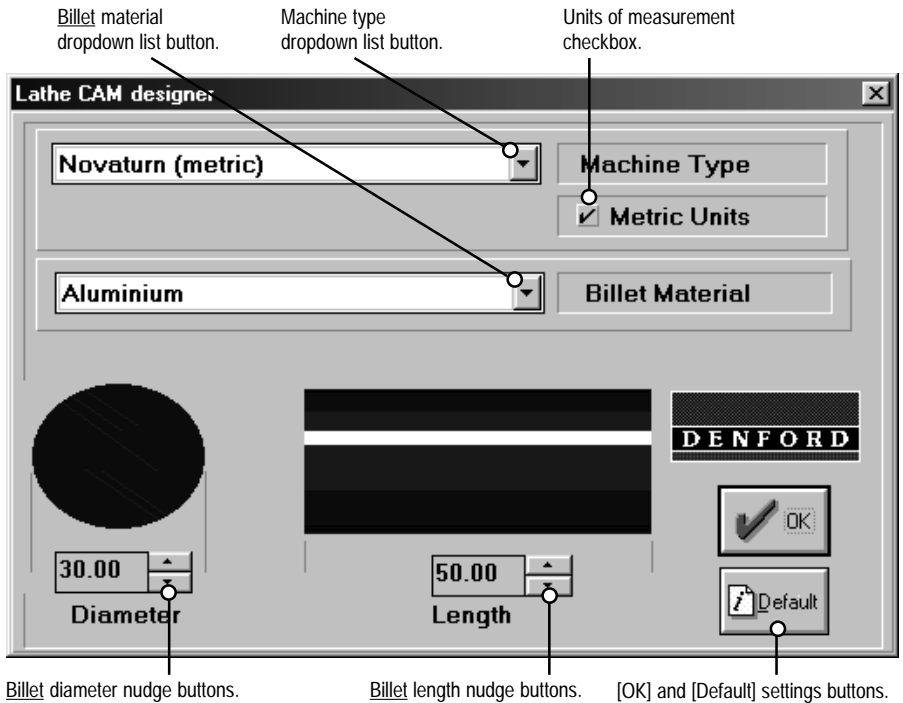
---

Note

Most CNC turning machines are different in the way they read or load G-code instructions. Therefore no specific instructions can be given within this manual. Please check the machine manufacturer's manual for exact details.

# Material Window.

The material window is the first window displayed when the LatheCAM Designer program is started.



**Jargon Buster** - □ X

The term Billet refers to the piece of material that will be machined.

# Material Window.

---

Various options are available in the material window:

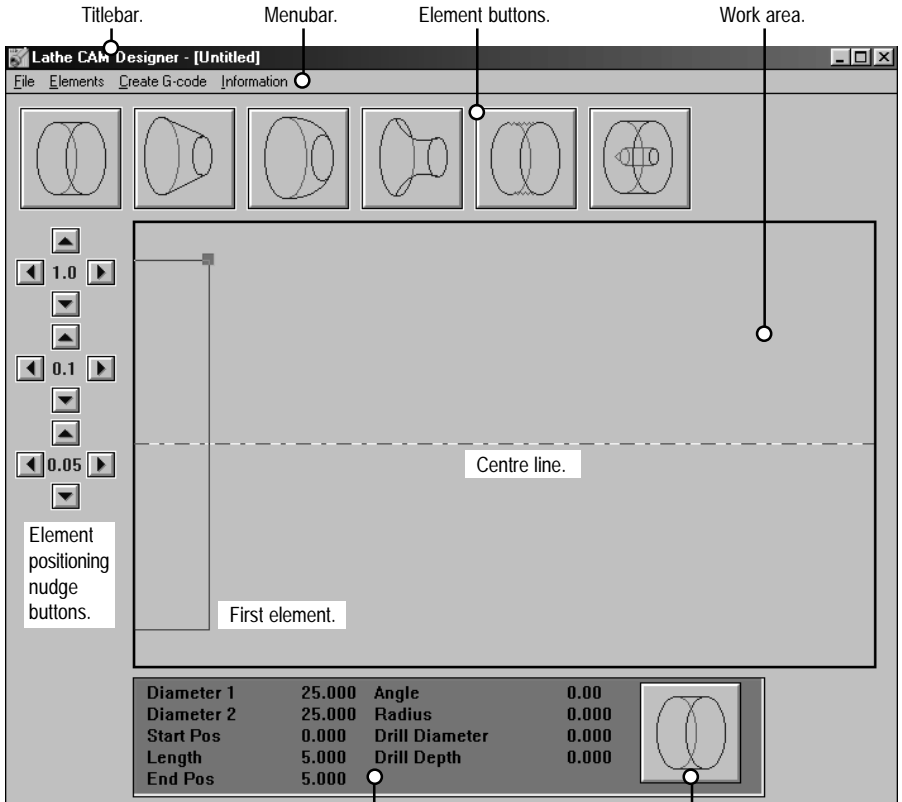
- 1) **Machine Type** - Click the arrow button next to the 'Machine Type' title to display the dropdown list, then click on the name of the machine you intend using for parts manufacture.
- 2) **Units of Measurement** - A tickmark in the 'Metric Units' checkbox indicates Metric measurement (millimetres) are selected. No tickmark indicates Imperial measurement (inches). Note that this checkbox is only used for display purposes - The Units of Measurement are set from the options in the 'Machine Type' dropdown list.
- 3) **Billet Material** - Click the arrow button next to the 'Billet Material' title to display the dropdown list, then click on the name of the material used for your billet.

Note - Selecting a particular material will load a set of specific machining parameters. Their values can be edited from the design window, using the 'Create G-code|Set G-code parameters...' menu. The units of measurement used in the set of machining parameters must match the units of measurement used in the machine type (chosen in section 1), i.e. they must be both Metric or both Imperial.

For more information see the G Code Parameters and Advanced setup sections on pages 24 and 29.

- 4) **Diameter** - Click the up and down nudge buttons to define the 'diameter' of the billet in the units of measurement previously selected. This will be the X axis movement of the machine.
- 5) **Length** - Click the up and down nudge buttons to define the 'length' of the billet in the units of measurement previously selected. This will be the Z axis movement of the machine. This figure should not include the additional material required for holding the real billet in the lathe chuck.
- 6) **[OK] / [Default]** - Click the [OK] button to save the current settings and open the design window. Click the [Default] button to reset all the options in the materials window.

# Design Window.



## Screen Layout.

### Work area.

This is the area where all element blocks are placed and manipulated. This working area represents the diameter and length of the billet. Elements are added from left to right, with the left hand side of the work area representing the end of the billet held in the machine tool chuck. The dashed line shows the centre line of the billet.

### Menubar.

Click the required menu title to display the dropdown list. Move the cursor down the list, highlighting each option. Click the highlighted option to select it.

### Titlebar.

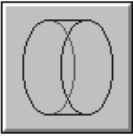
The titlebar shows the name of the program 'Lathe CAM Designer' along with the name of the file currently being edited (if saved previously).

# Element Buttons.

---

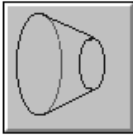
Shortcut keys:

[1]



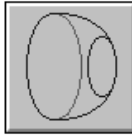
Parallel

[2]



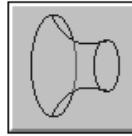
Tapered

[3]



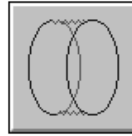
Convex

[4]



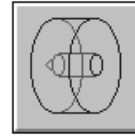
Concave

[5]



Thread

[6]



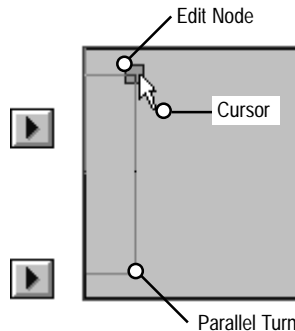
Drill

There are 6 element buttons, each representing a different shape that can be used in the work area.

To add a new element to your design, simply click on the button required, use the appropriate short cut key, or select the element name from the 'element' menu.

The shape of each element is changed by dragging the edit node. These are the faint squares, visible at the top edge of each element block. The selected element node is always highlighted using a purple square.

Element shapes can also be adjusted using the element positioning nudge buttons (see pages 10 and 18).



## [1] - Parallel.

This will only allow you to draw a rectangle on the screen, representing a true cylinder with parallel sides. Diameters 1 and 2 always remain the same.

## [2] - Tapered.

This allows you to alter the diameter 2 of the cylinder, thus giving a cone shape. Diameter 2, the right hand diameter, may only be larger than diameter 1 by a small amount. The difference in diameters is restricted by the angle set within the 'LCD.ini' file and is related to the back angle of your turning tools.

## [3] - Convex, [4] - Concave.

These allow you to place radii on your work, restricted to 90 degree arcs.



# Element Buttons.

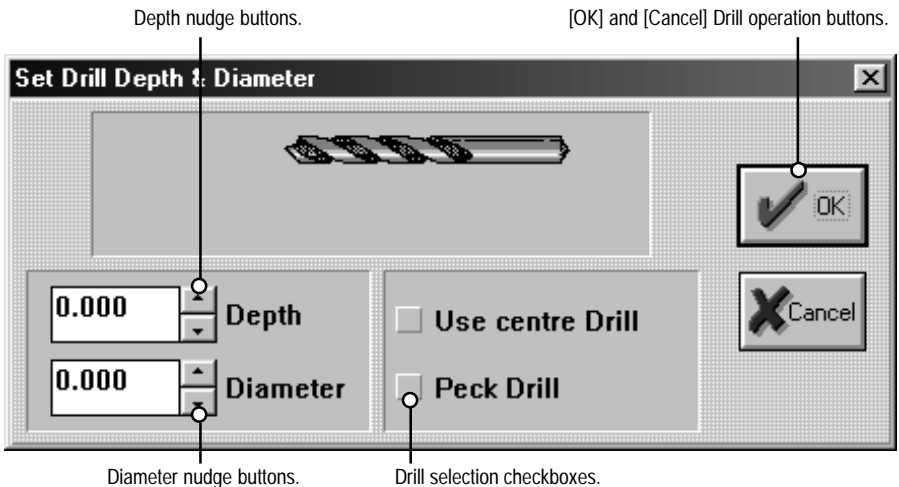
## [5] - Thread.

This allows you to place a thread on your design. You should ensure that the elements either side of the thread will allow the threading tool enough clearance (the program cannot detect how much clearance your machine will require). However, the program will automatically make an undercut to left of the thread.

## [6] - Drill.

This allows you to drill into the right hand side of your design. When the drill element is selected, a dialogue box will appear. The dialogue box will allow you to set both the diameter and the depth of a drilled hole, using the nudge buttons.

### Drilling dialogue box.



### Centre Drill.

Clicking the centre drill checkbox will place a centre drill operation at the end of your design, used for the accurate location of turning centres held in the machine tool tailstock, or as a pilot hole for the main drilling operation.

### Peck Drill.

During the drilling cycle swarf can build up in the drill flutes. This causes the drill bit to rise in temperature and wear prematurely. Clicking the peck cycle checkbox periodically withdraws the drill bit, which helps to remove the build up of swarf.

# Data Panel.

## Data Box.

<b>Diameter 1</b>	<b>18.000</b>	<b>Angle</b>	<b>0.00</b>	
<b>Diameter 2</b>	<b>11.000</b>	<b>Radius</b>	<b>3.500</b>	
<b>Start Pos</b>	<b>27.000</b>	<b>Drill Diameter</b>	<b>4.000</b>	
<b>Length</b>	<b>3.500</b>	<b>Drill Depth</b>	<b>20.000</b>	
<b>End Pos</b>	<b>30.500</b>			

Numerical data.

Selected element icon.

### Selected element icon.

This icon shows what type of element is highlighted. The icons displayed look identical to the buttons used at the top of the screen. As you select different elements the icon will change.

### Numerical data.

This window shows the exact position and size of the selected element. Each heading means:

- 1) **'Diameter 1'** - The diameter of the left hand side of the element.
- 2) **'Diameter 2'** - The diameter of the right hand side of the element.
- 3) **'Start Pos'** - The absolute position of the left hand side of the element, measured from the left hand end of the billet.
- 4) **'Length'** - The length of the element.
- 5) **'End Pos'** - The absolute position of the right hand side of the element measured from the left hand side of the billet.
- 6) **'Angle'** - This is the angle of the slope from diameter 1 to diameter 2. If the figure is negative then diameter 1 is smaller than diameter 2. The maximum negative angle is set by a value held in the file 'LCD.ini'.
- 7) **'Radius'** - This will only have a value for convex and concave elements, showing the radius of the arc on the highlighted element.
- 8) **'Drill Diameter'** - This displays the drill diameter. This is the same value entered when the drilling dialogue box was displayed.
- 9) **'Drill Depth'** - This displays the drill depth. This is the same value entered when the drilling dialogue box was displayed.

# Element Positioning Nudge Buttons.

---

## Element positioning nudge buttons.

These allow the elements to be shaped and positioned with a very high degree of accuracy.

To change the shape of a selected element, simply click on the required element positioning nudge button. There are three degrees of movement accuracy available using the nudge buttons.

The shortcut keys, shown below, can also be used to change the selected element shape.

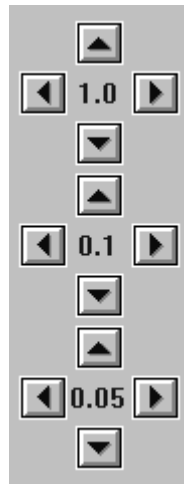
If you use the combination **[Arrows + Control + Shift]** then you can nudge 0.001.

### Shortcut keys:

For 1.0 movement use:  
**[Arrows]**

For 0.1 movement use:  
**[Arrows + Shift]**

For 0.05 movement use:  
**[Arrows + Control]**



# File Menu.

---



Click the 'File' menu title to display the dropdown list. Move the cursor down the list, highlighting each of the nine options. Click the highlighted option to select it.

## 'New'

Click this option to start a new design. The screen is cleared after giving you the option to save any previously designed work.

Shortcut keys **[Ctrl + N]**.

## 'Open...'

Click this option to open a previously saved LatheCAM Designer file. Choose the directory and folder from the 'Directories:' box, select the '.lcd' file from the 'Files:' box, then click the [OK] button to load the file.

Shortcut keys **[Ctrl + O]**.

## 'Save'

Click this option to save any changes made to a previously saved LatheCAM Designer file. LatheCAM Designer files are saved using the file extension '.lcd'.

Shortcut keys **[Ctrl + S]**.

## 'Save As...'

Click this option to save the current LatheCAM Designer file with the option to place the file in a specific directory and folder with a user defined filename. Choose the directory and folder from the 'Directories:' box, type the name of the file in the 'Filename:' dialogue box, then click the [OK] button to save the file. Filenames must not be longer than eight characters.

continued....

---

# File Menu.

---

## 'Refresh Screen'

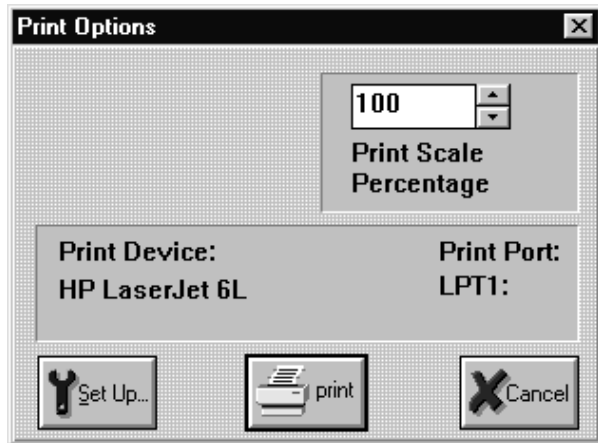
Click this option to redraw the design window.

Shortcut keys [Ctrl + R].

## 'Print'

Click this option to print the work area of the design window. The 'Print Scale Percentage' option of the 'Print Options' window allows the printed size of the work area to be changed. When set to '100%', the work area is printed as large as possible on the default selection printer paper.

Shortcut keys [Ctrl + P].



## 'Printer setup'

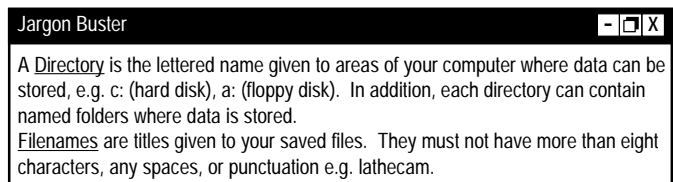
Click this option to configure your printer options.

## 'Alter Billet Size'

Click this option to display the Material window, where the billet size can be changed. Note - all data will be lost unless saved before selecting this option.

## 'Exit'

Click this option to close the LatheCAM Designer software. The program is closed after giving you the option to save any previously designed work.



# Elements and Information Menus.

---

Elements	
Parallel	1
Tapered	2
Convex Curve	3
Concave Curve	4
Screw Thread	5
Drill	6
Delete	Del

Click the 'Elements' menu title to display the dropdown list. Move the cursor down the list, highlighting each of the seven options. Click the highlighted option to select it.

- 'Parallel' ..... Click each
- 'Tapered' ..... of these
- 'Convex Curve' ..... options to
- 'Concave Curve' ..... add the
- 'Screw Thread' ..... named
- 'Drill' ..... element to
- 'Delete' ..... your design.

Click this option to remove the currently selected element.

Information
About Lathe CAM Designer

Click the 'Information' menu title to display the dropdown list. Move the cursor down the list, highlighting the single option. Click the highlighted option to select it.

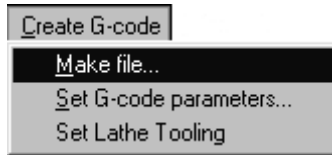
## 'About LatheCAM Designer'

Select this option to display information about the LatheCAM Designer software:

- Software version number.
- Language installed.
- Units installed.
- Dongle serial number.

Click the [OK] button to close the window.

# Create G-code Menu.



Click the 'Create G-code' menu title to display the dropdown list. Move the cursor down the list, highlighting each of the three options. Click the highlighted option to select it.

## 'Make file...'

Select this option to convert your LatheCAM file into a G-code file, a series of letters and numbers that make up the language used by CNC machine tools. G-code files are saved using the file extension '.fml'.

The 'File Save As' window will appear. The default directory is 'c:\denford\data'. Choose the directory and folder from the 'Directories:' box, then type in the name of the file in the 'Filename:' dialogue box. Filenames must not be longer than eight characters. Click the [OK] button to begin the conversion process.

The G Code Generator window will be displayed, showing a confirmation of the chosen filename and location. Click the checkboxes to select the 'part-off', 'face-off' or 'use coolant' options. A tickmark appears when each option is selected.



## Jargon Buster

A Directory is the lettered name given to areas of your computer where data can be stored, e.g. c: (hard disk), a: (floppy disk). In addition, each directory can contain named folders where data is stored.

Filenames are titles given to your saved files. They must not have more than eight characters, any spaces, or punctuation e.g. lathecam.

A parting-off operation removes the finished design from the remainder of the billet, still held in the machine chuck.

Face-off refers to the turning operation used to cut the billet end at 90 degrees to the centreline, before the design is machined.

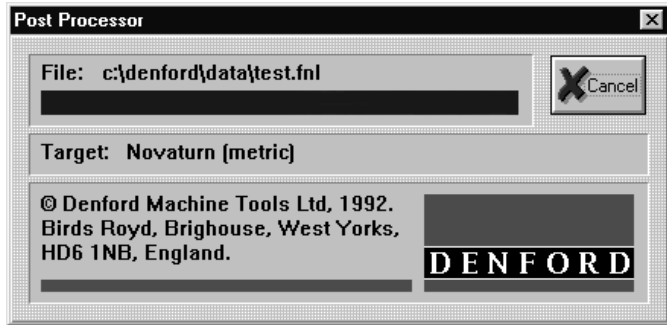
Coolants can be used to prevent the cutting tools from overheating during the machining process.

# Create G-code Menu.

---

'Make file...' option continued....

Click the [Create G-code] button to begin the conversion process. The machining paths for your design will be calculated and plotted on-screen, according to the values set in the G-code Parameters. Following this, the 'Post Processor' window will be displayed, indicating the progress of the file conversion and the final location of the G-code file.



'Set G-Code parameters'

Select this option to configure the machining parameters relative to the material used for the billet.



continued....



# Create G-code Menu.

---

## **Billet Material option.**

The billet material option loads a set of machining parameters appropriate for the material chosen. Click the arrow button next to the name of the material to display the dropdown list, then click on the name of the material you intend using for parts manufacture. If the machine units are set to Metric (as selected from 'Machine Type' in the opening material window), the machining parameters must also be defined using Metric units. If the machine units are set to Imperial, the machining parameters must also be defined using Imperial units.

## **Machining Parameters.**

Click the up and down arrow nudge buttons to specify the value of the following machining parameters, appropriate to the material and units of measurement currently selected:

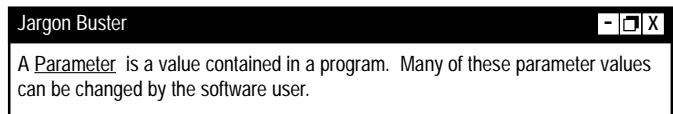
- Feed Rate (the speed at which the tool moves through the material, measured in millimetres per minute or inches per minute).
- Finishing Cut (the amount of material left before applying the final cut giving good dimensional accuracy and surface finish quality, measured in millimetres or inches).
- Roughing Cut (the amount of material removed in one roughing pass, measured in millimetres or inches).
- Threading Cut (the maximum amount of material removed in one pass, measured in millimetres or inches).
- Surface Cutting Speed (the ideal material surface cutting speed, measured in metres per minute or feet per minute. The actual cutting speed varies according to the diameter of the billet).

## **[Add...] option.**

You can specify your own material and parameters by clicking on the [Add...] button, typing a name for the new material, then adjust the set of machining parameter values for the new material. The settings for new materials are saved in the 'LCD.ini' file.

Any material selected or added will be used as the default in the 'Material' window, the next time the LatheCAM Designer software is run.

For more information see the Advanced setup section on page 29.



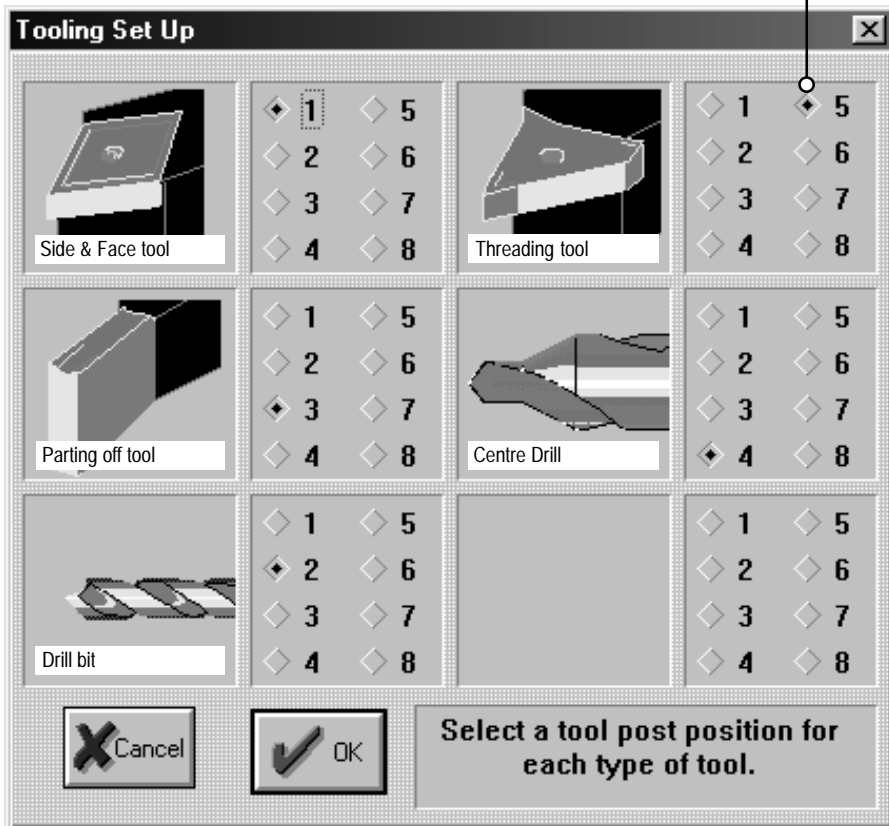
# Create G-code Menu.

## 'Set Lathe Tooling'

This menu will select a dialogue box that will allow you to match your CNC machine toolpost numbers with the LatheCAM tooling setup, i.e. if tool 1 on the CNC machine is a roughing tool, then tool 1 in LatheCAM Designer must be a roughing tool. This information is needed by the software so that when a tool change is required it can enter the correct number in the G-Code file.

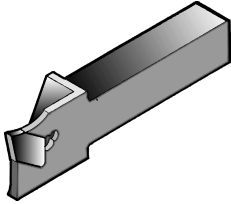
For each tool profile, click the grey diamond marker next to the tool number required. A black bullet is displayed to indicate the current number selected for each tool. Click the [OK] button to confirm the setting of all tool profiles.

Selected number for each tool shown by black bullet marker.



# Tool Types.

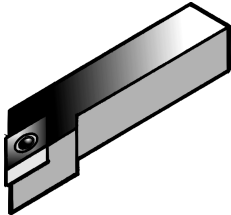
---



## **Parting tool.**

Used mainly to remove work from the end of the billet, but on occasion it is used to make small undercuts.

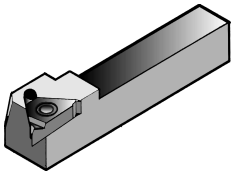
---



## **Roughing / Cutting tool.**

This is a right handed cutting tool. It is used on the outside of the billet and carries out the majority of the work.

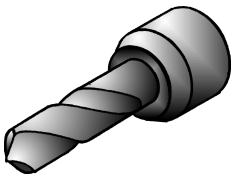
---



## **Threading tool.**

The threading tool has only one function due to its sharp steeply angled cutting edge. This tool is able to create many different depths of thread.

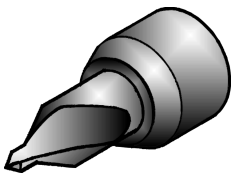
---



## **Twist Drill.**

The twist drill is used to make holes in the centre of the billet. The drill bits are available in a variety of lengths and widths. On a lathe the drill bit remains stationary as the billet revolves.

---



## **Centre Drill.**

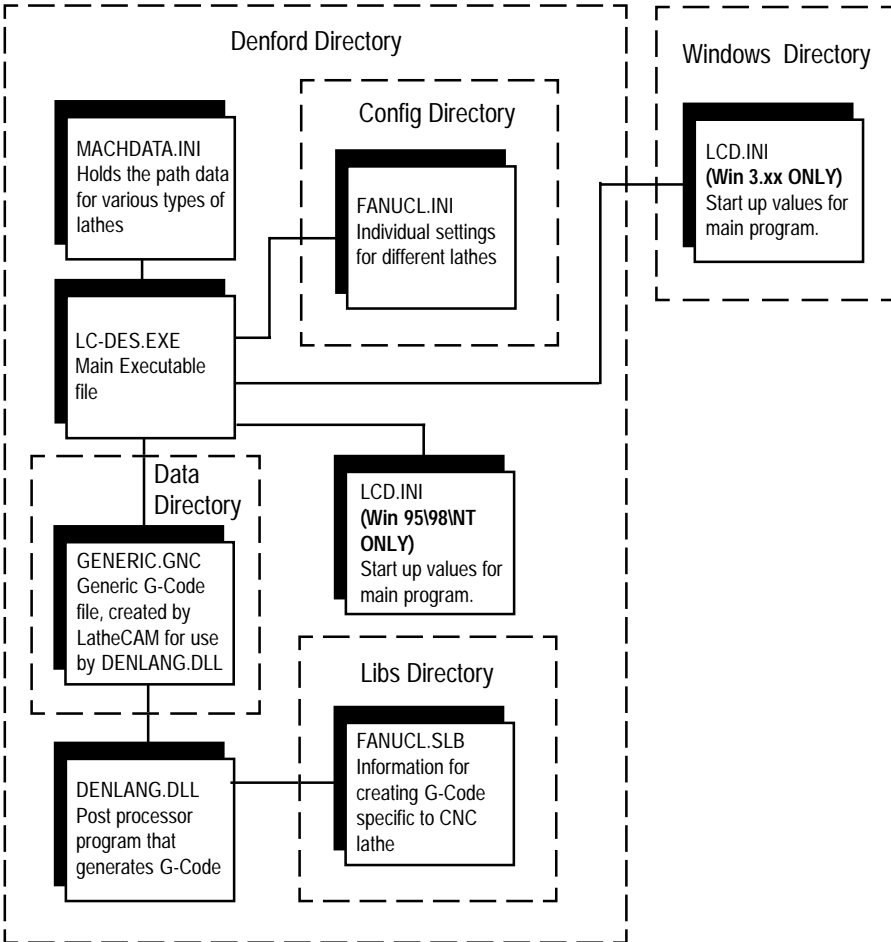
This is a small rigid drill bit that has the function of making a small pilot hole, used for guiding the main drill bit to the centre of the billet or locating a turning centre held in a lathe tailstock.

---

# Program Structure.

LatheCAM Designer uses and creates several files while it is running. The diagram below shows the position and function of these files. Note that the location of the 'LCD.ini' file depends on whether a Win 3.xx install disk or Win 95\98\NT install disk has been used.

## Position and purpose of the main LatheCAM Designer files.



# Advanced Program Setup.

---

All of the program features are setup automatically by the installation program. It is possible to alter some program settings manually, to suit your particular needs. The program default values are stored in the 'LCD.ini' file. This can be found in your Windows directory (Win 3.xx users) or Denford directory (Win 95\98\NT users). The following is an example of this file and a description of the contents.

## [Settings]

Material Type=Aluminium - Most recently used material.  
Length=50.0 - Most recently used length.  
Diameter=30.0 - Most recently used diameter.  
machine=Novaturn (metric) - Most recently used machine.  
type=1

## [Path Data]

GNC Name=c:\denford\data\generic.gnc - A temporary file generated by LatheCAM.  
LCD Files=c:\denford\data - LatheCAM files directory.  
GCode Files=c:\denford\data - G-code files directory.

## [Feed Rates]

Steel(Metric)=60.000 - Feed rates are how fast the tool moves through the material, defined in millimetres per minute for (Metric) settings and inches per minute for (Imperial) settings.  
Brass=70  
Plastic=150  
Steel(Imperial)=2.3  
Wax=350  
Aluminium=140.005

## [Materials]

Steel(Metric)=120 - These are the surface cutting speeds, defined in metres per minute for (Metric) settings and feet per minute for (Imperial) settings. The actual cutting speed varies according to the diameter of the billet.  
Brass=100  
Plastic=300  
Steel(Imperial)=120  
Wax=500  
Aluminium=170

## [Finish Cut]

Steel(Metric)=0.5000 - Finish cut, this is the amount of material left before applying the finishing cut, defined in millimetres for (Metric) settings and inches for (Imperial) settings.  
Brass=0.5  
Plastic=0.5  
Steel(Imperial)=0.02  
Wax=0.5  
Aluminium=0.5000

## [Roughing Cut Increment]

Steel(Metric)=1.0000 - These values are the maximum amounts taken out in any roughing cut, defined in millimetres for (Metric) settings and inches for (Imperial) settings.  
Brass=3  
Plastic=4  
Steel(Imperial)=0.08  
Wax=4  
Aluminium=0.500

Continued on the next page....

# Advanced Program Setup.

---

continued from the previous page....

## [Thread Cut Increment]

Steel(Metric)=0.0600  
Brass=0.06  
Plastic=0.06  
Steel(Imperial)=0.008  
Wax=0.088  
Aluminium=0.0610

- The maximum amount the tool removes on any single pass, defined in millimetres for (Metric) settings and inches for (Imperial) settings.

## [Thread]

Pull Out Angle=60  
Thread Angle=60  
Finish Allowance=0.03  
XOffSet=1  
ZOffSet=5  
XOffSet (Inches)=0.04  
ZOffSet (Inches)=0.04  
Diameter OffSet=0.05  
Diameter OffSet (Inches)=0.001  
Thread Undercut=2

- Angle at which the tool leaves the billet.  
- Pitch of the thread.  
- Material left before the final cut.  
- X and Z offsets define the starting point for the tool, used as an allowance to let the drive motors spin upto full speed before cutting takes place.  
- Diameter offsets define the amount of 'play' in the thread.  
- Specifies type of thread undercut.

## [Thread Lookup]

0=0.25  
1=0.4  
2=0.5  
3=0.7  
4=0.8  
5=1  
6=1.25  
7=1.5  
8=1.75  
9=2.00  
10=2.5  
11=3.0  
12=3.5  
13=4.0  
14=4.5  
15=5

- The thread lookup table is used to define an appropriate thread pitch for the thread diameter selected.

## [Thread Lookup (Inches)]

0=0.05  
1=0.05556  
2=0.0625  
3=0.07143  
4=0.07692  
5=0.08333  
6=0.09091  
7=0.1  
8=0.11111  
9=0.125  
10=0.14286  
11=0.16667  
12=0.2

- The thread lookup table is used to define an appropriate thread pitch for the thread diameter selected.

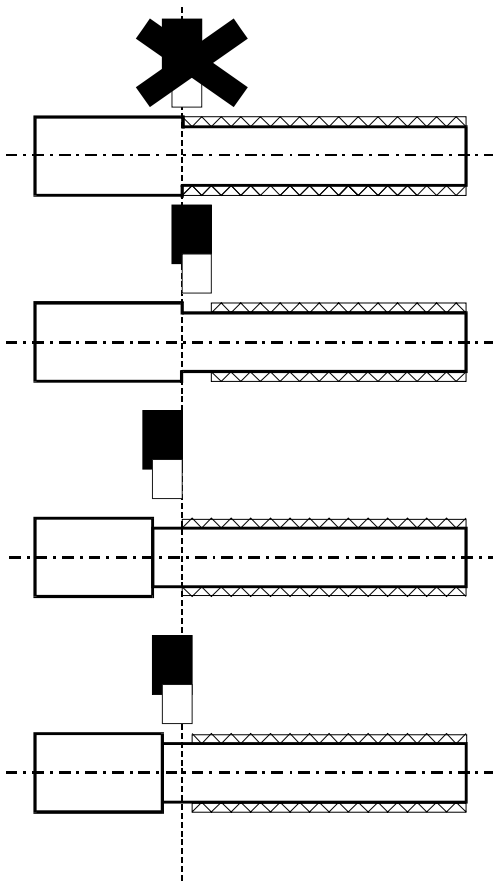
## [Drill]

Use Centre=1  
Peck Drill=1

- Use (1) or do not use (0) centre drill.  
- Use (1) or do not use (0) peck drilling.

# Advanced Program Setup.

---



---

## Thread Undercut options.

---

[ 0 ] No undercut.

---

[ 1 ] A small section of thread is removed at the end of the thread.

---

[ 2 ] A piece of material is removed before the thread has been cut.

---

[ 3 ] Material is removed from both the thread and the workpiece.

---

To gain access to the undercut options use a text editor and open the 'LCD.ini' file. Undercut options are placed under [Thread]  
e.g. Thread Undercut=1

---

# Network Installation of LatheCAM Designer.

---

## To install LatheCAM Designer on your server computer:

Install the LatheCAM Designer software to the required hard drive on your server computer.

Find the file 'LCD.ini', located in the 'Windows' or 'Denford' directory.

The 'LCD.ini' file will contain a section similar to the following:

```
[Path Data]
GNC Name=P:\Denford\data\generic.gnc
LCD Files=P:\Denford\data\
GCode Files=P:\Denford\data\
```

where 'P' is your server computer hard drive.

In addition to checking that these directories exist in the locations listed, check that the following rules apply:

- 1) 'GNC Name=' must point to a read/write file (it is a temporary file created for each student during the G-Code generation process).
- 2) 'LCD Files=' must point to a read/write directory (it is the directory where a students design files are kept).
- 3) 'G Code Files=' must point to a read/write directory (it is the directory where a students CNC files are kept).
- 4) Workstation computers must have access to the main program file 'LCD.exe' located in the 'Denford' directory.

Normally, these three entries must be reconfigured to point towards the students own personal directory area. For example, a student having a personal directory of 'N:\Mywork\' would have these entries changed in the server computer 'LCD.ini' file to read:

```
GNC Name=N:\Mywork\generic.gnc
MCD Files=N:\Mywork\
G Code Files=N:\Mywork\
```

---



# Network Installation of LatheCAM Designer.

---

**To install LatheCAM Designer on each workstation computer:**

Setup a shortcut to the main program file 'LCD.exe', located in the 'Denford' directory on your server computer.

Check that the 'WIN.ini' file, located in the 'Windows' directory, on each individual workstation computer running LatheCAM Designer has the following:

```
[Denford Machine Tools]
Directory=P:\Denford\
```

where 'P' is the Network drive mapped to the 'Denford' folder on your server computer.

For specialist networks (e.g. RM) contact Denford Customer Services if you experience any problems.

---

# How To...

---

## **Adding an Element:**

Click the appropriate element button to place the new shape into the work area.

## **Selecting an Element:**

Click the square edit node, located on the element you want to select. The edit node will change to a purple colour, when the element is selected.

## **Resizing an Element:**

- 1) Select the element you want to change.
- 2) Use the mouse or nudge buttons to resize the element.

## **Removing Elements:**

- 1) Select the element you want to remove.
- 2) Click the right mouse button.

## **Adding a New Material Type:**

- 1) From the menubar, click "Create G code | Set G code Parameters".
  - 2) The "Feeds" window will appear.
  - 3) Click the [Add] button.
  - 4) Enter a name for the new material type, then click [OK].
  - 5) Enter the desired feed and cut settings, then click [OK].
-

# How To...

---

The information listed below is common to all Windows-based software. If, after following the instructions, you are still having problems changing directories, please refer to your Windows Manual.

## Creating G codes and Saving LatheCAM Designer Files:

- 1) Open LatheCAM Designer.
- 2) Create a design.
- 3) Click "Create G code | Make File" to create a G code file, or "File | Save As" to save the LatheCAM design.
- 4) When creating a G code or saving the LatheCAM design, the name of your file is entered in the "Filename:" dialogue box.  
The location where the file will be saved is stated opposite the "Directory:" title. Using default settings this should read **c:\denford\data**.
- 5) Click the [OK] button to start creating the G code or save the LatheCAM design file.

## Changing Directories:

To change the directory to read **c:\denford\data**, when not displayed as listed above:

- 1) Using the "Directories." panel of the "File Save" window, click the dots in the square brackets [..] until the directory is displayed as **c:** drive. The current directory is stated opposite the "Directory:" title, directly under the filename dialogue box. Whenever you click on the dots in the square brackets, you will be taken back ONE directory level. If you do not understand directory structures, please refer to your Windows Manual.
  - 2) Once the Directory is displayed as the **c:** drive, double click **[denford]** in the "Directories." panel. The directory should now read **c:\denford**
  - 3) Double click **[data]** in the "Directories." panel. The directory should now read **c:\denford\data**
-

# Troubleshooting.

---

**Q. When trying to start the software, I receive an error stating "Dongle not present". The software does not open.**

- A1. If using the keyed version, check to see that your security key (dongle) is plugged into the parallel port and that the port is functioning correctly.
  - A2. If using the flash (keyless) version, contact Denford for assistance.
- 

**Q. After creating a design, I cannot create a G-code file.**

- A. If using the keyed version, check to see that your security key (dongle) is plugged into the parallel port.
- 

**Q. After creating a G code file, approximately 12.5mm (0.5") of material has been added to the part length.**

- A. When setting the material size in LatheCAM Designer, specify the machineable area and NOT the actual physical billet size. When creating the G code, the post processor will automatically add material to the length for chuck compensation.
- 

**Q. After creating a G code file and attempting to manufacture the part, the cutter is trying to remove too much material in one pass.**

- A. Lower the cut settings, particularly the roughing cut. From the menubar, select "Create G code | Set G code Parameters".
-

# Technical Support.

---

Denford Limited provides unlimited Technical Support on this software.

Technical Support is only available to registered users. If your software was not registered with Denford Limited at the point of sale, e-mail or fax your registration details to Denford Limited, or your authorised Denford shipping agent as soon as possible.

When you request Technical Support, please be at your computer, with your computer and software documentation to hand. To minimise delay, please be prepared to provide the following information:

- Dongle Serial Number.
- Registered user's name / company name.
- Software name and version number (found in the 'Information|About LatheCAM Designer' menu).
- The wording of any error messages that appear on your computer screen.
- A list of the steps that were taken to lead up to the problem.

## Contact Details:

Denford Limited

Birds Royd, Brighouse, West Yorkshire, HD6 1NB, UK.

Telephone: 01484 712264

Fax: 01484 722160

ISDN: 01484401157:01484401161

E-mail: [service@denford.co.uk](mailto:service@denford.co.uk)

Technical Support: Monday to Friday 8.30am - 4.30pm GMT

For USA please contact:

Denford Inc.

815 West Liberty Street, Medina, Ohio 44256, USA.

Telephone: 330 7253497

Fax: 330 7253297

E-mail: [service@denford.com](mailto:service@denford.com)

Technical Support: Monday to Friday 8.30am - 4.30pm Eastern

<http://www.denford.com>

# Glossary.

---

Billet .....	The piece of material that will be machined.
CAD .....	(Computer Aided Design) An application program that allows a designer to draw and edit on a computer screen, before transferring to another media.
CAM .....	(Computer Aided Manufacture) The use of computer driven robots and machines to help with manufacture.
CNC .....	Computer Numeric Control
Command line .....	The command line is the text entry in a dialogue box.
Computer configuration .....	The collection of computer hardware that makes up a particular computer system.
Cursor .....	The marker on the computer screen, that shows the position of the pointing device.
Dialogue box .....	A program window that opens to allow text to be entered.
Data .....	The numbers, digits, characters and symbols operated on by a computer.
Directory .....	The lettered name given to areas of your computer where files can be stored, commonly referred to on a computer by assigned letters, e.g. c: (hard disk), a: (floppy disk).
Disk .....	A magnetic storage device for external data.
DOS .....	<u>D</u> isk <u>O</u> perating <u>S</u> ystem. A program which controls the operation of a disk drive.
Execute .....	Run a computer program, or carry out a single instruction.
Feedrate .....	The rate at which the tool is moved towards, along and into the material.
File .....	An organised collection of related computer data and records.
Filename .....	File names are titles given to your saved files. They must not have more than eight characters, any spaces, or punctuation e.g. millcam.
Floppy Disk .....	A storage device for a computer. It can be removed and used to transport data.
Folder .....	Computer directories can contain named folders where files are stored.
Hard copy .....	Computer output on paper or similar media.
Hard Disk .....	A storage device for a computer. It cannot normally be removed from the computer, but it is very fast and can hold a large amount of data.
Hardware .....	Computer equipment.
Icon .....	An icon is a small picture, displayed in Windows. When you double click the icon, the program associated with that icon will run.
Mouse .....	A hand-held device that moves around a flat surface and transmits details of its position to the computer.
Network .....	A system of connecting several microcomputers so that they can share common facilities such as printers and hard disks.
Operating system .....	A program that controls the computer from when it is first switched on.
Parameter .....	A value contained in a program. Many of these parameter values can be changed by the software user.
Post-processor .....	A program that takes a generic code and processes it in order to produce a file understood by a machine tool.
Program .....	The set of instructions that makes the computer carry out the desired processes.
Program group .....	A program group, or group window is where a selection of icons can be stored together in the Windows environment.
Software .....	The programs used in a computer.
Windows .....	An easy to use environment that uses icons, a mouse and windowed user areas as the computer operating system.

# Index.

---

## B

Billet	
Billet	5
Diameter	14
Length	14

## C

CAD/CAM	5
Centre Drill	17, 27
CNC	5
Command Line	8
Contact Details	37
Contents	3
Coolants	23
Create G-code Menu	23

## D

Data Panel	18
Design Window	15
Dialogue Box	8
Dongle	
Fitting	7
Serial Number	22

## E

Edit	
Billet Size	21
G-Code Parameters	24
Edit Element	
Node	10, 16
Nudge Buttons	19
Shortcut Keys	16
Element	10
Concave	16
Convex	16
Drill	17
Parallel	16
Tapered	16
Thread	17
Element Buttons	16
Element Menu	22
Element Positioning Nudge Buttons	19
Exit Program	21

## F

Face-off	23
File Menu	20, 21
Flash	7

## G

G-code	5
Parameters	24
Save	23
Set Tooling	26

## H

Help	37
------	----

## I

Icon	8
Information Menu	22
Install	
Network	32
Windows 3.xx	8
Windows 95/98/NT	9

## M

Material Window	13, 14
Menubar	15

## N

Network	32
New File	20
Numerical Data	18

## O

Open File	20
-----------	----

## P

Parallel Turn	10, 16
Parted-off	23
Parting Tool	27
Peck Drill	17
Print	21
Program Group	8

## Q

Quick Start	10
-------------	----

## R

Refresh Screen	21
Roughing / Cutting Tool	27

## S

Save	
G-code File	23
LatheCAM File	20
Screen Layout	15
Selected Element Icon	18
Setup	
CNC Machine	12
G-code Parameters	24
Lathe Tooling	26
Program Install	8
System Requirements	6

## T

Technical Support	36, 37
Thread Undercut	31
Threading Tool	27
Titlebar	15
Twist Drill	27

## U

Units of Measurement	14
----------------------	----

## W

Windows 3.xx	8
Windows 95/98/NT	9
Work Area	15

# Notes.

---