DENFORD

EDUCATION SUPPORT SERVICE

Lathe CAM Designer
Users Manual

Notes

Contents

1 Conventions used in this Manual

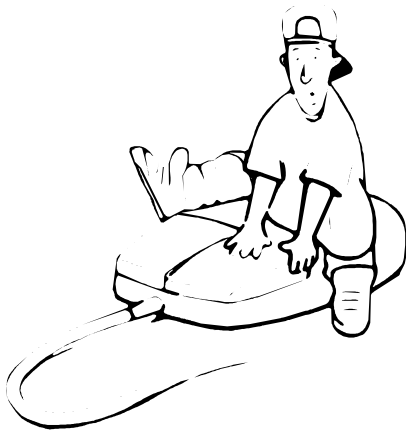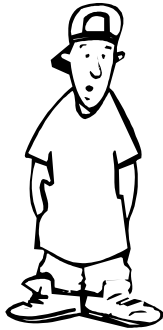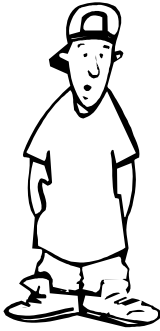|	
<u>Underline</u>	This is used to show key words. The full definition of these terms is given in Denny's Help Box.
Italics	Used to show menu and button selections. To pull down a menu move the pointer over the title and click once with the left mouse button. To select something from the menu move the pointer on to what is required, then click with the left button again. To select a button move the pointer over it and click once with the left mouse button.
[ ]	Square brackets contain any text that must be typed in.
[bold]	Square brackets with bold text show individual keys to press, e.g. [ENTER].
+	If a number of keys need to pressed in sequence they are shown with plus signs.
	This is used to show important notes.
	This shows a right button mouse press.
	This show a left button mouse press.

2 Introduction

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

The design is created graphically 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. Lathe CAM Designer is co-ordinate based, allowing extremely accurate movements of the blocks by either; keyboard arrow keys or a mouse. A block may be deleted at any time.

A '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 whilst machining.

Introducing
Denny Denford

Denny's Help Box

"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."
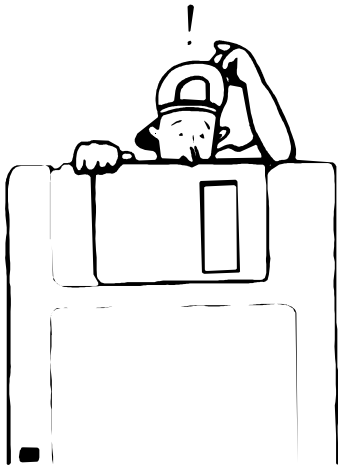
Introduction

Any design drawn can be drilled and <u>parted-off</u> .

Designs can be saved and loaded at any time, so a library of useful parts can soon be built up.

Lathe CAM designer has been developed to be as simple as possible to use. This, it is envisaged, will enable the use of CNC turning machines by a much wider range of ages and abilities than was previously practical. It will allow your CNC turning machine to be used as a 'everyday tool' rather than one reserved for special occasions.

Denny's Help Box

"<u>Parted-off</u> means to cut the finished work, from the remaining material with a special cutter."

3 Setting Up

Lathe CAM designer requires that your computer has:

- a hard disk,
- Windows version 3.10 or above installed,
- a minimum of 1 megabyte of memory.

Installation
Whilst running windows on your computer:

1. Insert the Lathe CAM Designer floppy disk into drive a:.

2. From program manager select *run* from the *file* menu.

3. A <u>dialogue box</u> will appear. Type in the command line box [a:install] and press [enter].

4. This will run the Denford install program. Simply pressing [enter] will install Lathe CAM

Denny's Help Box

"A <u>dialogue box</u> is a box that appears in windows and allows you to input data."

Setting Up

Designer on to your hard disk. It will also create a <u>group window</u> and place the Lathe CAM Designer <u>*icon*</u> in this group window.

Note.

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.

Denny's Help Box

"An <u>*icon*</u> is a small picture, displayed in windows, when you click twice on it the program associated with that <u>icon</u> will run."

Lathe CAM
Designers icon

"A <u>group window</u> is where a selection of icons can be stored together in the windows environment."
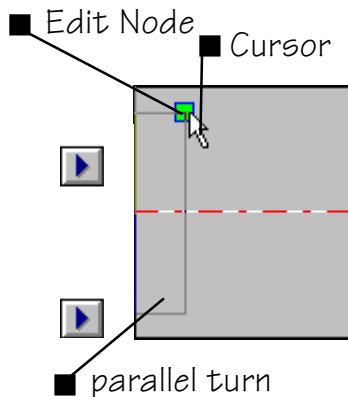
4 Quick Start

Denny's Help Box

"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 change the shape."

Edit Node

Cursor

parallel turn

This section will enable you to become familiar with the features of Lathe CAM Designer. It will allow the user to 'get going', without referring to the reference section.

1. Run Lathe CAM by on the Lathe CAM Designer icon. This will make the program run and display the start screen.

2. The start screen allows you to select the billet size, both the diameter and length. For now we will leave these as default values, so on the 'OK' button.

3. You will now be presented with the design screen. The grey area represents the side of the work billet and the dashed line is the centre of the billet. The rectangle in the work area is the first element, a parallel turn.The green square is called the _edit node_ (a green _edit node_ identifies the selected element). You can alter this first, selected element by moving the cursor over the _edit node_ pressing and holding the left mouse key and moving the mouse pointer.

Quick Start

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

| Parallel | Tapered | Convex | Concave | Thread | Drill |

Element Buttons
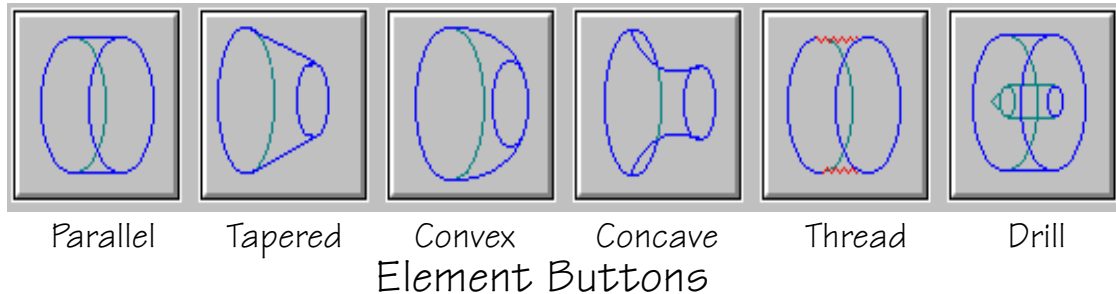
5. Each element may be highlighted by clicking on its edit node once with the . The information about each element is given in the lower window. The icon in the information window shows you what type of element it is, the other data is numerical and gives the exact location and size of the element.

Quick Start

File
- New
- Open...
- Save
- Save As...
- Print...
- Printer Setup...
- Alter Billet Size
- Exit

Create G-code
- Make file...
- Set G-code parameters...
- Set Lathe Tooling

6. Pressing [delete] on the keyboard will remove the highlighted element. To delete the whole drawing simply go to the *new* menu within the *file* menu.

7. When you have finished designing your shape, go to the <u>make file</u> section within <u>create G-code</u> menu. The code produced, is the code needed by your CNC turning machine to make your designed part.

8. The final step is to run your G- code on your machine and manufacture your part.
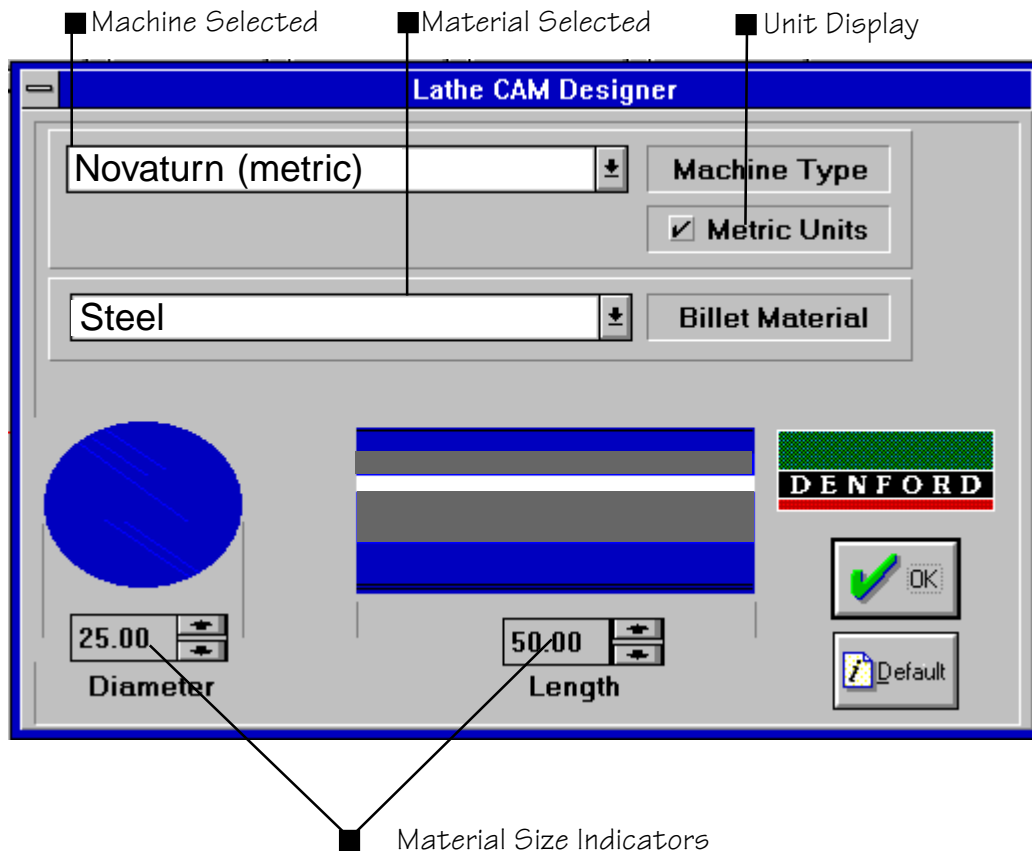
Note.

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.

5 Reference

The material size screen appears when the program is started. From this screen changes to the selected Lathe, size and type of billet can be made.

Machine Selected ■ Material Selected ■ Unit Display ■

Lathe CAM Designer

Novaturn (metric) ⬦ Machine Type

☑ Metric Units

Steel ⬦ Billet Material

DENFORD

✔ OK

📄 Default

25.00 ⬦
Diameter

50.00 ⬦
Length

Material Size Indicators

Reference

Screen Layout

Title bar

Menu bar

Element buttons

Lathe CAM Designer - [Untitled]

Information   File   Elements   Create G-code

Centre line

1.0

0.1

0.05

Diameter 1	20.000	Angle	0.00
Diameter 2	20.000	Radius	0.000
Start Pos	0.000	Drill Diameter	0.000
Length	5.000	Drill Depth	0.000
End Pos	5.000		

Nudge buttons

Selected element icon

Work area

Numerical data

Reference

Work area: This is the area where the element blocks are placed and manipulated. The area represents the diameter and length of the billet. The dashed line shows the centre line of the billet.
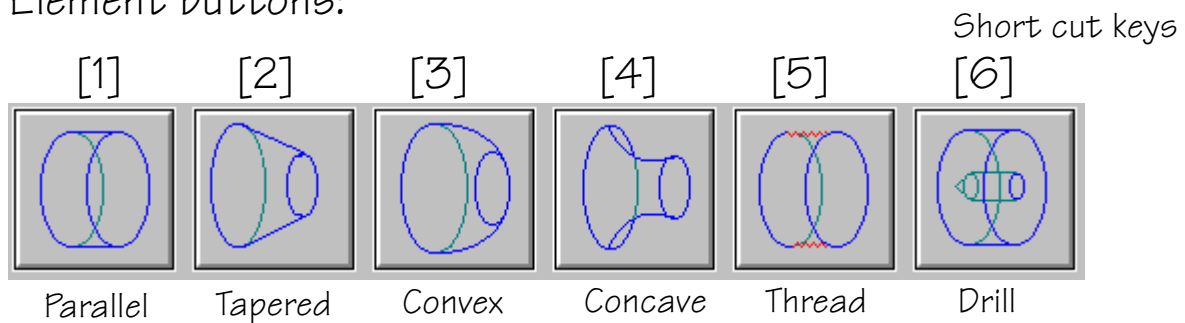
Menu bar: Select the menu required using 🖱 and then select the item by using 🖱.

Title bar   The title bar shows the name of the program 'Lathe CAM Designer' and the name of the file you are working on, if it has been saved.

Element buttons:

Short cut keys

| [1] | [2] | [3] | [4] | [5] | [6] |

| Parallel | Tapered | Convex | Concave | Thread | Drill |

There are 6 types of elements. They can be selected by; 🖱 on the button, using the short cut keys 1 to 6, or by selecting the element from the *element* menu:

Reference

Parallel: This will only allow you to draw a rectangle on the screen, representing a true cylinder with parallel sides. The diameters 1 and 2 will always be the same.

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. (See Advanced Setup)

Convex, Concave: These allow you to place radii on your work, which are restricted to 90 degree arcs.

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. (if required see -Advanced Setup)

Reference

Drill: Once you have selected drill, a dialogue box will appear. The dialogue box will allow you to set both the diameter and the depth of a drilled hole.

Drilling dialogue box
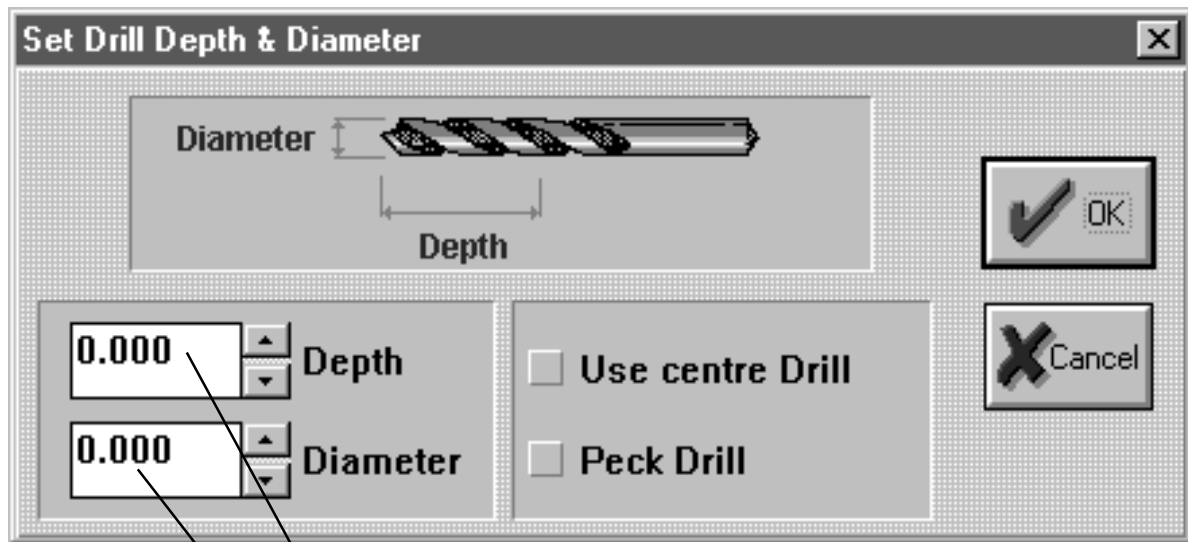
Diameter box ■   ■ Depth box

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 peck cycle periodically with draws the drill bit which helps to remove the build up of swarf

Reference

```
Diameter 1   40.000   Angle          0.00
Diameter 2   40.000   Radius         0.000
Start Pos    0.000    Drill Diameter 0.000
Length       5.000    Drill Depth    0.000
End Pos      5.000
```

Data Box

Select element icon: This icon shows what type of element is highlighted, the icon will 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:

Diameter 1: The diameter of the left hand side of the element.

Diameter 2: The diameter of the right hand side of the element.

Start Pos: The absolute position of the left hand side of the element, measured from the left hand end of the billet.

Reference

Diameter 1	40.000	Angle	0.00
Diameter 2	40.000	Radius	0.000
Start Pos	0.000	Drill Diameter	0.000
Length	5.000	Drill Depth	0.000
End Pos	5.000		

Data Box

Length: The length of the element.

End Pos: The absolute position of the right hand side of the element measured form the left hand side of the billet.

Angle: This is the angle of the slope from diameter 1 to diameter 2. If the figure is negative then diameter 1 is smaller then diameter 2. The maximum negative angle is set by a value held in the file LCD.INI.

Radius: This will only have a value for convex and concave elements, and shows the radius of the arc on the highlighted element.

Drill diameter: This displays the drill diameter. It will be the same value that was entered when the drilling dialogue box was displayed.

Drill depth: This displays the drill depth. It will be the same value that was entered when the drilling dialogue box was displayed.

Reference

Nudge buttons: These allow very high accuracy when editing the shape of elements. You may use the 🖱 on the on screen buttons or use short cut keys. The short keys use the arrow keys plus other keys to obtain different degrees of accuracy.

Short cut keys

[ARROWS]

[ARROWS] + [SHIFT]
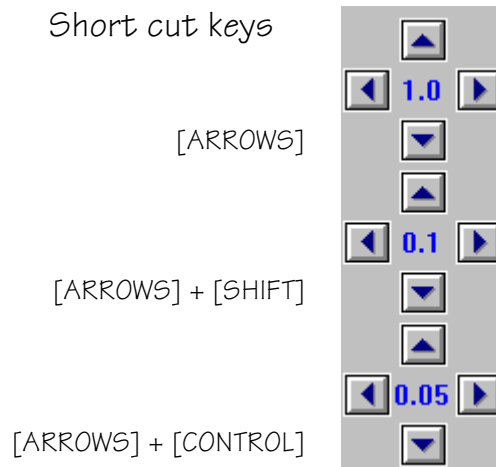
[ARROWS] + [CONTROL]

If you use the combination [ARROWS]+[CONTROL]+[SHIFT] then you can nudge 0.001.

Reference

```
Create G-code
  Make file...
  Set G-code parameters...
  Set Lathe Tooling
```
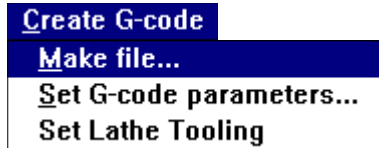
Create G-Code: This is a pull down menu that has 3 sub menu choices:

● <u>make file:</u>   Selecting this option will prompt you to enter a file name that can be used to store your G-code under.(this will later be used to control your CNC turning machine). Once you have entered a valid <u>filename</u> and 🖱 *on the 'OK' button, you see the generate G-code dialogue box, just* 🖱 *on the 'Create G-code' button. If you wish part off your component then you should first* 🖱 *on the 'part off component' tick box. If you wish to face off the component then* 🖱 *on the 'face off bar' tick box.*

```
c:\denford\data\example.fnl          [G create code]

  ☐ Part Off Component              [X Cancel]
  ☐ Face Off Bar
  ☐ Use Coolant
```
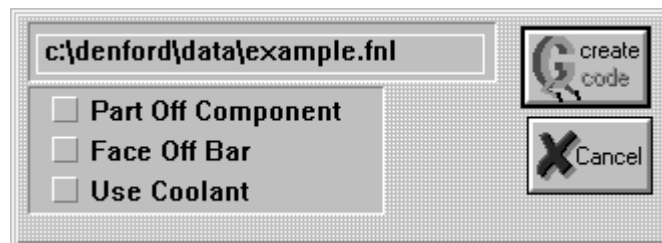
Reference

● Set G-Code parameters: This menu will select a dialogue box that will allow you to choose what sort of material you will be machining. You can select a material which will load cutting parameters appropriate for the material. These parameters can also be altered individually. These are feed rate, finishing cut, roughing cut, threading cut and the surface cutting speed. For more information see Advanced setup.
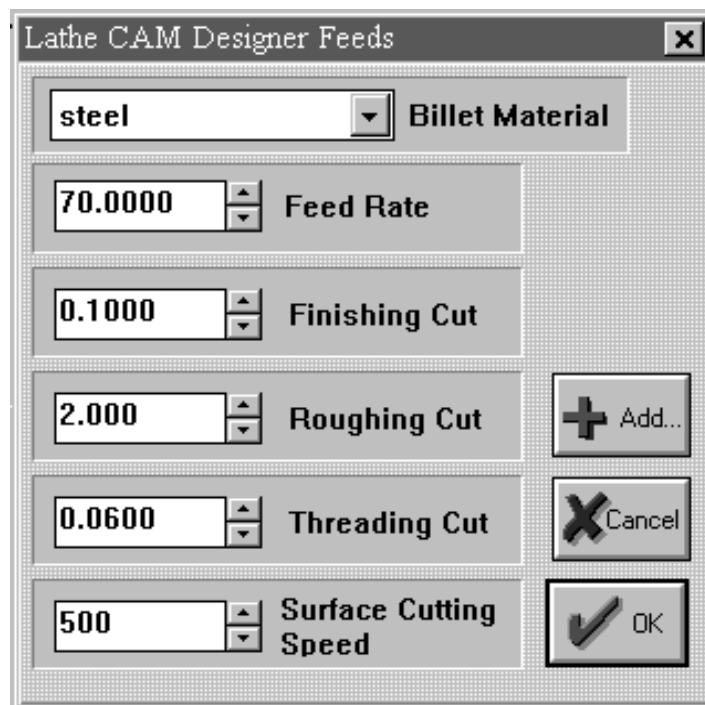
Reference

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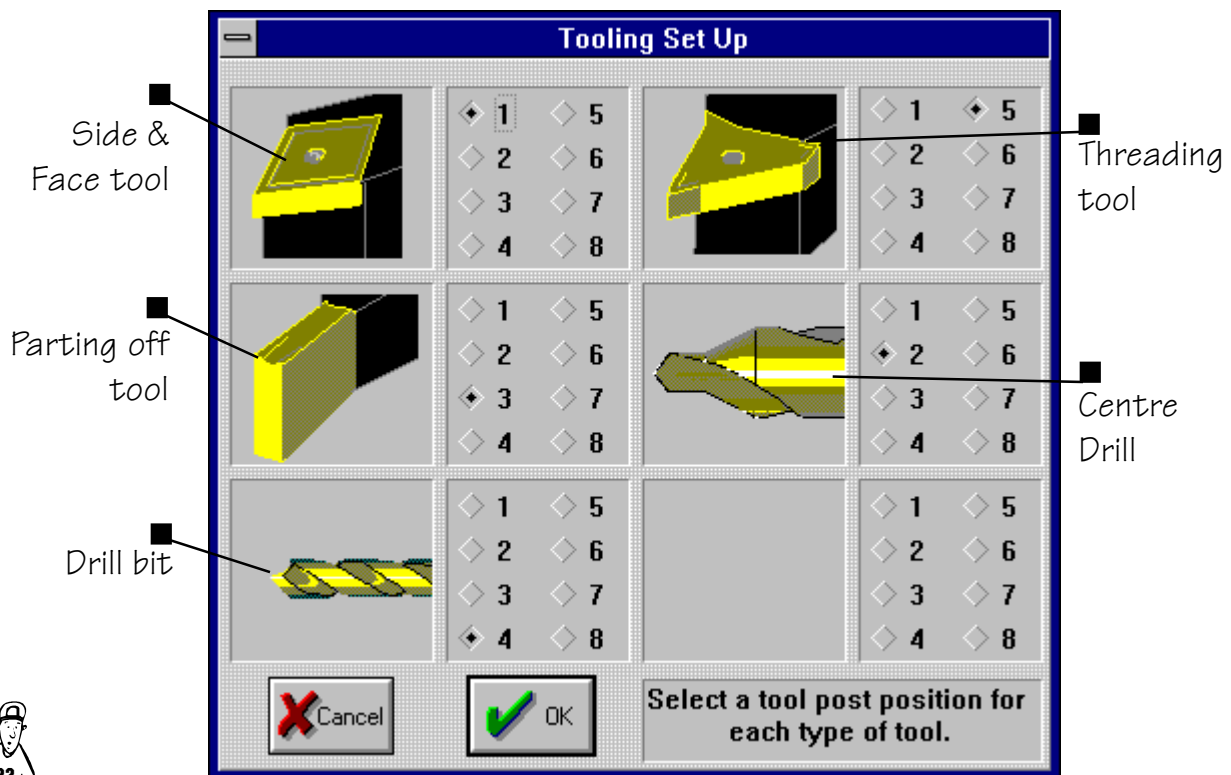then tool 1 must be a roughing tool. This information is needed by the software so that when a tool change is required it can enter the correct number in the G-Code file.

Tooling Set Up

Side & Face tool

Parting off tool

Drill bit

Threading tool

Centre Drill

Select a tool post position for each type of tool.

Cancel   OK

Reference
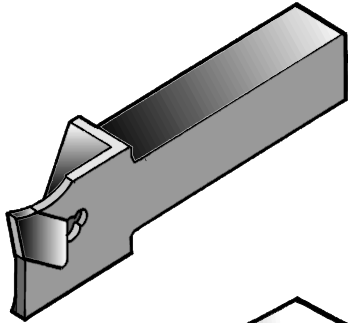
Parting tool- used mainly to remove work from the end of the bilet, but on occasion it is used to make small undercuts
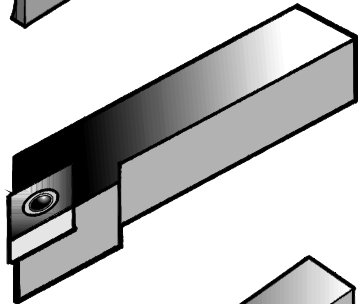
Roughing / Cutting tool - This is a right handed cutting tool. It is used on the outside of the billet and carries out the majority of the work.
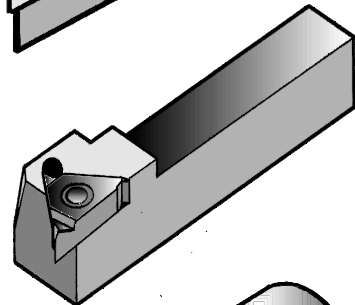
Threading tool - The threading tool has only one function due to its sharp steeply angled cutting edge. This tool is able to create many different depths of thread.
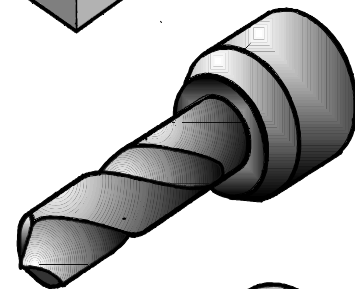
Drill - The drill is used to make holes in the centre of the billet. The drill bits are available in a variety of lengths and widths. On a lathe the drill bit remains stationary as the billet revolves.
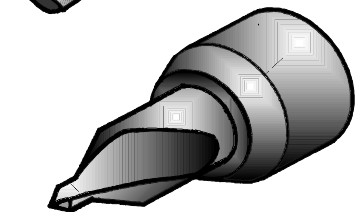
Centre Drill - This is a small rigid drill bit that has the function of making a small pilot hole. This guides the main drill bit to the centre of the billet.

6 Program Structure

Lathe CAM Designer uses and creates several files while it is running. The diagram below shows the position and function of these files.

Position and purpose of the main Lathe CAM Designer files

Denford Directory

MACHDATA.INI
Holds the path data for various types of lathes

LC-DES.EXE
Main Executable file

GENERIC.GNC
Generic G-Code file, created by Lathel CAM for use by DENLANG.DLL

DENLANG.DLL
Post pocessor program that generates G-Code

Config Directory

FANUCL.INI
Individual settings for different lathes

Libs Directory

FANUCL.SLB
Information for creating G-Code specific to CNC lathe

Windows Directory

LCD.INI
Start up values for main program.

7 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 file LCD.ini. This can be found in your windows directory. The following is an example of the file and a description of the contents.

[Settings]
diameter=25   -Most recently used material diameter.
length=50   -Most recently used material length.
material type=Steel   -Most recently used material type.
Machine=Starturn PC (metric)   -Most recently used machine.
Graphics Speed=1   -The speed at which the G-codes are generated
Parting Feed rate %=50   -The feed rate at which the parting tool advances
                                       (This is a percentage of the cutting tool feed rate)

[Path data]
Gnc name=C:\denford\data\GENERIC.GNC -A temporary file generated by Lathe Cam
Lcd files=c:\denford\data\   -Lathe Cam Designer files directory
g-code files=c:\denford\data\   -G-code files directory

[Materials]
Steel=120
Steel English=120
Brass=100
Plastic=300
Wax=500
Aluminium=170

These are the maximum cutting speeds for specific materials. The cutting speed is variable dependant on material type and the diameter of the billet

Advanced Program Setup

[Feed Rates]
Steel=70.00
Brass=140
Plastic=250
Steel English=2.3
Wax=300.00
Aluminium=140.00

Feed rates are how fast the tool moves through the material in mm per minute.

[Finish cut]
Steel=0.1000
Steel English=0.0200
Brass=0.5
Plastic=0.5
Wax=0.5000
Aluminium=0.5000

Finish cut , this is the amount of material left before the finishing cut

[Roughing cut increment]
Steel=2.0000
Steel English=0.08
Brass=3
Plastic=4.00
Wax=4.0000
Aluminium=3.0000

These values are the maximum amounts taken out in any roughing cut .

[Thread cut increment]
Steel=0.06
Steel English=0.08
Brass=0.06
Plastic=0.0.06
Wax=0.0880
Aluminium=0.0600

The maximum amount the tool removes on any single pass.

Advanced Program Setup

[Thread]
pull out angle=60   -Angle at which threading tool leaves billet
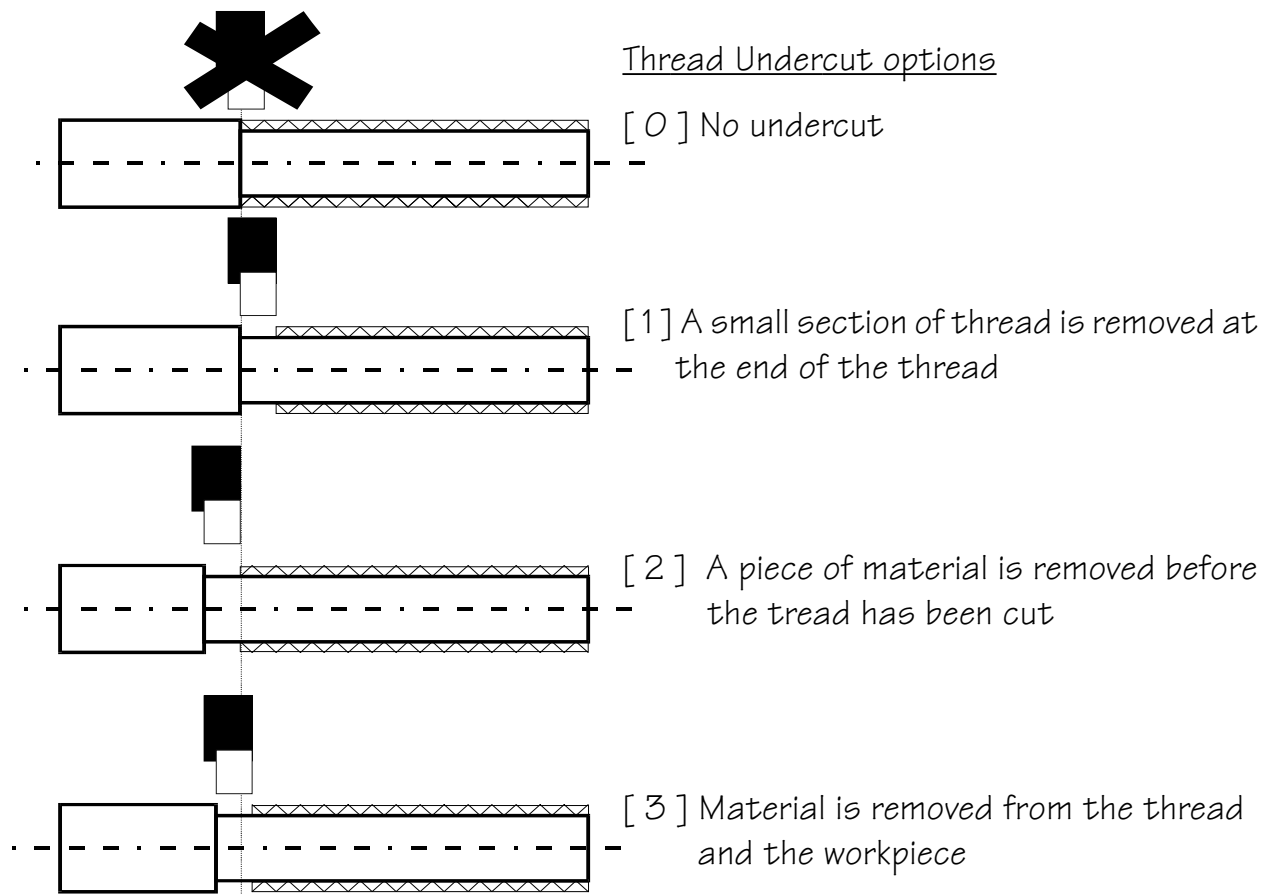Thread angle=60   -Pitch of the thread
Finish allowance=0.03 -Material left before the final cut
Thread Undercut=2   -Specifies which thread undercut you require

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 tread has been cut

[3] Material is removed from 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 [settings] e.g. Thread Undercut=1

8 Glossary of Terms

Arc	*Portion of a circle.*
Centre Drill	*A rigid drill bit used to make a hole which will guide the main drill.*
CNC	*(Computer Numeric Control). A system or machine that is controlled by a computer*
Default Settings	*Restore the machines initial settings.*
Floppy Disk	*A storage device for a computer. It can be removed and used to transport data.*
Face off Bar	*Takes rough edges off the end of the billet.*
Hard Disk	*A storage device for a computer. It cannot normally be removed from the computer, it is very fast and can hold a large amount of data.*
Memory	*The place where the computer stores data internally.*
Part off Component	*Cuts work area from the billet.*
Roughing cut	*Centres billet and removes the bulk of the material to make the approximate profile of the design, leaving enough material for the finishing cut.*