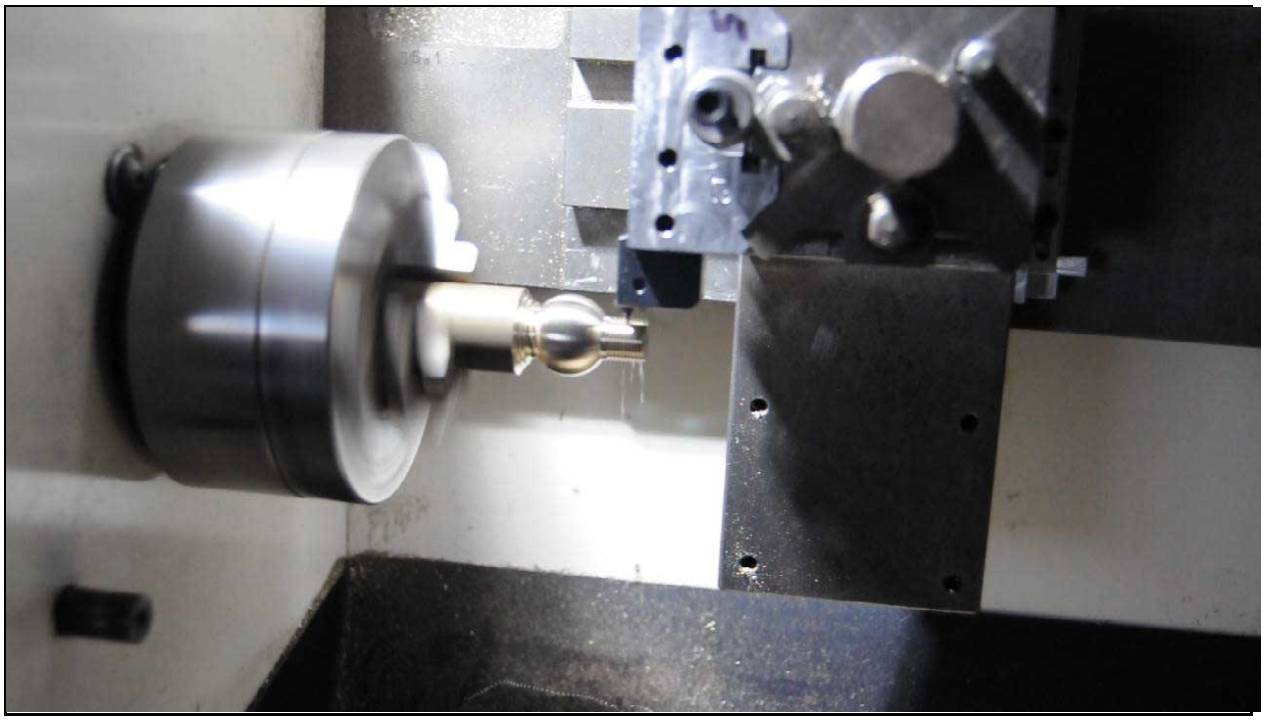
EziLathe USER MANUAL

Tools for the CNC Lathe



Version 1.6 June 2017

Table of contents

Introduction

Ezilathe (Tools for the CNC Lathe) is a program, designed to ease the production and debugging of G-code programs for the CNC lathe. It is written by a user of a CNC Lathe running Mach3 (Copyright Artsoft) for users who are similarly set-up, who have found good support for lathe work to be a bit "Thin on the ground". Code is produced that is both compact and "Commented", and as such can be easily fine tuned by hand if required.

Safety Emphasized

The Ezilathe program is used to produce and/or optimize G-code programs for use on CNC lathes. Potential dangers exist for both personnel and machine tools, if the work is not carried out in a safe manner. No guarantee is given that G-code generated is safe to run on any CNC machine, It must be checked carefully before use, and then initially by “Dry Run – without tooling” before any metal is cut. Speeds and Feeds, Excessive Gouging, and collisions especially in rapid feed are all potential dangers in the use of CNC machinery.

This Version of Help / Manual is Correct for Ezilathe Version 1.6.0.0

Welcome

As a user of CNC equipment in the pursuit of my hobby (Model Engineering), I am obviously into "Shortcuts" to eliminate the mundane work, and concentrate on the more interesting jobs. Even in programming a CNC Lathe, the mundane tasks creep in, it quickly becomes apparent that a higher level of "Automation" would be welcome in the production of programs.

Fully featured "CAM" systems are easily obtainable for the mill etc, but few exist for the lathe. Low cost packages like the "Vectric" programs are easily justified to support a mill, however for the hobby lathe, even that level of expense is difficult to justify. For the lathe, the use of "Wizards" can get you some of the way (especially with a bit of cutting / pasting), but still leaves many places where hand coding / calculation is required.

With the availability of no or low cost cad systems as "Graphical Calculators", it seems only logical to use this data to assist in the production of CNC programs as efficiently as possible. It can sometimes be difficult to get access to the data required, often just picking the lines etc and handwriting the data on paper drawings ! The first use for "EziLathe", was to automate the listing of all the useful data to paper copy for transfer to CNC. It has grown a bit since then, as other repetitive tasks come to hand.

I fully intend to continue to develop / debug this program for at least my own use. Other users are welcome, and are invited to provide feedback on bugs , suggestions, etc.

What's New (Version 1.6)

The usual bug fixes, and some additional error messages / warnings added. Additions to aid in compatibility with a greater number of CNC systems. DXF processing now has two additional operating modes that have radius compensation with (as yet) a limited number of tool configurations. The new modes are Trepanning, and Grooving, both with two operating modes to suit tool / work piece.

Overview

The "Ezilathe" program has a number of tools built in to assist in producing / modifying G-code for Mach3 Lathe and simulating the result :-

- **G-code Editor.** This is a simple text editor where programs can be loaded or produced with the tools supplied and "put together". It has a number of arithmetic functions to make global or local changes in the G-code, as is often needed. The functions can be used to convert (metric/imperial, diameter/radius) or modify (Change lengths/diameters keeping rough cuts in step, to produce a new, but similar component). As these functions are also useful for milling, a "Y" axis is included for milling support.

- **Favorites List.** A place where your favorite routines may be held, so they can be copied/pasted into the program editor and re-used as required. If you find as I do that certain fragments of G-code are usefully re-used, keep a copy here, for easy recall.

- **Simulator.** Displays a high resolution graphical representation of the G-code program. This simulator is linked into the editor, avoiding searching for the code that produces the feature in question. An inbuilt Tool Library is linked to the simulator to enable visualization of the finished part as produced with the tool profiles and see the effects of “Gouging” of the workpiece. The simulator allows an overlay of DXF data over the simulated finished part to confirm accuracy of the G-code program. The Step Through Simulator Mode allows stepping through the program, a line at a time to see on screen each cut as it progresses.

- **DXF Processor.** This enables retrieval of entity coordinate data from Cad via DXF files. Up to 400 Entities may be processed into G code. A graphical representation as well as a complete printout of Co-ordinate data is available. DXF data may also be used to produce the G-code program or fragment directly with little or no additional work required via the in-built DXF - G code processor. Code is generated that follows the DXF profile and generates all roughing cuts as required. Some modes also include Radius compensation.

- **Speeds and Feeds Calculator.** Is included to assist in the correct use of tooling. An estimate of power requirements is available to further assist in optimizing the operation of the CNC lathe.

- **Thread Calculator.** used to calculate the cutting parameters (with radius or sharp tools) and G76 / G32 Threading Calculator, producing code fragments, that can quickly and easily be integrated into the program currently in the editor.

- **Cad 4 Lathe.** A Simple Cad window designed to produce basic drawings that can then be processed as DXF files with the DXF Processor. It is intended for simple components only, such as those that are usually programmed via the "Wizards". Use this when opening your cad program accounts for more than 50% of the drawing time. **New - allows export to Cad systems via DXF.**

This Function is the latest addition to Ezilathe, and is still being developed to a full working state, and some features are currently under review.

Getting Started

"EziLathe" is a stand alone program, that does not require any unusual prerequisites or an installation program, simply extract the zip file to the working directory, copy the required data files from the sample directories (Imperial or Metric) if required and away you go.

Working Directory = C:\Ezilathe (There are internal pointers to this directory, for the placement of data-files etc).

The zip file (Ezilathe.zip) contains the following main files :-

Main Executable	= Ezilathe.exe
Windows Help File	= Ezilathe.chm

Two sample directories are included that include sample data files as listed below for a Metric or Imperial setup.
The Material Properties file (basic version) is included in both sample directories if required.
For Initial installation, it is recommended that the total contents of the appropriate sample directory be copied to the working directory.
(This has to be done manually to avoid overwriting your working data files on future updates).

A PDF Manual is also available if required (Ezilathe.pdf)

The program will run using default values, that should be verified, before serious usage. The program options may be changed via the main menu (Main Menu > Options).
The current options are discussed in Program Modes.

With use, the program updates a number of data-files that are created automatically if missing :-

Favorites.rtf	= Default File for Favorites. (Saved on Operator Request)
Lathes.txt	= Data on up to 4 lathes in Speeds / Feeds (Updates on Program Exit)
Lathetdat.txt	= Data on up to 99 Lathe Tools (This data is saved on operator request)
Ezilathe.cfg	= Options and Editor settings (Updates on program exit)
Speeds.txt	= Editor settings for Speeds / Feeds (Updates on Program Exit)
Threads.cfg	= Editor settings for Threading (Updates on Program Exit)

Most of the data files (the ones that update on program exit) are just saving your last inputs in the various controls around the place. However, to get full value out of the Simulator and CAM processor, details of your lathe and lathe tools will need to be entered early on.

It is recommended that archive copies of at least the important files are kept, just in case. Recreating the lathe tool data or the favorites can be time consuming as these will get quite large with use. Note that all data files are simple text files, so can be viewed with Notepad (or by EziLathe) or any text editor.

International Settings
There are two data files that use CSV format (Lathetdat.txt and Materialm.txt). The default comma separator is not suitable where a comma is used as the Decimal Separator. Whereas Windows settings cover most areas, Before running program for the first time, the two datafiles need the commas replaced with the new field separator, and the 4th line in Ezilathe.cfg needs to be changed from "Z" to "SEP = " + new field separator i.e. "SEP =;" Please advise any issues, as not been tested here.

"EziLathe" can if required, be the G code editor used under MACH3. Simply bring up the menu item Config > General Config in Mach3 and change the editor listed by browsing to C:\Ezilathe\Ezilathe.exe

System requirements

"EziLathe's" minimum system configuration includes :-

- Windows XP, Windows 7 or Windows 8, 8.1 & 10 - 32 or 64 bit in all systems listed
- Ram 512Mb for XP, 1Gb for 7,8,10
- 40Mb of free disk space
- 1024 x 768 screen resolution or higher.

Getting help

Context sensitive off-line help is available via windows std help interface Press "F1" or any help buttons.

A manual in PDF format is also available (Ezilathe*.pdf), that can be printed if required (same as help file, but in more convenient format).

Conventions

The Lathe has a "Z" Axis (Bed) and an "X" Axis (Cross slide).
The Graphic Screen (and Cad systems) have an "X" Axis (Left to Right) and a "Y" Axis (Up / Down).

To avoid confusion, I have labeled things this way.
So the Screen "X" is the Lathe "Z" and the Screen "Y" is the Lathe "X"
At least it's a different state of confusion.

"***Value to Process"*** - Labels on program features are shown In this text format.

"New V1.6" - Highlights New Features.
"Mod V1.6" - Highlights Modified Feature.

Terminology Used in this Help File or User Manual.

Full Program - A complete G-code program including initialization of active G codes and Program exit code. When generated, the program will replace the contents of the editor. It is expected that this program can be run (as is) with or without additional code added.

Program Fragment - A section of code to produce a feature. Usually generated from Tool Change to return to Tool Change Position. When generated, the fragment is **Added** to the editor contents at the end of the file, from where it can be re-located, if required, forming one part of the program.

Material Properties

The materials data file is basically just the Mach3 Mill data file "Metricated and Extended" to suit "EziLathe".

Material Name,Hss,HSStin,Carbide,CarbideTin,Kp
Aluminum,152,198,259,305,0.9
Soft Steel,29,40,76,76,1.2
Medium Steel,23,35,66,84,2.1
Hard Steel,8,20,38,66,3
Brass,70,99,168,212,0.82
Bronze,61,84,137,168,1.5
Soft Cast Iron,28,34,69,82,0.96
Hard Cast Iron,8,20,40,60,1.94
Other,30,30,30,30,2

All surface speed data is in Meters / Min. Data is all checked as reasonable from the "Machinery Handbook".

Kp Field is a factor from the "Machinery Handbook" for cutting power calculations. Note that this power figure is at the tool tip, Motor power can be twice this depending on efficiency of lathe transmission / coolant use etc, but should provide a useful guide once you get experience with it.

This file can be edited / extended using any text editor, but note the format must remain intact (not a comma out of place).

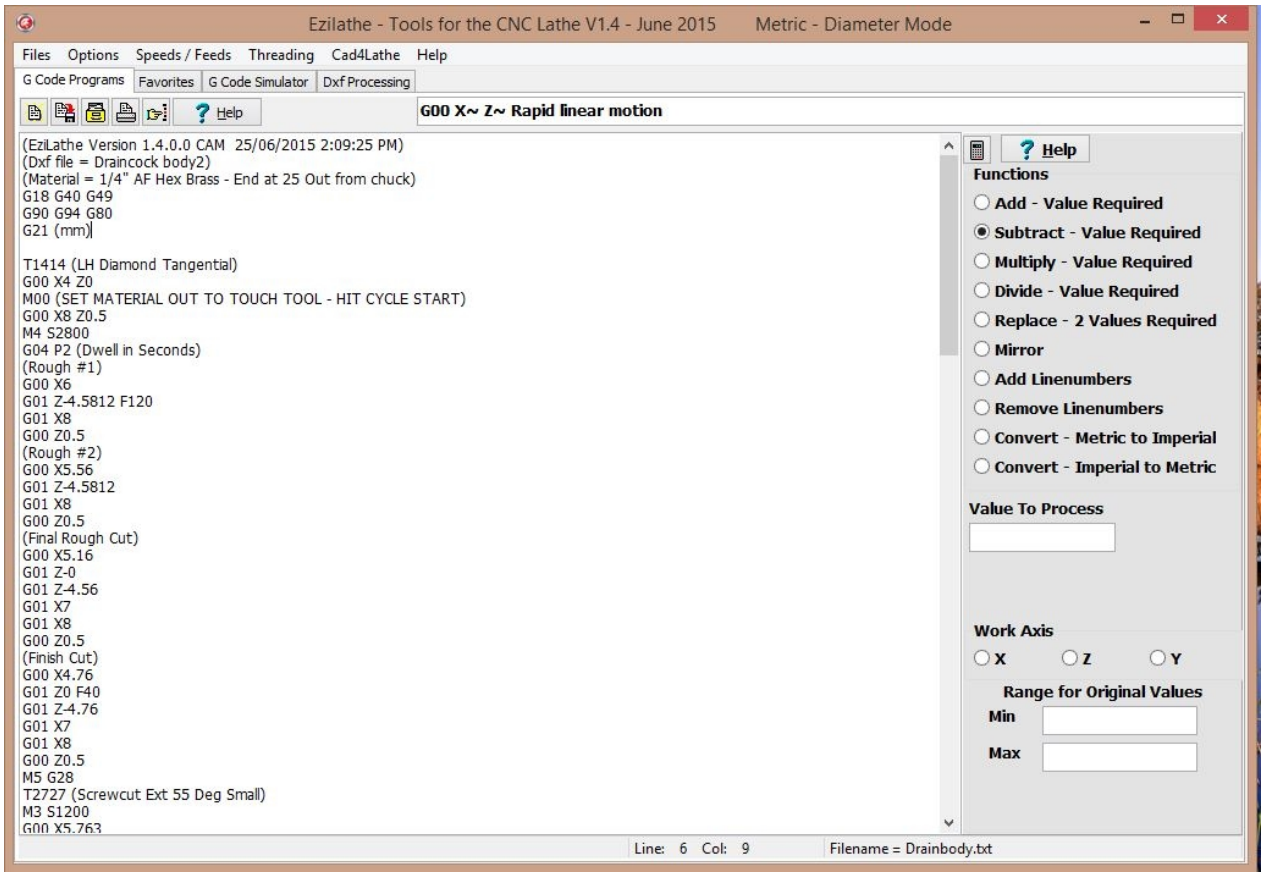
Only the One version of the data file is used for both metric and Imperial Modes.

Description

EziLathe - Tools for the CNC Lathe is a collection of useful routines brought together, so they can share data and resources.

These routines have developed over time in use by me for my own work. This development has predominantly been in Metric and Diameter mode. As a result, the program thinks and acts in Metric. The Imperial mode is purely cosmetic, and as such may contain as yet unnoticed issues (I hope not). Please be aware that 99% of the program is not effected by this mode, seeing only units of length. Speeds/Feeds and Threading are really the only areas effected.

User Interface

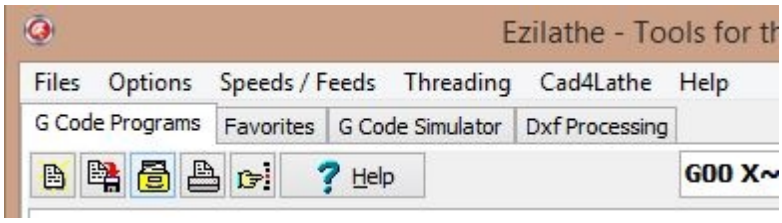


G-Code Editor and Functions Bar

"EziLathe" is a typical Windows program using the Following Elements (From Top to Bottom)

1. Title bar showing Version number and current operating modes.
2. Main Menu - A number of Global options are available here, as the headings for sub-menus or stand alone menu Items.
3. Page Control - 4 Selectable page tabs that lead to the main program functions.
4. Working Area - The Main part of the screen, may be a text editor or a graphical display, with or without an additional Tool-bar, Page control or Side bar.
5. Status bar - A Collection of information panels. Always dedicated to the current Page.

Main Menu



The Main Menu consists of 5 Items :-

- **Files** Menu Header - Three fields
    1. **Exit** = Close Program.
    2. **New Gcode** = Create a new blank G code program in Editor (Use when not using DXF processor).
    3. **Recent** = Opens Recent G-code Files dialog, to enable selection of recently used files.

- **Options** Header - Three Check-boxes.
    1. **Radius Mode** - Checked = Radius Mode. Un-checked = Diameter Mode.
    2. **Metric** - Checked = Metric. Un-checked = Imperial.
    3. **"New V1.6" Reverse Arc Generation** - Checked = Reverses Arcs in DXF processor. Un-checked = Generate Std Arcs.

- **Speeds / Feeds** - Opens the Speeds / Feeds Dialog Box. (Also includes Lathe Parameters)

- **Threading** - Opens the Threading Dialog Box.

- **Cad4Lathe** - Opens the Drawing Editor

- **Help** Menu Header.
    1. **Help** - Open Windows Help (This File)
    2. **About** - Opens About Dialog Box.

Program Modes

Program modes may be changed via the Main Menu. - **Options**.

Radius Mode

All "X" axis co-ordinates are specified as a Radius. E.G. X6.00 refers to a point at 6 Radius (12 Diameter). This should Match the mode set in Mach3.

Diameter Mode

All "X" axis co-ordinates are specified as a Diameter. E.G. X6.00 refers to a point at 3 Radius (6 Diameter). This should Match the mode set in Mach3.

Note :-

- DXF Data is ALWAYS in effect Radius mode (As displayed in graphic), and will convert as required for code generation. A Line at X (Y) 6.00 in the DXF graphic will produce the G-code X6 in Radius mode and X12 in Diameter Mode.
- A drill entered as 12 Diameter in the Tool editor is still that in both modes i.e. 12 diameter / 6 radius.
- A Workpiece entered as 12 Diameter in the Speeds/Feeds or a workpiece entered as 12 Diameter is also that in both modes.

Metric or Imperial Modes

For the most part, Metric or Imperial does not matter, as the program works essentially in Units. However ! It is Strongly recommended you stick with one only.

Note :-

- Tool editor works in units. A 4mm drill is very different to a 4 inch drill (Would look strange in the simulator). All sizes would need to change if this mode is changed.
- Feeds/Speeds inputs work in the ruling units and are taken as M.M. or Inches as appropriate. A 12mm workpiece will produce very different results to a 12 inch workpiece.
- Default settings under the DXF system will also need changing manually (60 feed looks good in mm / min, but a bit fast in inches / min.
- Dxf system works in Units. If required Scale up/down in CAD system first.
- G20/G21 have no effect in "Ezilathe", these are only useful on the machine itself.

If you have a program that does not match your settings, all the above may be adjusted easily in the editor using the functions (on the sidebar).

Reverse Arc Generation

If "Ezilathe" generates arcs that go the wrong way for your lathe (For whatever reason), "Ezilathe" can be made to produce (and Simulate) reverse arcs to suit by checking this check.box.

G code Editor

The G code editor is a typical basic text editor under Windows, with the required editing functions controlled via Speedbuttons or a Pop-up Menu.
G code can be loaded from disk or generated via the Dxf system, or typed in from scratch. Once G-code is here, it can be simulated and adjusted as required.

"New V1.6" Equations (As per Mach3 equations) may be entered manually. This can be useful in some programs to account for variables as required (such as tool width etc). Parameters (also as per Mach3) and controls such as "Block Delete". Equations can be checked via the pop-up menu.

It is best to save often as the program grows, to avoid having to rework it, if things go wrong.

The additional functions of this editor are available to allow quick and efficient adjustment of existing CNC programs while keeping all cuts in step.

File Open Button

Opens a G code Program in the Editor. The file is opened via a standard file open dialog box.
By default file selection is filtered to Text Files (.txt), Unless overridden in **"G code File Extension"** (See under Lathe Details)

File Save Button

Saves the G code Program in the Editor. The file is saved via a standard file save dialog box, to give the option to change the file name.
By default file selection is filtered to Text Files (.txt), Unless overridden in **"G code File Extension"** (See under Lathe Details)

The default extension is added where no .ext is entered

Recent Files Button

Opens the Recent Files Dialog box to re-load a G code file from the last 10 files accessed.
The dialog box is context sensitive, displaying only the relevant file types for the current page.

Print Button

Prints the G code Program in the Editor. The file is printed via a standard printer dialog box, that gives the capability to select a printer and modify its printing options.
By default, the file is printed to the default system printer.

Text Search Button

Opens a standard Text Search Dialog, to allow searching of the contents of the Program editor.
This is useful for manually adjusting a value. eg Search for "F" to manually adjust the feeds set.

The text to be searched for is selected and the cursor positioned. Click on the attached value to change, avoiding the search text.

Please note - Searches are conducted in the forward direction only. the direction check box has no effect (Don't blame me !).

G-code Prompt Box

A Pull-down list of G-code commands is also located on the tool-bar, for quick reference. This is a quick reference only to help when reviewing a G-code program.

Pop-up Menu

Select Items in Editor then Right-Click in Editor to Activate the Pop-up Menu. Left-click on menu to select action.

Menu Items are :-

- ***"Cut"*** - Cut Selected Item.
- ***"Copy"*** - Copy Selected Item.
- ***"Paste"*** - Paste to cursor position.
- ***"Select All"*** - Selects All text in the Editor.
- ***"Delete"*** - Delete Selected Text
- **"New V1.6" "Evaluate Equation"** - Evaluates an equation if Highlighted

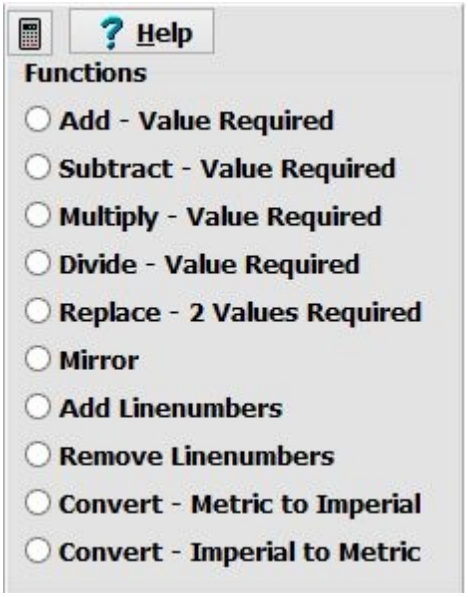
Warning ! If nothing is selected in the editor, then all is assumed to be selected.
For *"**Delete"*** this is a quick way to clear the editor.

Functions

The Functions Box Consists of a number of Available Options, with an Execute button above. Simply Select the required function, Fill in the values required in the boxes below, Select the section of code in the editor (if required) and press the Execute button.

Note ! If No code selected, then function is applied to all code in the editor.

Input boxes below the functions are available as appropriate for the function selected.



For many functions, an Axis will be required. This box will only be visible when an Axis is required.
A "Y" axis is also included, as this area of the program is also useful for Milling Machine operations.

Add, Subtract Value

These functions typically are used to shift, extend or reduce a feature keeping values in step (Finish and rough cuts)

Requirements

Must have numeric value in "***Value to Process"***

Must have an Axis selected in ***"Work Axis"***

Optional

Specify a range of values in "***Range for Original Values"***

Selection

Restricted to selected text range only, unless no text selected (in which case, whole file is scanned).

Multiply, Divide Value

These functions typically are used to scale up or scale down a feature.
Often used on "X" axis to convert between diameter and radius modes.

Requirements

Must have numeric value in "***Value to Process"***

Must have an Axis selected in "***Work Axis"***

Optional

Specify a range of values in "***Range for Original Values"***

Selection

Restricted to selected text range only, unless no text selected (in which case, whole file is scanned).

Replace

This function typically is used to change a specific value, such as distance from final rough cut to finished size.

Requirements

Must have numeric value in "***Replace Old Value"***

Must have numeric value in ***"With New Value"***

Must have an Axis selected in "***Work Axis"***

Optional

None

Selection

Restricted to selected text range only, unless no text selected (in which case, whole file is scanned).

Mirror

This function typically is used to turn the 'Job' around in "Z", often in conjunction with a shift (Add or Subtract).

The Mirror line is always through zero.

Requirements

Must have an Axis selected in "***Work Axis"***

Optional

None

Selection

Restricted to selected text range only, unless no text selected (in which case, whole file is scanned).

Add, Remove Line numbers

These functions add or Remove Line-numbers.

Requirements

None

Optional

None

Selection

Restricted to selected text range only, unless no text selected (in which case, whole file is scanned).

Convert Metric or Imperial

These functions Convert a G code Program Metric to Imperial or Imperial to Metric.

Note that these functions are sensitive to the G codes in play, and should not effect integer values.
Where G20 or G21 are included in the file, they are also changed as required.

Requirements

None

Optional

None

Selection

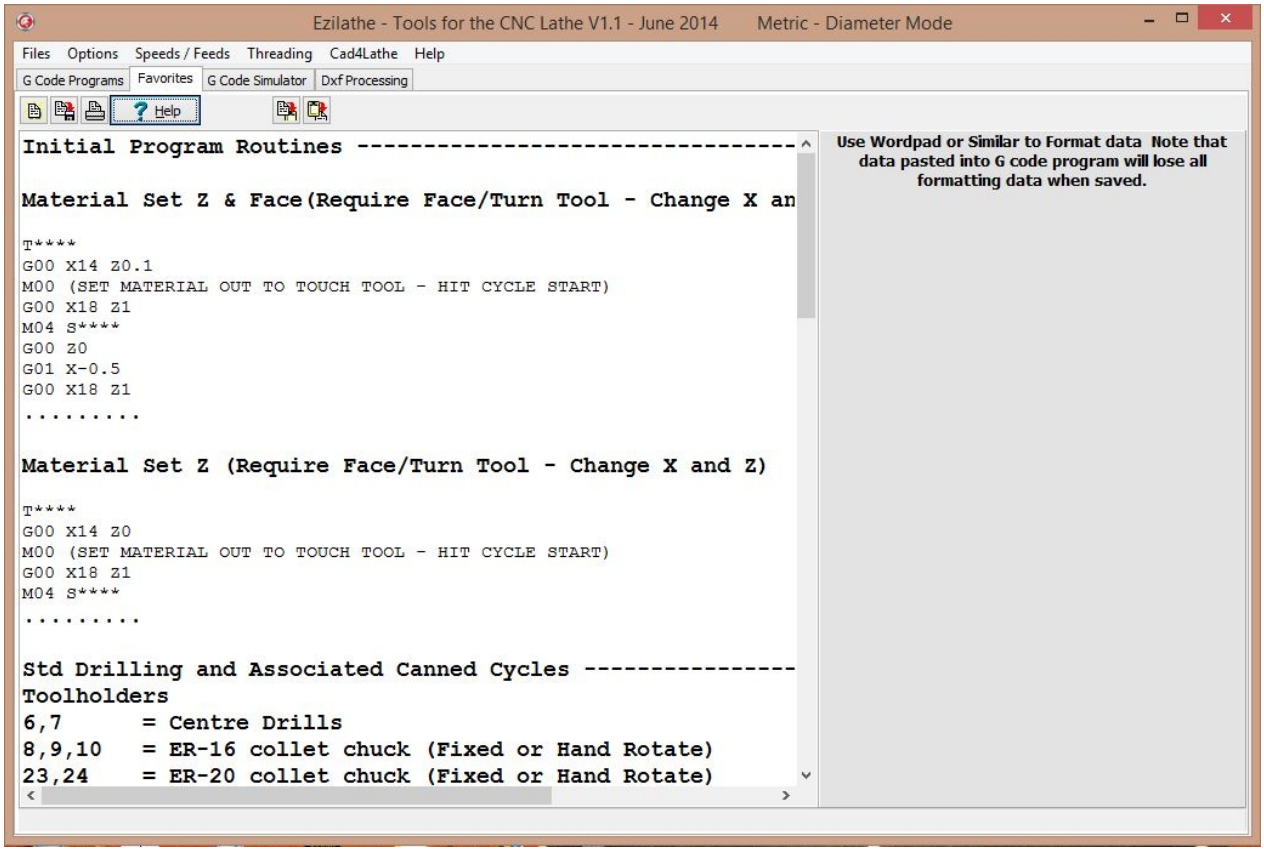
Whole file is converted, whether or not anything is selected.

Execute

Execute the selected function on the editor contents.

Note ! If no Text is selected, then the function is applied to all the editor contents.

Favorites page



This is essentially a Rich-Text editor, of the simplest kind. Start your file in Wordpad or similar, to make it look the way you want, and then just Copy/Paste away. How you use this is entirely up to you. For my part, I just store Code fragments (Blocks from tool change to tool change), some need virtually no changes to be reused.

Simply copy from here and paste to the editor. As new useful fragments are produced, copy/paste to Favorites and add a title. Please note that all text formatting is lost in data that is pasted into the editor (Simple-Text only, as is MACH3).

The File Favorites.rtf is the default, loading automatically on program start. You can have as many favorites files as you like, but stick with .rtf as this is the default extension in the file dialog boxes.

File Open Button

Opens a favorites file in the Editor. The file is opened via a standard file open dialog box.
By default file selection is filtered to Rich-Text Files (.RTF).

File Save Button

Saves the Favorites file in the Editor. The file is saved via a standard file save dialog box, to give the option to change the file name.
By default file selection is filtered to Rich-Text Files (.RTF).

The default extension = .rtf for file names. (where none entered)

Print Button

Prints the Favorites file in the Editor. The file is printed via a standard printer dialog box, that gives the capability to select a printer and modify it's printing options.
By default, the file is printed to the default system printer.

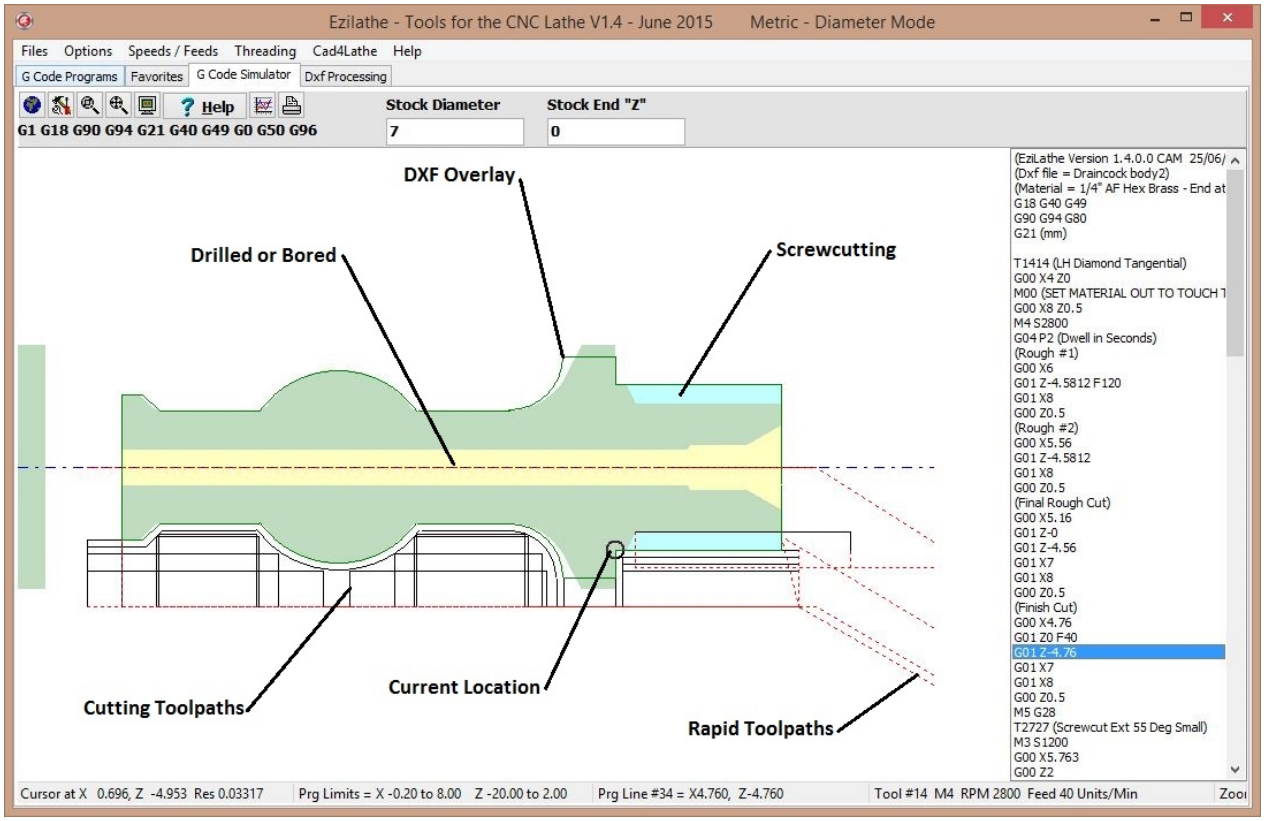
Copy Button

Copies the selected text to the clipboard (Duplicated by ***"Copy"*** in Pop-up Menu)

Paste Button

Pastes the contents of the clipboard to the Favorites editor (Duplicated by ***"Paste"*** in Pop-up Menu)

G code Simulator



The Simulator is available to quickly check the results of edits in the G-code editor and compare DXF data (Where available) to resulting part program. Two modes are available View mode (as above, the initial mode) and Simulate Mode (as described under Simulate mode).

In View Mode, the code may be stepped through line by line in the Program List-box (Right hand side of screen). A Circle highlights the current point in the graphic (graphic always displays the finished job, complete with tool paths), and status of program is displayed at top of screen and within the Status-bar. The Highlighted line in the List-box is also highlighted in the program editor, to assist in changing the code. You can move within the program list-box, using mouse or arrow keys (list-box must be the currently active control). The point of interest can also be picked on the screen (At a Start or End point) to synchronize the program list-box.

The simulator dis-regards some commands (that would have a visual effect) that are rare or unusual in lathe work. They may be added later if required.
Commands such as :-

- Nose radius Compensation (G40 - G42).
- Scaling and Coordinate systems (G50 - G59).

Where available, the DXF profile may be overlaid on graphic, as a further check of accuracy (as shown above).

Tool Display

The Tool may not be Displayed, but you can see where they have been !

One of the features of the simulator is portray the effect of tool shape on the workpiece. The graphic above shows clearly the deviation from programmed tool-path and the finished size of the workpiece. The white area between the tool-path (in black) and the finished workpiece (in green) represents "Gouging". Caused in

this example by using a 55 deg Diamond shaped carbide tip with 0.8mm (0.032") radius, in a straight tool. You may wish to compensate in your dxf, Fiddle the program, change tool or put-up with it (in this case, I quite like it).
To prove the point, simply disable the tool in the Tool editor, and regenerate graphic (a Zoom will do) to see it cut with a zero width sharp tool (wish I had one).

This is the same tool used, if you do not have the tool defined in the Tool Editor.

Load Button

Loads and Processes the data in the G code editor. If any changes have been made in the editor, the data must be reloaded.

Will cancel simulator mode, and return to View mode.

Open Toolform Button

Opens the Lathe Tool Editor Dialog box. Open Tool Form.

Zoom Window Button

Zoom Window.
1. Pick first point on screen to define first corner of rectangular window.
2. Move the mouse to the diagonally opposite corner of the rectangular window, a rectangular box will stretch out with mouse movement.
3. Pick second point on screen to define the second corner of the rectangular window.
4. Move the mouse to move the rectangular window if required.
5. Pick on screen again to execute the zoom.

Note ! To back-up 1 step, right-click on screen.

Not Available in Simulator Mode.

Zoom Extents Button

Zooms back to the original graphic image size i.e. Zoomed to the Extents. Horizontal and Vertical Scroll-bars are no longer required.

Not Available in Simulator Mode.

Mode Button

Toggles between View only mode or Full Simulator Mode.

View DXF Button

Overlay the Simulator graphic with the DXF graphic if Available. This button is not enabled if no DXF data loaded.

Available in all Modes.

Print Worksheet

Prints a Job worksheet. The worksheet consists of the Simulator graphic, Program extents, Spindle speed range, Tooling list and program header (1st 16 lines of G-code).

Once a program is produced, and simulated, a work sheet can be printed to present the data required to setup your CNC lathe. All the data presented is extracted from the G-code program. The only data not "officially" present in the program are material specs and size, initial set distance of bar out from the chuck or collet. It is usually considered good practice to add this data (and anything else useful) as comments to the beginning of the G-code. As such, the first 16 lines of G-code are included in this printout.

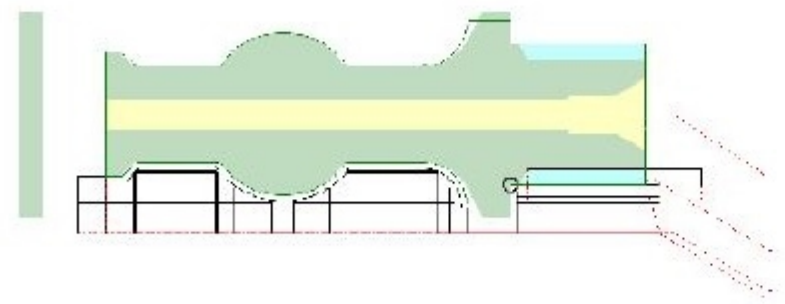
Program Running Times have been added in Version 1.4 (Approximation only). This includes Times for each tool in list. (Helps to know when you can disappear for a while).

Ezilathe V1.4.0.0 Metric - Diameter Mode

Date :- 25/06/2015 2:46:38 PM

Program Directory :-C:\Mach3\GCode\Lathe Prgs

File :- Drainbody.txt



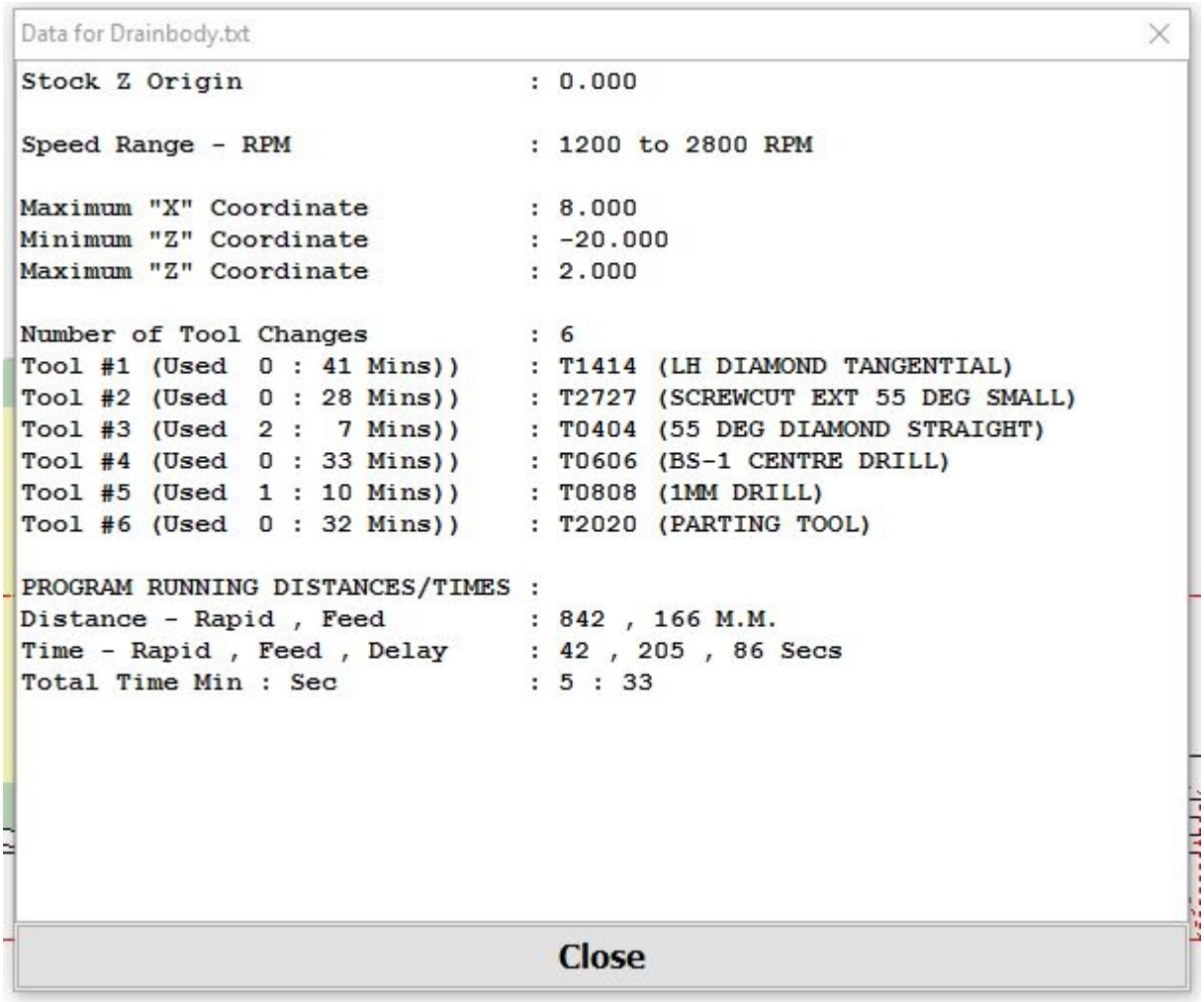
Material		: REFER DRAWING / PROGRAM HEADER
Stock Z Origin		: 0.000
Speed Range - RPM		: 1200 to 2800 RPM
Maximum "X" Coordinate		: 8.000
Minimum "Z" Coordinate		: -20.000
Maximum "Z" Coordinate		: 2.000
Number of Tool Changes		: 6
Tool #1	(In Use 0 : 48 Mins)	: T1414 (LH DIAMOND TANGENTIAL)
Tool #2	(In Use 0 : 34 Mins)	: T2727 (SCREWCUT EXT 55 DEG SMALL)
Tool #3	(In Use 2 : 13 Mins)	: T0404 (55 DEG DIAMOND STRAIGHT)
Tool #4	(In Use 0 : 40 Mins)	: T0606 (BS-1 CENTRE DRILL)
Tool #5	(In Use 1 : 16 Mins)	: T0808 (1MM DRILL)
Tool #6	(In Use 0 : 39 Mins)	: T2020 (PARTING TOOL)
PROGRAM RUNNING DISTANCES / TIMES :		
Distance - Rapid , Feed		: 1628 , 166 M.M.
Time - Rapid , Feed , Delay		: 81 , 205 , 86 Secs
Total Time Min : Sec		: 6 : 12

PROGRAM HEADER :

```
Line # 0  = (EziLathe Version 1.4.0.0 CAM  25/06/2015 2:09:25 PM)
Line # 1  = (Dxf file = Draincock body2)
Line # 2  = (Material = 1/4" AF Hex Brass - End at 25 Out from chuck)
Line # 3  = G18 G40 G49
Line # 4  = G90 G94 G80
Line # 5  = G21 (mm)
Line # 6  =
Line # 7  = T1414 (LH Diamond Tangential)
Line # 8  = G00 X4 Z0
Line # 9  = M00 (SET MATERIAL OUT TO TOUCH TOOL - HIT CYCLE START)
Line # 10  = G00 X8 Z0.5
Line # 11  = M4 S2800
Line # 12  = G04 P2 (Dwell in Seconds)
Line # 13  = (Rough #1)
Line # 14  = G00 X6
Line # 15  = G01 Z-4.5812 F120
Line # 16  = G01 X8
```

Quick Report

Opens a simple dialog displaying on screen, key data from the printout, such as times etc.



Stock Diameter Input

When a program is initially loaded in the editor, this value is assumed from the "X" Axis limit found. The Stock diameter under DXF processing is used if available.
This can be overwritten manually if required, and will remain until a new program loaded.
The Clearance Diameter is based on this and the **"Clearance"** input (Under DXF Processor), as such controls where the Cuts originate, and the diameter used for tool return.
This can be changed for each segment of a program to reflect what features need to be cleared during rapid moves to and from the cut.

Stock End "Z" Input

When a program is initially loaded in the editor, this value is assumed to be 0.
If the Gcode program suggests other than "0" is correct, then a warning will be issued via a message box.
This can be overwritten manually if required, and will remain as entered until a new G code program is loaded. Generally all DXF files should start at Z = 0.

Graphic Area

Workpiece is displayed from the perspective of the operator of a conventional lathe and displayed to fit the display window. The Workpiece (in green) is shown finished (Yellow areas are drilled or bored, and Blue area is a screwcut thread). Toolpaths in Red are Rapid (G00) otherwise shown in black. A DXF overlay is shown as dark green lines, if applicable).

The current mouse position is shown in the status-bar panel #1 (together with the resolution per pixel in units).

Various other data items are shown in other Status-bar panels, when lines in the list box are selected. Note that when the Program list-box is synchronized (position marked with a circle) the Cursor position is relative to this point, otherwise the absolute position is displayed.

Program List-Box

To the right of the graphic area is the program list box. This is a read only listing of the loaded program. When Active (has focus, a line highlighted) enables navigation (using mouse or arrow keys) through the program in both directions. A Highlight follows in the editor to assist in finding the line there. The active G codes are shown in the box above the graphics, and other modes are shown in the status bar including the co-ordinates. The point referred to in the program is also highlighted in the graphic with a black circle.

You can move up or down in this list-box using mouse or arrow keys when list-box has focus.

Status Bar Panels

Panel #1 = Current cursor location from a Lathe point of view (X,Z axis). Res refers to the current resolution of 1 pixel in the graphics area (Changes only when display is zoomed or program is reloaded).

Panel #2 = Limit of travel for current program (Excluding tool change location).

Panel #3 * = List-box Line Number + Axis location after execution.

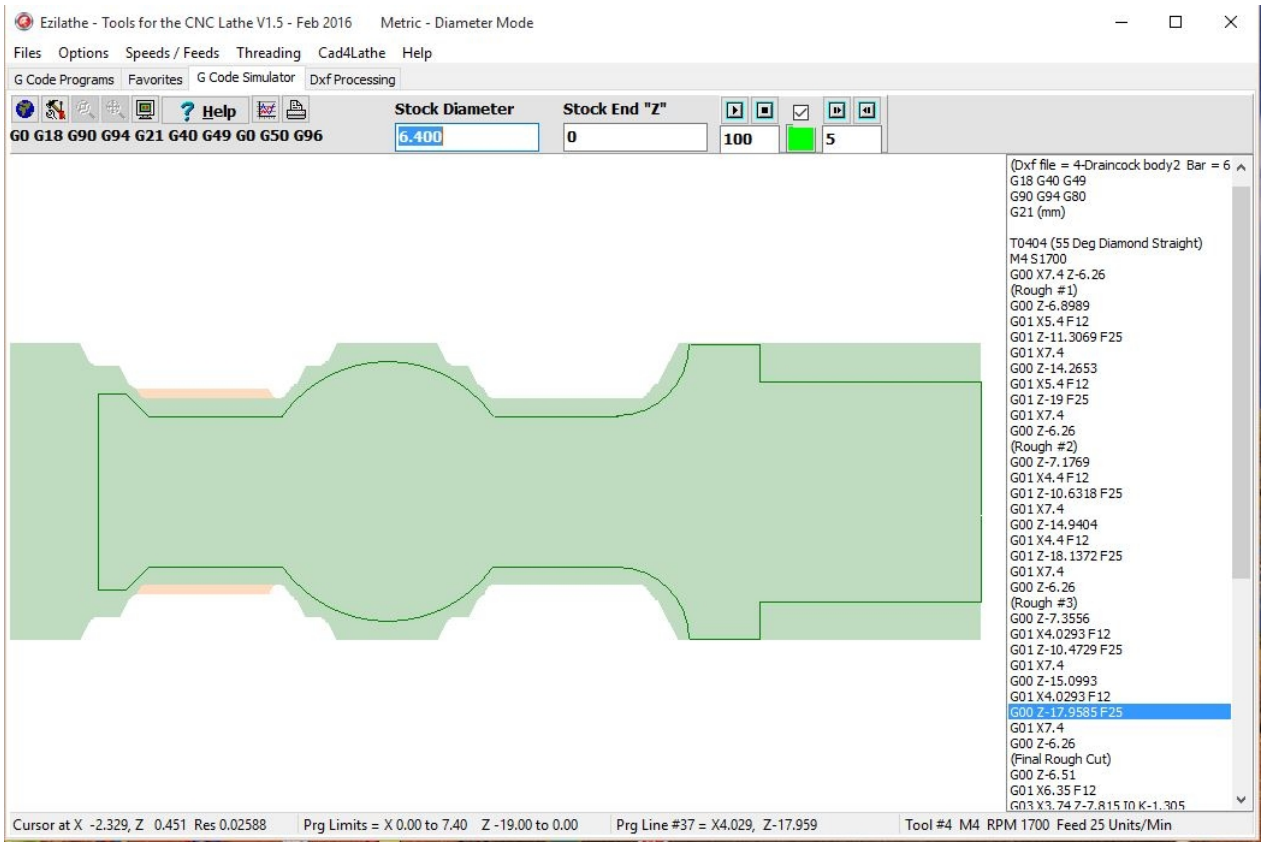
Panel #4 * = Status at List-box Line - Tool No, Spindle Rotation, Spindle Speed, Current Feed.

Panel #5 = Current Zoom Status.

Top panel (Under buttons) * = Current active G codes.

* Display is current only when List-box line is highlighted.

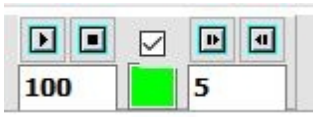
Simulator Mode



The Full Simulator mode allows you to, starting with the blank only, step through the program, and see the cut take place on screen. The DXF Overlay may be added at anytime to give an appreciation of the finished part. This mode is entered from the View only mode using the "**Mode Button**" . The Mode button, as well as the Load button may be used to exit this mode.

This mode is useful for highlighting points in program execution where the tool contacts the work piece in rapid traverse. The material will be shown in "Red" where contact takes place (This highlight is also available in View mode, but can be lost to later cuts). This can be seen above where a cut has been changed to G00 from G01.

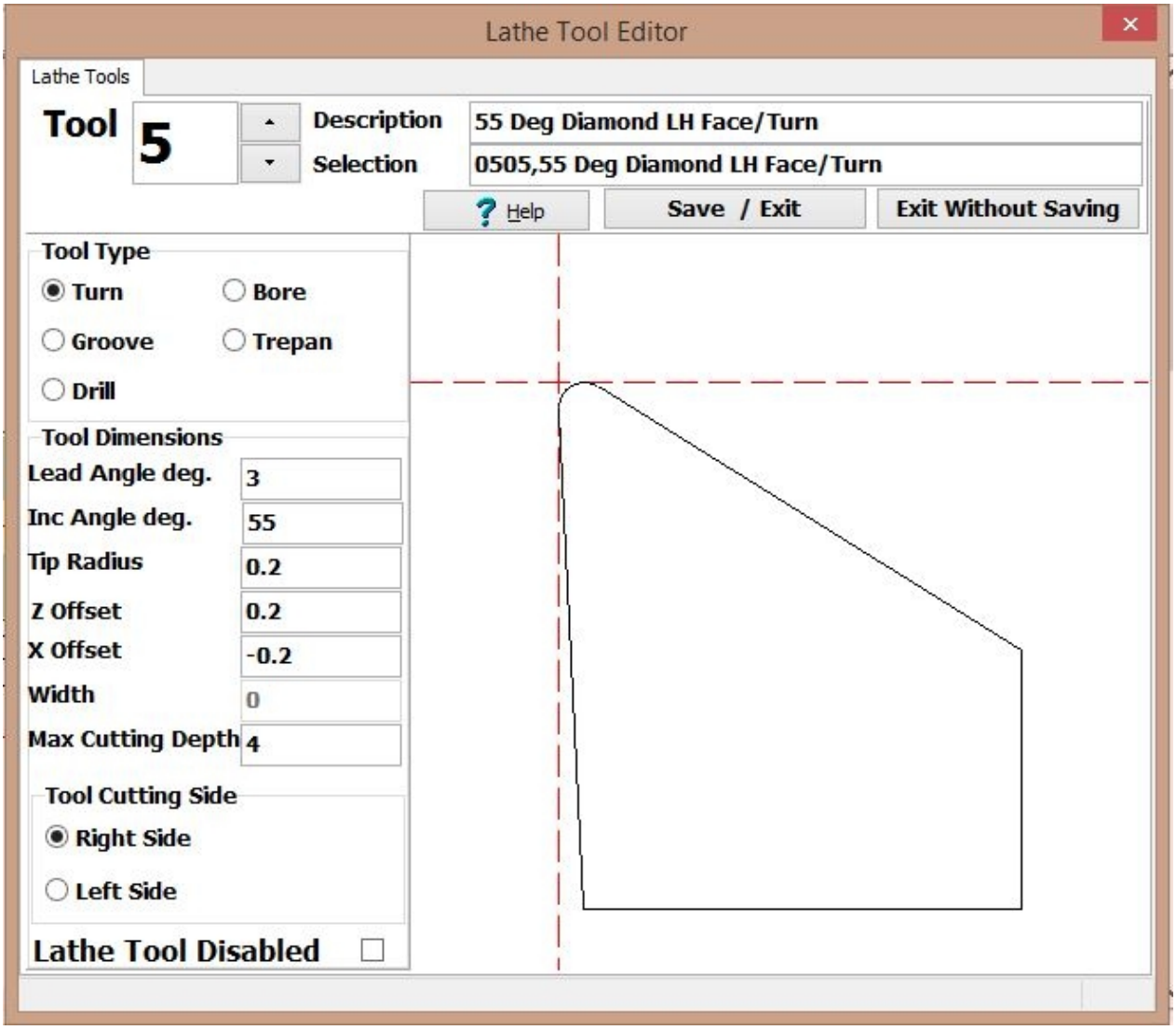
When in Full Simulator mode, a control panel will be visible, that allows for automatic or manual stepping through the program (Manual stepping may be in either direction).



The Buttons are (From Left to Right) - Start Auto , Stop Auto , Manual Step Forward, Manual Step Backwards
The Checkbox should be checked for a single pass through, or un-checked for a continuous run.
The first edit box (100 = default) is Step advance speed (1000 = 1 second).
The second edit box (1 = default) is the displayed feed. (in msec per pixel if you need to know)
If required, enter zero here for instant view, but less intense processing, on low power systems.

The Green "LED" advises the state - Yellow = Auto Mode Green = Ready (for input) and Red = Busy.

Tool Editor



The Lathe tool editor supplies tool data to the Simulator, where the effect of tool shape may be viewed, and to populate the selection box of Tools used in DXF processing.

To use Radius Compensation (Where Available) the tool must be correctly entered. No gouging avoidance is programmed, however you will be missing out on the graphic supplied by the simulator if tools are not correctly entered. Only "Std Turn" mode can be used without specific tool information.

To set-up a tool library is very simple, with instant feedback of edits in the graphic window.
Changes to tools can be permanent (Edit the tool data, then press ***"Save /Exit"*** button - this will save changes to disk) use when setting up or adding / editing tools.
The ***"Exit without Saving"*** button can be used to try "what if" changes or Disable / Enable tools on a temporary basis. (Changes are in force for session only, and are lost when "EziLathe" is closed).

Note That Changes made here, are also immediately reflected in the tool selection boxes, (under DXF processing and Threading).If New Tool is entered, File must be Saved to show in tool selection boxes.

Tool Number Input

This Integer number may be entered directly into the box, or use the Up / Down Control.

Limits - 1 to 99 only.

Usage - Reference number for library only. (But should reflect the Tool number)

Tool Description Input

Description of tool, sometimes useful to describe a chuck rather than the tool itself.

Limits - Limited to 40 length max.

Use - Just local in editor.

Tool Selection Input

Format = Integer , Description

Limits - Limited to 40 length Total.

Use -
Integer - Must compile to a number used by the system as the tool number between 1 and 99. the format used may be nn (i.e. 04) as tool number only or nnnn (i.e. 0404) as tool number and offset number (Generally the same). If the Tool number entered does not compile to an integer, only the simulator view will be effected (No display of actual tools). The compiled value may be confirmed in the status-bar under the simulator.

The Comma is required to separate the two values.

Description - This is the bracketed description inserted as a comment in G-code when a tool change is generated via the DXF Processor.

E.G. Selection Field = **0505,55 Deg Diamond LH Face/Turn** shows up in the compiled program as **T0505 (55 Deg Diamond LH Face/Turn)**

Tool Type Check-group

The type of tool is selected from here. The tool type effects where a tool may be used, and the data used to define it. Currently there are 5 types :-

- **Turn** - Normal external turning tools like the LH face / turn tool shown above. (My lathe is back to front, that's why RH face / turn labeled as LH !)
- **Groove** - Grooving or Parting tools. **"New V1.6"** Now includes Radius corners, Max Radius = button tool. Only type allowed in CAM system for Grooving.
- **Drill** - Std drills, Center drills, Countersinks etc.
- **Bore** - Boring tools - only type allowed in CAM system for boring.
- **Trepan** - As Groove, except turned 90 deg towards chuck. Only type allowed in CAM system for Trepanning.

Tool Dimensions Inputs

This group inputs used to describe lathe tools are all real numbers, and may be Positive or Negative (Positive assumed, if no sign entered).
Input availability will vary depending on tool type selected. Inputs may be replaced, or dis-enabled dependent on tool type.

The inputs are :-

- ***"Lead Angle deg."*** - The angle in degrees of the Leading face of the tool. Positive value gives clearance. The same 55 deg diamond insert in a straight holder would have a value of -27.5 here.
- ***"Included Angle deg."*** - The included angle of the Tip. A square insert would have a value of 90 degrees here.
- ***"Tip Radius"*** - Entered in units. This is the radius at the cutting tip.
- ***"Z Offset"*** - The controlled point is at the Tip of a sharp tool (Zero offset required) or at the center of radius otherwise (This will require an offset = to the radius in most cases to place the controlled point at the normal location).
- ***"X Offset"*** - As for Z offset, except X Axis.
- ***"Width'*** - The Width of the cutting edge, usually Groove or Trepan tools.
- ***"Max Cutting Depth"*** - Mainly effects the size of the tool when displayed in editor, and the sweep of the tool in the Simulator. Set too small as a turning tool, could leave bits of green material on the screen. Set to large as a boring tool, could take out the other side of the hole. This input should be manipulated so tools like screwcutting tools will have an accurate projected width (Helps with collision detection in the Simulator).
- ***"Diameter - Body"*** - Diameter in Units of a Drill, O.D. of a center drill etc.
- ***"Included Angle - Tip"*** - Included angle of tip in degrees. Used for Drills , Countersinks etc. On two diameter tools this stays with the body (60 deg for center drill). The end angle is assumed at 118 deg.
- ***"Diameter - Tip"*** - The smaller of the two diameters on a two diameter tool such as a center drill.
- ***"Length - Tip"*** - The extension of tip beyond the body on two diameter tools e.g. center drill.
- ***"Min Bore Diameter"*** - Used exclusively with Boring tools, This is the minimum diameter of the initial hole that a boring tool may fit This size is enforced in CAM system, too large may not fit the finished size, too small, and the tool will "Crash" on the lathe.

Tool Cutting Side Check-group

Controls the side of the tool where the "Controlled Point" is situated.

Typically most turning tools are Right Hand (for a conventional lathe). Parting tools are generally entered as left hand to avoid consideration of tool width in parting off operations.

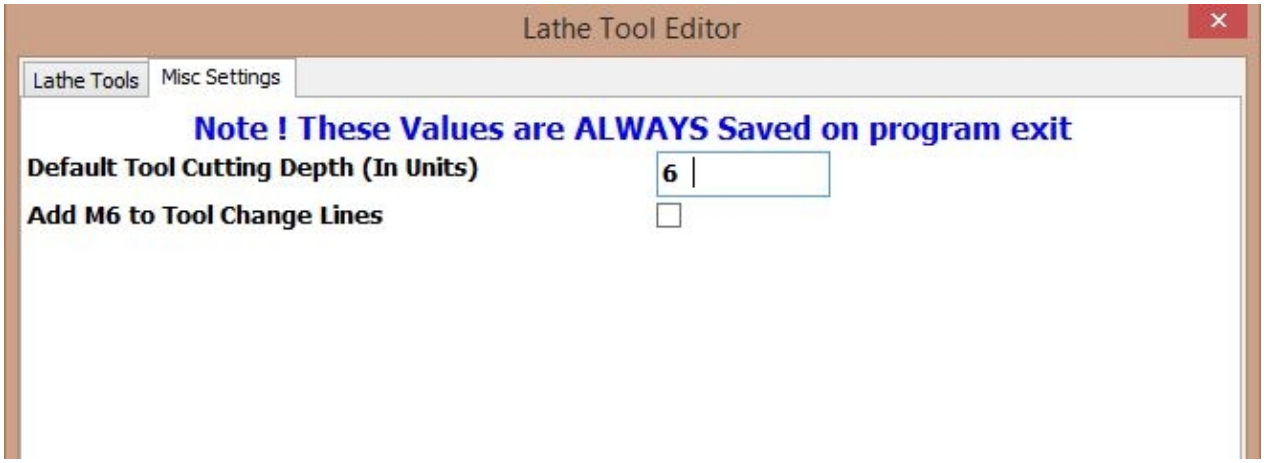
Not enabled for drills.

Lathe Tool Disabled Check-box

If Checked then a tool is disabled. This is a quick way to compare the cut to a Zero width tool. Incomplete inputs should be left disabled.

The state of this does not effect code generation in any way.

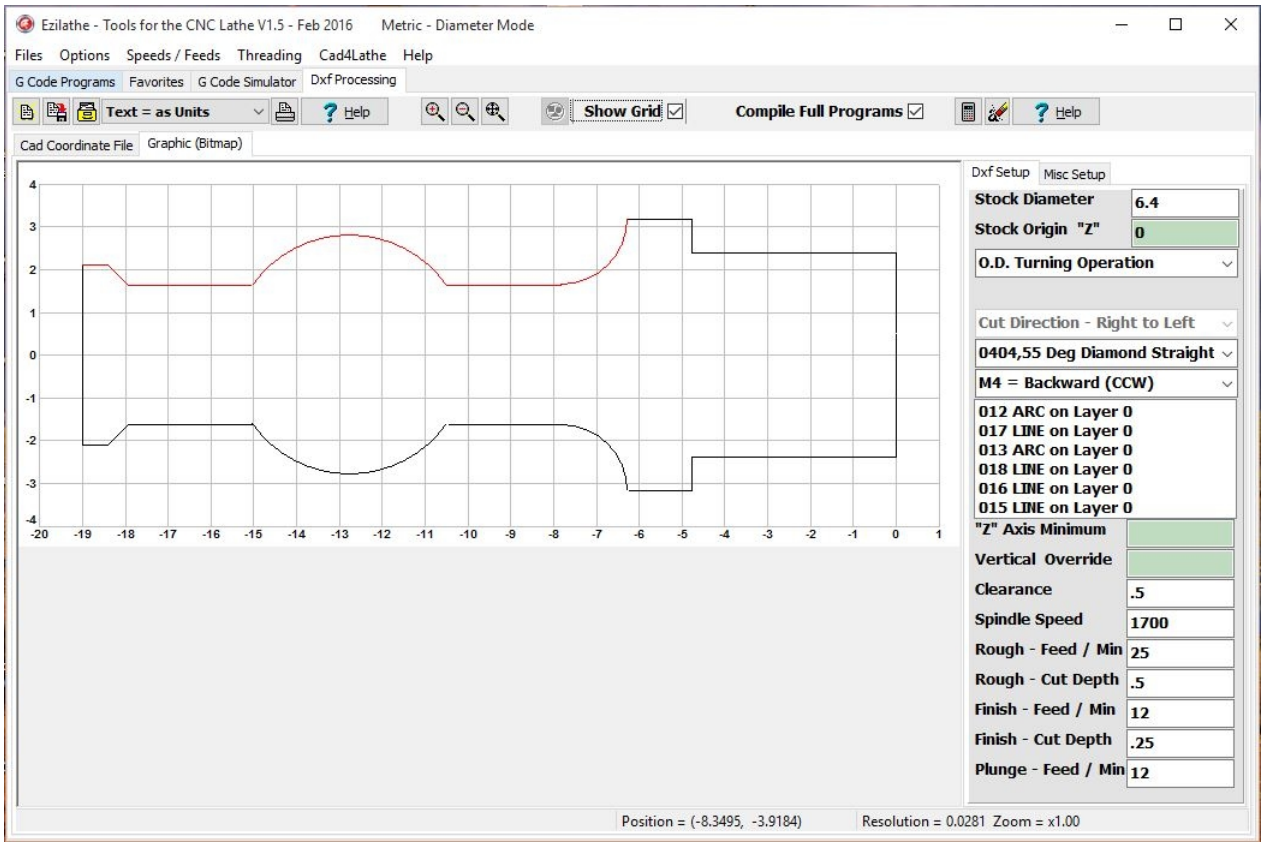
Misc Settings Page



The Misc Settings Page tab allows for additional Global settings for tooling to be entered as Required.

- Default Tool cutting depth (In Units) - When the currently simulated tool is unavailable or disabled, the default tool is used. This is a vertical line (on screen) of length as entered in units (mm or inch dependent on metric or imperial mode). The Default tool is zero width, sharp nose and with a cutting depth = to length entered. Note that this (like any tool cutting depth in simulator) only effects the way the job looks when simulated. Too small, and strips of blank will remain on screen. Too large, and boring tools will appear to wipe out the other side of a bore. Try 6 to 8mm or 0.25 to 0.312 inches as a reasonable starting point.
- Add M6 to Tool Change Lines - Checked - Tool changes include "M6" i.e. T0404 M6 (55 deg diamond R.H). Unchecked - No M6 i.e. T0404 (55 deg diamond R.H.).

Dxf Viewer



The DXF viewer is one of the most useful functions within "EziLathe". Adding the DXF processor extends the value even further. Between the graphic screen (above) and the Cad Co-ordinate page (Text screen next) all entities within a drawing can be identified. Entities can be selected in the graphics screen (red when selected, otherwise black) and their id number read from the ***"Selections List-box"*** The same ID number as on the text page. Up to 400 entities can be loaded from the DXF file (Data is truncated above this limit)

To ease the burden of finding what you want on the graphic screen, the display may be filtered to display only entities on the selected layers refer to Layers Check-box under Cam processor section.

As with the Simulator, the mouse position is continually updated in panel #1 of the status-bar.

DXF files

DXF files are produced by many CAD systems, the format was after all developed for inter-application transmission of data. Over time, the format has continued to develop, and branch out into binary and text formats. "EziLathe" does not need anything sophisticated.

File Type. - Text based .DXF files. I have yet to find an incompatible DXF file from R12 up to R2013.

Supported Entity Types

- **Lines**
- **Arcs**
- **Polylines** - Line and Arc segments.
- **LWPolylines** - (Light weight Polylines) Lines and Arc segments.

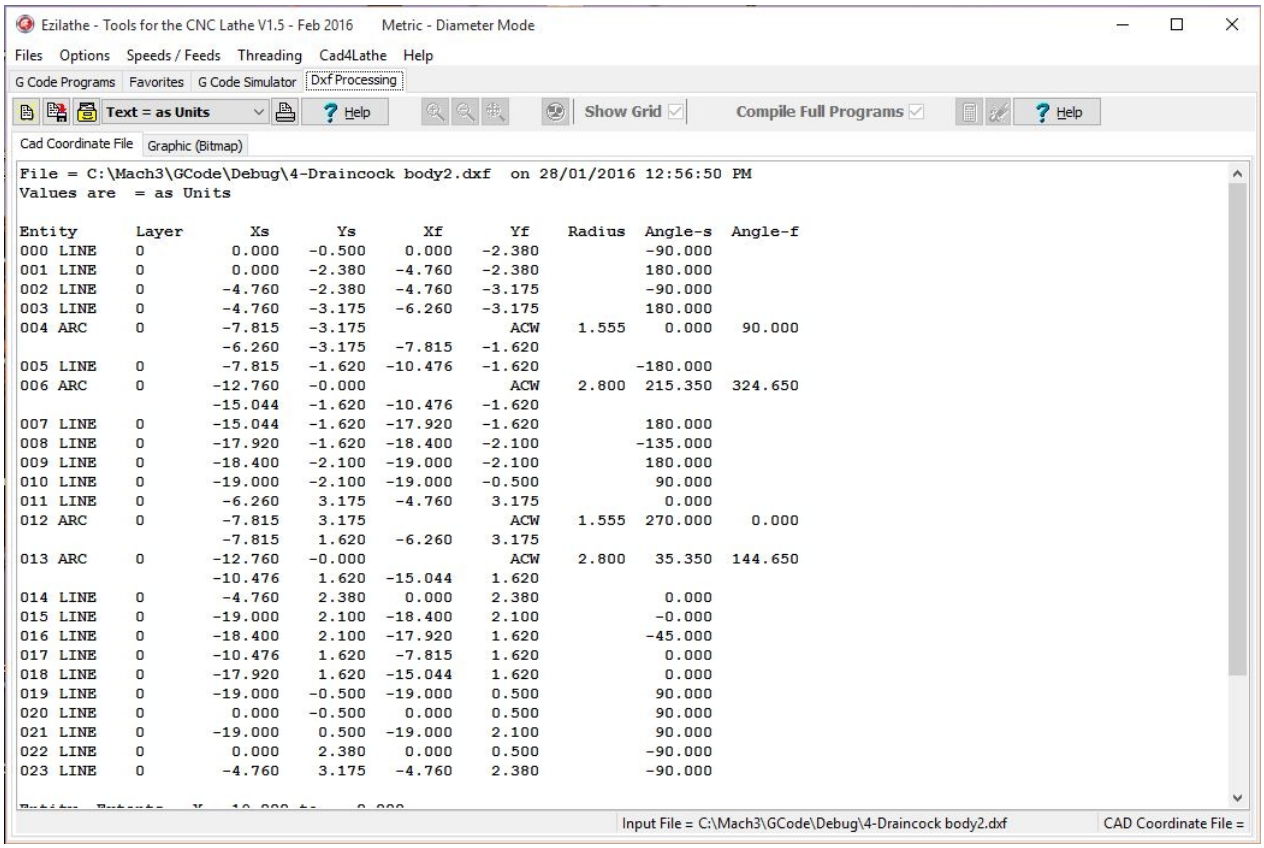
Not Supported Entity Types

- **Splines** - Unlikely to be supported or needed.

Notes

- I tend to prefer Polylines / Lwpolylines, that are joined. This gives the ability to offset / move whole profiles for radius compensation etc. For complex profiles, Polylines can make life easier, as the whole sequence can be selected as 1 entity, and still allow de-selection of unwanted segments.
- It is recommended that you keep your stock end at Z = 0 and Work towards Z minus (Chuck end). Your DXF should reflect this (X axis to CAD) . Only Positive X values may be selected for CAM, so keep to Positive 'Y' axis in CAD.
- Use layers so you can control visibility if required, but "Ezilathe" only uses Layer names up to 6 characters in length only.
- Additional entities are fine, "EziLathe" will only use as the toolpath what you select. You can often find that an original profile as well as an adjusted profile are useful in the cad data to confirm the accuracy of any variations you have set up (radius compensation etc).

Cad Co-ordinate Page



This page contains a listing of all entities entities within the DXF file, The Axis listed are named as is conventional for the CAD system, so in effect Listed X axis = Lathe Z axis, and Listed Y axis = Lathe X axis.

The listing may refer to polylines, containing a group of vertex (of type Arc or Line). Polylines may be open or closed, and the sequence is terminated by "SEQEND".
Note that the Start of a polyline includes it's number (e.g. #0) used for polyline selection. E.G. "**POLYLINE 1 - CLOSED SEQUENCE #0".**

The Columns list data as follows :-
1. Entity - Polyline or Entity ID + Entity Type
2. Layer - Layer name or Vertex type
3. Xs - X Start co-ordinate - for 1st line of Vertex Arc or Arc this is the X center of arc.
4. Ys - Y Start co-ordinate - for 1st line of Vertex Arc or Arc this is the Y center of arc.
5. Xf - X Finish co-ordinate.
6. Yf - Y Finish co-ordinate.
7. Radius - Radius of Vertex Arc or Arc.
8. Angle-s - Start Angle of Vertex Arc or Arc. Angle of a Vertex Line or Line.
9. Angle-f - Finish Angle of Vertex Arc or Arc.

Note Angles are degrees running Counter Clockwise from Horizontal (0 deg) to the Right (East). 90 deg is Up (North).

File Open Button

Opens a Dxf file in the Viewer. The file is opened via a standard file open dialog box.
By default file selection is filtered to Dxf Files (.dxf).

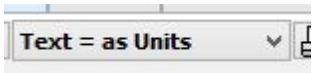
File Save Button

Saves the Dxf processed data. The file is saved via a standard file save dialog box, to give the option to change the file name.
If the graphics page is active, then it's contents are saved as type Bitmap (.bmp)
Otherwise the text data is saved as a text file with the extension .ccf (Cad co-ordinate File)

Recent Files Button

Opens the Recent Files Dialog box to re-load a file from the last 10 files accessed.
The dialog box is context sensitive, displaying only the relevant file types for the current page.

Text Value Converter



This selection box contains 3 options that effect how dimensional data is displayed in the text window.

- Text = as Units - Displayed as values in DXF, in the same Units.
- Text = MM to Inch - DXF dimensions are multiplied by 0.03937 (i.e. MM converted to inches)
- Text = Inch to MM - DXF dimensions are multiplied by 25.4 (i.e. Inches converted to MM)

This can be useful for checking DXF in native drawing units, if changed for G coding.

Print Button

Prints the Dxf processed data. The file is printed via a standard printer dialog box, that gives the capability to select a printer and modify it's printing options.
If the graphics page is active, then it's contents are printed, otherwise the text data is printed.

By default, the file is printed to the default system printer.

Zoom Windows Button

Zoom Window.
1. Pick first point on screen to define first corner of rectangular window.
2. Move the mouse to the diagonally opposite corner of the rectangular window, a rectangular box will stretch out with mouse movement.
3. Pick second point on screen to define the second corner of the rectangular window.
4. Move the mouse to move the rectangular window if required.
5. Pick on screen again to execute the zoom.

Note ! To back-up 1 step, right-click on screen.

Zoom Extents Button

Zooms back to the original graphic image size i.e. Zoomed to the Extents. Horizontal and Vertical Scroll-bars are no longer required.

Select Polyline Button

Allows the selection of a Polyline sequence by it's number as listed on the text page.
The polyline is selected in the correct direction for processing, segments may be removed manually if required.
(Click on Segment on Graphic Screen or Right-click list-box to select delete from pop-up menu.

Note - This button is disabled if no polyline sequences exist in the DXF input file.
If only 1 polyline sequence exists, then this is automatically selected without showing the dialog box.

Show Grid Check-box

If Checked, a grid is shown on the graphics page.

Full Programs Check-Box

In the DXF processor, if this box is checked, then a full G-code program is produced, that will clear all existing content of the G-code editor (after confirmation if data already exists).

If this box is not checked then a code fragment is produced, that is simply added to the end of the editor, leaving existing content intact.
This Code Fragment can be cut/pasted to the required location in the program that uses multiple tool changes.

Compile Code Button

Compiles the G code Program from the inputs presented, if no errors are present. For a full description of the processes involved, Go to Dxf Processor.

Clear Selections Button

Clears all selected entities in DXF processor.

Status-bar Panels

Graphics Page

Panel #1 = Current mouse position.
Panel #2 = Resolution (Units per screen pixel)

Text Page

Panel #1 = Input File Name (*.DXF)
Panel #2 = Output File Name (*.CCF)

DXF Processor

The in-built DXF processor takes the DXF data loaded in the viewer, and with a minimum of set-up can produce a complete G-code program, or a code fragment for insertion into a G-code program. Up to 400 entities may be processed into G code. This code is designed to be easily adjusted if required to arrive at the finished part (The Gcode is compact, and commented to assist with easy manual editing if required). Some profiles are not practical to machine with this approach, without adjusting the DXF file to suit (Additional lines within the DXF to lead the cut from 1 section to another).

Processing Modes - 6 modes are available to cater for differing requirements

- **O.D. Turning Operation** - The Basic Mode. The code produced is simply toolpaths that finish up at the selected profile, no tool radius compensation is carried out. Can use any Tool except Trepan or an undefined tool.
- **I.D. Boring Operation** - As above, a Basic mode for boring. Assumes nothing about a hole being present, but must have a defined Boring tool that fits the Dxf profile to be bored (Also accounts for Clearance as set).
- **"New V1.6" Trepan Multiplunge** - Roughing is carried out by Overlapping Plunges to Finish cut (Requires Tool Overlap entered, but Roughcut is Dis-regarded). Trepans a groove in the face of a component, using a defined Trepanning Tool. This mode compensates for the width of the tool, and currently is only correct for full radius tools (Button tools). Useful for Flywheels etc. Use with care, as very new. Logically you would cut from the outside in to minimize the clearance required on the outside face of the tool.
- **"New V1.6" Trepan Plunge/Traverse** - As Above, except Roughing is carried out by plunging to Roughcut Depth and Traversing across. Does not need Tool overlap field entered.
- **"New V1.6" Grooving Multiplunge** - Roughing is carried out by Overlapping Plunges to Finish cut (Requires Tool Overlap entered, but Roughcut is Dis-regarded). Grooving is carried out using a defined Grooving tool. This mode compensates for the width of the tool, and currently is only correct for full radius tools (Button tools). Useful for deep / wide grooves, but does not suit multiple narrow grooves due to the fixed pitch of the roughing cuts.
- **"New V1.6" Grooving Plunge/Traverse** - As Above, except Roughing is carried out by plunging to Roughcut Depth and Traversing across. Does not need Tool overlap field entered. This Mode is ideal for components that do have multiple fixed pitch grooves. Cuts automatically align to component grooves. Where Tool and groove are close to the same width, the finish cut can be reduced practically to zero (say 0.001) to ensure that depth is roughed out near to full depth.

Note that the new modes are "Very New" and have limited use to date. Use with Care, but do try.

Comments / Tips

- Stock size - The diameter of the stock entered controls (in conjunction with "Clearance" as input) the X position of rapid returns, as well as the number of rough cuts used. The stock origin "Z" usually = DXF Max, and are both preferably zero. This can be altered for special effect, but is not usually useful. The program will issue a warning if the two do not match.
- Initial facing cuts - so many choices here!!! The default processing if the vertical end is selected is plunge into center and face out to meet the od cut. You are better off to not select the end, and deal with the initial facing cut manually in G code or on the lathe (or do not face the end, as often not required).
- Final end of cut - The DXF profile usually ends abruptly, leaving a vertical shoulder on the material. Depending on the tool used, this can cause an excessive cut right at the shoulder if a Vee shaped tool is used (see example below). The effect is clearly visible in the simulator, but I have not felt the need to fix this yet, however this could change.

Entity Selection

Entities must be selected in order in the cut direction, If out of correct sequence, the results will be interesting, but not very useful. Entities are selected using the mouse on the graphic screen, simply place the cursor on the entity to be selected, and select using the left mouse button. Entities that are not

selected, are not processed in any way, but are available for display in the "Simulator" DXF overlay.

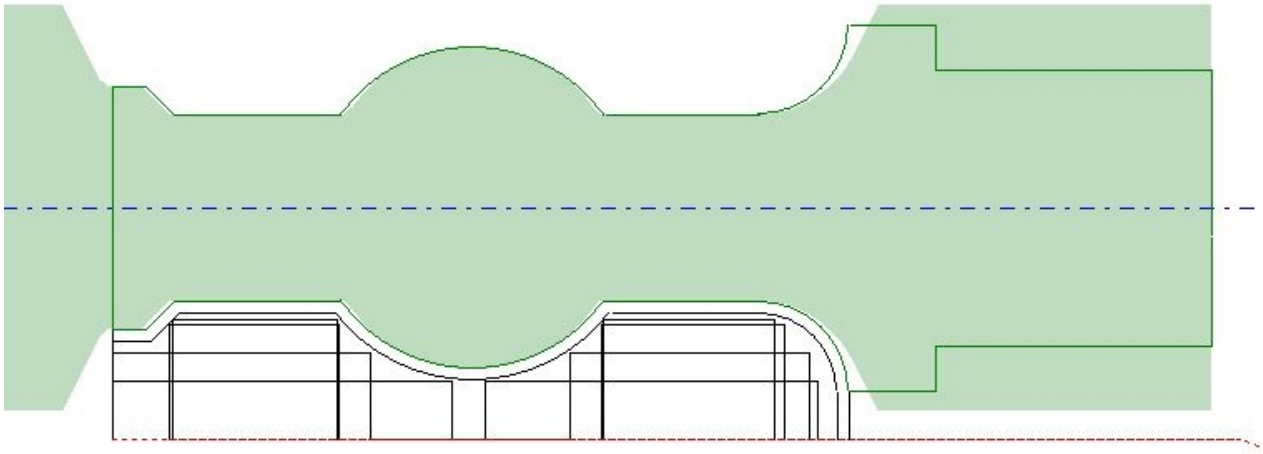
As entities are selected, a note appears in the selection list-box, and the entity is displayed in red in the graphics screen.
If there is a gap between selected entities, the program will simply cut a straight line across the gap (That's one reason to select entities in order). Sometimes, this will produce undesirable results, Lines or arcs may be added to lead the cut in the desired direction.

Re-ordering or Removing from Selections
Entities may be selected in the ***"Selections List-box"***. If Right-clicked, a Pop-up menu appears with the options [***"Move Up","Move Down","Remove from List","Delete Entity from Graphic"***]. Left-click on the required option to complete the action. This may need multiple moves if some distance involved. Entities may be removed from selection by re-selecting in the graphics screen.

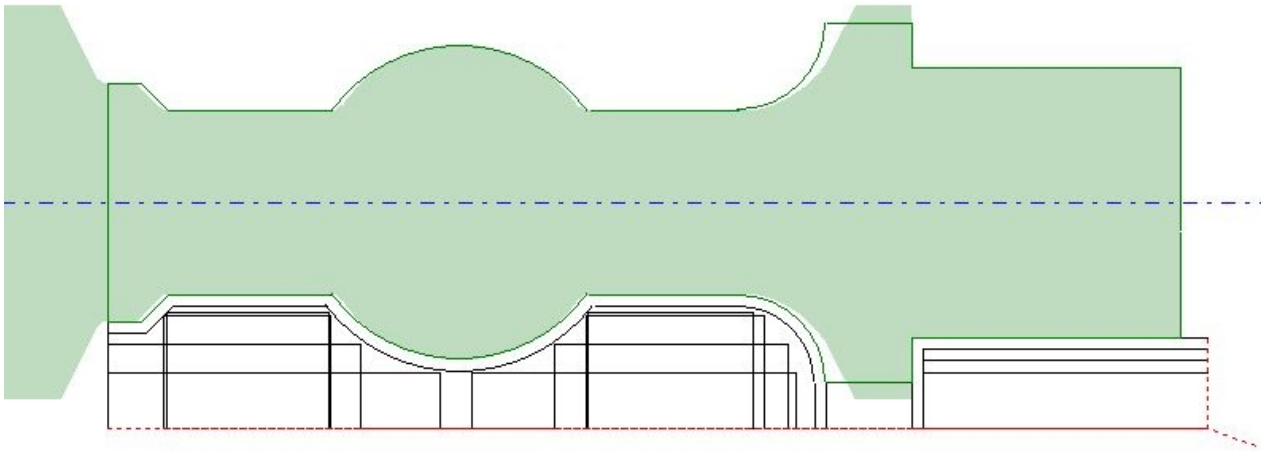
A Typical Job (Using O.D. Turning Mode).

The steps can be done in any order, and assembled in a more appropriate order later (I tend to do the best bits first).

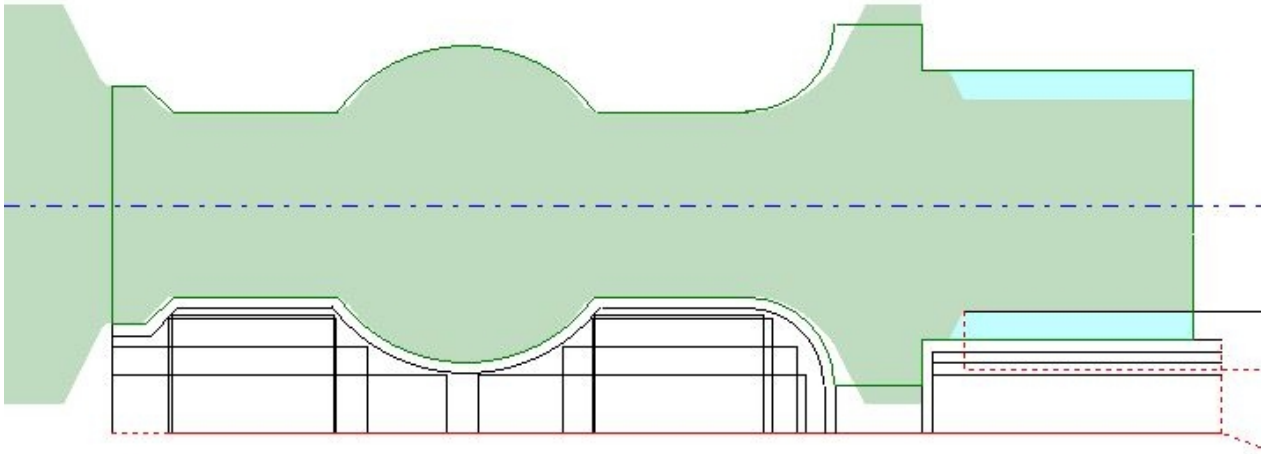
Step #1
After data entry into the Cam Setup control, Set ***"Compile Full Programs"*** Check-box to Checked, to Clear the editor (If Reqd) and add all the Program header data.
For the first step, I used a straight 55 deg Diamond tip tool, best for plunging in to take cuts. The Stock Origin is at Z 0.0, the computer automatically plunged in at a reduced feed to start the cuts, as the cutting started at Z -6.350, well away from the stock end.



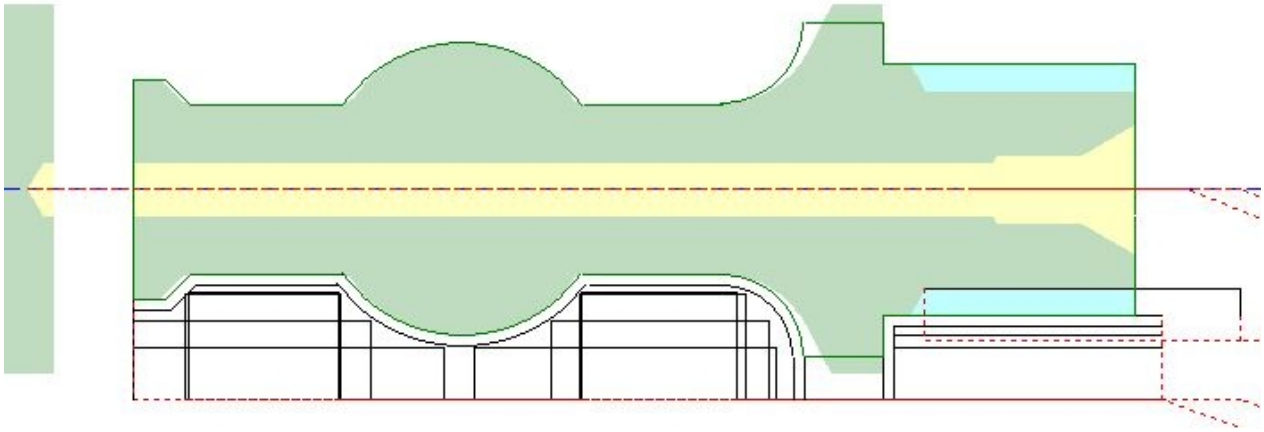
Step #2
Compile Full Programs Check-box Un-checked, so that this step is added to the end of Step #1.
For This second step, I used a Sharp Face/Turn Tool. This is cut from the end, so no plunging cuts. All entities from Step #1 are cleared from selection, and the new entities selected.



Step #3
Using the Threading and G76 calculator, a program fragment was produced to cut the 3/16" x 40 M.E. thread to the end of the workpiece. When the tooling is correctly set-up in the Tools Editor, clearance from screwcutting tool to shoulder can be confirmed visually to close limits.



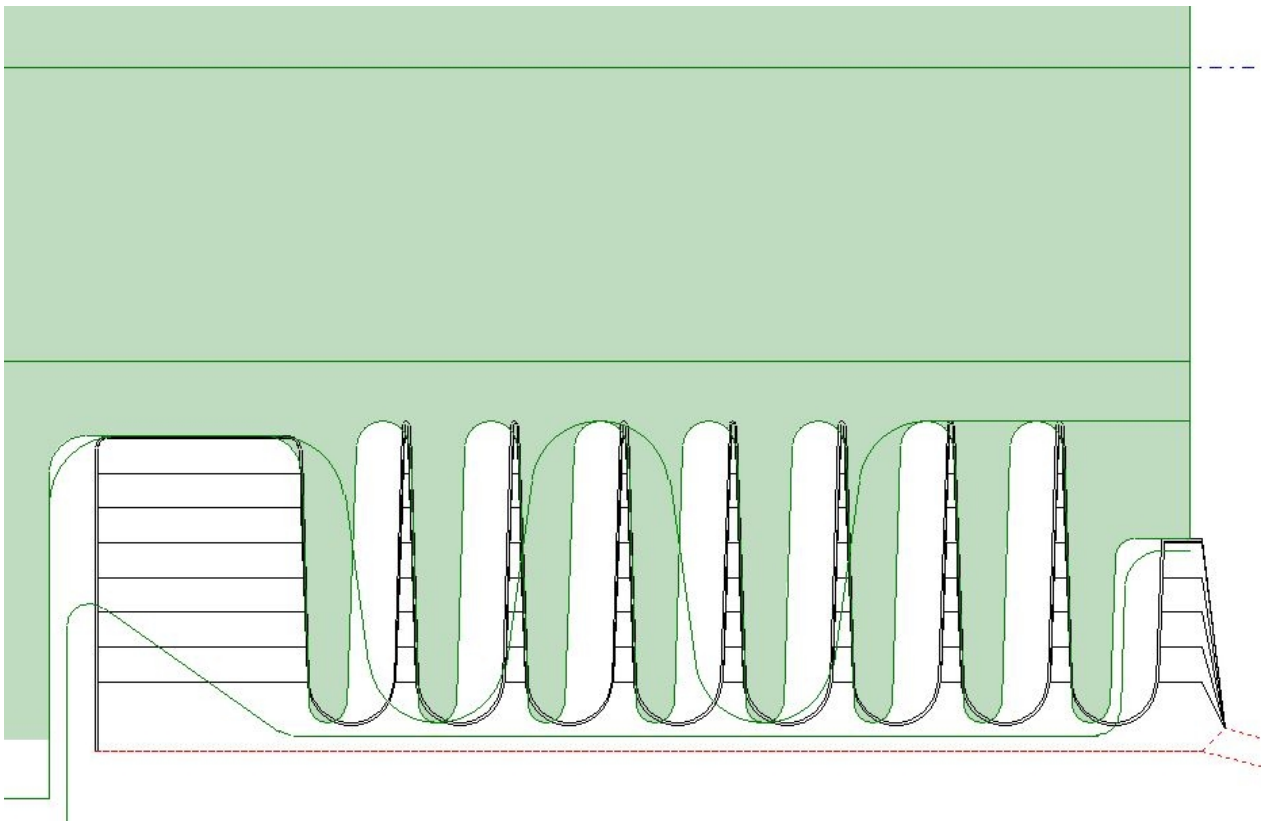
Step #4
Cut and Paste from "Favorites" - Center Drill - Drill through - Part off. Manually add speeds into favorites fragments, adjust drill depth, and part off co-ordinates. Re-order initial steps to a more suitable order, i.e. Start with Step #2, then Steps #3 and #1 (Looked a bit fragile to cut in previous order). Job Done ?



Step #5
Not Quite Finished. I see the end of the drill "Peeping". I need a flat face here, as i will not be facing off, so pull the drill back 1mm (Parting tool is 1.5mm wide). Final result can be seen under "G code Simulator". Copy from the favorites a couple of lines that position the facing tool as a material stop before the spindle start-up. Includes a M0, for opening the chuck / sliding the material out to touch the tool at Z 0.0. Now the job is done. In case you are wondering, the part of the job left not machined, is a hexagon.

This job was run on a CNC lathe (running Mach3) exactly as depicted here, without any further edits.

An Example of Grooving - Plunge / Traverse.



This is Gerry Howell's Vee Twin Cylinder (Nice engine). After cutting down to just above the fins with straight cuts (The Line in the DXF using O.D. Turn) and resetting the Stock diameter to suit, the Grooving is done to finished profile from the DXF. Note the Toolpaths are offset automatically compared to the actual cut. No Additional changes were required to run the program. The Additional Profiles in the DXF were additional Debugging Profiles.

DXF Setup Page

The DXF setup page can be seen under Dxf Viewer.

Note ! Green inputs are optional, may be left clear.

Stock Diameter Input

Enter the Stock Diameter. Note that the tool will return at clearance value above this Diameter, and Roughing cuts are calculated from Stock Diameter.This Dia can be adjusted for each operation, for efficiency.

Stock Origin "Z" Input

The End of the Bar. It is recommended that this is always Z = 0.0
If left clear, 0.0 is assumed.

A warning will be issued if this input does not match the DXF profile.

Operation Combo-box

Two options here :-
1. **"O.D. Turning Operation"** - the usual for most turning operations.
2. **"I.D. Boring Operation"** - Used for boring only. Assumes the initial hole exists. (Copy / Paste from "Favorites" as required)
3. **"New V1.6" "Trepan - Multiplunge"** - Groove is roughed using multiple overlapping plunges to rough depth. Final rough / finish cuts are traversed.
4. **"New V1.6" "Trepan - Plunge / Traverse"** - All Rough cuts are Plunged Progressively to Finishcut by Roughdepth and Traversed across face.
5. **"New V1.6" "Grooving - Multiplunge" -** Groove is roughed using multiple overlapping plunges to rough depth. Final rough / finish cuts are traversed.
6. **"New V1.6" "Grooving - Plunge / Traverse"** - All Rough cuts are Plunged Progressively to Finishcut by Roughdepth and Traversed along Z Axis.

Initial Hole Diameter Input

Visible and applicable only when I.D. Boring Operation is selected. A pre-drilled is assumed when boring.

The Initial Hole Diameter must be small enough to fit within the required bore (from selected entities from DXF), and be big enough to allow the boring tool to enter (and have the entered Clearance value). An error is raised if values entered are invalid.

E.G. using a boring tool entered with a minimum bore diameter entered as 10, and a clearance of 0.5 - an Initial Hole Diameter of 10.5 is the required minimum here to fit.

Tool Overlap %

Visible and Applicable only when Trepan or Grooving Multiplunge modes are selected.
Specifies the amount of advance per plunging Rough cut as a Percentage of tool width.

Cut Direction Combo-box

Two Options Here :-
1. Cut Direction - Right to Left
2. Cut Direction - Left to Right

Cut direction into chuck or away from chuck. Usually Right to Left.

This Box is currently locked to Right to Left. Reason = Not worth the Trouble.

Tool Combo-box

This Box allows selection of the tool to use for the current job.
It is populated from the Tool Editor, and will reflect latest additions only after Saving the Tool File.

Rotation Combo-box

Two Options Here :-
1. M3 = Backward (CCW)
2. M4 = Forward (CW)

Selections List-box

As mentioned previously, this List-box shows the currently selected entities (I.D., Type, and layer name)

If Right-clicked, a Pop-up menu appears with the options [***"Move Up","Move Down","Remove from List","Delete Entity from Graphic"***]. Left-click on the required option to complete the action. This may need multiple moves if some distance involved. Entities may be removed from selection by re-selecting in the graphics screen.

"Z" Axis Minimum Input

If entered this value must be Less than the Minimum Z co-ordinate from the DXF data.

If correctly entered, the cut will continue on from the end of the DXF data to the specified point, at X = to DXF End X.

Usually this value will be negative, so do not forget the "-" sign (I always do).

Vertical Override Input

If Entered, this value must be less than ***"Finish - Cut Depth"***

This value is the effectively the finish cut depth for vertical shoulders. This reduced cut can often be required when the tool is "Plunged" into a shoulder, prior to facing outwards during the finish cut.

Clearance Input

For External Turning, this value is the distance above stock diameter used for all rapid traverses of the tool.

For Boring operations, this value is the distance above "Initial hole diameter".

Spindle Speed Input

This value is spindle speed used after the toolchange.

Value can be exported from ***"Speeds / Feeds"*** if required.

Rough - Feed/Min Input

This Value is the Roughing feed used for all rough cuts, except when plunging.

Value can be exported from ***"Speeds / Feeds"***.

Rough - Cut Depth Input

This value is the Maximum depth of cut for Roughing. The program will generate code to produce cuts to this depth as far as possible, only the last cut is adjusted to maintain the finishing cut depth. If you prefer, due to minimal depth of the last numbered rough cut (Not the final rough cut), it can be deleted. The code is commented to simplify fine tuning of the code.

Finish - Feed/Min Input

This Value is the Finishing feed used for the Finishing cuts, except when plunging.

Value can be exported from ***"Speeds / Feeds"***.

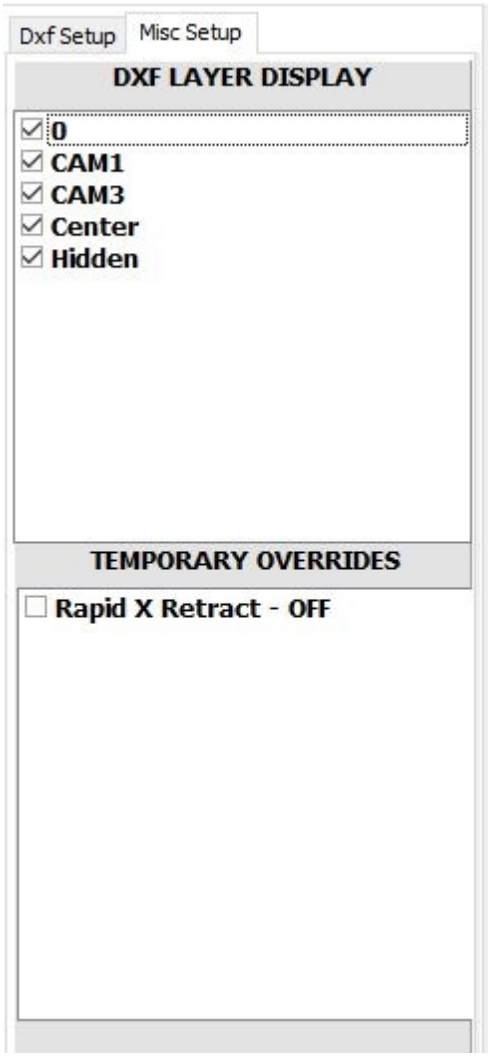
Finish - Cut Depth Input

This Value is the Standard finish cut depth, This can be reduced at vertical shoulders by ***"Vertical Override Input"***

Plunge - Feed/Min Input

This is a slow feed used when Plunge cuts are used. If not entered, the Finish Feed is used, even on roughing cuts.

Misc Setup Page



Layers Check-box

Layers in the DXF file are listed here. By Default, all are checked i.e. will display. Un-check to not display the selected Layer.

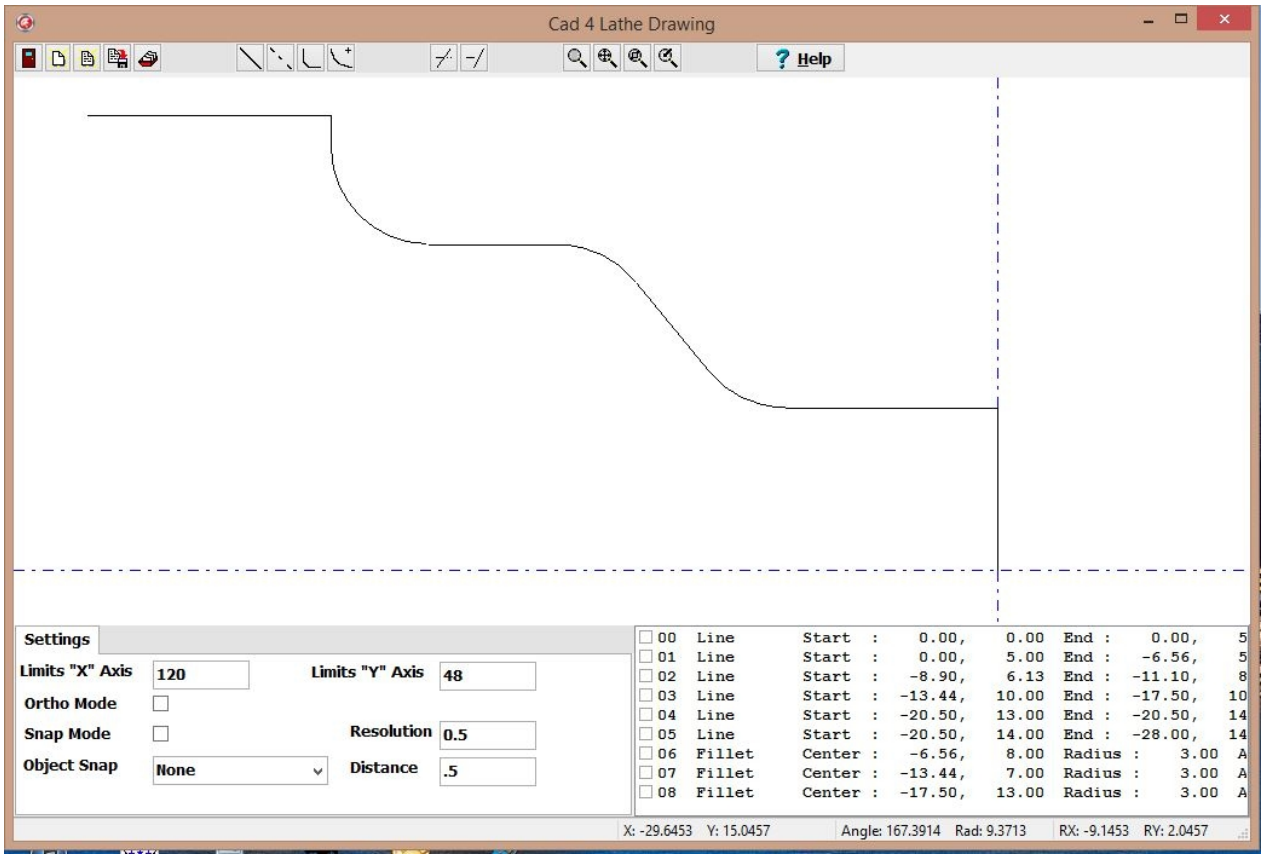
Temporary Overrides

"New V1.6"

These Temporary Overrides may be used to control various settings within the DXF Processor. Note that these settings are not saved, but return to un-checked when the program is closed.

1. Rapid X Retract Off. - At the end of a cut the tool is retracted at feed speed in X until clear of the job, This is safe particularly when facing tools reach a vertical face, avoiding cutting in rapid. If you are happy, when Checked this will retract the tool in rapid, saving time. Use with care.

Cad 4 Lathe



This Function is currently under development and is subject to change. Functions may not be implemented completely yet, but there are sufficient features operational to allow for simple turned parts to be drawn for DXF processing.

Cad4Lathe is designed as a quick and simple way to draw a few lines / arcs, that can then be exported into the DXF processor to produce the G-code. It is not intended to replace even the simplest Cad system, but can replace the more usual "Wizards" often used for simple programs. Cad4Lathe is designed to operate only within the bounds of recommended practice for Lathe work, and as such does not consider entities in the X plus or Y minus areas when calculating the drawing extents.

There is no provision for adding presentation entities (such as text, dimensions), and it is not planned. If it is required to check or document the program offline, use the DXF graphic and text printouts. The drawing file format is not standard, and cannot be accessed with other Cad systems.

Entities can be produced by selecting the co-ordinates on screen (Ortho, Snap and Object Snap modes help) or entering co-ordinates via input boxes.

Entity Draw or Modify commands when entered bring up the "Command" tab that supplies useful data and prompts for the command in question. A text edit box will be available if required The Command tab replaces the "Settings" tab (visible above).

If while a command is active, the settings page is required, it is accessible, by clicking on the "Settings" tab (Access to drawing modes etc). The command tab will return, when the mouse pointer clears the Settings tab.

Whenever a Drawing is Exported to the DXF processor, an entities only DXF file is created in the EziLathe System Directory (C:\Ezilathe\Tempdxf.dxf). This file may be imported into any CAD system.

Settings editor

The Settings editor allows the entry of a variety of modes and settings.

Note Axis are as per CAD / DXF i.e. X = Horizontal and Y = Vertical.

"Limits "X" Axis" - Enter a value to define the initial "X" size of the drawing, to allow for the maximum size likely to be drawn.

"Limits "Y" Axis" - Enter a value to define the initial "Y" size of the drawing, to allow for the maximum size likely to be drawn.

"Ortho Mode" - Ortho Mode allows only horizontal or vertical lines to be drawn by selecting co-ordinates on screen. Checked = Ortho Mode.

"Snap Mode" - Snap mode allows only co-ordinates that are rounded to ***"Resolution"*** by selecting on screen. Checked = Snap Mode.

"Object Snap Mode" - Object Snaps allow the selection of features on previously drawn entities. Object snap mode is selected from a series of options in a Pull-down list. The current modes are :- None, Endpoint, Center. To snap to a feature, you must move the mouse to within the set distance of the feature and have a relevant command active (Where more than one feature is possible, the closest one to the picked point will be selected). A colored marker will show the currently selected feature, if applicable.

Note that these modes apply only to on screen selection. Data entered in edit boxes is not effected by these modes.

Keyboard / Popup

Pop-up Menu.

"Enter" Terminates a text entry, or if no text entered (Null input) terminates a multi entity command such as Lines or Trim etc.

"Undo" Undo Last Edit.

"Delete" Deletes a selected entity. (same as placing a check mark in the entity list box).

"Cancel" Cancels the current command.

Keyboard.

Return or Enter key = as ***"Enter"*** from Pop-up menu.

Space bar = as ***"Enter"*** from Pop-up menu.

Escape (Esc) key = as ***"Cancel"*** from Pop-up menu.

Arrow keys = Move up / down the control list (Usually edit boxes).

Basic Controls

Closes Cad4Lathe window (Data remains intact, and available when the window is re-opened)

Starts a new Drawing.

Opens an Existing Drawing via Std File Open Dialog box.

Saves the Current Drawing via Std File Save Dialog box.

Exports the Current drawing to the DXF Processor and closes the Cad window. (Data remains intact, and available when the window is re-opened)
An Entities only Dxf file (Tempdxf.dxf) is also created in your Ezilathe directory (Usually C:\Ezilathe).

Entity Draw Controls

Co-ordinates may, when required be entered by picking on screen (Using Ortho, Snap or Object Snap modes as required) or as text in the command line.

For text entry, Co-ordinate data may be entered as Absolute data or Relative data (to the last point entered as listed in the location box). Prefix the co-ordinates with "@" for relative data.

Cartesian Co-ordinates = Z , X e.g. -12.7 , 9.52 or @-10 , 0 for relative co-ordinates.
Polar Co-ordinates = Radius < Angle e.g. 12.7 < 150 or @10 < 180

Line Button - Lines will continue to be drawn until Finish button (on Line Panel) is pressed (or Enter selected from Pop-up menu).

Construction Line Button - Construction Lines will continue to be drawn until Finish button (on CLine Panel) is pressed (or Enter selected from Pop-up menu).
Construction Lines are not exported via the Export Function.

Fillet Button - Constructs a fillet of input radius between 2 entities. Current Fillet Radius is displayed in the Location box. To accept the current radius without change, simple pick the 2 lines without entering a radius. Note that entry of a 0 radius, will effectively trim or extend the entities to an intersection.
Currently only pairs of Lines may be Filleted.

Arc Button - Constructs an arc at input radius and center between 2 points. The order of input (on screen or text) is Arc Center, Radius, Start Angle (from center), End Angle (from center).

Entity Modify Controls

The first step in these commands is to select a boundary line, followed by any number of lines to trim or extend.

Trim Button - Trims entities that intersect the boundary line.

Extend Button - Extends entities to boundary line if an intersection is possible.

Currently Under Review, Not all combinations of entities operational.

Zoom Controls

Zoom All (To Limits as set in Data Editor)

Zoom Extents (of drawing entities)

Zoom Window.

1. Pick first point on screen to define first corner of rectangular window.
2. Move the mouse to the diagonally opposite corner of the rectangular window, a rectangular box will stretch out with mouse movement.
3. Pick second point on screen to define the second corner of the rectangular window.
4. Move the mouse to move the rectangular window if required.
5. Pick on screen again to execute the zoom.

Note ! To back-up 1 step, right-click on screen.

Zoom Previous (Zoom to last view)

Edit Window

Selecting a drawing entity on screen or in the entity list box, while not in a command, will bring up the Edit tab.

The edit tab will contain the controls and prompts relevant to the entity to be edited.

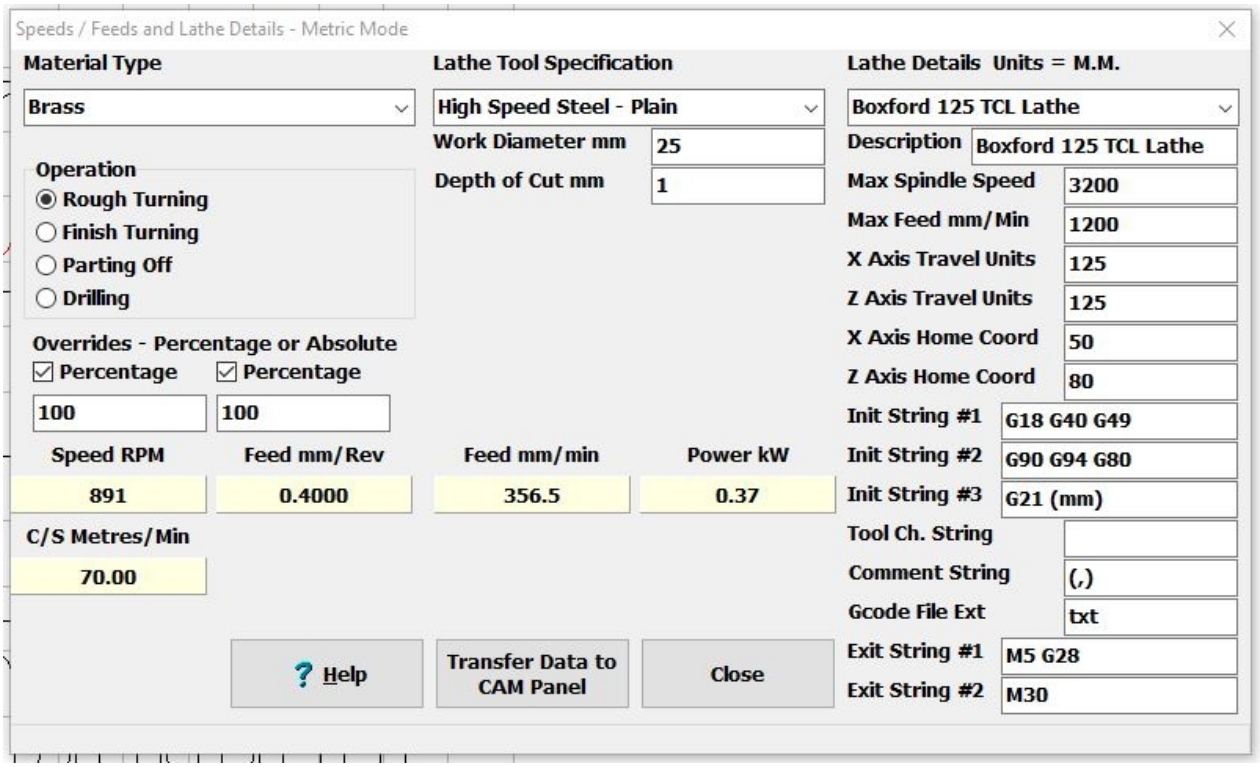
Deleting Entities

Entities may be deleted by "Checking" within the Entity List box, or selecting on screen and selecting ***"Delete"*** from the Pop.up menu.

Entities may be un-deleted by removing the check mark against it's listing in the entity list box.

Deleted entities are removed from the list when the program is opened next time.

Speeds/Feeds



Speed & Feeds calculator

This is a stand alone calculator for Speeds / Feeds (Some data may, however be exported to the Dxf processor)
The basic data is taken from the Materialm.txt data file, that can be user modified to more closely match your requirements.

The combination of Material Type, Lathe tool specification and work diameter are used to calculate the spindle speed required.
This speed is adjusted (slightly) by the operation involved.

Using the same data as above, feeds can be approximated for most applications, but feeds are much effected by other considerations.
The calculated feeds take into account the flexibility of the workpiece, reducing the feed as the workpiece diameter reduces.
However without "seeing" the length of the job (effect on Rigidity), these feeds can only be a rough guideline only.

A pair of overrides are available to tailor the output to your needs (Speed and Feed both can be overridden as a percentage, or an absolute value)

Power calculations are again guidelines only. Power is that used for cutting only, and takes no account of transmission losses in the spindle drive. A mental derate should be applied to your motor power to account for these losses. Taking that into account, the figures seem realistic.

The last of the Output boxes ***"C/S Meters / Min" or "C/S Feet / Min"*** just shows the actual cutting speed achieved at the ***"Speed RPM"***

Lathe Details

There is an allowance for data on up to 4 lathes to be entered here, and the active lathe will be showing in the edit boxes. Not all this data is currently used, but may be in the future.

Data that is currently used :-

"Max Spindle Speed" - The limit of Spindle Speed used in the Speeds / Feeds Dialog box.

"X Axis home Coordinate" - The Home position as used when G28 is executed *. (In approx program coordinates, not machine coords)

"Z Axis home Coordinate" - The Home position as used when G28 is executed *. (In approx program coordinates, not machine coords)

"Init String #1..3" - These 3 Strings are used in Full Program Mode as Initialization values for the Program's Active G-codes.

"Tool Ch String" - This string if Length > 1 adds a G code line after all Tool changes that consists of this string.
This could be used to Turn Coolant On (M7, M8).

"New V1.6" "Comment String" - Comment Start , Optionally Comment End. By default as Mach3 i.e. (,). If using system other than Mach3, Enter the Comment Start , Comment End (If Required) to suit your system. Note the comma is required between start and end if an end is required.

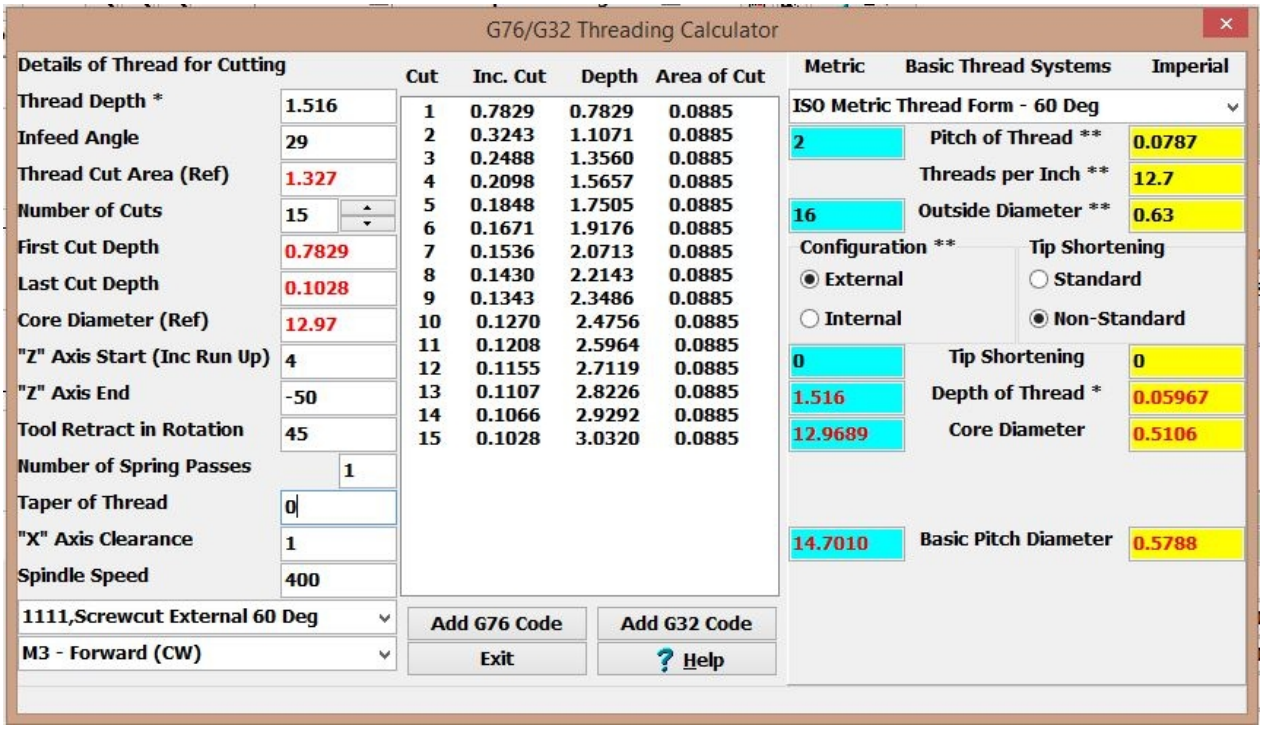
"New V1.6" "G code File Extension" - If using system other than Mach3, then your G code files may require a different extension. Enter here without the ".". default = txt.

"Exit String #1" - This string terminate all program fragments. Usually "M5 M9 G28" (Stop spindle, Coolant Off Go to tool change location) .

"Exit String #2" - This string terminate the Full Program. Usually "M30" (Program End and Rewind).

*** Note -** Home position is only used to calculate Rapid travel distance to home only - Does not effect lathe execution. (Used for Program Run Time)

Threading



This Dialog box is designed to assist in the production of threaded parts on the CNC lathe. Firstly enter your thread details, then the parameters required to actually cut the thread. The cuts are calculated to maintain a constant area of cut for maximum efficiency and consistency in threading.

You may enter thread data as you wish in metric or imperial, the data is all kept in sync. For the G76 / G32 section, data is taken from the required side (Yellow = Imperial or Blue = Metric) depending on the EziLathe mode (Metric or Imperial).

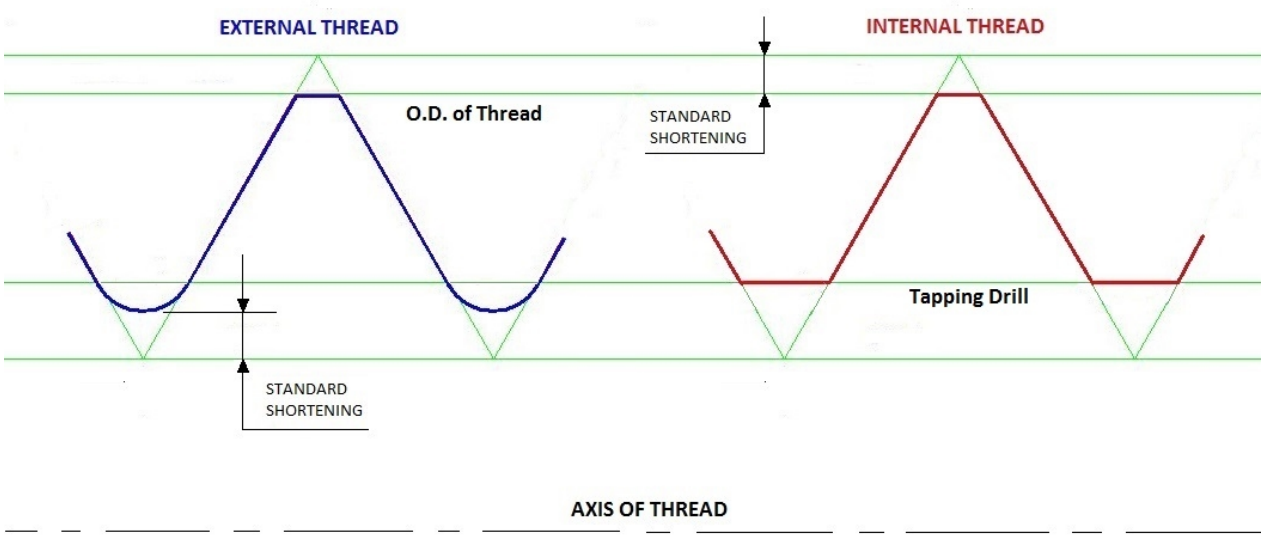
Edit boxes with red text are read only. Thread depth, Infeed angle are entered from the Thread details, but may be changed manually in the G76 / G32 Calculator if required.

Thread Calculator

The inbuilt thread calculator is designed to quickly calculate the cutting parameters that can be used in the G76 Thread calculator to actually cut the thread.
Even in the metric world we still cut Imperial threads, so both metric (Blue boxes) and Imperial (Yellow boxes) are available, and are kept in sync.
Data here is taken from the applicable basic thread forms as listed in the Machinery Handbook.

Knowing what Thread form to use and an Outside Diameter and Threads per Inch (or Pitch) is all that is required.

Metric or Imperial Mode only effects where the G76 / G32 Thread Calculator gets it's data (Blue box or Yellow box).



The calculator can adjust for Non-Standard Tip Radius or Flat on your screw-cutting tool. Tip Shortening controls the Distance from the standard maximum depth of cut to the apex of the sharp "V" of the thread form. Pick Non-Standard Shortening, and enter value in the edit box below.

e.g. when using a sharp "V" screwcutting tool, pick Non-Standard Shortening, and enter "0" in the edit box below.

G76 / G32 Threading Calculator

The G76 / G32 Threading Calculator allows the generation of a code fragment for screw-cutting in the lathe. This code is added to the end of the program in the editor, where it can be Cut / Pasted to where you require it. The code produced includes the normal code for the initial tool-change to the final return to tool change location.

Once all the data required is entered for your thread, and a number of cuts entered (use the arrow buttons, or type the number), the G76 and G32 buttons become available. The details of the cuts to be taken are displayed in the central box. The Cut Number, The Incremental cut, Depth reached and the area of each cut is displayed. By default, the cutting uses a constant area of cut. The optimum number of cuts depends on the thread, material, tooling used, and the lathe. For my type of work, I generally wind up the number of cuts until the **Final** incremental cut is in the range of 0.1 to 0.5mm (0.004 to 0.002"). Works for me anyway, If using Inserts, follow the manufacturers recommendations.

Inputs Required (For the Basic Thread) :-

- ***"Thread Depth"*** - This is the depth supplied from the Basic Thread Systems side, However it can be manually adjusted here.
- ***"Infeed Angle"*** - This is generally slightly less than 1/2 the thread angle for the recommended modified flank cutting (or 1/2 the angle for flank cutting). This is fully discussed in other documents.
- ***"Thread Cut Area (Ref)"*** - Read Only. This is the Area of the Vee form cut.
- ***"Number of Cuts"*** - See Note 1. This can be entered directly, or the up / down arrows used.
- ***"First Cut Depth"*** - See Note 1. The Number of cuts may be based on this.
- ***"Last Cut Depth"*** - See Note 1. The Number of cuts may be based on this.
- ***"Core Diameter Ref"*** - Read Only. Actual Core diameter as produced with the tool used. Knowing this can avoid unexpected appearances of thread on thin wall workpieces.

Note 1 - These 3 inputs interact together to maintain a constant volume removal of material per pass (important to avoid power fluctuations and the effects). Changing any one of these will recalculate all.

Inputs Required (For the Job)

- ***"Z" Axis Start (Inc Run Up)"*** - The cut should start a couple of pitches clear of the job to allow the axis movement to stabilize. A thread starting at Z = 0.0 might see an input of 4mm (0.160") here.
- ***"Z" Axis End"*** - This is the end co-ordinate of the thread. This is not the Length of thread, it is the co-ordinate of the Z axis.
- ***"Tool Retract in Rotation"*** - Called Chamfer in some places, but is the spindle rotation angle used for the tool to dis-engage from the workpiece. (45 or 1/8 turn is often used)
- ***"Number of Spring Passes"*** - The number of additional passes (with no "X" axis change) to clean-up the effect of any spring in the workpiece. Use Sparingly especially on some materials that may workharden.
- ***"Taper of Thread"*** - Entered in degrees of included angle for Taper threads, like Pipe Threads.
- ***"X" Axis Clearance "*** - The Distance above the Thread Outside Diameter (or Below the Thread Core diameter for Internal threads) used for the "X" axis when tool is retracting.
- ***"Spindle Speed"*** - Enter the Spindle Speed here (Not imported from other areas).

Note on Taper Threads - BSP / NPT Threads may have a taper of whatever. This is entered as the included angle. Outside Diameter is entered at the small end (including run-up), so may be smaller than you think.

Revision History

Initial Upload to Mach3 forum V1.1.0.0 on 22 June 2014

V1.1 +

- Threading - Minor corrections to input processing.
- Functions - Minor improvements to Metric/Imperial and line number functions.
- Dxf processor - Major corrections to arc processing, and corrected display of arc direction on text page.
- Cad4Lathe - Continued Additions / Corrections.
- Cad4Lathe - On Processing, File Tempdxf.dxf (entities only) created for import into external cad systems.
- Simulator - Warning added - "Program Stop" (M02 / M30) in body of program (Display stops at this point). Also warning added for missing program stop.

V1.2 (Uploaded to Mach3 forum November 2014.)

- Dxf processor - Warning issued if Stock origin "Z" and DXF profile do not match.
- Simulator - Now includes a true "Simulator" in addition to the static "View" mode. Generates a "Red" area when tool contacts material in rapid motion (G00).
- Simulator - Warning issued if Material origin and G code do not match.
- Simulator - Additional G codes simulated, including G52 Origin shifts and subroutines.

V1.2.1 (Uploaded to CNCzone March 2015)

- Minor bugs removed from DXF processor (Handling of vertical lines and gaps in profile).
- Additional page added to tool editor. (1 item so far - Length of cut for default tool)
- Additional error protection in tool editor. (Tool number formalized in selection field to aid output in correct format)
- Number of tools available increased to 99 in tool editor.

V1.3.0.0 (Uploaded April 2015)

- DXF processor - More bug removal, and restructure for easier debugging in the future.
- G32 threading added and threading editor improved.
- Tooling editor improved. Bugs fixed, and additional protection added.
- Printed Worksheet now added to Simulator.

V1.4.0.0 (Uploaded June 2015)

- Recent files dialog box added to allow easier repeat file selection. (available where all file types are loaded)
- Zoom window function added to simulator, to avoid multiple zooms to get in close. (Replaced zoom In / Out).
- Program run times added to simulator printout.
- Ongoing bug fixes, especially in DXF processor - Imperial mode.

V1.5.0.0 (Uploaded Feb 2016)

- DXF Processor - More bug removal.
- DXF Processor - Selection of Polyline sequences now available.
- DXF Processor - Improved selection re-ordering with addition of "Move To" function.
- Simulator - 1 pass check box added to Simulator panel.

V1.6.0.0 (Uploaded June 2017)

- DXF Processor - More Bug Removal (Hopefully no additions) and New Modes Added.
- Simulator - Parameters and Equations added.
- Simulator - Relative dimensions displayed when Line is selected
- Tool form - Radii added to Trepan and Groove tools
- Simulator - Now "Block Delete" Character "/" recognized, and controlled via check box
- DXF Processor - Add Temporary Overrides.
- Main - Add support for Reversed Arcs.

Wish list (I will get to it sometime)

- Cad4Lathe - Yet to be Finished (but getting there..slowly).
- Simulator - Tool Display (Currently you can see the progress of the cut, but not the tool).
- Taper threads - MAY use input dia as small end and reduce to suit automatically for threading run-up.

Final Thoughts

Safety Emphasized

The "EziLathe" program is used to produce and/or optimize G-code programs for use on CNC lathes. Potential dangers exist for both personnel and machine tools, if the work is not carried out in a safe manner. No guarantee is given that G-code generated is safe to run on any CNC machine, It must be checked carefully before use, and then initially by “Dry Run – without tooling” before any metal is cut. Speeds and Feeds, Excessive Gouging, and collisions especially in rapid feed are all potential dangers in the use of CNC machinery.

I just thought i would catch those who always check out the last page First.

Happy CNC'ing.