

Using the Tormach PCNC Duality Lathe

Programmer's and Operator's Guide to the Duality Lathe



© 2008 Tormach® LLC All Rights Reserved

Questions or comments?
Please email us at:
tech@tormach.com



PCNC Duality Lathe Manual
Part Number 31023 – Rev A1-2

Contents

1.	Preface	7
1.1	Manual Overview.....	7
1.2	Safety.....	7
1.2.1	Safety Publications	7
1.2.2	Operator Safety.....	7
1.2.3	Electrical Safety.....	9
1.3	Performance Expectations	9
1.3.1	Cutting Ability.....	9
1.3.2	Understanding Accuracy	10
1.3.3	Resolution, Accuracy and Repeatability of the PCNC.....	10
1.4	Scope and Intellectual Property.....	10
1.5	Nomenclature.....	11
2.	Unpacking & Installation	13
2.1	Unpacking & Preparing for Use	13
2.2	Manual Mode Installation.....	14
2.2.1	Setting the Duality Lathe onto the Chip Tray	14
2.3	CNC Installation	14
2.3.1	Mounting the Duality Lathe to the PCNC 1100.....	14
2.3.2	Electrical Connections	15
2.3.3	Quick Change Tool Post Installation	17
2.3.4	Installing the Machine Control Software.....	18
3.	Familiarization with Lathe Controls.....	19
3.1	Overview	19
3.1.1	Headstock.....	19
3.1.2	Running Gear	19
3.1.3	Tailstock.....	19
3.1.4	Carriage.....	19
3.2	Headstock and Spindle.....	20
3.2.1	3-Jaw Chuck and other Work Holding.....	20
3.2.2	Backgear.....	21
3.2.3	Speed, Direction and E-Stop	21
3.2.4	Running Gear for Manual Threading and Self-acting Feed.....	22
3.2.5	Spindle Indexing Clamp.....	22
3.3	Tailstock.....	23
3.4	Carriage and Manual Toolpost.....	23
3.4.1	Controls.....	23
3.4.2	Toolpost options	24
3.4.3	Configuration for CNC Operation	24
3.5	Quick-change CNC Tool-post and Tool-holders	25
4.	Making your First CNC Part.....	27
4.1	First Part Overview	27

4.2	Setting up the Stock and the Tool	27
4.2.1	Mechanical setup	27
4.2.2	Software familiarization	27
4.2.3	Setting the tool center height	29
4.2.4	Setting work coordinates	30
4.3	Loading and Previewing the code	32
4.4	Cutting the Bishop	33
5.	Manual turning	35
5.1	E-Stop circuitry.....	35
5.2	Adjusting Tool Bit Height.....	35
5.3	Turning	35
5.3.1	Turning manually.....	35
5.3.2	Turning with Power Feed	36
5.4	Facing.....	36
5.5	Turning Angles	37
5.5.1	Compound Slide Rotation	37
6.	Routes from an Idea to a Part	39
6.1	Introduction	39
6.2	Using Visicycles.....	39
6.2.1	Turning a length to size	40
6.2.2	Facing the end of stock	41
6.2.3	Facing and setting Z to zero	41
6.2.4	Storing data for multiple operations	42
6.2.5	Cutoff Visicycle.....	42
6.2.6	Chamfer Visicycle	42
6.2.7	Thread Visicycle.....	42
6.3	Using Code Written by Wizards.....	43
6.3.1	Turning Wizards	44
6.3.2	Threading Wizard	46
6.3.2.1	Thread depth.....	46
6.3.2.2	Tool change	46
6.3.2.3	Cut depths and spindle speed.....	46
6.3.2.4	Multiple start threads.....	47
6.3.2.5	Threading bores	47
6.3.2.6	Settings.....	47
6.4	Using CAD/CAM	47
6.4.1	TurboCAD/CAM.....	48
6.4.2	Alibre and SprutCAM.....	49
7.	On-Screen Software Controls	51
7.1	Screen selection and Reset	51
7.2	Loading Family	51
7.3	Program Running	51
7.4	Axis position family.....	52
7.5	Toolpath and Modes	53
7.6	Jog Family.....	53

7.7	MDI.....	54
7.8	Feed Family.....	54
7.9	Spindle Family	54
7.10	Visicycles.....	55
7.10.1	To Family	55
7.10.2	Cutting Parameters.....	55
7.10.3	Current position	56
7.10.4	Commands.....	56
7.11	Offsets Screen.....	57
7.11.1	The Coordinate Equation.....	57
7.11.2	Setting Offsets	58
7.12	Settings Screen	58
7.12.1	IK Mode	58
7.12.2	Center Height	59
7.12.3	Lathe/Mill Exchange.....	59
7.12.4	Imperial Speed Calculator	60
7.13	Diagnostics screen	60
8.	Setting up a Job and Tools	61
8.1	Tool Center Height	61
8.1.1	Setting tools in holders.....	61
8.1.1.1	Setting with Quick Change post on Duality Compound Slide	62
8.1.1.2	Setting with no QC post on Compound Slide	62
8.1.2	Setting Y axis on the PCNC	63
8.2	Tool Table	63
8.2.1	The Tool Table	63
8.2.2	Setting X and Z Offsets	64
8.2.2.1	The X and Z work offsets with Master Tool.....	65
8.2.2.2	Setting Tool Table Entries.....	65
8.2.3	The virtual cutting point.....	66
9.	Milling work in the lathe.....	69
9.1	The spindle clamp	69
9.2	Maintaining coordinates moving from Turn to Mill.....	69
9.2.1	Principle of changeover.....	70
9.2.2	Distance between tools.....	70
9.2.3	Moving to changeover position	71
9.2.4	Setup in mill	71
9.2.5	Returning to Lathe	72
9.2.6	Summary.....	72
10.	Part-programming Language Reference.....	73
10.1	Some definitions.....	73
10.1.1	Linear Axes	73
10.1.2	Scaling input.....	73
10.1.3	Controlled Point.....	73
10.1.4	Co-ordinated Linear Motion	73
10.1.5	Feed Rate.....	74
10.1.6	Arc Motion	74
10.1.7	Coolant.....	74
10.1.8	Dwell.....	74
10.1.9	Units.....	74

10.1.10	Current Position	74
10.1.11	Selected Plane.....	74
10.1.12	Tool Table	74
10.1.13	Tool Change	75
10.1.14	Path Control Modes	75
10.2	Interpreter interaction with controls.....	75
10.2.1	Feed and Speed Override controls	75
10.2.2	Block Delete control	75
10.2.3	Optional Program Stop control.....	75
10.3	Tool File	75
10.4	The language of part programs	75
10.4.1	Overview	75
10.4.2	Parameters.....	76
10.4.3	Coordinate Systems	76
10.5	Format of a Line	76
10.5.1	Line Number.....	77
10.5.2	Subroutine labels	77
10.5.3	Word	77
10.5.3.1	Number.....	77
10.5.3.2	Parameter Value.....	78
10.5.3.3	Expressions and Binary Operations	79
10.5.3.4	Unary Operation Value	79
10.5.4	Parameter Setting.....	79
10.5.5	Comments and Messages	80
10.5.6	Item Repeats.....	80
10.5.7	Item order	80
10.5.8	Commands and Machine Modes.....	81
10.6	Modal Groups	81
10.7	G Codes.....	82
10.7.1	Rapid Linear Motion – G00.....	83
10.7.2	Linear Motion at Feed Rate – G01.....	84
10.7.3	Arc at Feed Rate – G02 and G03	84
10.7.3.1	Radius Format Arc.....	84
10.7.3.2	Center Format Arc	85
10.7.4	Dwell – G04	85
10.7.5	Set Coordinate System Data Tool and work offset tables - G10.....	85
10.7.6	Plane Selection - G17, G18, and G19	86
10.7.7	Length Units - G20 and G21	86
10.7.8	Return to Home - G28 and G30.....	86
10.7.9	Reference axes - G28.1	86
10.7.10	Threading - G32.....	86
10.7.11	Nose Radius Compensation - G40, G41, and G42.....	87
10.7.12	Scale factors G50 and G51	87
10.7.13	Temporary Coordinate system offset – G52.....	88
10.7.14	Move in Absolute Coordinates - G53.....	88
10.7.15	Select Work Offset Coordinate System - G54 to G59 & G59 P~	88
10.7.16	Set Path Control Mode - G61, and G64	88
10.7.17	Canned Cycle – High Speed Peck Drill G73.....	89
10.7.18	Canned Cycle – Threading G76.....	89
10.7.19	Canned cycle – Turning G77.....	90
10.7.20	Canned cycle – Facing G78.....	90
10.7.21	Cancel Modal Motion - G80.....	90
10.7.22	Canned Cycles - G81 to G89.....	90
10.7.22.1	Preliminary and In-Between Motion	91
10.7.22.2	G81 Cycle.....	91
10.7.22.3	G82 Cycle.....	92
10.7.22.4	G83 and G83.1 Cycles.....	92

Contents

10.7.23	Set Distance Mode - G90 and G91	93
10.7.24	G92 Offsets - G92, G92.1, G92.2, G92.3	93
10.7.25	Set Feed Rate Mode - G94 and G95	94
10.7.26	Set spindle speed mode – G96 and G97	94
10.7.27	Set Canned Cycle Return Level - G98 and G99	94
10.8	Built-in M Codes	94
10.8.1	Program Stopping and Ending – M00, M01, M02, M30.....	94
10.8.2	Spindle Control – M03, M04, M05.....	95
10.8.3	Tool change – M06.....	96
10.8.4	Coolant Control – M07, M08, M09	96
10.8.5	Re-run from first line - M47	96
10.8.6	Override Control - M48 and M49.....	96
10.8.7	Call subroutine - M98	96
10.8.8	Return from subroutine	97
10.9	Macro M-codes	97
10.9.1	Macro overview	97
10.10	Other Input Codes	97
10.10.1	Set Feed Rate - F.....	97
10.10.2	Set Spindle Speed - S.....	97
10.10.3	Select Tool – T	97
10.11	Error Handling	98
10.12	Order of Execution	98
11.	Tool Tip Radius Compensation	99
11.1	Introduction to Compensation.....	99
11.2	How compensation is specified	100
11.3	Potential difficulties	101
12.	Warranty, Specification and Maintenance	103
12.1	Intended Use Statement	103
12.2	Support.....	103
•	This manual – ALWAYS the first place to check!!	103
•	Related documents found at: http://www.tormach.com/documents.htm	103
•	Our website at: www.tormach.com	103
•	Email to: info@tormach.com	103
•	Telephone Tormach at: 608-849-8381	103
•	Fax Tormach at: 209-885-4534.....	103
12.3	Outside of the Scope of Intended Use	103
12.4	Specification	104
12.5	Maintenance.....	104
12.5.1	CNC mode.....	104
12.5.2	Manual mode.....	105
12.5.2.1	Adjustments.....	105
12.5.2.2	Lubrication	107
12.5.3	Acknowledgements.....	108
13.	Appendices	109
13.1	Appendix 1 – Exploded Parts Views.....	109
13.2	Appendix 2 – Advanced Manual Turning Operations	110

Preface

13.2.1	Threading	110
13.2.2	Change Gears	110
13.2.3	To change the gears	111
13.2.4	Threading Dial.....	111
13.2.5	Tool and Setup.....	112
13.2.6	Threading Process.....	113
13.2.7	Acknowledgements.....	113
13.3	Revision history.....	114
14.	Index.....	115

1. Preface

1.1 Manual Overview

This manual is intended for users of the PCNC Duality Lathe. This lathe is an accessory to the Tormach PCNC1100 Mill and readers are assumed to be familiar with the documentation for that machine. Details of generic operations may not be covered in this manual. If you have questions of a general nature that are not covered in this manual, be sure to check your PCNC 1100 mill manual.

The Duality Lathe can be used: (a) for conventional manual turning both when it is mounted on the mill table or when it is freestanding on the bench and (b) as a CNC lathe. Additionally as the spindle can be clamped and milling operations can be performed on work mounted in its chuck.

The Duality Lathe and its tooling can be mounted and de-mounted very quickly in an accurately repeatable position thus simplifying setup operations.

Some users may wish to experiment with the lathe manually before going to CNC operation and this manual is laid out with this in mind. The chapter introductions indicate which parts you may wish to omit if you are going for CNC operation.



1.2 Safety



This manual offers guidance on safety precautions and techniques when using the Duality Lathe. Because Tormach does not know the details of your workshop or other local conditions, no responsibility can be taken for its performance or any damage or injury caused by its use. It is the owner's responsibility to ensure there is an understanding of the implications of the system integration and operation and to comply with any legislation and codes of practice applicable to country or state. If there is any doubt, seek guidance from a professionally qualified expert rather than risk injury.

Any machine tool can be potentially dangerous and automated tools are not different. The PCNC1100 and the Duality Lathe can deliver forces strong enough to break tools, bones, and skin.

The safety of any application using the Duality Lathe is ultimately the responsibility of those performing setup and operation. Please do not take this responsibility lightly.

Always use common sense when using a power tool. Review this section carefully in conjunction with any safety instructions shipped with the Duality Lathe.

1.2.1 Safety Publications

Tormach recommends the following publications for assistance in enhancing the safe use of this machine.

- Safety Requirements for The Construction, Care and Use of Drilling, Milling and Boring Machines (ANSI B11.8-1983). Available from The American National Standards Institute, 1430 Broadway, New York, New York 10018.
- Concepts and Techniques of Machine Safeguarding (OSHA Publication Number 3067). Available from The Publication Office – O.S.H.A., U.S. Department of Labor, 200 Constitution Avenue, NW, Washington, DC 20210.

1.2.2 Operator Safety

CAUTION: Read all instructions and warnings before using the Duality Lathe.

ABIDE BY THE FOLLOWING:

1. When using electric tools, machines or equipment, basic safety precautions should always be followed to reduce the risk of fire, electric shock, and personal injury.
2. Do not attempt jobs that are beyond its capacity.
3. Check the workpiece after placing it in the chuck or other work holding device.
NOTE: Be sure the workpiece is secure before powering the Duality Lathe.
4. Do not run this Duality Lathe without knowing the function of every control key, button, knob or handle. Refer to this manual or contact Tormach if any function is not understood.
5. Do not run the Duality Lathe in the CNC mode without the E-Stop integration feature connected.
6. Keep work area clean. Cluttered areas invite injuries.
7. Do not use flood coolant with the Duality Lathe.
8. Wear ANSI approved eye protection at all times.
9. Wear appropriate safety shoes.
10. Never use compressed air to remove chips or to clean the Duality Lathe.
11. Avoid contact moving parts. Before operating the Duality Lathe, remove all jewellery including watches and rings, neckties and any loose-fitting clothing. Tie back hair.
12. Keep hair away from moving parts.
13. Take off gloves before operating the Duality Lathe.
14. Never operate with an unbalanced workpiece.
15. Remove all tools (wrenches, chuck keys, etc.) from the mill spindle and Duality Lathe before operation.
16. Use adequate and secure work clamping.
17. Never operate the Duality Lathe after consuming alcoholic beverages or taking strong medications.
18. Stop the Duality Lathe spindle and ensure the computer control software is stopped before:
 - Changing a tool.
 - Changing parts or adjusting the work piece.
 - Changing the headstock gear ratio.
 - Clearing away chips or oil.
 - Making adjustments to the part, fixture, or taking measurements.
 - Removing protective shields or safeguards.
19. Do not reach for the part, tool or fixture around a guard.
20. Keep work area well lit.
21. Keep the computer area clear of clutter.
22. Avoid being pinched in places where the table, saddle or spindle head create “pinch points” while in motion.
23. Always use proper feeds and speeds, as well as depth and width of cut, to prevent tool breakage.

24. Use proper cutting tools for the job. Do not use dull or damaged cutting tools.
25. Never use longer or larger tools than necessary.
26. Be aware, chips and dust from certain materials can be flammable. Fine dust from normally non-flammable materials can be flammable or even explosive.
27. Be aware, chips and dust from certain materials can be toxic. Vapours from certain overheated materials can be toxic. Always check a Materials Safety Data Sheet (MSDS) of suspect materials. Refuse machining work requests of unknown materials.
28. If in any doubt, seek guidance from a professionally qualified expert rather than risk injury
29. Do not use the Duality Lathe or power tools in damp, wet or poorly lit locations.
30. Do not use tools in the presence of flammable gases or liquids.
31. All children should be kept away from the work area.
32. Guard against electric shock. Prevent body contact with grounded surfaces such as pipes, radiators, ranges, and refrigerator enclosures.
33. Stay alert while operating the Duality Lathe.
34. Keep proper footing and balance at all times.
35. Use only identical replacement parts when servicing.
36. Be certain every operator understands the operation and safety requirements of the Duality Lathe before operating. It is the responsibility of the owner of the Duality Lathe to provide and ensure point of operation safeguarding per OSHA standards.
37. Be sure all switches are OFF when not in use and before plugging in to a wall outlet.
38. Do not use inappropriate attachments in an attempt to exceed the tool's capacity.
39. Check for damaged parts. Before using any tool, any part appearing damaged should be carefully checked to determine it will operate properly and perform its intended function.
40. Check for alignment and binding of all moving parts.
41. Check for broken parts or mounting fixtures and any other condition affecting proper operation. Do not use the Duality Lathe if any switch does not turn off and on. Any damaged part should be properly repaired or replaced by a qualified technician.

1.2.3 Electrical Safety

The Duality Lathe is designed for operation from a 115 volt AC supply and is fitted with a suitable three-prong plug; the third (round) prong is the ground. Plug this cord only into a three-prong receptacle. Do not attempt to defeat the protection the ground wire provides by cutting off the round prong. Cutting off the ground will result in a safety hazard and void the warranty.

Caution: Do not modify the plug in any way. If unsure about the connections, call a qualified electrician.

1.3 Performance Expectations

1.3.1 Cutting Ability

The Duality Lathe benefits from the stiffness of the PCNC1100 and the mass of the head-mounted tool post. This means that you can expect to get a better surface finish in CNC mode than in manual turning.

The overall metal removal rate is limited by the spindle power of the lathe and by the cutting methods used. The control software has features for you to assess the load you are putting on the spindle in your turning. Metal removal rates will be increased through the use of sharp tooling and high positive rake angles.

As with all machining of "difficult" materials you will encounter difficulties on the Duality if you use too small a depth of cut or too small a feedrate. If the tool rubs rather than cuts the material will become hardened and very difficult to machine. This problem is more acute with work hardening materials such as stainless steel.

1.3.2 Understanding Accuracy

While a machine tool may seem absolutely rigid, the truth of the matter is that everything has some elasticity. Related to elasticity is the compressibility of components such as ball nuts and bearings. Preloading of bearings and ballscrews can remove the physical open space between moving parts, but the technique cannot eliminate compressibility. The key to achieving maximum accuracy is understanding and controlling the magnitude and direction of forces. Maximum accuracy is achieved when the forces are minimized, as occurs in a finishing cut. Maximum repeatability is achieved when the forces are repeatable, both in magnitude and direction.

1.3.3 Resolution, Accuracy and Repeatability of the PCNC

The minimum discrete position move is 0.0001", this is the resolution of motion command. Machine accuracy is closely related to ballscrew accuracy. Our ballscrews are accurate to 0.0006" per foot, but considering all the other factors that come into play, we prefer to keep accuracy expectations to 0.0013" per foot. Repeatability should be better than 0.001" per foot. In turning the effect on the part diameter will, of course, be twice these values.

Machining is a mix of science, skill and art. These numbers represent the accuracy and repeatability of machine motion, not those of the finished work. The caveat in stating accuracy and repeatability is that these factors depend on the techniques used by the machinist. A skilled machinist can deliver accuracy that exceeds the accuracy specified by the machine builder, while an inexperienced machinist may have difficulty delivering the expected accuracy. With this understanding, we cannot tell you what accuracy you will be able to achieve in your own work. Nevertheless, the accuracy specified by a machine builder remains an important reference point.

1.4 Scope and Intellectual Property

This document is intended to give enough details about how to install, setup and use your Duality Lathe. It assumes that you are familiar with the PCNC1100 mill and have appropriate experience and/or access to training for any Computer Aided Design/Manufacture software that you intend to use with the machine.

Tormach LLC is dedicated to continual improvement of its products, so suggestions for enhancements, corrections and clarifications will be gratefully received.

Tormach LLC, LittleMachineShop.com and John Prentice assert their right to be identified as the authors of this work. The right to make copies of this manual is granted solely for the purpose of training courses related to, evaluation of and/or use of the Duality Lathe. It is not permitted, under this right, for third parties to charge for copies of this manual beyond the cost of printing.

Tormach LLC wish to thank LittleMachineShop.com for permission to use material from their publications and draw users' attention to the wide range of accessories for lathes, many of which are suitable for the Duality Lathe, which are sold by that company.

Every effort has been made to make this manual as complete and as accurate as possible but no warranty or fitness is implied. The information provided is on an "as is" basis. The authors and

publisher shall have neither liability nor responsibility to any person or entity with respect to any loss or damages arising from the information contained in this manual.

Windows 2000, Windows XP and Windows Vista are registered trademarks of Microsoft Corporation. If other trademarks are used in this manual but not acknowledged please notify Tormach LLC so this can be remedied in subsequent editions

1.5 Nomenclature

This manual uses the following typographical nomenclature:

Software control

Refers to a Control Software “soft” control. (i.e., a Windows control on the PC screen).

Hardware Control

Refers to a physical button or switch on the Operator’s Panel of the machine.

G-code (e.g., G01X34.8)

Used to show G-code programs.

Key name (e.g., *Enter*)

Tells you to press the indicated key.

2. Unpacking & Installation

This chapter contains information which is applicable to users who wish to use manual and/or CNC operation. All users should read 2.1. Manual lathe users should read 2.2 and CNC users should read 2.3

2.1 Unpacking & Preparing for Use

Upon receipt, carefully unpack the Duality Lathe and inspect to ensure damage did not occur in transit and to account for all parts. Should any damage be apparent, or parts are missing, please contact Tormach immediately.

With an assistant, lift the Duality Lathe onto a sturdy surface or workbench to prepare it for use.

The Duality Lathe will arrive coated with grease to protect it from corrosion during shipment. Follow this procedure to remove the grease:

Wipe most of the grease off with rags or paper towels after spraying with a petroleum based solvent (e.g. White Spirit).

Coat the machined surfaces with a thin oil or WD40.

The machine should have been packed with:

1. 4 Rubber Feet
2. 3 External Jaws (for 3-Jaw Chuck)
3. Quick Change Tool Post
4. E-Stop Interface Kit
5. Connection Cable
6. 4 Hex Keys
7. Open End Wrench – 8 x 10mm
8. 1 Chuck Key
9. Open End Wrench -14 x 17mm
10. 1 gear set
11. 1 Plastic Oil Container
12. Turning and Facing Tool Holder
13. Cut-off Tool Holder
14. Turning and Boring Tool Holder
15. Boring Bar Holder
16. No.2 Morse Taper Centre
17. Morse Taper #2 Holder
18. E-Stop Jumper Block
19. Spare Spindle Drive Fuse
20. CD for install of Duality Lathe Software
21. 2 Plastic Handles w/Nuts and Bolts
22. Instruction Manual

Check these items.

All hex keys and wrenches necessary to carry out the various adjustments are supplied together with a chuck key for the 3-Jaw chuck and the spare fuse.

For transit purposes, the cross-slide feed handle has been mounted in reverse. To remove it, unscrew the hex socket head screw securing it to the cross-slide screw and mount it in the correct orientation. Check it moves freely, evenly, and smoothly.

Unpacking and Installation

Attach the plastic handles to the rims of the manual feed and tailstock feed hand wheels respectively, ensuring the nuts are tight and the handles spin freely about the bolts without excessive end play.

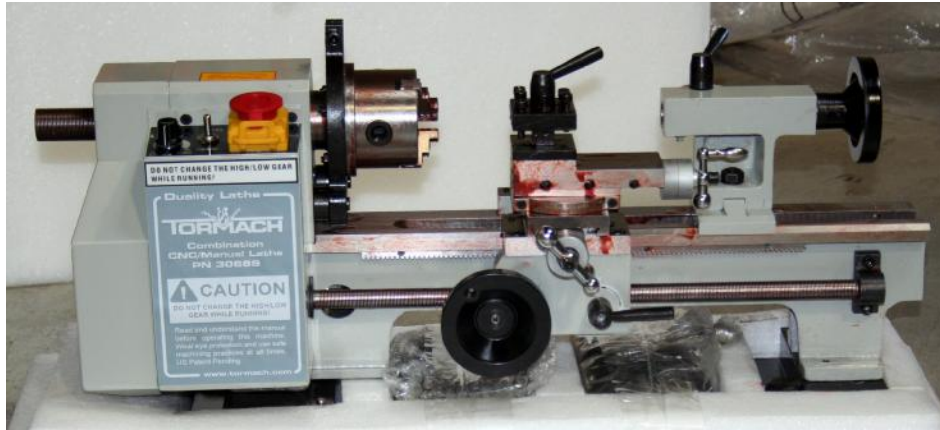


Figure 2.1 – Front of lathe as packed

The machine as shipped is illustrated in figures 2.1 and 2.2.

The carriage/saddle, cross-slide, and compound slide adjustments are all factory set to ensure smooth movement in both directions. However, the adjustments may have been upset during transit (indicated by stiff or erratic movement). Instructions for adjusting are given later.



Figure 2.2 – Rear of lathe showing back gear

2.2 Manual Mode Installation

2.2.1 Setting the Duality Lathe onto the Chip Tray

Clean the chip tray and ensure that the Duality Lathe's mounting feet are clean.

1. Orient the headstock end of the Duality Lathe on the side of the chip tray with 6" of surface beyond the bolts mounted to the padded feet.
2. Carefully place the Duality Lathe on the chip tray, allowing the pin guides to fall into the slots.
3. Slide it around a little bit to make sure it is sitting flat.

2.3 CNC Installation

2.3.1 Mounting the Duality Lathe to the PCNC 1100

1. Clean the PCNC 1100 table. Be sure to clean inside the T-slots. Apply a thin layer of lubricant.
2. Be sure the underside of the lathe mounting feet are clean
3. For maximum machining length capacity, line up the headstock end of the Duality Lathe's base plates with the inside edge of the PCNC 1100 table's side coolant gutter. This is shown in figure 2.3.

Note: If the extended guard is installed on the PCNC1100, the Duality Lathe should be placed on centerline to avoid conflict.

Unpacking and installation

- Carefully place the Duality Lathe on the PCNC 1100 table, allowing the pin guides to fall into the T-slots closest to the operator.

Note: Be sure the eccentric adjustment screw is neither too far up, nor down.

- Slide it around a little bit to make sure it is sitting flat and pull the Duality Lathe forward on the table so the pins locate on the front machined face of the front T-slot..

- Place the 3 T-nuts into the appropriate places and lightly tighten the nuts onto the studs.

- Place a dial test indicator on the PCNC 1100's spindle head. Set the dial to 0.000"

Jog the mill X axis to run it along the edge of the Duality Lathe's bed to check if the Duality Lathe is parallel to the table travel. The dial should stay within ± 0.003 " as it runs along the bed.

- If it is not, slacken the mounting nuts, carefully turn the eccentric adjustment bolt on the tailstock end of the Duality Lathe to adjust position and recheck until the lathe is correctly positioned.
- Fully tighten down the nuts when the Duality Lathe is square.



Figure 2.3 – Lathe position on mill table



Figure 2.4 – Aligning the bed when first mounting lathe

2.3.2 Electrical Connections

Mains Power

Check that the Duality Lathe is turned off (figure 2.5) – red E-stop button locked down. Plug the power cord into a grounded 115 volt AC outlet.

E-Stop

When mounted on the mill the Duality Lathe **MUST** be operated with emergency stop integration. In this mode the lathe and the mill are acting as one integrated machine. Proper and safe operation requires that their control systems be integrated as well through the E-stop integration cable. Initial use requires that the E-stop Integration Kit be installed in the mill. The E-stop Integration Kit provides the socket required to connect the E-stop Integration Cable. . Once it is installed the E-stop buttons on both the mill



Figure 2.5 – E-Stopped

Unpacking and Installation

and lathe will need to be "out" for either machine to work. Thus you can E-stop both machines with one hit on either of the buttons if something is wrong during machine operation.

1. On the back of the headstock, there is a 6" cable with a E-Stop Feedback Block (p/n 30786) plugged on it. (figure 2.6)
2. Unplug the dummy connector from the cable.
3. Safely store the dummy connector.
4. Find the E-stop Integration Cable (p/n 30787).
5. Attach the 5' Integration Cable to the 6" cable on the lathe.
6. Remove the Estop Feedback Block from the plug on the right hand end of the PCNC1100 control cabinet.
7. Take the free end of the 5' Integration Cable and plug it in where you removed the Feedback Block.. (figure 2.7)

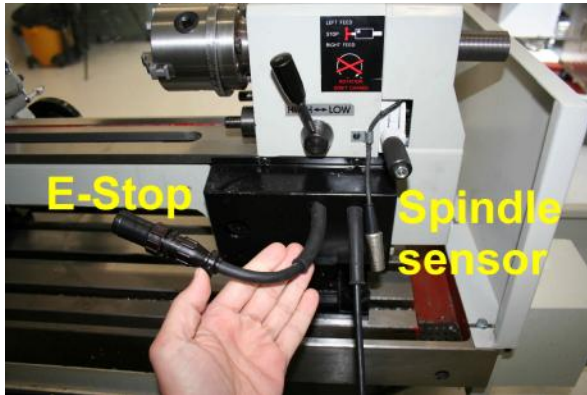


Figure 2.6 – E-Stop & Spindle cables, Lathe end

Spindle Sensor Cable

The Sensor Cable allows the machine control software to measure the spindle's instantaneous angle and speed.

1. Attach the Sensor Cable to the small cable found on the backside of the headstock next to Power Feed Lever. (figure 2.8)



Figure 2.7 – E-Stop cable, Mill end



Figure 2.8 – Connecting Spindle sensor, Lathe end



Figure 2.9 – Mill accessory socket

2. Take the opposite end of the Sensor Cable and attach it the plug marked "Accessory" on the PCNC1100 control panel.

2.3.3 Quick Change Tool Post Installation

Note: Before beginning installation, carefully examine the Quick Change Mounting Bracket. There are two pieces comprising the bracket; a $\frac{3}{4}$ " thick plate and a $\frac{1}{4}$ " plate. Undo the screw holding them together.

1. Make sure the nose of spindle is clean and smeared with oil.
2. Slide the $\frac{1}{4}$ " plate onto the spindle nose. Its clamping screw should be on the right hand side as you face the machine with the hex socket visible. Align the clamping slit to be parallel with the length of the table by eye. A steel rule held in the slot will help you do this.
3. With the plate touching the underside of the spindle head tighten the clamping screw.
4. Now slide the $\frac{3}{4}$ " plate onto the nose. Its clamping screw faces backwards and the tool post will be at the front on the left hand side. Figure 2.10 shows this part being put in place.



Figure 2.10 – Tool post clamping plates



Figure 2.11 – Locating screw

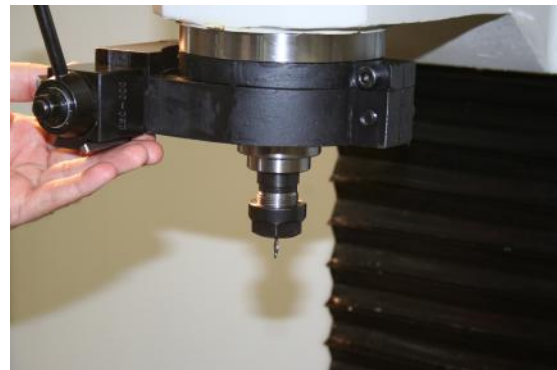


Figure 2.12 – Mounted tool post

5. While the thick plate is still loose insert the screw to fix it to the thinner plate and tighten it finger tight (figure 2.11).
6. Finally tighten the split-clamp screw on the $\frac{3}{4}$ " plate (figure 2.12).

Note: You should remove the lower, larger plate when you remove the Duality Lathe from your mill, but not the smaller upper plate. The upper plate is an alignment fixture that will allow you to re-mount the tool post accurately without the need for a dial indicator. The $\frac{3}{4}$ " side should be removed during standard milling operations.

7. Set a dial indicator off PCNC1100 table. Place an empty tool holder on the Quick Change Tool Post and run the dial along vertical surface by jogging the mill head up and down.
8. The dial should stay within $\pm .002$ ". If it is out of alignment, loosen the nut holding the tool post and tap it into adjustment. Tighten the nut and recheck the adjustment.

2.3.4 Installing the Machine Control Software

Included with the Duality Lathe is a CD containing the machine control software. To load the software:

Make sure the PCNC1100's mill machine control software is installed.

1. Load the CD containing the Duality Lathe's machine control software into the PC connected to the PCNC.
2. A welcome box will appear. Click **Next** to continue, or **Cancel** to stop installation.



Figure 2.13 – Software installer Welcome screen

3. If **Next** was clicked, the End User License Agreement box will appear. Click **Yes** to continue.
4. The Read Me Information box will appear. Click **Next** to continue.
5. The Select Destination Directory box will appear. This will allow a directory to be chosen to store the software files. Click **Next** to continue with the default C:/PCNC3 folder unless you have a non-standard mill installation. In that case use the folder name containing your mill control software..
6. The Start Copying Files box will appear. Click **Next** to continue.
7. The Setup Finished box will appear. Click **Finish** to complete the installation. You should have a DCNC logo on your desktop to allow you to run the Duality Lathe control software.

3. Familiarization with Lathe Controls

This chapter describes the mechanical controls of the Duality Lathe. If you are initially only going to do CNC turning you may wish to skip the description of the carriage and slides.

Familiarity with the machine controls is essential for safe working. Please study this chapter in conjunction with your machine.

3.1 Overview

3.1.1 Headstock

The motor provides a drive to the spindle via an internal tooth type belt. Spindle speed is variable, and is regulated by the Speed Control Knob located on the Duality lathe control panel.

The spindle is provided with an internal No.3 Morse taper to accommodate a center for use with a face plate or turning clamp.

The spindle has 6 holes drilled in its flange to accommodate a range of fixtures such as a Face plate or 4-jaw chuck.

The 3-jaw Self Centering Chuck is mounted on the Spindle Flange. To remove the chuck, simply remove the three securing nuts to the rear of the flange allowing it to be pulled free together with the three mounting studs.

Three external jaws are also supplied which extend the capacity of the chuck.

3.1.2 Running Gear

The Running gear is not used in CNC operation. It is protected by a cover on the left hand end of the lathe which is removed by unscrewing the two securing hex-head cap screws.

The gear train transmits drive from the spindle to the lead screw. By operating the Auto Feed lever on the apron, which engages a nut with the lead screw, drive is transmitted to the carriage/saddle and consequently the cutting tool. This provides a power feed for manual thread cutting and turning operations. The rotational speed of the lead screw, and hence the rate of feed of the cutting tool, is determined by the gear configuration.

The drive to the lead screw may be disconnected by operating the tumbler lever. Which is also used control the direction of feed.

3.1.3 Tailstock

The tailstock is used to support long work or to hold a chuck for drilling operations. The tailstock spindle has an internal No.2 Morse taper.

3.1.4 Carriage

The carriage is not normally used during CNC Turning, however it can be used while the lathe is mounted on the mill if there are no automated operations going at the same time. Some operators may prefer to do some final finishing operations manually if they have not bothered to program them. It carries the cross-slide onto which is mounted the compound (or top) slide with a tool post.

The position of the tool is controlled by turning the cross-slide feed handle, which moves it across the lathe, and the carriage or manual feed handle, which moves it longitudinally i.e.

parallel to the spindle axis. Additionally, the compound slide feed handle may be used to move the tool by small amounts along the axis or at an angle to it.

A 4-way tool post carries 8 hex socket head screws which are used to secure a cutting tool in any desired position. Four tool bits may be mounted for quick and easy changes. The tool post is rotated by slackening the lever on its top a sufficient amount so the post can be lifted slightly and then turned to the desired position. An optional quick-change tool post is available as an accessory which replaces the 4-way post and allows the same tools to be used for both manual and CNC turning with very little resetting being required.

3.2 Headstock and Spindle

The headstock and its spindle are responsible for rotating the work while it is being machined. Power is derived from an electric motor with a variable speed electronic drive.

3.2.1 3-Jaw Chuck and other Work Holding

Round work is generally held in a 3-jaw self centering chuck. Small diameter work is clamped using the inside face of the normal, inside jaws. These jaw can be removed and replaced with the outside jaws that are supplied if you need to hold larger diameter work. The Inner jaws can also be tightened with one or more of the steps inside tubular work or into a pre-machined bore. The two jaw sets are illustrated in figures 3.1 and 3.2



Figure 3.1 – Inner jaws of chuck



Figure 3.2 – Outer jaws pf chuck

Note: when you exchange the jaws you must insert them into the correct slots in the chuck body and the jaw marked "1" must engage with the start of the scroll-thread in the body first, followed, in turn by the jaws marked "2" and "3". If, after you have changed jaws, they do not close properly onto the center of the bore then you have inserted them incorrectly.

The chuck is "self-centering" in the sense that all the jaws move together. You cannot rely on work clamped in a self-centering chuck being perfectly concentric with the centerline. This does not generally matter provided you do not remove the work from the chuck and then replace it.

A 4-jaw chuck is available as an accessory. As its jaws move independently it can be used to hold square or rectangular stock and to adjust the position on an existing turned surface to be concentric with the centerline. A 4-jaw chuck is not often used with CNC machining processes.

The chucks are bolted onto a flange at the nose of the spindle. You may wish to attach custom accessories such as a collet holder or a faceplate to this flange.

A dead-centre can be mounted in the #3 Morse taper in the nose of the spindle for turning "between centers".

3.2.2 Backgear

The spindle has a two speed gear drive. In low gear speeds range from 100 to 1100 RPM. In high gear speeds range from 225 to 2500 RPM. Use the low gear for maximum torque and high gear for the smoothest quietest running.

The backgear is changed using a lever on the rear of the headstock (figure 3.3)

This lever must **not** be moved when the spindle is rotating.

Ensure that the gear is fully engaged by rocking the chuck to and fro while moving the lever.



Figure 3.3 – Backgear speed change lever

3.2.3 Speed, Direction and E-Stop

The speed on the spindle is continuously variable by an electronic drive. It is set by a control knob (figure 3.4) that is adjusted by the operator. When running with the CNC software the current spindle speed is displayed on the screen together with a bar-graph that compares the actual speed with the speed requested in the part-program.

A three position toggle switch between the speed control knob and the E-Stop switch selects Reverse-Off-Forward motion. Reverse running is not required for CNC operation but is used for some threads during manual screwcutting. In reverse rotation the spindle speed is cut in half.

The E-Stop button cuts off power to the lathe spindle motor and, when integrated with the mill, all power to the mill steppers and spindle. The knob locks in the down position. It is released by pushing backwards, parallel to the panel. Figure 3.5 shows it just after release.



Figure 3.4 – Spindle Control Panel



Figure 3.5 – Released E-Stop

3.2.4 Running Gear for Manual Threading and Self-acting Feed

The leadscrew can be driven from the headstock spindle through a user configurable change gear train (figure 3.6). The drive to this train and the direction of rotation is controlled by the tumbler control lever at the back of the headstock (figure 3.7). In CNC operation the leadscrew is not used so the tumbler lever should be in the central (neutral) position. Details of setting up the change gears for manual threading and power feed are given in Appendix nn.



Figure 3.6 – Change gears



Figure 3.7 – Tumbler gear lever

3.2.5 Spindle Indexing Clamp

The spindle can be clamped so it cannot rotate. Its position can be set using a built-in protractor scale. This feature allows CNC milling operations to be performed on work mounted in the lathe chuck. Examples might include machining a keyway in a shaft that has been turned, cross drilling a shaft or milling flats onto a custom designed fastener.

The assembled clamp is shown in figure 3.8. Figure 3.9 shows the front clamp shoe being positioned on its pin. Figure 3.10 shows details of the indexing protractor.

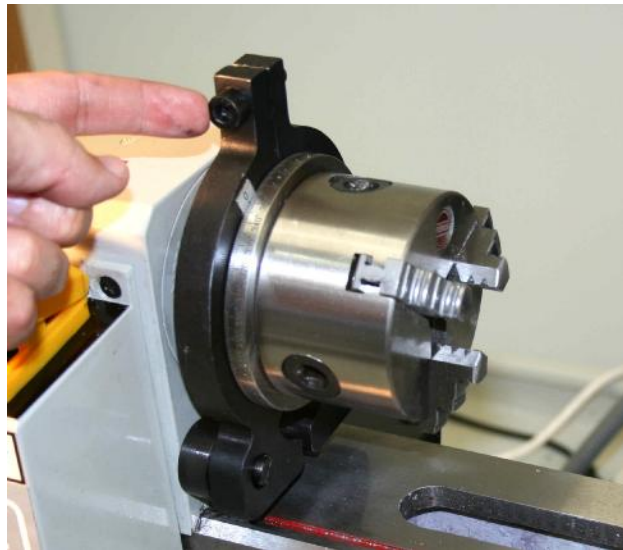


Figure 3.8 – Spindle Clamp In Place



Figure 3.9 – Clamp shoe installation



Figure 3.10 – Spindle lock index

3.3 Tailstock

The tailstock is used for supporting long workpieces and for manual drilling and reaming operations.

The handwheel advances and retracts the barrel. Its barrel has a #2 Morse Taper for holding the center or a drill chuck. The taper will be ejected if the barrel is fully retracted into the tailstock casting.

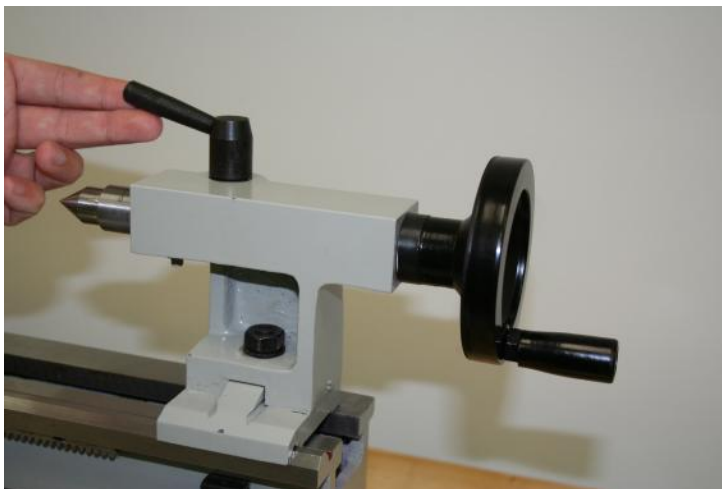


Figure 3.11 - Tailstock

The barrel is locked in position using the handle indicated in figure 3.11.

The tailstock is locked to the bed using the nut visible on its base. It may be removed from the lathe by sliding it off the right-hand end of the bed.

The front/rear alignment of the tailstock can be adjusted to ensure the tailstock is on the centerline of the spindle. See Appendix for details.

3.4 Carriage and Manual Tool Post

3.4.1 Controls

The carriage carries the cross-slide and the cross slide carries the compound-slide. The tool post is mounted onto the compound slide (figure 3.12).

The large black knob controls movement of the carriage along the bed of the lathe. The black lever engages the half-nuts with the leadscrew for screw cutting and self-acting feeds. Up is disengaged and down is engaged. The knob cannot be used to move the saddle when the half-nuts are engaged.

Lathe Controls

The lower chromed handle controls the cross-slide and hence the diameter of the work being turned. It has a thimble graduated in units of 0.01".

The upper chromed handle controls the compound slide. This can be set at any angle relative to the bed of the lathe. It is used for cutting short tapers or chamfers, feeding a screwcutting tool down the flank of the thread being cut or, when set parallel to the bed, to turn a particular length to size. It has a similar thimble to the cross-slide.

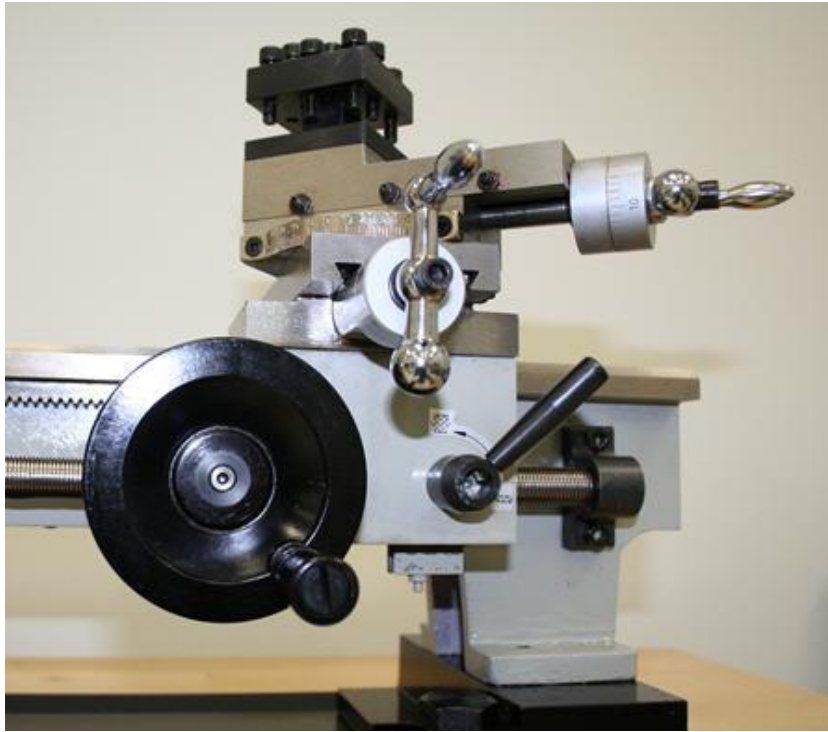


Figure 3.12 – Saddle controls

3.4.2 Toolpost options

As supplied the Duality lathe has a conventional 4-way tool post on the compound slide. With this it is necessary to use packing shims to bring the tools to the height of the centerline of the spindle.

A tool post similar to that used in CNC operation can be mounted in place of the 4-way post. This allows tools to be set to height by a simple adjusting screw and used interchangeably for CNC and manual turning. The tool holders are described below. The CNC quick change tool post is PN 31050. The CNC tool post can be identified by the fact that the locking wedge that holds the tool in place moves away from the operator handle as the tool is locked into place. The manual quick change tool post is PN 30704 and can be identified by the fact that the locking wedge that holds the tool in place moves toward the operator handle as the tool is locked into place.

Note: The tool **holders** are fully interchangeable between the two tool posts. With care the CNC post can be used on the carriage or the manual post be used for CNC but this is not recommended.

3.4.3 Configuration for CNC Operation

If you do not intend to do manual turning then you can get more working space and less chance of crashing the CNC turning tool if you partially or totally dismantle the saddle.



Figure 3.13 – Removing Compound Slide

The easiest option which requires no tools is to remove the top-slide by turning its handle counter-clockwise until it is free of its dovetail slides. This is shown in figure 3.13.

The option which gives the greatest access is to remove the entire carriage. To do this:

- Remove the tailstock.
- Remove the two cap-screws retaining the leadscrew bearing bracket and slide the bearing off the leadscrew.
- Wind the carriage to the right using the handwheel then carefully slide it to the right by hand until it is free of the bed.
- Replace the leadscrew bearing bracket.

Carefully store the removed components, taking care that no loose gib strips are lost.

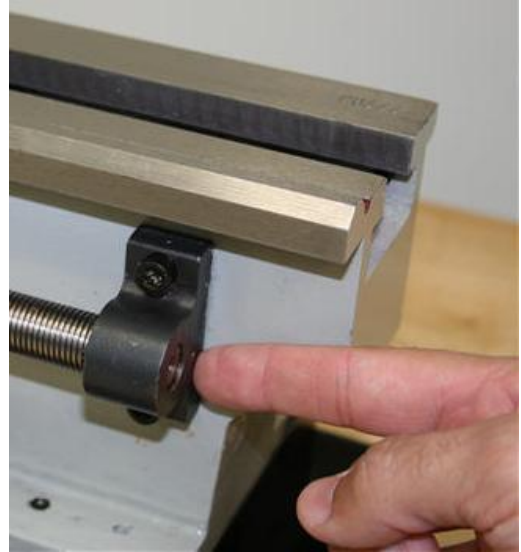


Figure 3.14 – Leadscrew bearing bracket

3.5 Quick-change CNC Tool-post and Tool-holders

The installation of the CNC tool post on the mill spindle nose has been explained in chapter 2. Individual tools are clamped into tool holders and these mount on a dovetail slide on the tool post. Tool holders are available for a range of different tooling from conventional turning tools to boring bars and Morse taper drills. Figure 3.15 shows some of these holders.

Notice that each holder has a knurled ring which can be locked off with a lock-nut. This ring bears on the body of the tool-post so setting the position of the tool relative to the centerline of the lathe. All tool-holders should be adjusted so that the distance from the underside of the ring to the top of the cutting tip is the same for all tools. This means that when one is set dead on center, by the process given in section xx.xxx, then all tools will be on the centerline.

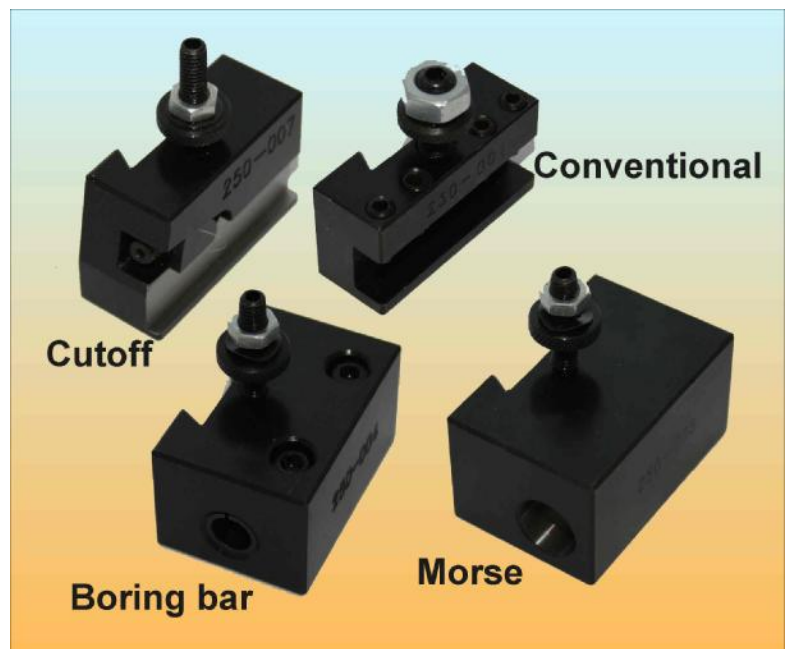


Figure 3.15 – QC Tool-holders

4. Making your First CNC Part

If you are familiar with manual turning your curiosity has probably got the better of you by now and you will have done some simple turning on the Duality in manual mode. Although there are many details of the CNC machine that are yet to be explained, you are now ready to make your first CNC part.

4.1 First Part Overview

We are going to present you with some recipes so you can make your first turned CNC part. In this chapter we will skip over explaining why you do things and take some shortcuts. You will work faster and get much better accuracy and finish when you understand the principles of setting up the machine having read later chapters.

FirstTurnPart.nc is a file on your release CD which contains the part program (i.e. G- and M-code instructions) for turning a chess bishop. You will need a short length of $\frac{3}{4}$ " diameter aluminum bar. 4" long is about right.

The machining is illustrated in the Duality Lathe introductory video at www.tormach.com so you can get an idea of what to expect.

4.2 Setting up the Stock and the Tool

4.2.1 Mechanical setup

Insert the stock into the 3-jaw chuck with the standard (internal) jaws. Tighten the chuck using all three square sockets for maximum security and accuracy.

Insert the left hand knife tool (figure 4.1) into a quick change tool holder and slide this forward onto the dovetails of the tool-post. Lock it with the tool-post handle (figure 4.2).



Figure 4.1



Figure 4.2 – Tool in tool-holder and tool-post

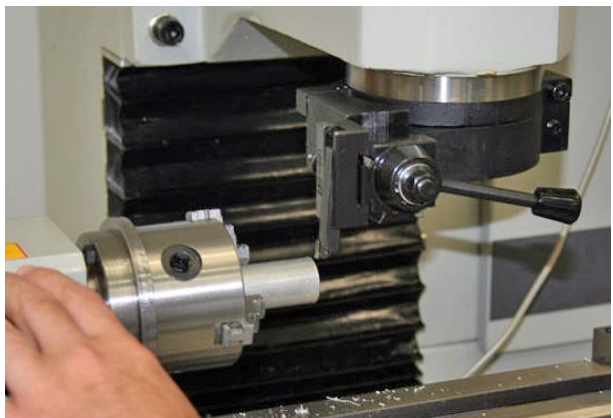


Figure 4.3 – Workpiece and tool

4.2.2 Software familiarization

Next you have to ensure that the software knows where the tool is relative to the stock.

Lathe Controls

Double-click the Duality lathe icon on the desktop.

The main screen of the lathe control software will be displayed (figure 4.4).



Note: On the first run of the system you will be asked to accept the disclaimer. You can indicate that you will always accept it with the check box.

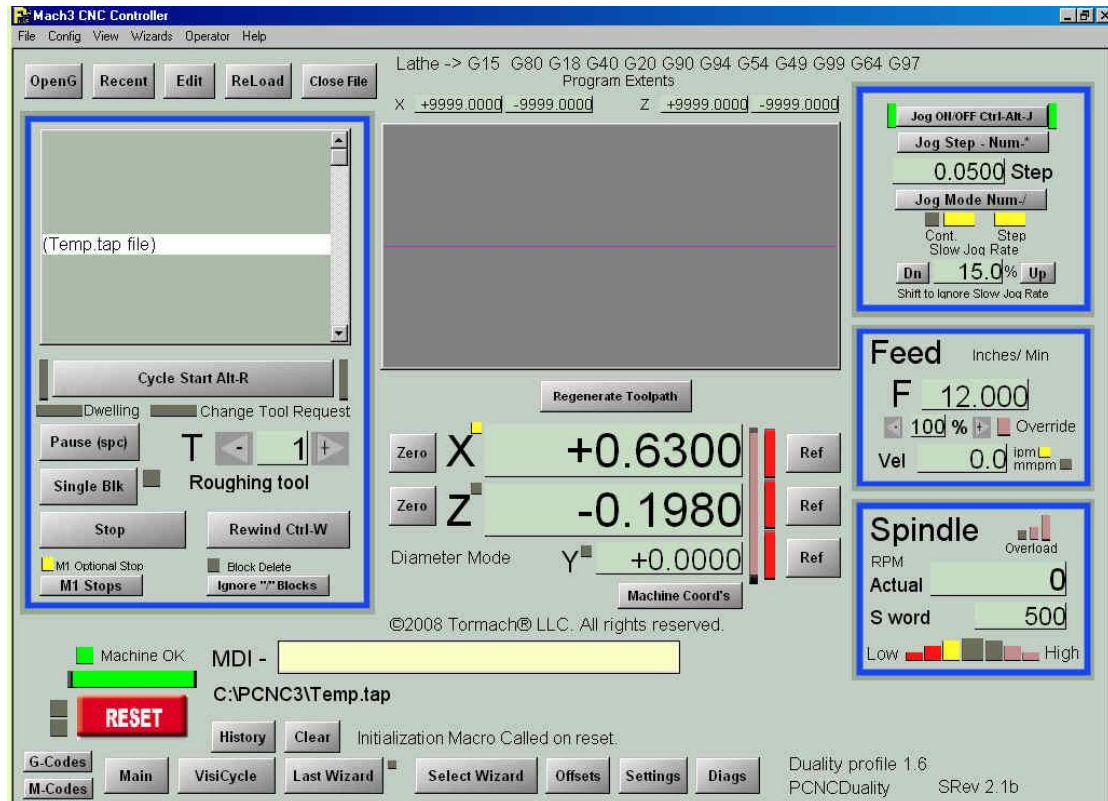


Figure 4.4 – Control Software (Main screen)

The two main axes on a lathe are X and Z. The position of the tool relative to the centerline of the lathe is called Y. It is only used during setup.

The directions in terms of the movement of the tool are shown in figure 4.5. In fact of course the Z and Y motions are the table moving but you should get used to thinking of the tools motion.

You need to pretend you are standing beside the lathe on the table.

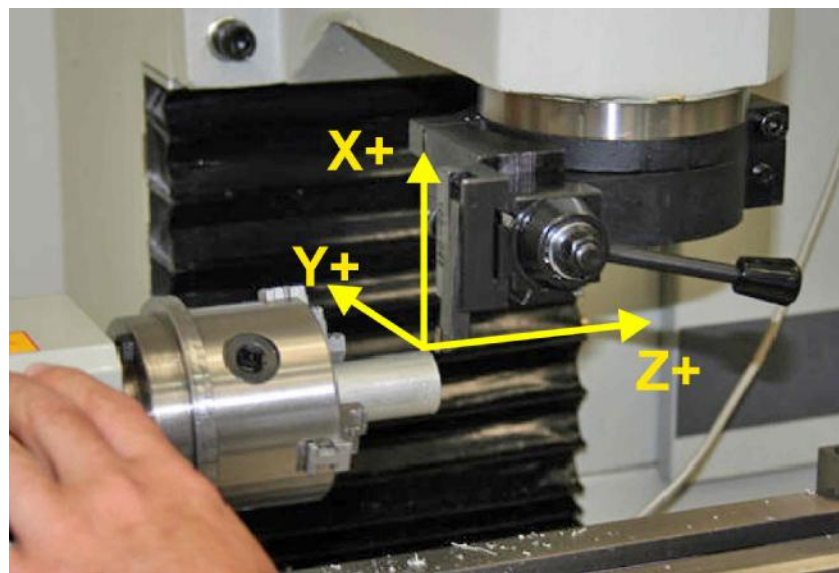


Figure 4.5 – Direction of axes in terms of tool motion

This may seem odd because it is different than the mill, but it is an industry standard and required in order to have your Duality Lathe conform to industrial standards. It may be easier

Making you first part

to digest if you imagine your mill turned 90 degrees counter clockwise, where the mill Z spindle becomes the lathe spindle.

Make sure that the tool is well clear of the chuck and practice jogging the axes until you are thoroughly familiar with which axes is which and which direction is positive.

You will find that you do more jogging when setting up the lathe than is need for setting up the mill. We strongly recommend the Tormach Jog/Shuttle controller (p/n 30616) because it provides a combination of rapid jog motion plus precision fine motion. It is essentially an electronic handwheel for the lathe.



Figure 4.6 – Spindle motor control

When you are confident jogging it is time to run the spindle. Choose Low Gear (see Chapter 3). Start the spindle in the Forward direction (figure 4.6). Use the speed control knob to adjust the speed using the digital readout (DRO) on the screen (figure 4.7).

At present it does not matter that you have not requested a speed using the S word from a Wizard or part-program. When you have done this then the Low/High bar graph will compare the current actual speed and the requested one and make it easy for you to adjust the knob achieve the right speed.

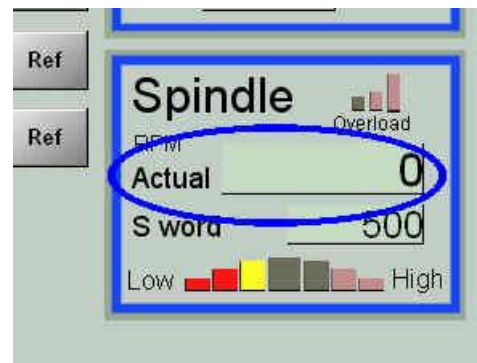


Figure 4.7 – Actual speed DRO

4.2.3 Setting the tool center height

The relationship of the cutting tip to a diameter of the work is important. If the tip is too "high" then the cutter will tend to rub or dig-in to the work. If it is too "low" then the cutting angles are wrong.

The terms "low" and "high" relate to tools in a conventional lathe tool-post view (a) in figure 4.8. The tool in the Duality is rotated 90 degrees clockwise as in view (b). Low means below the diameter in the direction of rotation of the work.

This height of the tool is set in two different ways. The position of the tool-holder relative to the tool-post is adjusted by the knurled ring on the holder and the relationship between the tool-post and the centerline of the lathe is adjusted by moving the Y-axis of the mill.

We will see later how to adjust the knurled rings so every tool has the same center-height.

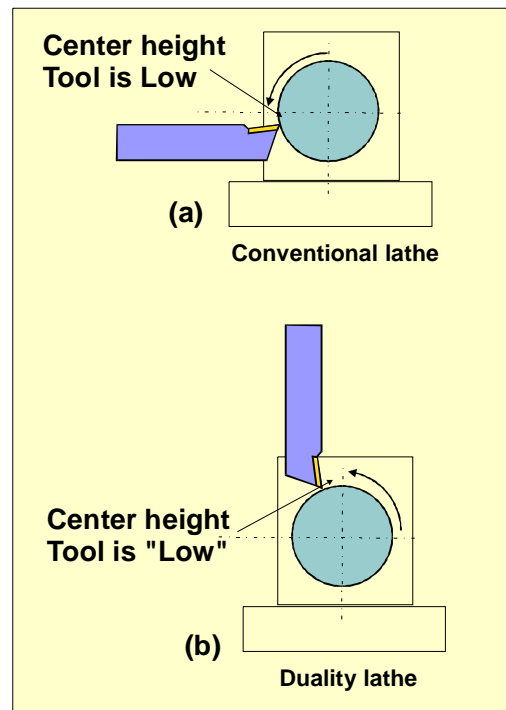


Figure 4.8 – Explanation of Center Height

But for now as we are only using one tool you can ignore the knurled ring setting.

First you should reference the Y axis to the home/limit switch. Endure that the tool is mill head is well above the lathe so that the tool cannot crash into anything when you move the Y-axis. Click the Ref Y button. The table will move away from you until the home switch is reached, the LED will turn from Red to Green and the Y axis will stop just off the switch.

Jog the tool so it is near the end of the 3/4" stock. Jog the Y-axis so the tool is obviously slightly low – e.g. as in figure 4.8. Zero the Y axis DRO

Now jog X and Z so the tool just touches the work. Zero the Z DRO at this position. Turn on the spindle and set it to around 1000 RPM.

Use the jogging controls to move the tool to face about 0.05" off the end of the bar. This should be done in one pass.

Because the tool is low, cutting may not be ideal and you will be left with a pip in the center (figure 4.10). Stop before as you reach the centerline in the X direction. With the spindle still running Jog the tool out of the way using first the Z axis and then the X axis. Do not move Y.

This will leave a neatly turned and parallel pip on the end of the bar. Measure the diameter of the pip.

Choose the Settings screen and enter the measured diameter into the Pip Diameter DRO (figure 4.11). Do not forget to press Enter to accept the value. Then click the screen button Measured pip diameter = to set the Y axis so its zero is on the centerline.

Ensure that the tool is clear of the work and press Goto Saved Centreline. The Y axis will move and the tool height will be correct whenever Y = 0.0.

4.2.4 Setting work coordinates

Finally we have to set the X-axis so its zero is the centerline of the lathe and set the Z axis so its zero is the end of the work.

For now we will only be using one tool so we will ignore the tool table. The procedure that follows is what you will do to set up the Master Tool when you do want to use the tool table.

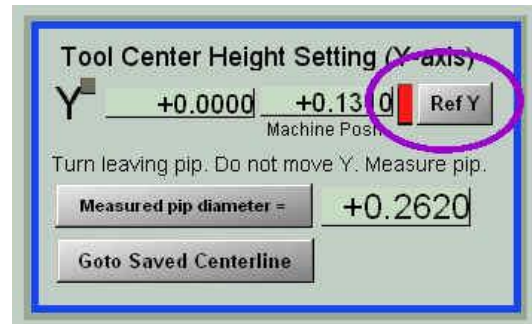


Figure 4.9 – Referencing Y axis



Figure 4.10 – Measuring "pip" to set center height



Figure 4.11 – Recording the Pip

Making you first part

We presume that you still have the $\frac{3}{4}$ " stock in the chuck. Display the Offsets screen (figure 4.12).

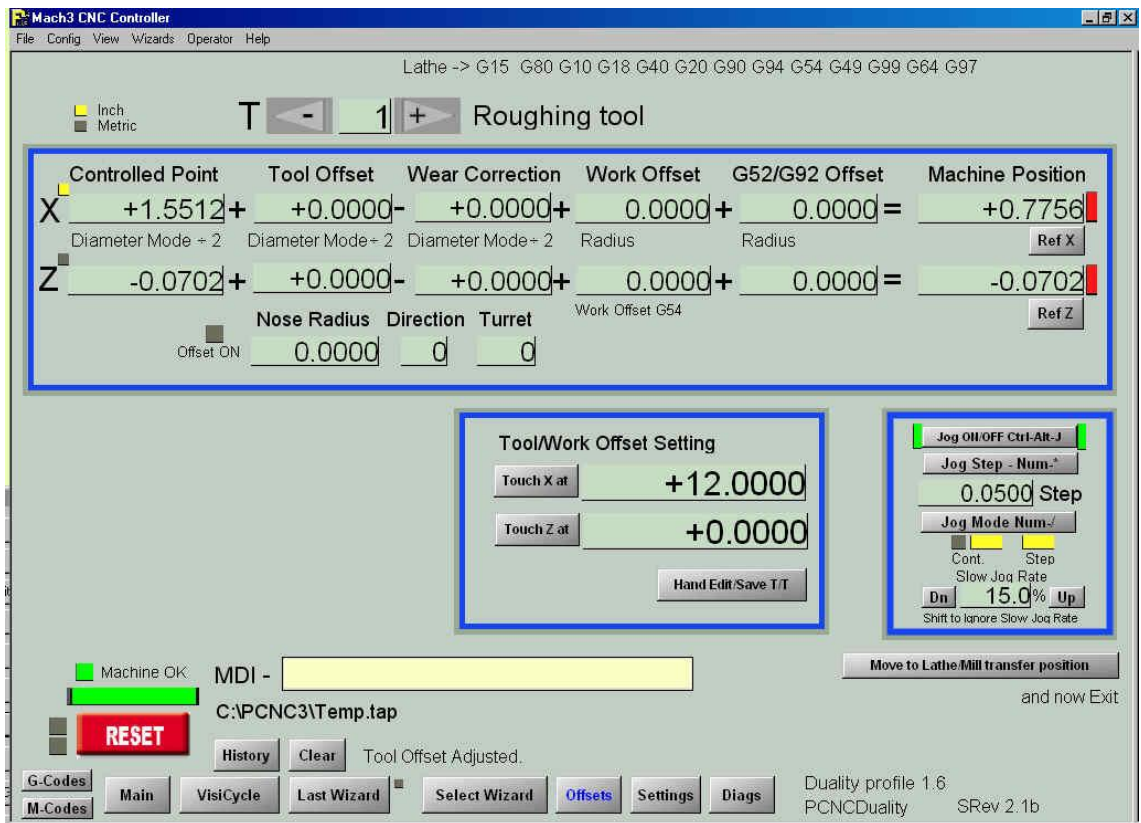


Figure 4.12 – Offsets Screen

This is similar in concept to the mill Offsets screen. The top section shows the calculation that the control software performs to convert the coordinates in your G-code to the absolute coordinates of the machine. You should use tool #1 for this FirstTurnPart and later always use it as the master tool.

Select tool #1 on the Offsets screen and make sure you left-hand knife tool is in the toolpost and it is set on the centerline.

Jog the X and Z axes so the tool is near the free end of the stock and the tip is clear of the corner.

Now using jogging, turn the stock down just enough to clean it up. A length of about $\frac{1}{2}$ " is fine. When you have done this move the tool out of the way using just the Z axis. **Do not** jog X. Stop the spindle and measure the diameter of the turned section of bar. Enter this measurement into the DRO next to the Touch X at button and click Touch X

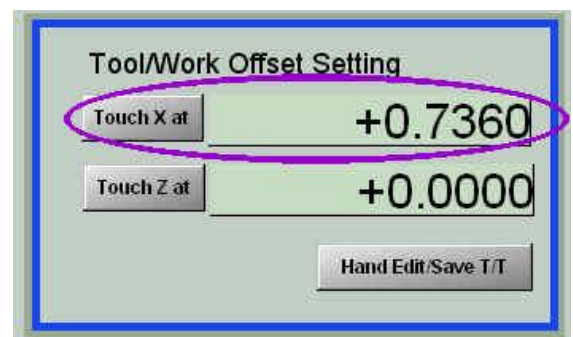


Figure 4.13 – Setting X offset with 0.736" dia.

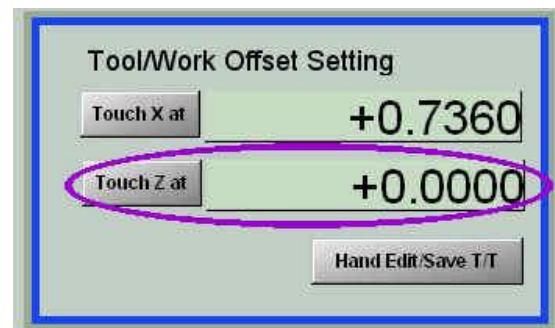


Figure 4.14 – Setting Z offset to Zero

at. This will set the X offset and the X axis DRO will read the diameter of the turned down stock (figure 4.13).

Now, using low speed jogging, move Z so the tool will take a light facing cut on the end of the bar. You will be able to remove the pip left when setting up the center height with the Y axis. When this is done, but **without moving** the Z axis, ensure that the DRO next to the Touch Z at button is set to 0.0 and click the Touch Z at button (figure 4.14). This will put a value into the Z Work Offset DRO and the Z axis DRO will read 0.0. All Z values in the part-program will be relative to the end of the stock.

Note: This looks very complicated when written out in detail. You will be able to do it in less than a minute once you are familiar with the techniques.

The other thing to remember is that you do not need to do everything each time you put new stock into the chuck.

(a) Provided you reference the Y axis whenever you load the Duality control software and use the Goto Centerline button then your tool height will be the same every time you use the lathe. The home switch and location of the Duality on the mill table are generally quite accurate enough for the tool center height..

(b) You do not have to reset the X axis offset unless you power down the PCNC1100. The method described is more accurate than relying on the home switch on the mill column.

(c) You do need to reset the Z axis offset every time you put stock in the chuck, It combines neatly with facing the stock end which you will probably want to do anyway.

We are ready to look at the FirstTurnPart code.

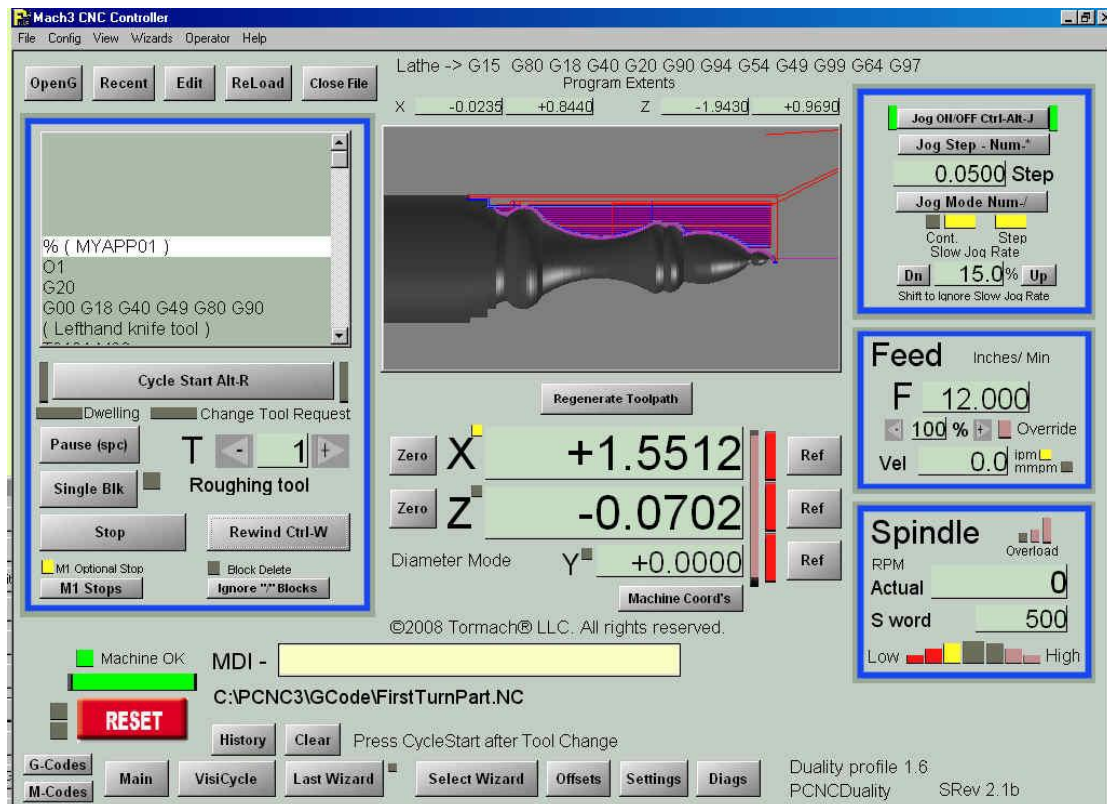


Figure 4.15 – Main screen with FirstTurnPart loaded

4.3 Loading and Previewing the code

The software installation will have copied this file from the release CD –
C:\PCNC3\GCode\FirstTurnPart.nc

Making your first part

Display the control software Main screen and use the OpenG button to load this file.

You will see that FirstTurnPart is a chess bishop.

You can zoom the toolpath by dragging with the mouse left button or rolling the scroll wheel and can pan the display dragging with the right button. Notice the display shows the path of the tool and the finished part rendered to show its 3D form.

Study the other main parts of the screen. The controls are generally similar to those on the PCNC1100 mill screens. The exception is the spindle control (figure 4.16)

The Actual DRO gives the speed measured by the Duality lathe and fed into the Accessory socket on the PCNC panel. Switch the spindle to Forward and use the control knob to alter its speed. You should see the Actual DRO respond as you change the speed.

The S Word DRO records the spindle speed requested by the part-program. The default is 500 RPM.

Notice the bar graph below the S Word DRO as you alter the speed from say 300 RPM to say 800 RPM. The graph give you a quick visual indication if the speed you have set is higher or lower than that requested by the part-program.

The final spindle feature is the Overload graph. This compares the current spindle speed with the long term average speed. If the speed is dragged down by an excessively heavy cut then the graph gives you early warning before the drive motor stalls.

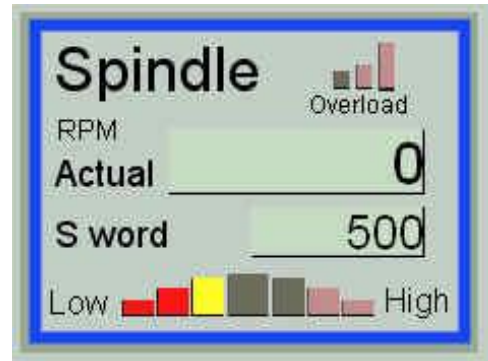


Figure 4.16 – Spindle control

4.4 Cutting the Bishop

You are now ready to manufacture the bishop. It will be cut at around 1200 RPM so start the spindle Forwards and set the Actual DRO to read 1200. Press Cycle Start

The software will run until the part-program calls for tool #1 when it will flash the Change Tool Request LED. As

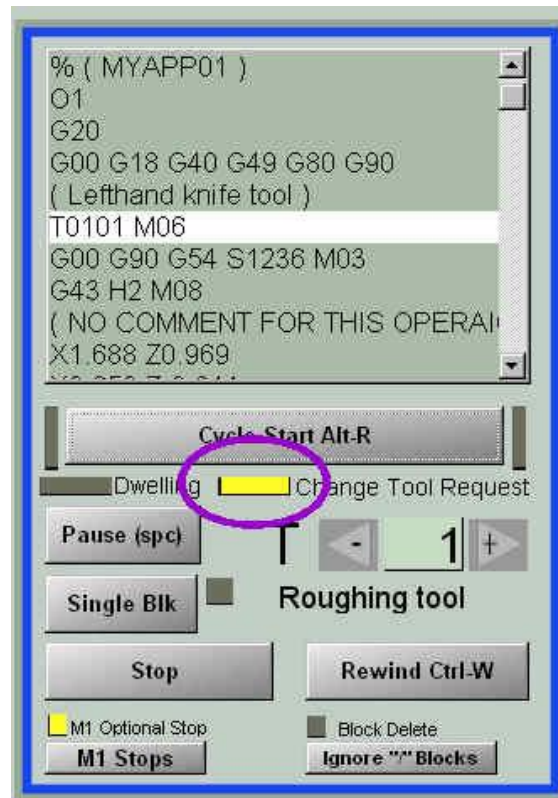


Figure 4.17 – Tool change request

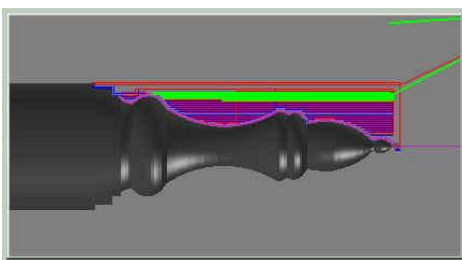


Figure 4.18 – Roughing cuts



Figure 4.19 – Finishing profile cut

Lathe Controls

we already have it in the tool post just continue by clicking Cycle Start. The Duality will rough out the profile of the bishop (figure 4.18). When the roughing is complete the finishing cuts follow the finished profile of the piece (figure 4.19)

The next operation should be to cut-off (part) the bishop from the stock. To keep FirstTurnPart simple we are only using offsets for one tool. You can either cut the part off with a hacksaw or use the cut-off (parting) tool with manual jogging. If you choose the latter technique, because the cut-off blade removes a wide chip, you need to be sure that the cut-off tool is on the correct center height and take very gentle cuts. Lubrication with a hand-help spray bottle of coolant or WD40 will help.

5. Manual turning

Thanks to LittleMachineshop.com for permission to base these instruction on using the Duality as a manual lathe on the documentation for their manual variant of the lathe.

5.1 E-Stop circuitry

If you are using the Duality lathe on the bench then you should use the E-stop jumper plug connected to its E-stop integration lead. Without this you will not be able to run the spindle

5.2 Adjusting Tool Bit Height

The cutting edge of the tool bit should almost always be set to the center height of the lathe spindle.

There are several methods for checking the height of the tool bit. Perhaps the simplest way is to place a thin strip of metal, such as a steel rule or feeler gage, between the work piece and the point of the tool bit. If the height is correct, the strip of metal will be held vertical. If the top is leaning toward you, the tool bit is too low. If the top is leaning away from you, the tool bit is too high. Using the standard tool post, you adjust the tool bit height using shims under the tool bit. You can get an economical set of shims, about the right size, at any auto parts store. Purchase a set of feeler gauges and remove the pivot pin.

The easy way to adjust the tool bit height is to get a quick change tool post. Virtually all quick change tool posts incorporate a mechanism for easily adjusting the tool bit height.

5.3 Turning

The most common use of a lathe is turning down the diameter of a work piece.

Follow these steps to turn the outside diameter of a work piece.

5.3.1 *Turning manually*

1. Put a tool bit in the tool holder and adjust the cutting edge to center height.
2. Angle the tool so that the front cutting edge forms a small acute angle with the axis of the work piece. If you are using carbide inserts then the tool holder will do this for you
3. Move the carriage so that the tool bit is near the right end of the work piece.
4. Turn the lathe on. Adjust the speed to an appropriate speed for the material and diameter you are working on. The LittleMachineShop.com Web site has a calculator to help you determine appropriate cutting speeds.
5. Using the cross slide feed handle, slowly advance the tool bit into the work until it just touches the surface of the work piece.
6. Move the carriage to the right so that the tool bit is past the end of the work piece.
7. Using the cross slide feed handle, advance the tool bit about 0.010".
8. Using the carriage hand wheel, move the carriage slowly to the left. As the tool bit meets the work piece, it starts cutting.

5.3.2 Turning with Power Feed

The Duality lathe incorporates a power carriage feed that can move the carriage either direction. This same power feed is used for turning and threading. For turning, the change gear train is configured with 20 tooth gears in positions A and C, and 80 tooth gears in positions B and D. This is the way the lathe comes from the factory, and is how you should reset it after threading. If you haven't changed the gearing, this is the way your lathe is configured.

To turn with power feed:

1. Put a tool bit in the tool holder and adjust the cutting edge to center height.
2. Angle the tool so that the front cutting edge forms an acute angle with the axis of the work piece.
3. Move the carriage so that the tool bit is near the right end of the work piece.
4. Move the tumbler lever to the forward position (figure 5.1).
5. Turn the lathe on. Adjust the speed to an appropriate speed for the material and diameter you are working on. The LittleMachineShop.com Web site has a calculator to help you determine appropriate cutting speeds.
6. Using the cross slide feed handle, slowly advance the tool bit into the work until it just touches the surface of the work piece.
7. Move the carriage to the right so that the tool bit is past the end of the work piece.
8. Using the cross slide feed handle, advance the tool bit about 0.010".
9. Push down on the power feed lever until the half nuts engage. As the tool bit meets the work piece, it starts cutting.
10. When the carriage has moved as far as you want, raise the power feed lever to disengage the half nuts. The carriage stops.



Figure 5.1 – Power feed tumbler lever

Note: Be sure to move the power feed forward/neutral/reverse lever which is behind the headstock to the neutral position when you have completed the turning operation.

5.4 Facing

Facing is cutting on the end (or face) of the work piece.

To face a work piece:

1. Put a tool bit in the tool holder and adjust the cutting edge to center height.
2. Angle the tool so that the side cutting edge forms an acute angle with the face of the work piece.
3. Move the carriage to the right so that the tool bit is past the right end of the work piece.
4. Ensure that the power feed forward/neutral/reverse lever is in the neutral position.
5. Push down on the power feed lever until the half nuts engage. You might have to move the carriage slightly so the half nuts will engage.
6. Turn the lathe on. Adjust the speed to an appropriate speed for the material and diameter you are working on. The LittleMachineShop.com Web site has a calculator to help you determine appropriate cutting speeds.

7. Using the compound rest feed handle, slowly advance the tool bit into the work until it just touches the surface of the work piece.
8. Move the cross slide back so that the tool bit is clear of the diameter of the work piece.
9. Using the compound rest feed handle, advance the tool bit about 0.005”.
10. Using the cross slide feed handle, advance the cross slide slowly. As the tool bit meets the work piece, it starts cutting.
11. Continue advancing the cross slide until the tool bit reaches the center.

5.5 Turning Angles

There are several methods of turning angles or tapers.

- For large angles of short length, such as a chamfer, turn the compound rest to the angle you want. Advance the tool across the work with the compound rest, and advance the tool into the work with the cross slide or the carriage.
- You can use the same method for small angles (usually called tapers) of a length less than the compound rest travel.
- For longer tapers, the work is usually placed between centers with the tail center offset from the centerline of the lathe.

5.5.1 Compound Slide Rotation

The compound slide rotates on the cross slide and you can position it at any angle. Position the compound slide so it moves parallel to the ways to make precise facing cuts.

Position the compound slide at 29.5 degrees for cutting standard threads.

To change the angle of the compound slide:

1. Using the compound slide feed handle, retract the compound rest until the locking socket head cap screws are exposed.
2. Loosen the two socket head cap screws.
3. Turn the compound rest to the desired angle.
4. Tighten the two socket head cap screws.

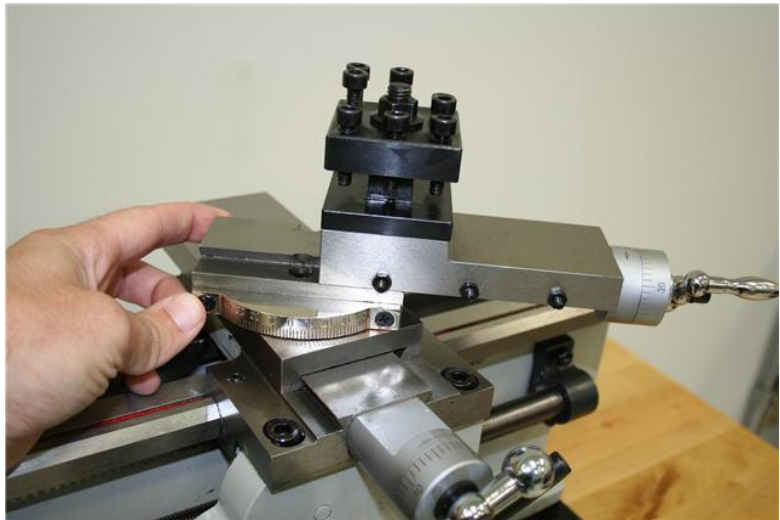


Figure 5.2 – Rotating the Compound Slide

6. Routes from an Idea to a Part

This chapter is concerned with CNC use of the Duality lathe. It discusses the ways that you can turn your design ideas into code or operations to control the Duality Lathe.

6.1 Introduction

The motion of the tool relative to the stock when you are turning under computer control on the Duality always comes from instructions often referred to as G-codes. While a complete CNC program can look rather complex in essence there are only three main G-codes concerned with motion. G00 moves between two positions at the full speed of the machine (Rapids). G01 moves in a straight line between two points and a specified speed (the feed rate). G02 (and G03) move in a circular arc between two points at a specified feed rate.

This chapter describes how you can produce a suitable sequence of G-codes, called a part-program, to turn your design idea into a turned part.

The most obvious route is to learn what the G codes do and to write the part-program yourself by specifying the coordinates of the start and finish of each move required. This is not particularly hard but can be very tedious as most operations, e.g. turning a length of bar down to a size, cannot be completed in one pass of the tool. In hand coding you have to decide what is the maximum depth of cut that the machine can take on your material and code each of the passes using this depth of cut. We do not dismiss hand coding but think you will find the techniques described in this chapter are faster and cause less errors and scrapped work.

The techniques recommended are:

- (a) **Visicycles** – built in simple operations like facing, turning to size, chamfering and threading. These just require you to specify the end coordinates of your cut and the depth of cut and feed you wish to use. The control software will cut with a suitable number of passes from the current position of the tool to the point you specify. You can use a Visicycle without disturbing a part-program that you have loaded into the control software. Thus they are ideal for simple jobs and for preparing stock for a part program written in any of the following ways
- (b) **Wizards** – Wizards allow you to choose from a visual menu of shapes that you want to turn and to specify the details by entering dimensions and cutting parameters into boxes (DROs) on the computer screen. The Wizard then writes a suitable part-program and loads it into the control software. You can use the code as it is, and/or save it for use next time you want to turn the same part. You can also combine the code from more than one Wizard by using a text editor.
- (c) **CAD/CAM** (Computer Aided Design/Computer Aided Manufacturing) is, as its name suggests, a two stage process where you define the geometry of the part by some sort of drawing package and then use a separate program (or sometimes module of the CAD program) to produce the G-code required.

Visicycles and Wizards are both built into the Duality control software. For CAD/CAM you will have to purchase additional software.

6.2 Using Visicycles

For this section, we will assume that you have set up one or more tools. Using a single tool is explained in Chapter 4. Setting up multiple tools is described in Chapter 8

The Visicycles screen is shown in figure 6.1

6.2.1 Turning a length to size

The top panel of the screen is where the cut of a Visicycle is to end. In the example in 1st Op this has been set as X = 0.5", Z = -1.5". Ignore 2nd Op and 3rd Op for now.

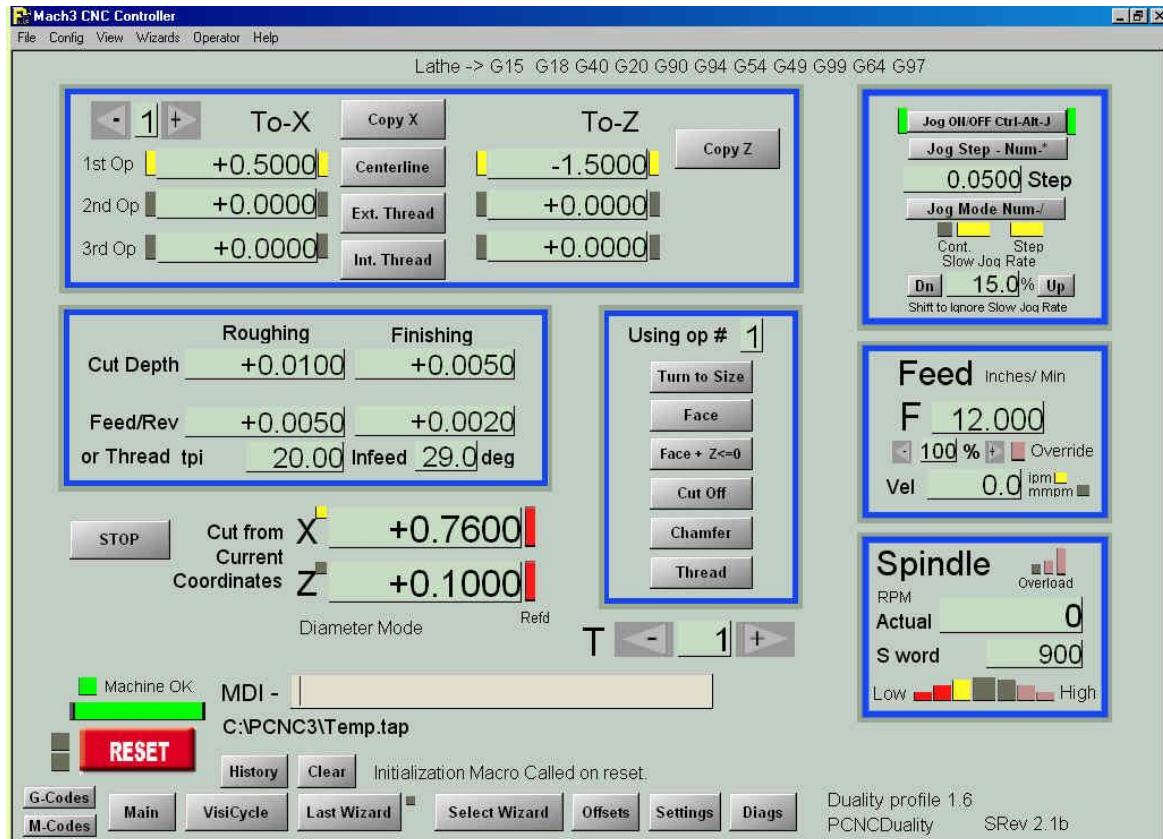


Figure 6.1 – Visicycles Screen

The current position of the tool is shown above the MDI (Manual Data Input) line. In the example it is X = 0.76" and Z = 0.1"

Suppose you have a length of ¾" diameter bar in the chuck and its end is set as Z = 0. The current position of the tool will be just clear of the end of the bar.

Set a reasonable spindle speed in the S DRO (e.g. 900 RPM in figure 6.1) and start the spindle and adjust its speed.

Remember you must press the *Enter* key to accept data when changing a DRO.

Next click Turn to size then the Visicycle will compute the moves to go from cutting air at a diameter slightly greater than the stock to end with a 1.5" length turned down to exactly ½" diameter. The cuts specified in the Roughing/Finishing panel will be used. The Turn to Size Visicycle will ask the operator to confirm that the data is correct (figure 6.2).



Figure 6.2 – Turn to Size confirmation

Check that the cut depths look reasonable and, as a sanity check, that the number of passes is OK. Once you OK the Visicycle you cannot stop it without using the E-stop button which will lose the tool position setup.

Notice that the feed is specified in units per revolution. In other words this will be the thickness of the chip. The depth of cut will be the width of the chip. When you find a suitable pair of values for a particular type of tool and material being cut you will not need to alter the data in Roughing/Finishing whatever diameter of work you are turning as the control software will read the speed you are turning at and set the right feedrate from it.

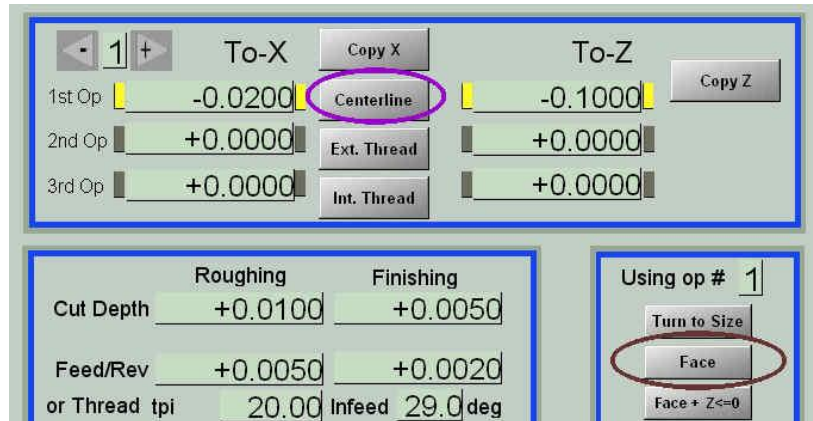


Figure 6.3 – Facing end of stock

When complete the Visicycle returns the tool to where it was at the start

6.2.2 Facing the end of stock

Figure 6.3 shows data set up for facing the end of stock. We are using the same start position as in Turning to Size (0.76, 0.1) but the end position is different. We have assumed that (a rather generous) 0.1" will be needed to clean up the end and we want to face to just beyond the centreline (X = -0.02"). This values is easily set by clicking the Centerline button.

Note: You may have noticed that, although the cut depth has not been changed the Facing Visicycle seems to be taking smaller cuts than Turn to Size (i.e. ten thou, as opposed to nearly 20 thou.) In fact the chip size will be similar because Turn to Size is reporting what is happening to the diameter in each pass.



Figure 6.4 – Facing confirmation

6.2.3 Facing and setting Z to zero

It is very common that after facing you will want to set Z = 0 at the newly faced end. This can be done by using the Face and Z <= 0 button instead of Face.

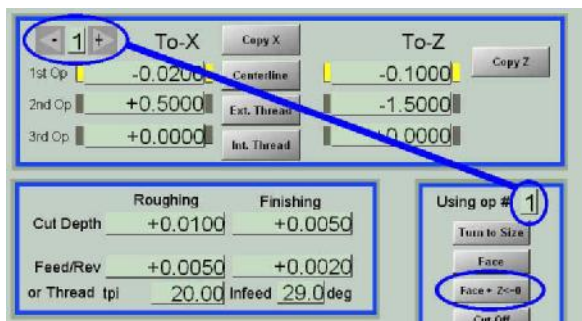


Figure 6.5 – Operation 1 = Face

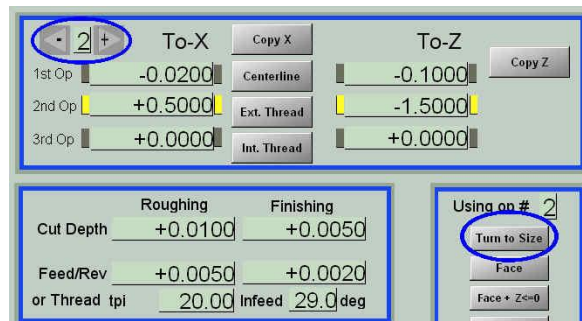


Figure 6.6 – Operation 2 = Turn to Size

6.2.4 Storing data for multiple operations

Suppose you are making a part that is designed for 1/2" stock but you have none. You can use the 3/4" you have but before each part's part-program is run you need to face the stock end and turn it down to 1/2". Thus there are two end positions you want to use. This is where the 2nd (and 3rd) operation data can be useful.

Figure 6.5 shows the Facing data in Operation 1 and the Turn to Size data in Operation 2. The data for the currently selected operation is highlighted by the screen LEDs. You can select the current operation by typing into either of the operation number DROs or by using the up/down arrows.

6.2.5 Cutoff Visicycle

The Cutoff Visicycle ignores the Z position in the current operation. For actually cutting off you will need an X values beyond the centerline (e.g. as set by the Centerline button). The Visicycle can also be used for grooving by using a suitable X value.

6.2.6 Chamfer Visicycle

This Visicycle cuts an angle from the current X,Z to the values in the current operation.

6.2.7 Thread Visicycle

This is not typically useful in preparing stock like the other Visicycles but it is provided as it is useful when producing a prototype part be a combination of manual turning using the Jog/Shuttle controller and Visicycles.

For Thread the Feed per rev data in the Roughing/Finishing panel is ignored. The thread data is given by the TPI and Infeed angle DROs. Figure

6.7 shows the data set in Operation 3 for a 1/2" 20 TPI thread 1" long. Using the Ext Thread button calculates the typical depth of thread for the given pitch and stores it in To X of the current operation.

The Cut Depth DRO is interpreted in a special way. It will be the depth of cut of the first pass. Each subsequent pass will be shallower. The depth is adjusted so as the width of the chip increases as the thread gets deeper so the chip thickness decreases to maintain a constant amount of metal removed for each pass.

Notes:

(a) The thread depth calculated by Ext. Thread and Int. Thread (for internal threads) assumes that the tool has the correct tip radius for the thread being cut. A "sharper" tool will need to cut deeper to get the required thread width on the pitch diameter. You may need to refer to traditional screw tables and do some experiments to get the desired fit of threads.

The screenshot shows the Tormach PCNC Duality Lathe control interface. At the top, there are fields for 'To-X' and 'To-Z' with 'Copy X' and 'Copy Z' buttons. Below these are three rows of data for '1st Op', '2nd Op', and '3rd Op'. The '2nd Op' row is highlighted with a blue bar. In the '2nd Op' row, the 'Ext. Thread' button is circled in blue. Below the operation data is a 'Roughing' and 'Finishing' panel. The 'Roughing' panel has 'Cut Depth' set to +0.0100 and 'Feed/Rev' set to +0.0050. The 'Finishing' panel has 'Cut Depth' set to +0.0050 and 'Feed/Rev' set to +0.0020. Below this is a 'Thread' panel with 'Cut from X' set to +0.5000 and 'Cut from Z' set to +0.1000. The 'Thread' button is circled in blue. On the right side, there is a 'Using op #' field set to 3, and a vertical stack of buttons: 'Turn to Size', 'Face', 'Face + Z<=0', 'Cut Off', 'Chamfer', and 'Thread'. The 'Thread' button is circled in blue.

Figure 6.7 – Threading data setup

(b) In Metric mode the screen changes to define a thread by its Pitch rather than TPI. This is shown in figure 6.8.

(c) 29° is the infeed angle recommended by most tooling manufacturers for 60° included angle threads (e.g. UNF, UNC, ISO)

	Roughing	Finishing
Cut Depth	+0.0100	+0.0050
Feed/Rev or Thread pitch	+0.0050	+0.0020
Infeed	29.0 deg	

Figure 6.8 – Threading in Metric (G21) Mode

6.3 Using Code Written by Wizards

A Wizard is a screen on which you enter data defining the operation you want to perform. The screen will have a button, typically labeled, Post Code which writes suitable G-code for the cuts you want into a file and opens that file as the current part-program.

The Wizards supplied with the Duality Lathe are contributed by the user community and so you may notice that there are inconsistencies in terminology in places.

If you find a need and can program using VB Script then you can write your own Wizards which can be run in the same way as those provided with the Duality control software.

The Select Wizard button brings up a list of the Wizards installed on your system. Double click an entry or select it and click Run (figure 6.9)

You will then be shown the Wizard data screen or in some case a pictorial sub-menu of features to be machined. An example is given in figure 6.11.

Once you have used a particular Wizard the latest Wizard used will be remembered until you close down the control software. The LED by the Last Wizard button indicates that this button is active (figure 6.10).

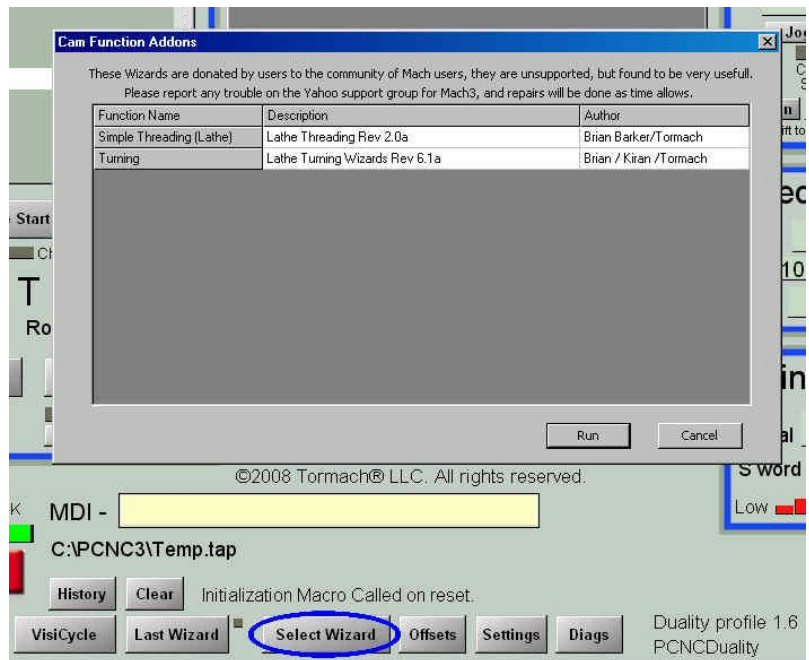


Figure 6.9 – Selecting Wizard to run

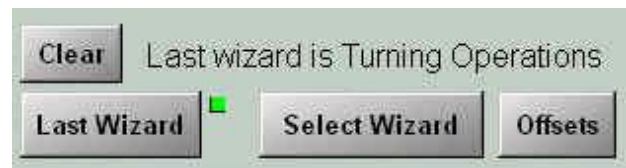


Figure 6.10 – Selecting the Last Wizard used

6.3.1 Turning Wizards

We will look at the ball turning Wizard as a typical example of the Turning Wizard set (figure 6.11).

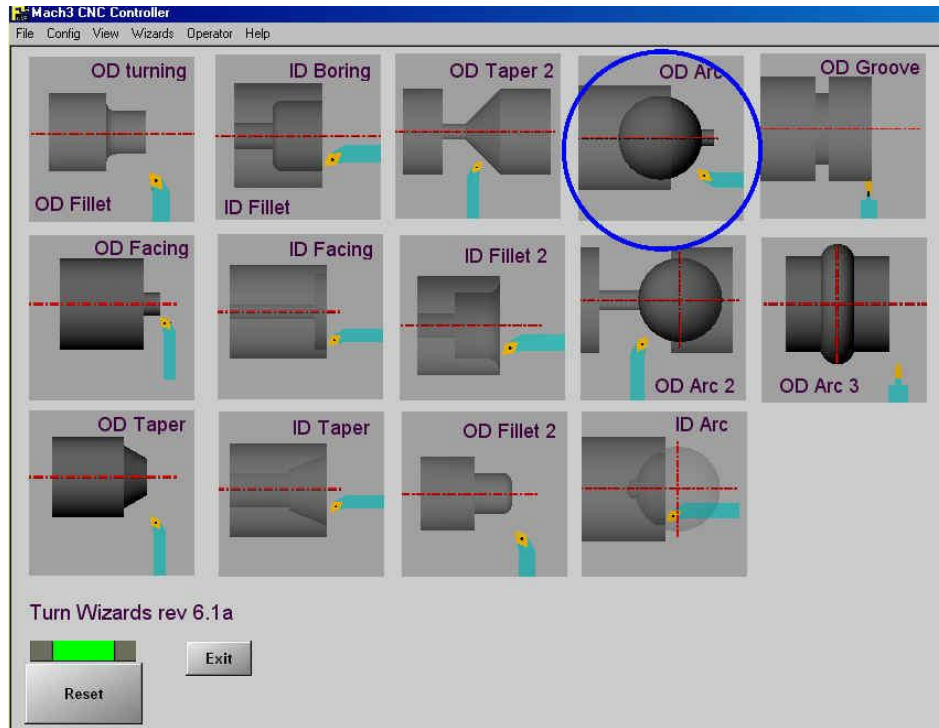


Figure 6.11 – Turn Wizard selection

Click the feature you want to turn. We will use the Outside Diameter Arc highlighted in figure 6.11. A screen like figure 6.12 will be displayed.

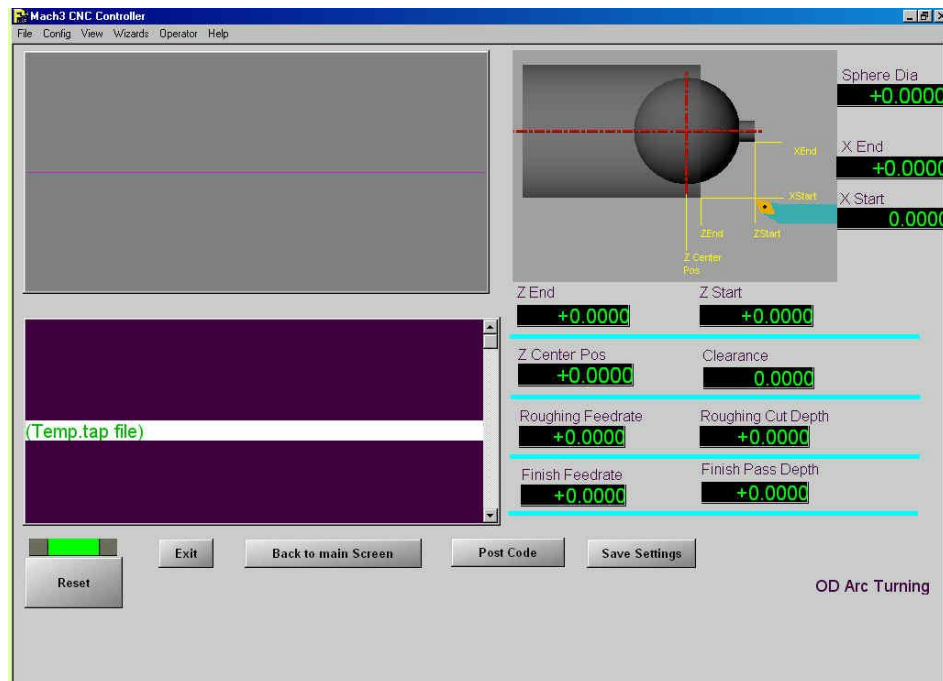


Figure 6.12 – Blank pro-forma for ball end

The meaning of the DROs should be obvious from the labels on the sketch. Assuming you have $Z = 0.0$ at the end of the stock then Z End and Z Center Pos. will be negative values.

Note: Do not forget to press *Enter* after typing a value into a DRO. Clicking the mouse into another DRO just discards the typed value.

Clearance is the gap allowed during rapid motion between cutting passes.

Assuming you are working in inch units then a Roughing feedrate of 6 and Finishing feedrate of 3 will be safe values. Depth of cut of 10 to 15 thou. will be OK. Because CNC operations are automatic you will probably find that you use more conservative cutting parameters on the Duality than you would when turning by hand.

When you have set up your data then you can save it as the basis for another run of the Wizard using the Save Settings button. If you do not do this then the next run of the Wizard will start with the values you started with this time – e.g. zeros for a new installation of the system. The saved values only relate to the screen you are on. Thus for example the internal arc option has similar names in its DROs but these will have their own set of saved values.

When you have entered all the data – and if desired saved it – click on the Post Code button. This will write the G-code part program required to machine your feature. The code will be displayed in the code window and the tool path will be previewed. Figure 6.14 shows this in detail. The G-code window has been scrolled to a cut near to the end of the end of the code which is highlighted in the toolpath display.

Click the Exit button to return to the screen from which you called the Wizard (usually the Main screen).

You can call up the editor to edit the G-code with the Edit button. If you wish to save the Wizard generated code for a repeat of this job then use the Save As function in the editor File menu.

Next you should start the spindle, set a suitable speed for the material you are machining and use Cycle Start to run the part-program.

Note: It is assumed that you will have put the correct tool in the tool post. The Wizard does not ask for any tool changes.

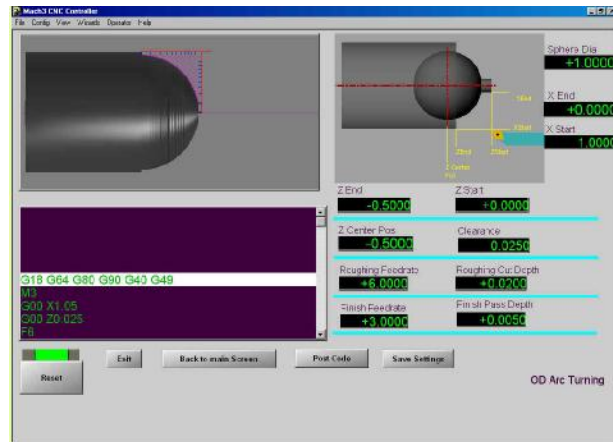


Figure 6.13 – The completed code

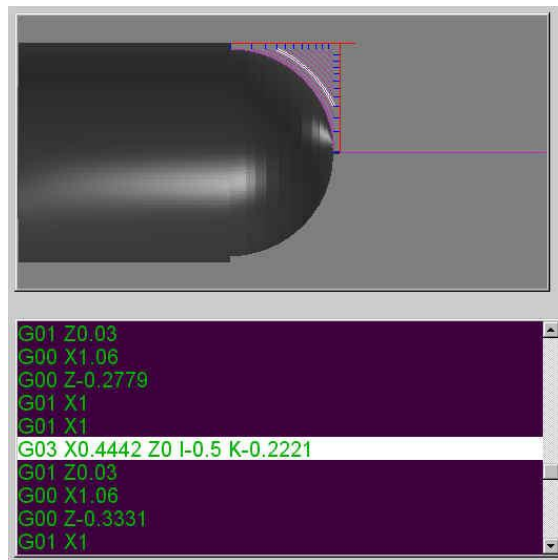


Figure 6.14 – The generated code and toolpath



Figure 6.15 – The completed feature

6.3.2 Threading Wizard

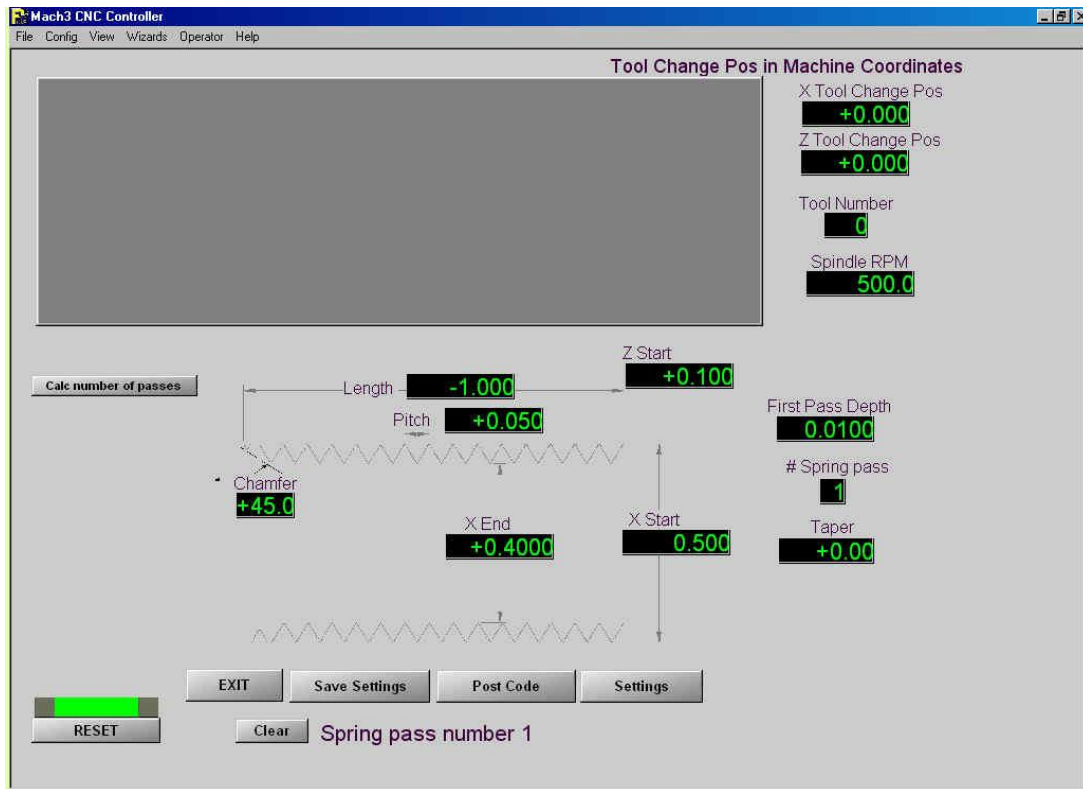


Figure 6.16 – Threading Wizard screen

The other standard Wizard allows you to cut a screw thread. Figure 6.16 shows the screen with some data set in it. There are some important things to notice.

6.3.2.1 Thread depth

You need to calculate the appropriate depth of cut so you can fill in the X End DRO. The value to use will depend on the pitch of the thread that you are cutting and the radius of the point of your tool. Thread tables will give you the diameter data for the recommended root radius. If your tool has a smaller radius the you will have to cut deeper to get the correct fit in a nut. Similarly a larger radius tip needs to be cut shallower and may, of course, interfere with the thread crests in the nut. Unless you are using a "full-form" threading insert (and this tends to be costly as you need a separate insert for each pitch of thread you will cut) you will need some trial and error to get the required fit with a suitable nut.

6.3.2.2 Tool change

This Wizard will insert tool-change code into the generated part-program. In the Duality version of the Wizard the Tool Change Pos. DROs are always ignored. Use jogging to position the tool post so you can change tool holders.

6.3.2.3 Cut depths and spindle speed

The First Pass Depth DRO is used to set the size of cuts. Each pass is calculated so it removes the same cross section of chip. In other words the infeed gets less and less as more and more of the thread is formed. This value is best set by trial and error to give between 20 and 30 passes. You can display the number of passes together with other data about the motion required to cut the thread by clicking Calc number of passes.

The Calc number of passes button will also tell you if your spindle speed is too fast for the pitch of thread being cut. It is important to set the actual speed of the spindle to the value given to the Wizard or you may get poor threads with the wrong pitch.

Set the Z Start DRO to be about three pitches clear of the end of the stock. The threads cut in air allow the Z axis to get up to speed.

6.3.2.4 Multiple start threads

If you wish to cut multiple start threads then this is done by cutting each thread at different Z positions. For example to cut a three-start thread at 8 tpi one calculates that the crests of each thread should be $\frac{1}{24}$ " (0.041667) apart. Thus the first thread might start at Z = 0.2", the second one at Z = 0.241667 and the third one at Z = 0.28333)

6.3.2.5 Threading bores

The Wizard will generate valid code for internal threading if X Start is less than X End. The toolpath display will not be realistic as it will not show the hole in the stock.

6.3.2.6 Settings

The settings button displays a screen with additional infrequently used values (figure 6.16). Two of these deserve further explanation.



Figure 6.17 – Threading Settings screen

Last Pass depth: should be set to define how big you want the final pass to be. This can be as small as you like unless you are threading a work hardening material where you may need a depth of cut that will ensure the tool is always under the skin.

Min path depth: serves the same function but applies to all passes including the last.

If you find that reducing the First Pass Depth on the main screen does not give you as many passes as you want then it is probably because Min Path Depth is overriding the depths calculated by the chip area algorithm.

The Back button returns you to the main threading screen.

Figure 6.18 shows a test piece for a spindle with a bearing and a pulley seat and thread for a retaining nut.



Figure 6.18 – Finished thread

6.4 Using CAD/CAM

A wide range of parts can be machined using Visicycles, Wizards or a combination of these tools. It is straightforward to use an editor to combine the programs written by different Wizards. There are nevertheless some good reasons for investing in CAD/CAM software and in learning how to use it.

Advantages include the ability to:

- Document designs using drawings and photo-realistic renderings
- Make small changes to part of a design without reworking all of it
- Use software to calculate stresses, deflections etc.

- Exchange designs with others (e.g. suppliers or customers)
- Organize the storage of designs to allow repeat manufacture months or years later.

Tormach provides two distinct CAD/CAM solutions.

In TurboCAD/CAM the Computer Aided Design and Computer Aided Manufacturing functions are closely integrated. A change in the geometry will immediately reflect into the G-code of the part program.

Alibre CAD and SprutCAM rely on passing a file from the CAD program into the CAM program. The advantage of having two pieces of software is that each are much more capable than a combined package. Thus the part can be modeled in CAD using a wide range of techniques (extrusion, revolves etc.) and there are many available machining strategies. A further advantage come from being able to define features in the part some of which will be produced by turning and some by milling.

The sections below aim to give a flavor of the use of CAD/CAM tools. Details can be found in the documentation of the software concerned.

6.4.1 TurboCAD/CAM

TurboCAD integrates drawing and turning very neatly. If you only use 2½ D milling you may find it meets all your needs. Figure 6.19 shows the start of the design the bishop used as the FirstTurnPart. The outline of the half-section is drawn and a rectangle used to define the size of the stock from which the part will be turned.

Next the tool to be used for turning is chosen and roughing toolpaths calculated. This is shown in figure 6.20.

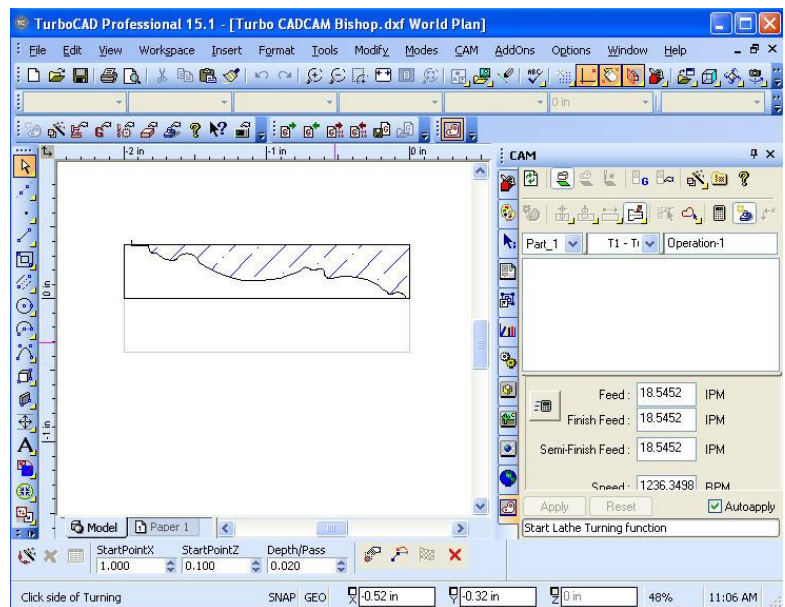


Figure 6.19 – Profile and stock in TurboCADCAM

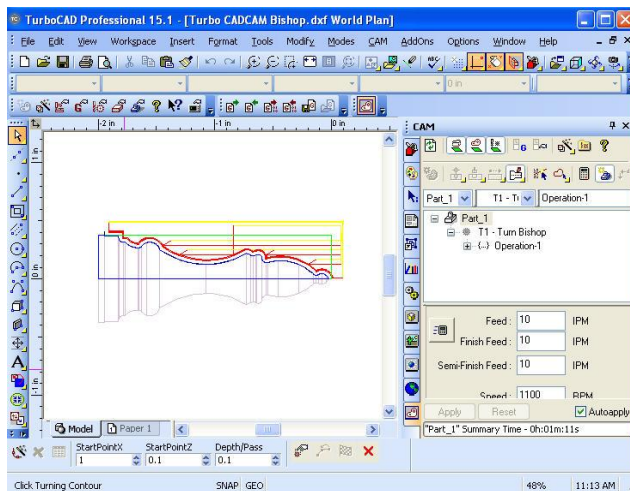


Figure 6.20 – Toolpaths calculated

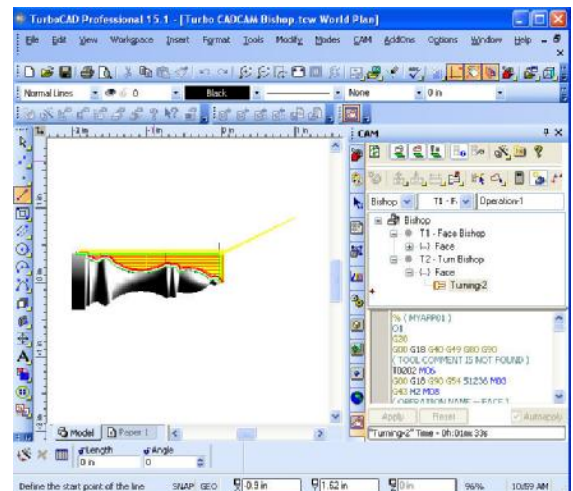


Figure 6.21 – Rendering of finished part

Figure 6.21 shows the rendering that can be produced to check the validity of the design.

Finally the part program that will implement this toolpath can be seen in figure 6.22.

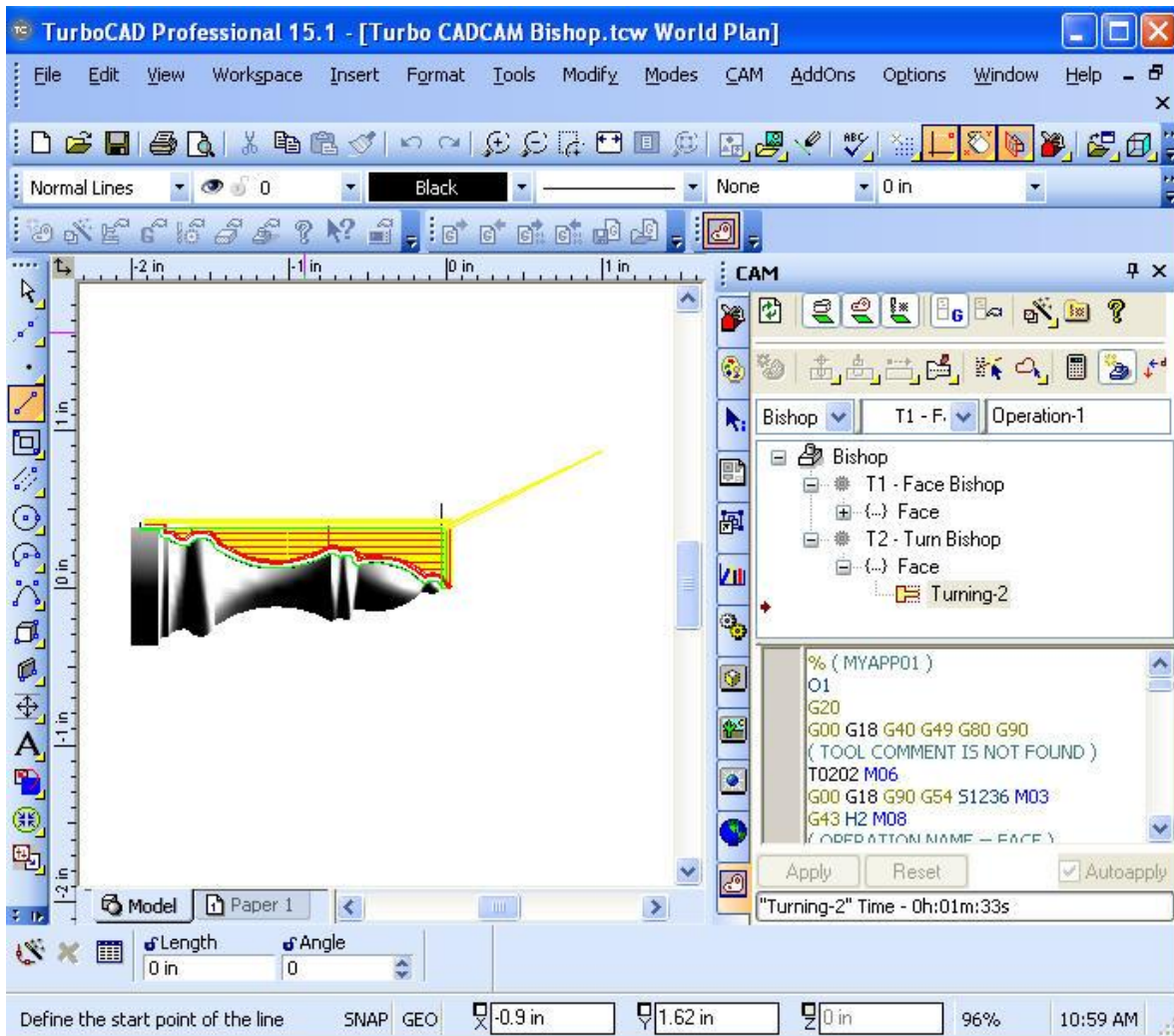


Figure 6.21 – Completed bishop showing generating G-code

6.4.2 Alibre and SprutCAM

Alibre is a 3D modeling package. Suppose we want to design a simple stepped pulley. This can be modeled as a series of cylinders which are produced by extruding a circle. Figure 6.23 shows

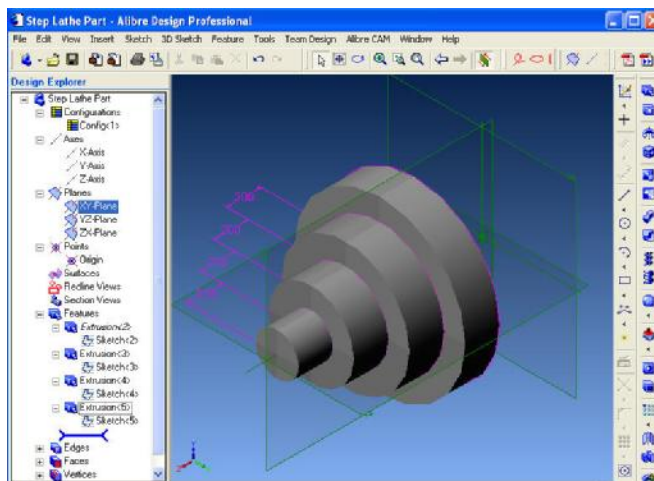


Figure 6.23 – Alibre model of pulley

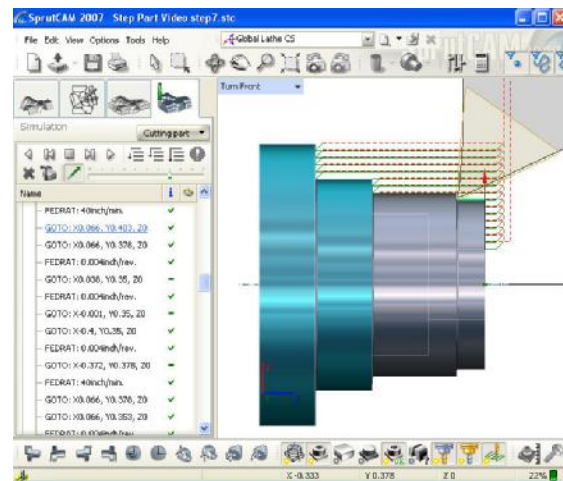


Figure 6.24 – Pulley model in SprutCAM

the model.

Figure 6.24 shows the simulation of the tool cutting it. On the screen, this is animated so you can see the exact sequence being followed.

Figures 6.25 thru 6.27 show the definition of machining of the bishop in SprutCAM.

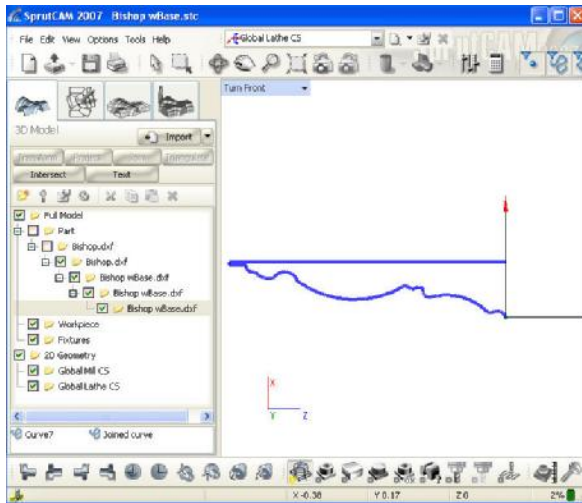


Figure 6.25 – Profile imported to SprutCAM

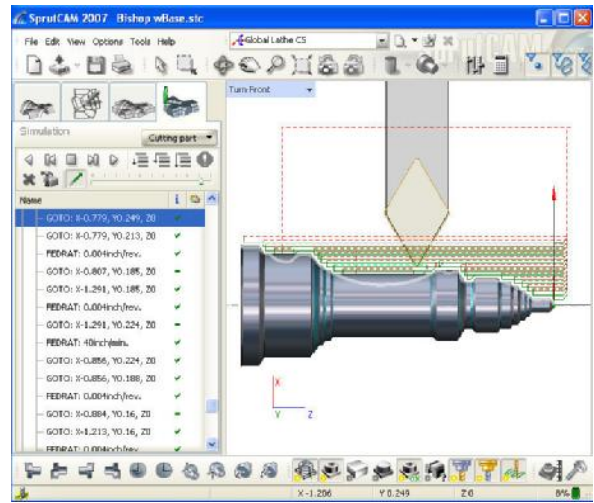


Figure 6.26 – Roughing toolpath and simulation

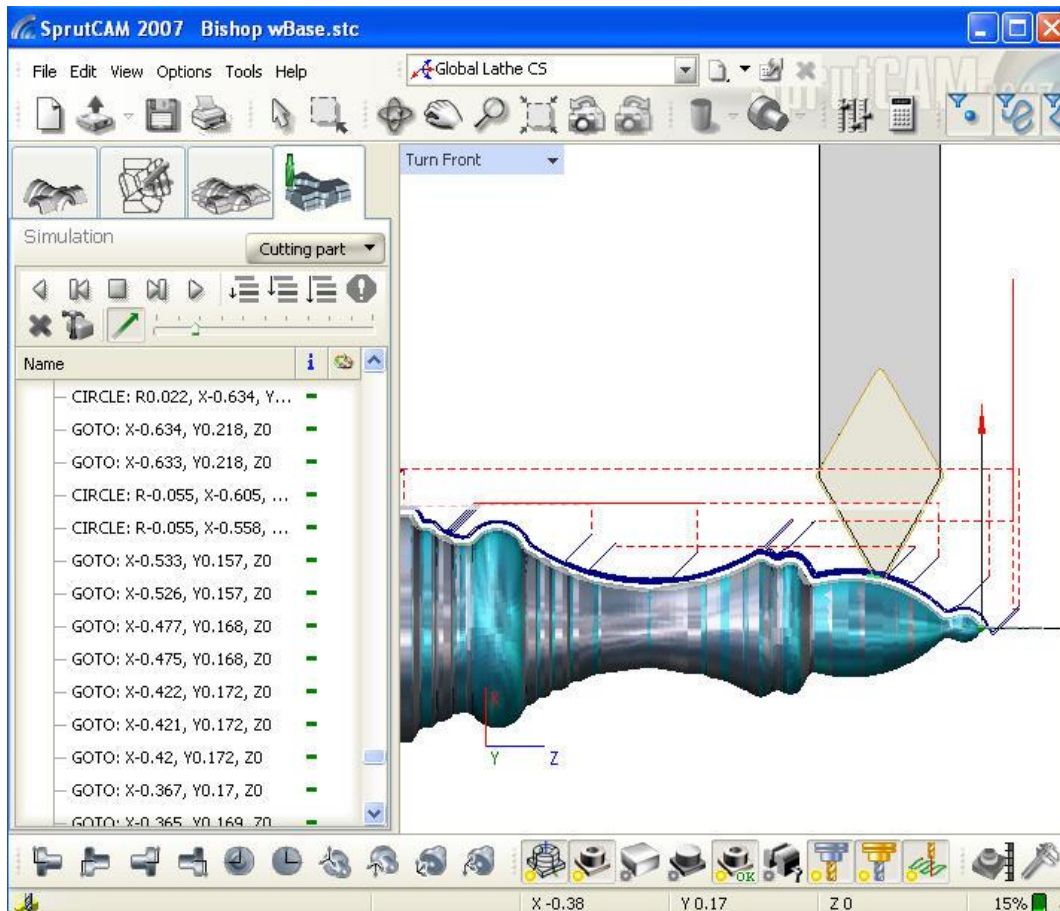


Figure 6.27 – Simulation of finishing cut

7. On-Screen Software Controls

This chapter describes the on-screen controls in the Duality Lathe control software. The controls are grouped into logical families. You may find it best to skim this chapter on first reading of the manual and return to it when you want to understand the details of some feature.

7.1 Screen selection and Reset



Figure 7.1 – Screen selection family

The main controls in this family are to allow the operator to switch from screen to screen.

The data on the right of these buttons gives the revision data for the Duality profile (.XML file) and the screen set itself. These should be quoted when reporting any bugs or problems.

Above the screen selection buttons are controls relating to errors. The last Error (or Warning) message is displayed here. In the example above this is a reference to trying to use the Last Wizard when none was remembered. The Clear button clears this message from the screen. The History button displays a list of recent messages.

Above the message line is the pathname of the currently loaded part-program.

The Reset button is used when the control software is booted or after an event that causes an E-stop. It will only reset if the PCNC is powered up and its start button has been pressed. The green bar LED indicates that the control software is running. If it flashes Red and Green and the square LEDs beside the Reset button are lit then the control software needs to be reset.

7.2 Loading Family

See figure 7.2.

OpenG displays a file open dialog for opening part-program (G-code) files.



Figure 7.2 – Program loading family

Recent displays a list of the G-code files recently loaded.

Edit opens the current part-program in Notepad for detailed corrections.

ReLoad opens the current part program again

Close File closes the current G-code file. You need to do this to re-run the post processor in a CAM program.

7.3 Program Running

For the program running controls see figure 7.3

This family includes the G-code window. This displays the currently loaded part program. The "current" line is highlighted. This can be changed by using the scroll bar on the right hand side of the window.

Cycle Start runs the part program from the current line.

Pause (aka Feed Hold) brings the motion to a controlled stop so you can continue by using Cycle Start again. It is unwise to use this function during complex operations like threading or drilling cycles.

Stop stops motion immediately. The axis drives will not use controlled acceleration so in general steps will be lost using Stop

Rewind makes the first line of the part program be the current line.

Single Blk is a toggle button with indicator LED to show when it is ON. With single block operation ON you must press Cycle Start for each block (line) of the part program.

M1 Stops is a toggle button with indicator LED to show when it is ON. When it is ON then each M01 command will stop the execution of the part program as if it was a M00. When it is OFF then M01 commands are ignored.

Ignore "/" Blocks is a toggle button with indicator LED to show it is ON. When it is ON and Block (line) whose first character is / (front slash) will be ignored.

The **Dwelling** LED flashes when a G04 – Dwell command is active.

The **Change Tool Request** LED flashes when the control software is waiting for the tool to be changed. Use Cycle Start to continue when the correct tool is in the post.

The **T** DRO gives the number of the current tool and its name (as set in the tool table). The value can be nudged with the arrow buttons or a new value can be typed into the DRO.



Figure 7.3 – Program running

7.4 Axis position family

Figure 7.4 shows the axis position family.

The current position of the controlled point is generally given in the X, Z and Y DROs. If the Machine Coords toggle button is clicked then the tall red LED will flash and the data that is displayed will be the absolute machine coordinates (i.e. with no tool or work offsets applied).



Figure 7.4 – Axis position family

The Zero buttons update the work offset so that the controlled point is zero for the chosen Axis.

The Ref buttons will reference the PCNC to its home switches and set standard machine coordinates. When an axis is referenced then the red LED by the Ref button will turn green. Take care when referencing that the axis motion will not cause a tool crash to any part of the machine. It is usually safest to reference the Z axis first.

The LEDs by the axis names indicate which, if any, axis will be jogged by the Tormach Jog/Shuttle controller.

7.5 Toolpath and Modes

Figure 7.5 show the toolpath information.

The stock is shown slightly larger than the position of the largest cutting movement. You can pan the display by dragging with the right mouse button and zoom by dragging with the left mouse button or by rotating the mouse scroll wheel.

The cuts that are to be made are shown in blue with the one corresponding to the current line (block) of G-code being highlighted.

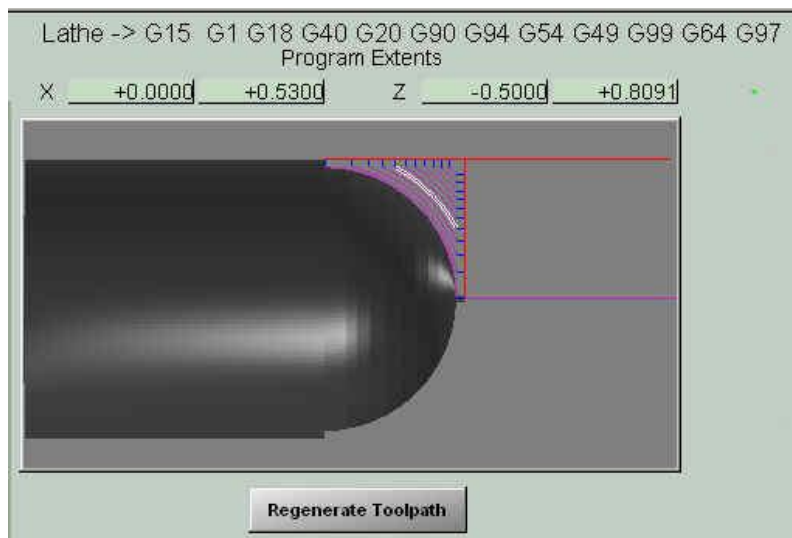


Figure 7.5 – Toolpath, modes etc.

After changes to the code and/or offsets it may be necessary to use the Regenerate Toolpath button.

The X and Z DROs shown the greatest positive and negative extents of movement of the controlled point for the loaded part-program. You can use them to check that there are no gross code errors which could, for example, crash the tool into the chuck.

The top line of codes shows the Modes that the software is currently in. You can cross check their meanings in chapter 9.

7.6 Jog Family

For jog family controls see figure 7.6

The PCNC can only be jogged when the LEDs by the Jog ON/OFF button are lit.

The Jog Mode button cycles between Cont. (continuous) jogging and Step jogging using the arrow keys on the keyboard.

Left/Right arrows move the Z axis, Up/Down arrows move the X axis and Pg Up/Pg Dn arrows move the Y axis.

In Cont. jogging, the axis moves for as long as the key is depressed at the percentage of the rapid rate (65 ipm) set in the Slow Jog Rate DRO. The Slow Jog Rate is overridden by holding down the *Shift* key; the axis will move a full rapid rate.

In Step jogging, the axis moves one step for each depression of the key. The motion is a Feed rate as set by F DRO in the Feed family.

The size of step is defined by the Step DRO. Any value can be typed into this or a preset cycle of values can be accessed by clicking the Jog Step button.



Figure 7.6 – Jog family

The jog family also controls the operation of the Tormach Jog/Shuttle dial. The Jog ON/OFF button must be toggled ON for any motion to occur. The inner wheel executes steps whose size is defined in the Step DRO. The maximum speed of the outer ring is defined by the Slow Jog Rate DRO.

7.7 MDI

The MDI line is activated by clicking the mouse in it or typing *Return*.

The MDI line allows you to type in any G-code commands for immediate execution. When the line is active a short history is displayed. You can use the up and down arrows to select a command from the history (figure 7.7).

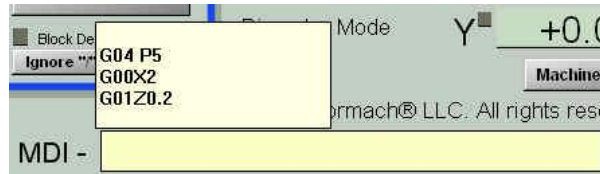


Figure 7.7 – Manual Data Input

7.8 Feed Family

The F DRO is set by an F word in a running part-program or MDI. It can also be set by typing a value into it (figure 7.8).

When the axes are moving it will display the motion that is taking place. The units depend on whether the Duality is in Metric or Inch units (G21 or G20) and if the feed mode is Feed per minute (G94) or Feed per revolution (G95).

The requested feed can be overridden by the value in the Override DRO. You can nudge this value with the arrow buttons or type a value from 1% to 299% into the DRO. If override is in effect the LED will flash to indicate this.

The Vel DRO displays the actual feed velocity. This may be useful in Feed Per Rev mode.

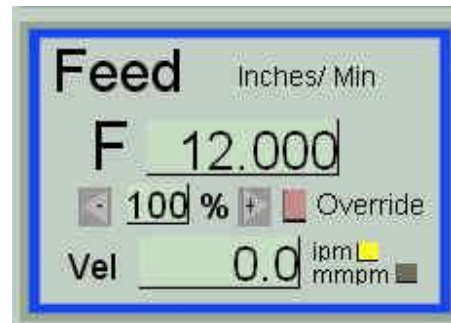


Figure 7.8– Feed controls

7.9 Spindle Family

The Duality spindle is controlled by the operator rather than the control software. The software does however have features to assist the operator (figure 7.9).

The S Word DRO is set by an S word in a running part-program or MDI. It can also be set by typing a value into it.

The Actual DRO displays the speed of the spindle that is measured by the index sensor which plugs into the PCNC accessory socket.

The software compares these two DROs and displays a Low/High bar graph to make it easy to adjust the knob on the Duality to the correct position. When the S Word and Actual speeds are nearly the same then the center green bar will be illuminated (figure 7.10).

The control software also compares the current speed with the long term average speed. If the current speed is significantly lower then this could be because the cut is too aggressive and the spindle is in danger of stalling. This is indicated by the Overload graph.

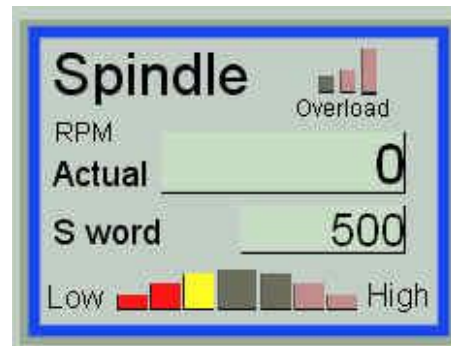


Figure 7.9 – Spindle controls

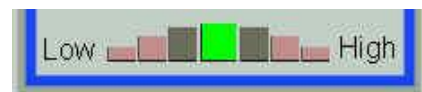


Figure 7.10 – Matching speed indication

7.10 Visicycles

Visicycles are simple operations that the control software automates without affecting any currently loaded part-program. This makes the particularly useful in preparing and setting up stock.

7.10.1 To Family

All Visicycle start at the current position of the tool and cut to the TO position.

Three TO positions can be set up at any one time. These will probably relate to different operations.

For example one might be for facing up the

end of new stock while another might be turning it down to the starting size used in the loaded part-program.

The operations are numbered 1 to 3. The current operation is indicated by LEDs beside the active data. You choose an operation number by typing the number into the DRO in the top left corner of by using its nudge buttons.

The buttons provide shortcuts for filling in the data.

Copy X and **Copy Z** copy the current values of the tool position into the selected operation TO DROs.

Centerline sets the TO X value of the current operation to a small negative value (i.e. tool just over the centerline as in facing or cutoff (parting) operations

Ext Thread sets the TO X value to the root diameter of a standard thread based on the given pitch. The current X position is used to define the thread nominal diameter.

Int Thread performs the same function as Ext Thread but for threads in a bore (e.g. nuts)

Note: The depth calculated may not be correct for the particular tool you are using. More details are given in the section on using Visicycles.

	To-X	To-Z
1st Op	-0.0200	-0.0500
2nd Op	+0.6500	-2.0000
3rd Op	+0.0000	+0.0000

Figure 7.11 – Visicycle - TO

7.10.2 Cutting Parameters

The Cut Depth parameters define the depth of cut to be used. They are distances (chip widths) so a diameter is reduced by twice the cut depth in each cutting pass. In threading operations the specified cut depth will be used for the first pass. In subsequent paths, as the chip gets wider, the depth is reduced so the amount of material being removed remains constant..

	Roughing	Finishing
Cut Depth	+0.0200	+0.0100
Feed/Rev	+0.0100	+0.0050
or Thread tpi	24.00	Infeed 29.0 deg

Figure 7.12 – Visicycle - Cutting parameter

The Feed/Rev parameters are used for all processes but threading. Visicycle always operate in Feed/Rev mode so a feed value chosen for a given material and cutting tip specification will not need to be altered as you adjust the spindle speed to allow for different diameters of work.

The Thread tpi/pitch parameter should be a "threads per inch" value entered for imperial threads and a "pitch in millimeters" for metric threads. This is to correspond to the conventional specification of threads in the two standards.

The Infeed should be the value suggested by the tool manufacturer. 29 degrees is often used for threads with a 60 degree included angle.

7.10.3 Current position

All Visicycles start from the current position as indicated by these DROs (figure 7.13)

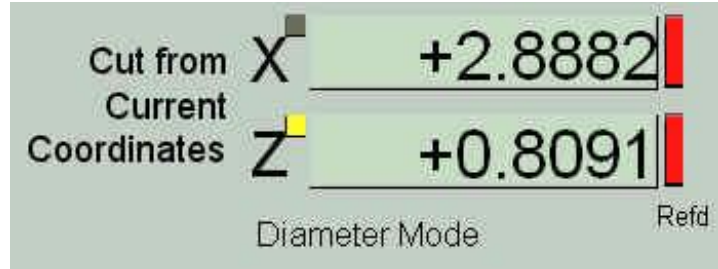


Figure 7.13 – Current position

7.10.4 Commands

When the machine is at the start position and the To position of the operation is defined the cut is planned by clicking an operation button (figure 7.14).

The Operation # DRO is a duplicate of the one in the To positions panel. Check that you have the correct data selected before clicking a button.

The Visicycle will calculate how many passes the operation will involve and display this. Check this carefully as you cannot abandon a Visicycle that is running without risking losing the machine position (figure 7.15)

Turn to Size will take cuts at reducing values of X and traversing Z for the current position to the To position of the operation. When the operation is completed then the tool will be returned at the original current position.

Face will take cuts at values of Z going from the current position until the To position. Each cut will go from the current X to the To-X for the operation. When the operation is completed then the tool will be returned at the original current position.

Face + Z ≤ 0 will operate as Face but at the end of the operation the tool will be returned to the original current X but Z will be the machined face of the stock. The Z work offset will be set so the end of the stock is Z = 0.0. This is the usual



Figure 7.14 - Commands

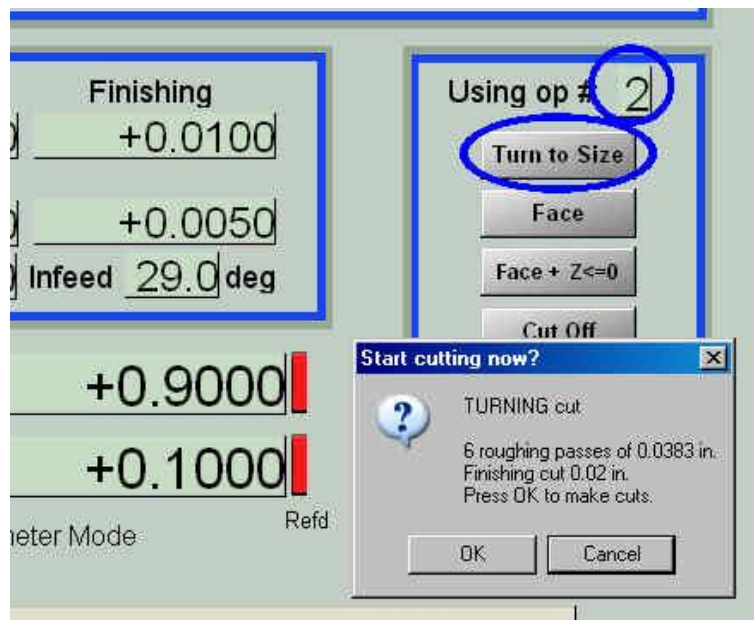


Figure 7.15 – Command confirmation dialog

facing command to use when setting up a job.

Cut Off will not move Z but will advance X from the current position to the To position. When the operation is completed then the tool will be returned at the original current position

Chamfer will cut a chamfer by diagonal cuts from the current point to the TO point for the operation. When the operation is completed then the tool will be returned at the original current position. Although called "chamfer" it can be a long taper if there is a big difference in Z points compared with X points.

Thread will cut a thread from the current point. The To-Z defines the end of the thread and the To-X its root diameter. The Current Z should allow cutting three threads in air before the cut enters the stock. This allows the Z axis speed to stabilize. Multiple start threads can be cut by starting subsequent threads with an additional starting Z position. You should add the crest-to-crest distance of the finished thread onto Z for each thread to be cut. When the operation is completed then the tool will be returned at the original current position.

7.11 Offsets Screen

7.11.1 The Coordinate Equation

Controlled Point	Tool Offset	Wear Correction	Work Offset	G52/G92 Offset	Machine Position
X +0.4134	+0.3224	+0.0000	0.1299	0.0000	+0.4978
Diameter Mode + 2 Diameter Mode + 2 Diameter Mode + 2 Radius Radius Ref X					
Z -1.5324	+0.4000	+0.0010	4.1230	0.0000	+2.9896
Work: Offset G54 Ref Z					
Nose Radius		Direction	Turret		
Offset ON 0.0000		d	d		

Figure 7.16 – Coordinate equation

The display (figure 7.16) shows how the control software converts from the coordinates of the controlled point (i.e. the coordinates used in your part-program, MDI line, Visicycle data etc.) into machine coordinates that are used for the actual positioning.

The Z calculation is easiest as one does not have to worry about diameter.

There are three offsets which can be used to define the controlled point – These are referred to as Tool, Work and G52/G92 offsets.

The Tool offsets are stored in the tool table. It is usual to consider tool #1 as the master tool with zero as its offsets. Then other tools will be described in terms of their difference from the master tool. For convenience the tool table stores the nominal offset from the master tool and a Wear Correction offset that can be changed as the tool tip wears.

The Work offset is generally set so that the master tool is on the centerline on the machine when X = 0.0 and at the end of the stock when Z = 0.0. The control software supports multiple work offsets but it is usual to always use the default G54 offset. Never use the G55 offset as it is used internally by the control software. Other offsets can be used in special circumstances such as special fixture used in place of a chuck.

The G52/G92 offset is seldom used in turning but can be helpful with special fixturing.

Figure 7.16 shows how the controlled point of Z = -1.5324 is converted to the machine position of Z = 2.9896.

Exactly the same principle is applied to X coordinates except that, as the Duality lathe operates in Diameter mode, some values are displayed as diameters. This means that the controlled point and tool X values are divided by 2 to turn them into distances when used in the offsets equation.

7.11.2 Setting Offsets

Full details of the workflow for setting up tools in the tooltable are given in chapter 8. Figure 7.17 shows the controls for doing this.

The operation of these controls depends of whether the current tool is tool #1 (the Master tool) or any other tool.

Master tool: The Touch X at and Touch Z at buttons update the current (usually G54) work offset system so that the controlled point is the value in the corresponding DRO.

Other tools: The Touch X at and Touch Z at buttons update the current tool table entry so that the controlled point is the value in the corresponding DRO

The Hand Edit/Save T/T button opens the tool table editor grid. We advise you to edit the tool table using the touch facilities or by typing into DROs in the Coordinate equation as these methods take account of whether you are in inch or metric mode. You do need to use this button to edit the names of the tools. This cannot be done any other way.

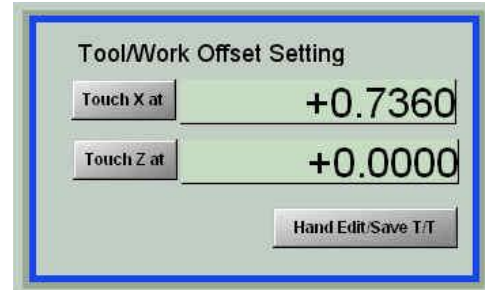


Figure 7.17 – Offset setting

7.12 Settings Screen

The settings screen has data which will only infrequently need to be examined or changed.

7.12.1 I/K Mode

G-code arcs (G02 and G03) can be specified in several ways. One is by using I and K words to define the center of the arc. Unfortunately there are two meanings for I and K. Some CNC controls give the center as an absolute position in I and K. Others use I and K as the distance of the center from the start of the arc.

The Duality control software normally works with the second (incremental) interpretation. If one tries to use G-code from a CAM system that produces absolute I/K values then the toolpath will show great big circles rather than neat arcs. Figure 7.18 shows the ball end in the wrong I/K mode. Figure 7.19 shows the buttons to change I/K mode.

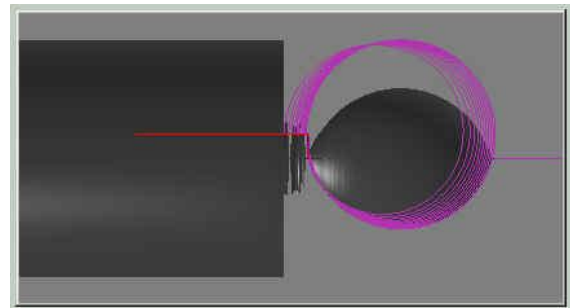


Figure 7.18 – Wrong I/K mode

Note: That if you are using code generated for another lathe you may also get the same sort of circles if the code is for a rear-mounted toolpost rather than a front-mounted one as is used on the Duality. If this happens then you can correct the part-program by using an editor to change all G02 commands (may be written G2) to G03 and all G03s to G02.

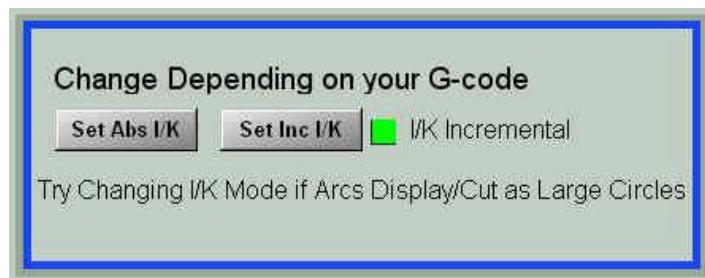


Figure 7.19 – Changing I/K Mode

7.12.2 Center Height

The Y axis is used to set the tool center "height". The detailed procedure is given in chapter 8. The controls used are explained here for completeness (figure 7.20)

The first line gives the controlled point and absolute machine position of the Y axis. It also shows if the Jog/Shuttle is jogging Y and if the axis is referenced. You are advised to reference (home) Y even if you do not do this on other axes as this means that you only need to set up tool center heights once.

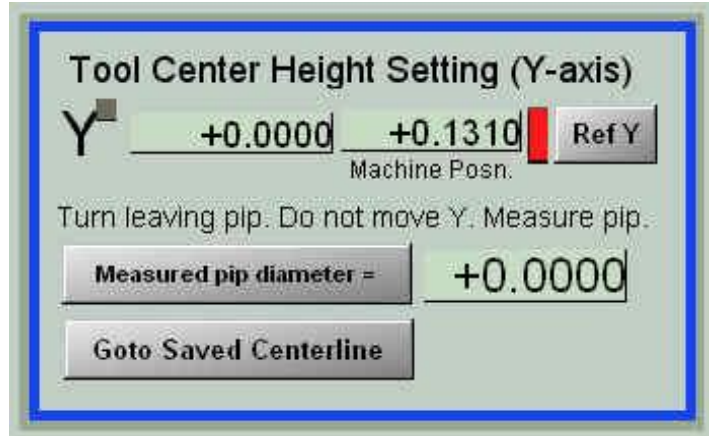


Figure 7.20 - Tool height setting family

In essence the process involves turning a pip in the end of a piece of scrap stock with the tool height set too low and measuring the diameter of the pip. This value is entered in the DRO by the Measured pip diameter = button and then that button is pressed to set the Y work offset.

The Goto Saved Centerline executes the code G00Y0. Take care when using it that the X axis is high enough to avoid any tool crashes.

The Y axis must be referenced for either of these buttons to work.

7.12.3 Lathe/Mill Exchange

The screens provide a technique for retaining the coordinates of a part when switching between using the mill control software and the Duality control software. **The workflow is explained in detail in chapter 8.**

The objective of these controls is to allow you to measure the offsets in three dimensions between the tip of the lathe master tool and the centre of the end of the mill master tool.

An edge finder (wiggler) is used in the mill spindle for the dimensions in the lathe Y and Z directions. The master mill tool is used to mill a piece of stock rotating in the lathe and the "turned" diameter is used to calculate the dimensions in the lathe X direction.

The edge finder diameter is entered in the DRO and when it is on the end of the stock Record Z offset is clicked. The edge finder is then touched on the periphery of the turned stock and Record Y offset is clicked.

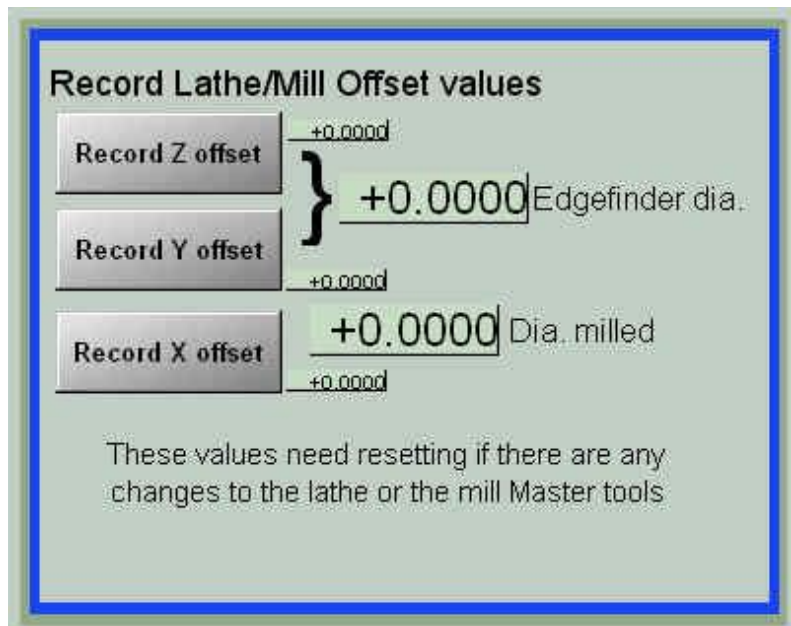


Figure 7.21 - Lathe/Mill tool offsets

Then the mill master tool, which must be able to cut, is put in the mill spindle. With both spindle running a small milling cut is taken on the stock in the lathe. The diameter is measured and put in the Dia. Milled DRO and Record X offset is clicked.

7.12.4 Imperial Speed Calculator

The settings screen has a simple calculator to convert surface speed (in feet per minute) into spindle RPM for a given diameter of work.

Enter the Cutting Speed Required from your tooling data and/or your experience with different materials on the Duality. Enter the approximate Work Diameter that you will be turning and click Calc RPM.

Imperial Speed/Feed Calculator

Cutting Speed Required (SFM) Work Diameter (in) **Calc RPM**

300 / (1.000 x Pi/12) = 1146 RPM

Don't Forget to Press <Enter> After Setting a DRO

Figure 7.22 – Speed calculator

7.13 Diagnostics screen

The diagnostics screen repeats many controls from other screens in one place to make it easy for support engineers to help you fault find. The layout is shown in figure 7.23

Mach3 CNC Controller

File Config Function Clg's View Wizards Operator Plugin Control Help

Lathe -> G15 G18 G40 G20 G90 G94 G54 G49 G99 G64 G97

G18 G64 G80 G90 G40 M3 G00 X1.06 G00 Z2.53 F6

Spindle Toggle
Cycle Start
Pause
Single
Stop
Rewind
Run
Tool Request
Dwell Active

Jog OH/OFF Ctrl-AR-J
Jog Step - Num- 0.0500 Step
Jog Mode Num-
Cont. Step Slow Jog Rate
On 15.0% Up
Shift to Ignore Slow Jog Rate

Outputs
OK Signal to Machine
Output Relay
English
Metric

Inputs
Emergency
X Limit and Home
Y Limit and Home
Z Limit and Home
Spindle Angle Sensor

Actual RPM +0.0000
Average RPM +0.0000

Machine OK
RESET
History Clear
Last wizard is Turning Operations
G-Codes Main VisiCycle Last Wizard Select Wizard Offsets Settings Diags
M-Codes

Duality profile 1.6
PCNCDuality SRev 2.1c

Figure 7.23 – Diagnostics screen

The controls that are unique to this screen are the LEDs recording the state of the outputs controlled by the software and the state of the inputs from the PCNC and the Duality lathe. You can for example check the connection of the spindle position sensor to the Accessories socket by rotating the lathe spindle by hand and watching for flashes on the Spindle Angle Sensor LED.

8. Setting up a Job and Tools

This chapter is about setting up standard tools and setting work in the Duality so that you can swap between tools and make multiple parts with minimum effort is measuring. It also explains how to switch between turning and milling of work held in the Duality lathe.

You may find the descriptions hard going at first reading. Even if you do not use the Tool Table when milling, we would encourage you to understand the concepts in order to get the best out of the Duality.

8.1 Tool Center Height

It is important that the tip of each cutting tool touches the work at the end of a plane through the centerline of the lathe. If the tool is too low then the cutting angle of the tip is wrong – in extreme cases moving from a positive to a negative rake. If the tool is too high then the front clearance angle is too small. The tool may rub and the tendency is to dig in and cause chatter.

Figure 8.1 shows what we mean by the tool being "low" when CNC turning on the Duality.

8.1.1 Setting tools in holders

The tool holders have a knurled ring which registers on the top face of the quick change tool post so that a tool is at the same height each time the holder is put into the tool post. (figure 8.2)

We recommend that you use throwaway tip carbide tooling with the Duality. This will ensure optimal cutting angles and the ability to replace blunt cutting edges without the need for resetting the tool which arises if you grind your own tools in HSS or brazed carbide.

We also advise that you have enough tool holders for the range of tools that you commonly use (e.g. General turning/facing, cut-off, threading, profiling, boring, drilling etc. If you have to remove a tool from its holder to insert another one then the advantages of being able to preset all tools is lost.

Clamp your tool firmly in the tool holder with the smallest overhang that will let you work on the range of diameters you encounter in your parts. The small overhang contributes to the rigidity of the machine. The tool shank is probably the "weakest link" in achieving stiffness and hence good accuracy and finish.

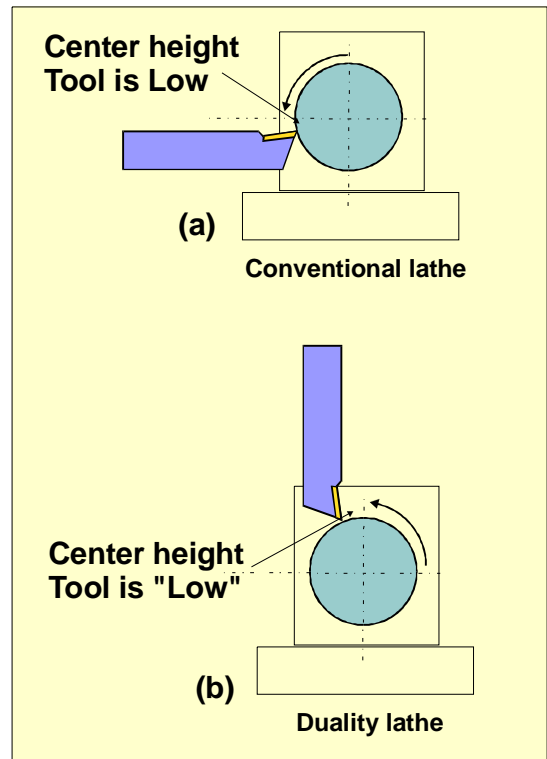


Figure 8.1 – Tool height showing what is Low



Figure 8.2 – Quick change tool holder

The different styles of quick change holder differ in detail on how the tool is clamped.

Choose a numbering scheme for your holders and mark them clearly with their number. You can use a diamond burr in the Proxxon high speed spindle or even a code with nicks cut by a small grinding disc. You need to be able to reliably pick out tool 1, 2, 3 etc. when they are required.

Insert the tool that will be your master tool in holder #1. Ideally the master tool should be able to cut any material you intend to use and be capable of facing and turning. Do not use your master tool in situations where it will wear or get broken. If you change the master tool then you will need to set up the offsets of the other tools again.

8.1.1.1 Setting with Quick Change post on Duality Compound Slide

Tool height setting is very easy if you have the optional quick change post on the compound slide.

Put the master tool holder (#1) onto the post. Adjust its knurled ring so that it is at the height of the lathe centerline. This can be done by trial and error facing of some scrap stock and seeing when the tool does not leave any center pip. Alternatively you can put a center in the headstock (or turn a point on some scrap stock in the chuck) and using a strong eye-glass align the height of the tip with the point.

When you are happy with the center height, lock off the knurled ring and recheck the height.

Now measure the height of the tip from the flat top surface of the bed. A height gauge is ideal for this but a steel rule is OK. Record this value. You can use it for setting all other tools and resetting the master – if this is ever needed.

Next set the height to all the other tools to be the same by using the height gauge or ruler from the bed to the cutting tip.

8.1.1.2 Setting with no QC post on Compound Slide

If you do not have a quick change post on the compound slide for manual turning then the setting of the master tool is arbitrary. Set its knurled ring to halfway along the thread and lock it.

You now must measure the distance between the lower surface of the ring and the tip of the tool. This is probably easiest done on a flat surface (e.g. a surface table) by measuring from the table to the ring and the table to the tip and subtracting the readings. This is illustrated in figures 8.3 and 8.4. {update pics here}

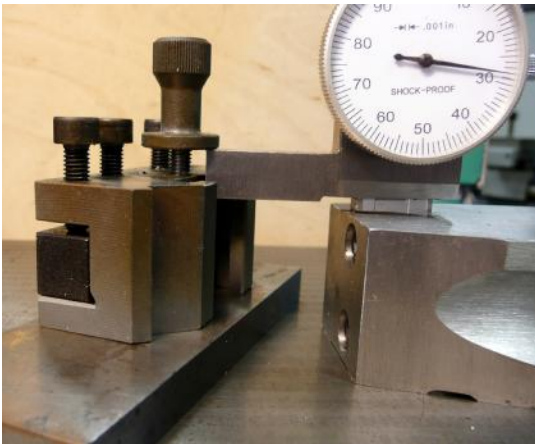


Figure 8.3 – Gauging locking ring

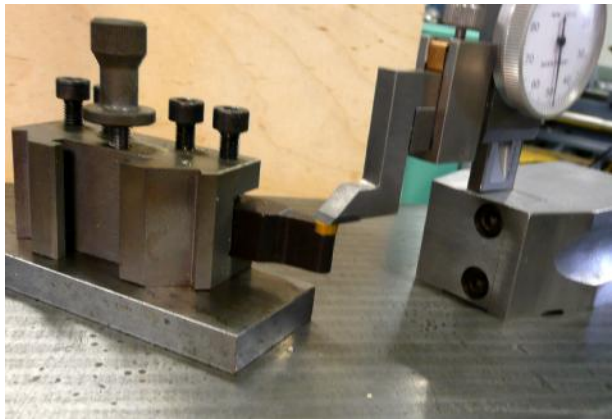


Figure 8.4 – Gauging tool tip

Now with all the other tool holders, adjust the knurled ring so the distance from the underside of the ring to the tip is the same as with the master tool.

8.1.2 Setting Y axis on the PCNC

When all the tools are set in their holders it merely remains to set the Y axis of the PCNC with the master tool so that it is on the centerline of the lathe.

Chuck a piece of scrap ½" bar in the lathe.

Working on the Main screen, jog so you can reference Y safely and click the Y Ref button. The home switches are not accurate enough for setting the X axis position but as the centerline is not critical to within a few thou. The home position is good enough here.

Use any tool that will do a facing cut. Jog the Y axis so the tool will be cutting low by about 0.1" (see figure 8.1) Using jogging, face the end of the bar. You will get a pip left on the end. Measure this as near to the faced end as possible. Calipers are better than a micrometer because the micrometer anvil will tend to measure the largest diameter rather than in the corner. The tool tip is low by half this diameter (figure 8.5).

Now go to the Settings screen and enter the pip diameter into the DRO by the Measured pip diameter = button.

Note: More detailed instructions are given in chapter 4 when you were making FirstTurnPart.

The machine work offsets are set so Y = 0.0 is the centerline for all tools whose knurled rings are properly set. If you click OK to Save Work Offsets when you close down the control software then this position will be remembered even when the system is switched off. Referencing Y and clicking Goto Save Centerline will get you to the correct position without having to repeat the pip turning process..

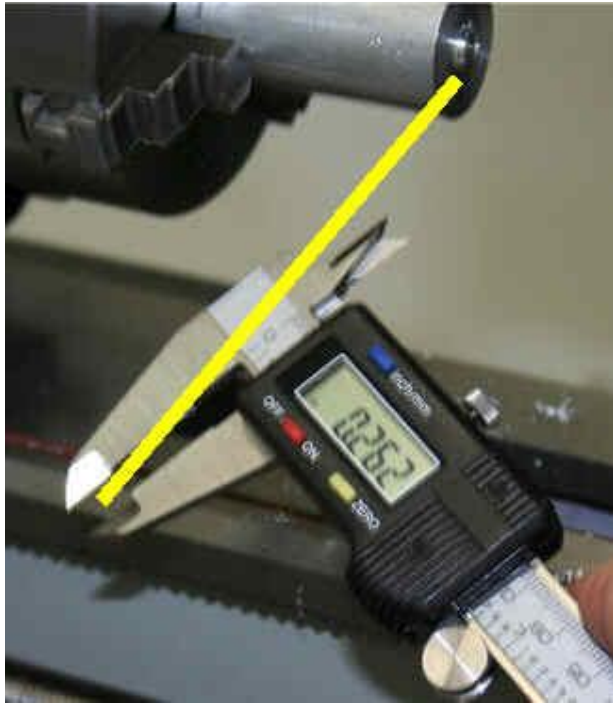


Figure 8.5 – Low tool pip

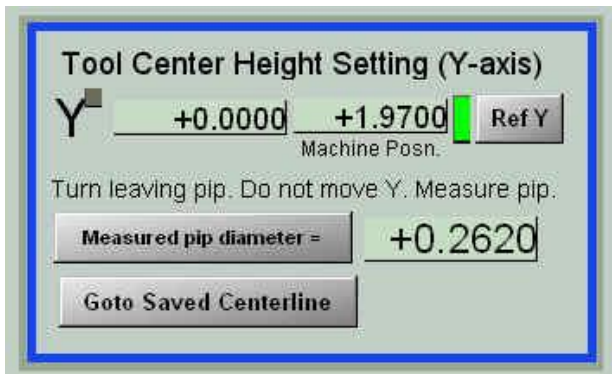


Figure 8.6 – Pip size in DRO

8.2 Tool Table

While it is straightforward to mount every tool so it has the same height for its cutting point, it is neither desirable nor practicable to make the X and Z positions of the point be the same. To understand this one only needs to think of the differences between a turning tool, a boring bar and a twist drill.

8.2.1 The Tool Table

The control software allows for these differences by taking the position of the cutting point of each tool, relative to the master tool, from a list of tools called the Tool Table. You can see the

raw data in the tool table by opening it with the Hand Edit/Save T.T button on the Offsets screen. A typical table is shown in figure 8.7.

Tool	Description	Tip Dir...	Tip Ra...	X Offset	Z Offset	X Wear	Z Wear	Turret...	Post
0	Ref. Tool	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000	
1	Roughing tool	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000	Front
2	Parting Tool	0.0000	0.0000	0.1612	0.4000	0.0000	0.0010	0.0000	Front
3	Threading tool	0.0000	0.0000	20.0000	30.0000	0.0150	0.0000	0.0000	Front
4	Profiling tool	0.0000	4.0000	17.7800	-93.4520	0.0000	0.0000	0.0000	Front
5	Empty	0.0000	0.0000	12.7800	0.0000	0.0000	0.0000	0.0000	Front

All Tool Entries are in your default setup measurement units regardless of G20/G1 modes.

Figure 8.7 – Tool Table editor

The tools are **numbered** from 0 to 253. Tool 0 is not used when running part programs. In the tool number column, the icons are only as a reminder about the tool shap for a human reader. They are not used by the control software.

The **description** is displayed with the tool number on most screens when the control software is running. This text can only be edited when viewing the tool table like this in the editor.

All other details can be edited using DROs on the Offsets screen. This is usually easier than using the raw data in the tool table editor. Values in the editor grid are always in inches irrespective of whether the control software is running in imperial or metric mode. Values on the Offsets screen are displayed in the current units.

The **Tip Direction** and **Tip Radius** are used in conjunction with the built in tool tip radius compensation feature (see chapter 11).

X Offset, **Z Offset**, **X Wear** and **Z Wear** are distances of the given tool's cutting tip relative to the master tool. Tool #1 is the Master Tool and its offsets must always be zero. The discussion of the Offsets Equation explains how these offsets are combined with Work offsets and, perhaps, G52/G92 offsets to convert from the part-program's coordinates to absolute machine positions.

The **Turret** entry relates to automatic tool changers and so is not used. **Post** refers to the tool post position and is always "Front" with the Duality.

While you have the tool table editor open enter the text descriptions of all the tools in the tool holders that you have marked up. The allocation of numbers does not matter though it must, obviously, agree with the numbering you are going to use in your part-programs.

8.2.2 Setting X and Z Offsets

Tool table entries only need to be setup when new tools are put into the holders or when wear of the cutting tip requires a correction to be provided.

Different techniques are used to set X from Z. X = 0.0 is a very definite fixed place on the lathe – i.e. the spindle centerline. Z = 0.0 will often vary for each piece of stock put into the chuck. The accuracy of setting in the X axis generally needs to be much greater than that in Z because diameters usually have much finer tolerances than lengths of bearing seatings, threads etc.

8.2.2.1 The X and Z work offsets with Master Tool

The first operation on loading the Duality system is to establish the settings of the G54 Work Offset system. Generally no other work offsets are used in turning. You need to do this prior to turning parts when you have your tool table setup or in preparation for setting up a tool table.

If you have the tool table setup and are going to machine a part chuck the stock for it. If you are setting up the tool table chuck a short length of ½" diameter aluminum bar with about 1" protruding from the chuck. This will be the setting piece.

Put tool #1 (the Master) into the tool post.

On the Offsets screen, select tool #1.

X offset: Using jogging, turn a short length down so it cleans up all the way round the bar. Measure the diameter of the turned portion. Say it was 0.487". Enter the value into the DRO beside the Touch X at button (figure 8.8). Notice that the X position DRO is a meaningless value; this is not surprising as we have only just started the system running.

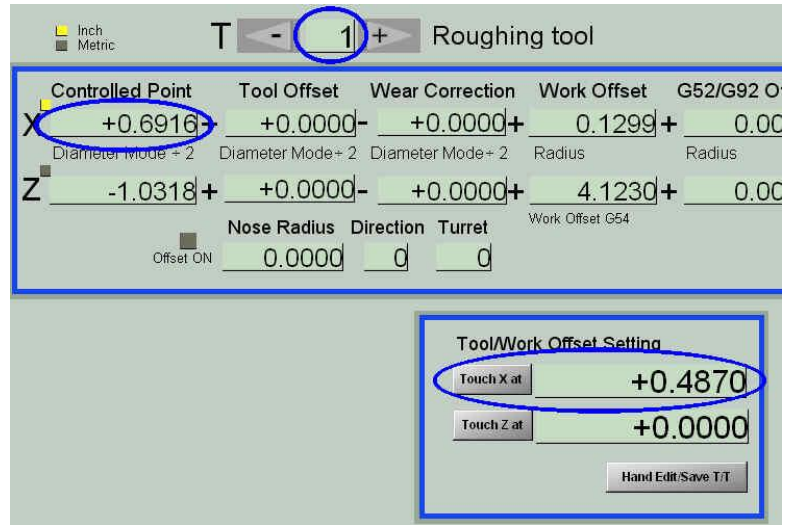


Figure 8.8 – Setting X offset for Master tool

Click the Touch X at button. If you are very sharp eyed you will see the X controlled point and Work Offset values change. The Controlled point has changed to your measured diameter and this has happened because the X work offset has been changed in the formula (figure 8.9).

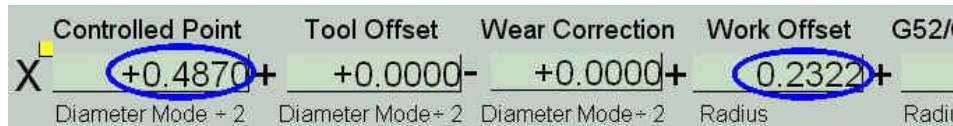


Figure 8.9 – After clicking "Touch X at"

To recap: the tool has just cut a diameter measured as 0.487" and we have made the control software's X DRO read that value by applying an X work offset.

Z offset: Using jogging face of the end of the stock. Ensure that the DRO by Touch Z at has zero in it and click the button. The Z Work Offset DRO will have been changed so the Z axis DRO reads zero.

The work offsets are now set for the master tool. If the tool table is set for your other tools then you can go ahead making the parts. The only thing you need to do for each bit of stock loaded in the chuck is to repeat the Z Offset setting by facing and clicking Touch Z at.

The next section explains about setting up tools in the tool table.

8.2.2.2 Setting Tool Table Entries

Before you do this you must have completed the section above to set the work offsets for the Master tool. Do not move the ½" diameter setting piece.

You repeat the steps below for each tool holder:

Put tool holder X (e.g. tool holder #2) in the tool post and select it as the current tool. Check its description is correct. If not you will have to edit the tool table contents as described above.

Take a light cut off the diameter for a length of ½" or so. Move the tool out of the way by jogging Z only. Measure the diameter. Enter this to the DRO beside the Touch X at button and click the button. The X Tool Offset DRO will be set to a value so the controlled point is the diameter you measured and entered.

Now jog the Z axis so the tool tip is against the faced end on the stock.

With a left hand knife tool this is easily done using the Rizla paper technique. You can use any of the other touching techniques in the PCNC1100 mill manual.

If the tool is a threading tool, a button tool etc. which does not have its cutting point on the left hand face then you need to use some ingenuity to position the cutting point a known distance from the faced end. Some suggestions for setting Z and X for unusual tools are given in figure 8.10.

Enter the distance from the cutting point to the faced end in the DRO by the Touch Z at button and click the button.

If you are going to use the control software's tool tip compensation then you should enter the tip radius and direction information in the relevant DROs. The codes for direction are indicated in figure 8.11.

That completes the setup of a tool. Go back to putting in the next tool holder and selecting it.

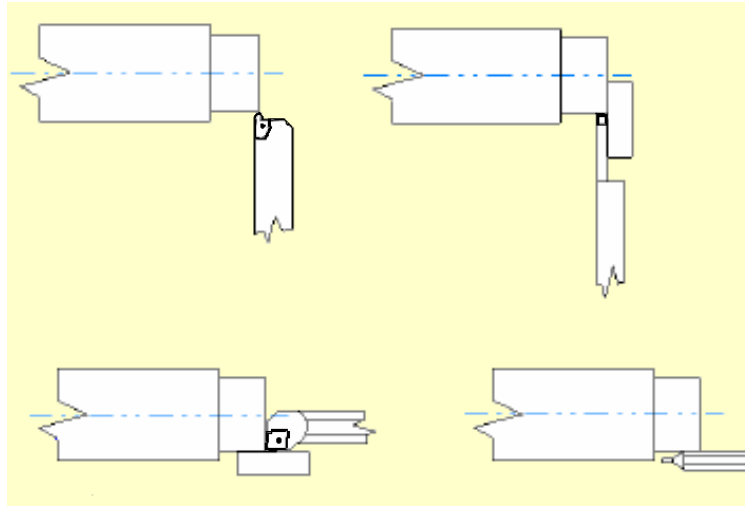


Figure 8.10 – Hints for measuring X and Z on unusual tools

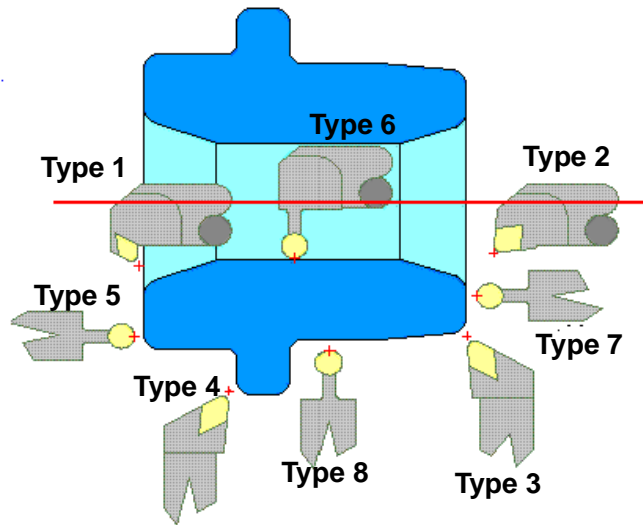


Figure 8.11 – Tool direction codes

8.2.3 The virtual cutting point

To fully understand what happens in touching tools and how this relates to facing, turning parallel and chamfering or profiling we need to look at an exaggerated view of a tip.

Figure 8.12 shows a tool tip with exaggerated radius on its tip. When it is used to face stock it cuts on the plane represented by line "A". When it is turning to a diameter it cuts on the plane

represented by line "B". In other words, so far as facing and turning is concerned, it is like a tip with zero radius at the intersection of the lines A and B. This is the virtual tip point.

As the diagram shows, it does not behave like the virtual tool when cutting at an angle. The radius on the tip means it cuts back from the virtual point. The difference depends on the angle of the direction in which it is cutting. The error is greatest at 45°

Tool tip compensation is all about allowing for this effect.. As with milling it is generally best to allow the CAM software to do the calculations which can be very difficult with a complex shape that will end up as a lot of short moves in the part-program.

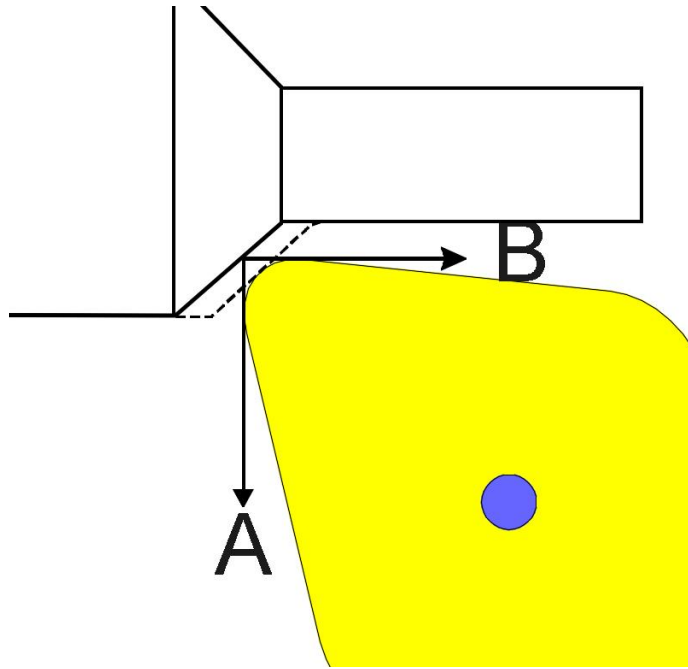


Figure 8.12 – Detail of tool tip

9. Milling work in the lathe

This chapter explains the techniques of milling parts held in the Duality chuck. The details require you to be confident in using offsets in the control software. You may find that parts of this description need to be read several times while experimenting on the machine.

9.1 The spindle clamp

In order to use a milling cutter on work in the lathe chuck one needs to clamp the lathe spindle. The Duality is supplied with a heavy duty clamp which includes an accurately calibrated scale so multiple features can be milled at exact angular spacing.

The clamp consists of two clamping bars which locate on pivots in the headstock casting. Figure 9.1 shows the installation of the front clamp bar. Figure 9.2 shows the graduated scale on the spindle and the index mark used to read the scale.

Figure 9.3 shows the locking screw for the assembled clamp.



Figure 9.1 – Spindle clamp



Figure 9.2 – Index mark



Figure 9.3 – Clamp locking screw

9.2 Maintaining coordinates moving from Turn to Mill

After turning a part, it is obviously possible to close the Duality control software, to load the PCNC 1100 mill software and to use any of the usual mill techniques to locate datum faces on the part so it can be milled. This would be tedious for machining many parts and facilities are provided so that the coordinates of the lathe spindle centerline and $Z = 0.0$ can be used when milling.

9.2.1 Principle of changeover

Three things are required to transfer between turning an milling.

- You need to have the tooling you want to use set up in the tool tables. Milling cutters will be in the PCNC 1100 software's tool table and turning tools in the Duality software's tool table.
- The control software needs to know the distance, in three-dimensional space, between the tip of the lathe master tool and the center of the cutting end of the master mill tool. This only needs to be done once when the machine is set up and then repeated if the mill or the lathe master tools are changed or the fixed tool post clamping ring is moved on the mill spindle nose.
- Before changing over the control software, the table and head needs to be moved to a fixed position known by both the Duality and the Mill software. At this position one piece of control software can be closed and the other opened.

Note: The process assumes that the mill master tool is an end mill rather than a test bar or the like.

9.2.2 Distance between tools

For this infrequent setup, you work on the Duality Settings screen. You need the Duality master tool, the milling master tool and an edge finder (wiggler) that can mount in the mill spindle.

You should be running the Duality control software. Turn and face a test piece as described in the section "The X and Z work offsets with Master Tool" in chapter 8.

Measure the diameter of the wiggler and enter this in the DRO of the controls shown in figure 9.4.

Put the wiggler in the mill spindle and using the manual spindle control run the spindle at a low speed so you can see the wiggler finding edges.

Jog the Duality to find the faced end of the test bar. Click Record Z offset.

Jog the duality to find the **back** "curved edge" of the test bar where it is turned to size. It is the side facing the column of the mill that must be measured. A wiggler is particularly effective for this measurement. The Tormach probe **can** be used provided its lathe "X" position is exactly on the lathe centerline so it probes a true diameter of the test bar. Click Record Y offset..

Stop the mill spindle and replace the wiggler with the mill master tool. Run the lathe spindle at a low speed (say 500 RPM) and, under manual control run the mill spindle at a suitable speed for the diameter of the master tool.

Jog the mill tool by eye so it is above the

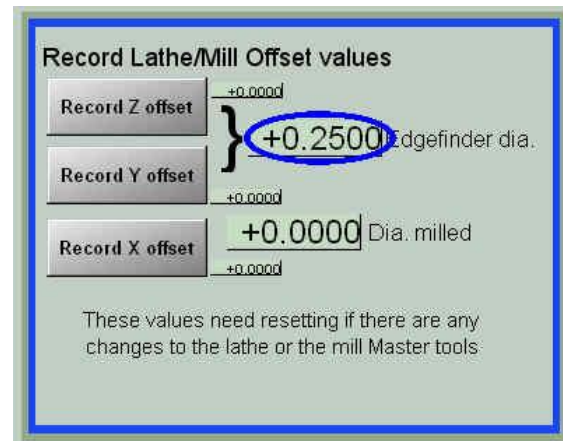


Figure 9.4 – Wiggler diameter

Fig 9.5

Photo of milling in rotating lathe spindle

lathe centerline. Carefully jog X down until the mill takes a very fine cut on the test piece. Backlash in the lathe drive will cause trouble if you take off more than a thou. or so (figure 9.5)

Stop the spindles. Do not move the lathe X axis but jog Y so you can measure the diameter of the milled bit of the test bar. Enter this in the Dia. Milled DRO and click the Record X offset button.

You have now recorded the distance between the lathe and mill master tools. You can move the axis and remove the mill master tool. These values will be saved even when you shut down the system and remain valid unless you totally remove the Duality tool post fixed ring or change either mill or Duality master tools.



Figure 9.6 – Milled diameter DRO

9.2.3 Moving to changeover position

You need to perform this step every time you want to close down the Duality control software and load the mill control software.

Jog the Duality axes so the tool is well clear of any parts of the lathe. On the Settings screen click the Move to Lathe/Mill Transfer Position button. The duality axes will move to the mutually known transfer position (figure 9.7).

Close the Duality control software clicking Yes if you are asked to Fixture Save and/or Tool Table save.

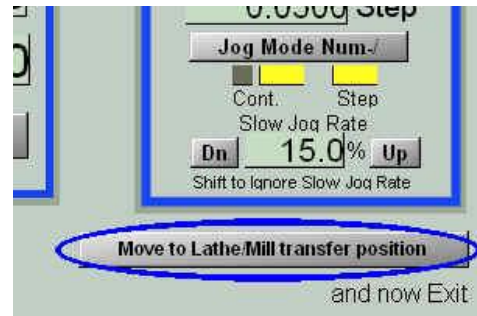


Figure 9.7 – Move to position

9.2.4 Setup in mill

Load the PCNC mill control software. Do **not do anything** to move the axes yet.

Check that the mill master tool is selected

On the Offsets screen (figure 9.8), click Set offsets to Lathe Zero. This performs two purposes: (a) The absolute current position of the machine axes are remembered (b) The mill work offsets are set so the controlled point of the current

(master) tool is Z = 3" above the lathe spindle centerline, Y = on centerline and X is the end of the stock (i.e. Z = 0.0 in the lathe coordinates).

Lock the Duality spindle in the desired position and perform the required milling based on X = 0.0 being the end of the turned stock and Y = 0.0 & Z = 0.0 being the lathe centerline.

You can use any of the mill tools set up in the mill tool table. With very short tools you might find that the Duality tool holder can crash with the lathe headstock. You can safely remove the main Duality tool post provided you do not move the upper position reference ring.

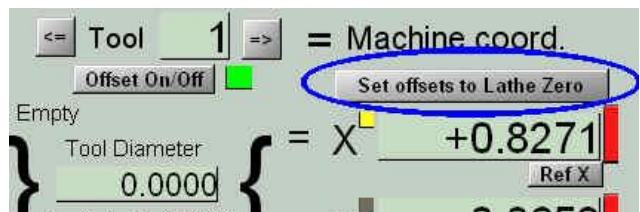


Figure 9.8 – Registering the lathe position

9.2.5 Returning to Lathe

Remove the milling tool to ensure that it will not crash with any part of the lathe. On the Settings screen (figure 9.10), click Move Machine to Lathe Zero. The axes will move to the absolute position remembered when you used Set offsets to Lathe Zero.

Do not do anything that moves the axes and shut down the mill control software. It does not matter if you save Fixture or Tool offsets.

Run the Duality lathe control software. The coordinates will be correctly set just as you left them. You will probably cutoff the completed part and proceed to load new stock, reset Z = 0.0 to the faced end and turn the next part.



Figure 9.10 – Moving to exchange position

9.2.6 Summary

The transfer process has several steps so we summarise them here.

- You need to be comfortable with the working of offsets in general and the mill and Duality tool tables in particular.
- You need to have discovered and stored the distance between the lathe master tool and the cutting end of the mill master tool. This is done on the Duality Offsets screen
- When you have completed the main turning operations you use the move to Lathe/Mill transfer position button on the Duality offsets screen.
- Without moving the axes, you close the Duality control software, open the mill control software and click the Set offsets to Lathe Zero button on the mill Offsets screen.
- You can then perform any milling operations with coordinates relative to the centerline of the lathe spindle.
- To switch back you use Move machine to Lathe Zero on the mill offsets screen and without moving the axes, close the mill control software and load the Duality control software.

If you miss any of the steps then there is a risk of crashing tools. You can minimise this risk by understanding what is going on in the process and performing your own sanity checks on the readings in the axis DROs after you have swapped functions but before you start running a part-program or Visicycle. It is quite easy to see if the tool position on the machine look consistent with the DROs.

10. Part-programming Language Reference

This section defines the language (G-codes etc.) that are understood and interpreted by the Duality Control Software.

Information on commands and on axes unused by the current Duality Lathe is sometimes included in this chapter for completeness.

10.1 Some definitions

10.1.1 *Linear Axes*

The X and Z axes form a standard right-handed coordinate system of orthogonal linear axes. Positions are expressed using coordinates in these axes. A spindle and the tailstock are mounted parallel to the Z axis. The centreline is generally made to be at $X = 0.0$.

Unlike most lathes the Duality can be moved in the Y axis. This is generally only used to set up the center-height of the tool.

10.1.2 *Scaling input*

It is possible to set up scaling factors for each axis. These will be applied to the values of X, Z, I, K and R words whenever these are entered. This allows the size of features machined to be altered and mirror images to be created - by use of negative scale factors.

The scaling is the first thing done with the values and things like feed rate are always based on the scaled values.

The offsets stored in tool and fixture tables are not scaled before use. Scaling may, of course, have been applied at the time the values were entered (say using G10).

10.1.3 *Controlled Point*

The controlled point is the notional point whose position and rate of motion are controlled. Commands like G00 and G01 specify by their X and Z words the new position of the controlled point and the Axis DROs display it.

10.1.4 *Co-ordinated Linear Motion*

To drive a tool along a specified path, a machining system must often co-ordinate the motion of several axes. We use the term "co-ordinated linear motion" to describe the situation in which, nominally, each axis moves at constant speed and all axes move from their starting positions to their end positions at the same time. This produces motion in a straight line, hence the word "linear" in the term. In actual motions, it is often not possible to maintain constant speed because acceleration or deceleration is required at the beginning and/or end of the motion. It is feasible, however, to control the axes so that, at all times, each axis has completed the same fraction of its required motion as the other axes. This moves the tool along the same path, and we also call this kind of motion co-ordinated linear motion.

Co-ordinated linear motion can be performed either at the prevailing feed rate, or at rapid traverse rate. If physical limits on axis speed make the desired rate unobtainable, all axes are slowed to maintain the desired path.

10.1.5 **Feed Rate**

The rate at which the controlled point or the axes move is nominally a steady rate which may be set by the user. In the Interpreter (unless inverse time feed rate (G93) mode is being used) the interpretation of the feed rate is as follows:

- For motion involving one or more of the linear axes (X, Z), the feed rate means length units per minute along the programmed linear XZ path.

10.1.6 **Arc Motion**

The pair of the linear axes (XZ) can be controlled to move in a circular arc in the plane of that pair of axes.

The feed rate during arc motion is as described in Feed Rate above.

10.1.7 **Coolant**

The control software supports flood coolant and mist coolant which may each be turned on independently. They are turned off together. The Duality Lathe is, in accordance with the advice from many carbide tip tool manufacturers and to avoid damage to the electronics in the headstock, designed to cut dry

10.1.8 **Dwell**

A machining system may be commanded to dwell (i.e., keep all axes unmoving) for a specific amount of time. The most common use of dwell is to break and clear chips or for a spindle to get up to speed. The units in which you specify Dwell are seconds.

10.1.9 **Units**

The setup units of the Duality Lathe are inches. Distances along the X and Z axes may be measured in millimetres or inches depending on the mode set by G20 or G21. Units for all other quantities involved in machine control cannot be changed. Different quantities use different specific units. Spindle speed is measured in revolutions per minute.. Feed rates are expressed in current length units per minute or in current length units per revolution of the spindle.

Warning: We advise you to check very carefully the system's response to changing units while tool and fixture offsets are loaded into the tables, while these offsets are active and/or while a part program is executing

10.1.10 **Current Position**

The controlled point is always at some location called the "current position" and the Control Software always knows where that is. The numbers representing the current position are adjusted in the absence of any axis motion if any of several events take place:

- Length units are changed (but see Warning above)
- Tool length offset is changed
- Coordinate system offsets are changed.

10.1.11 **Selected Plane**

There is always a "selected plane", which for turning is the XZ-plane of the machining system.

10.1.12 **Tool Table**

Zero or one tool is assigned to each slot in the tool table.

10.1.13 Tool Change

The control software allows execution of programs to stop so you can change tools by hand when required.

10.1.14 Path Control Modes

The machining system may be put into any one of two path control modes: (a) exact stop mode, (b) constant velocity mode. In exact stop mode, the machine stops briefly at the end of each programmed move. In constant velocity mode, sharp corners of the path may be rounded slightly so that the feed rate may be kept up. These modes are to allow the user to control the compromise involved in turning corners because a real machine has a finite acceleration due to the inertia of its mechanism.

Exact stop does what it says. The machine will come to rest at each change of direction and the tool will therefore precisely follow the commanded path.

Constant velocity will overlap acceleration in the new direction with deceleration in the current one in order to keep the commanded feedrate. This implies a rounding of any corner but faster and smoother cutting. The lower the acceleration of the machine axes, the greater will be the radius of the rounded corner.

10.2 Interpreter interaction with controls

The standard screens do not have controls for all these functions but they can be implemented if required on custom screens. Consult Tormach for assistance..

10.2.1 Feed and Speed Override controls

Mach3 commands which enable (M48) or disable (M49) the feed and speed override switches. It is useful to be able to override these switches for some machining operations e.g. threading. The idea is that optimal settings have been included in the program, and the operator should not change them.

10.2.2 Block Delete control

If the block delete control is ON, lines of code which start with a slash (the block delete character) are not executed. If the switch is off, such lines are executed.

10.2.3 Optional Program Stop control

The optional program stop control works as follows. If this control is ON and an input line contains an M01 code, program execution is stopped at the end on the commands on that line until the *Cycle Start* button is pushed.

10.3 Tool File

The control software maintains a tool file for each of the 253 tools which can be used.

Each data line of the file contains the data for one tool. This allows the definition of the tool tip shape, offsets and radius.

10.4 The language of part programs**10.4.1 Overview**

The language is based on lines of code. Each line (also called a "block") may include commands to the machining system to do several different things. Lines of code may be collected in a file to make a program.

A typical line of code consists of an optional line number at the beginning followed by one or more "words." A word consists of a letter followed by a number (or something that evaluates to a number). A word may either give a command or provide an argument to a command. For example, G1 X3 is a valid line of code with two words. "G01" is a command meaning "move in a straight line at the programmed feed rate," and "X3" provides an argument value (the value of X should be 3 at the end of the move). Most commands start with either G or M (for General and Miscellaneous). The words for these commands are called "G codes" and "M codes."

The language has two commands (M02 or M30), either of which ends a program. A program may end before the end of a file. Lines of a file that occur after the end of a program are not to be executed in the normal flow so will generally be parts of subroutines.

10.4.2 Parameters

The control software maintains an array of 10,320 numerical parameters. The first 1000 parameters will be undefined when the software is loaded but in advanced work they can be used as working variables in a part-program.

10.4.3 Coordinate Systems

The Duality Lathe has an absolute coordinate system and 254 work offset (fixture) systems. It is most unusual to employ other than the G54 work offset system when turning.

You can set the offsets of the fixture systems using G10 L2 P~ X~ Y~ Z~ A~ B~ C~. The P word defines the fixture to be set. The X, Y, Z etc words are the coordinates for the origin of for the axes in terms of the absolute coordinate system. You can select one of the first seven work offsets by using G54, G55, G56, G57, G58, G59. Any of the 255 work offsets can be selected by G59 P~ (e.g. G59 P23 would select fixture 23). The absolute coordinate system can be selected by G59 P0.

You can offset the current coordinate system using G52, G92 or G92.3. This offset will then be applied on top of work offset coordinate systems. This offset may be cancelled with G92.1 or G92.2. You are strongly advised to use G52 rather than G92 if you need an offset as its operation is much more intuitive.

You can make straight moves in the absolute machine coordinate system by using G53 with either G00 or G01.

10.5 Format of a Line

A permissible line of input code consists of the following, in order, with the restriction that there is a maximum (currently 256) to the number of characters allowed on a line.

- an optional block delete character, which is a slash "/" .
- an optional line number.
- any number of words, parameter settings, and comments.
- an end of line marker (carriage return or line feed or both).

Any input not explicitly allowed is illegal and will cause the Interpreter to signal an error or to ignore the line.

Spaces and tabs are allowed anywhere on a line of code and do not change the meaning of the line, except inside comments. This makes some strange-looking input legal. For example, the line g0x +0 . 12 34z 7 is equivalent to g0 x+0 .1234 z7

Blank lines are allowed in the input. They will be ignored.

Input is not case sensitive, except in comments, i.e., any letter outside a comment may be in upper or lower case without changing the meaning of a line.

10.5.1 *Line Number*

A line number is the letter N followed by an integer (with no sign) between 0 and 99999 written with no more than five digits (000009 is not valid, for example). Line numbers may be repeated or used out of order, although normal practice is to avoid such usage. A line number is not required to be used (and this omission is common) but it must be in the proper place if it is used.

10.5.2 *Subroutine labels*

A subroutine label is the letter O followed by an integer (with no sign) between 0 and 99999 written with no more than five digits (000009 is not permitted, for example). Subroutine labels may be used in any order but must be unique in a program although violation of this rule may not be flagged as an error. Nothing else except a comment should appear on the same line after a subroutine label.

10.5.3 *Word*

A word is a letter other than N or O followed by a real value.

Words may begin with any of the letters shown in figure 9.1. The table includes N and O for completeness, even though, as defined above, line numbers are not words. Several letters (I, J, K, L, P, R) may have different meanings in different contexts.

A real value is some collection of characters that can be processed to come up with a number. A real value may be an explicit number (such as 341 or -0.8807), a parameter value, an expression, or a unary operation value. Definitions of these follow immediately. Processing characters to come up with a number is called "evaluating". An explicit number evaluates to itself.

10.5.3.1 *Number*

The following rules are used for (explicit) numbers. In these rules a digit is a single character between 0 and 9.

- A number consists of (1) an optional plus or minus sign, followed by (2) zero to many digits, followed, possibly, by (3) one decimal point, followed by (4) zero to many digits - provided that there is at least one digit somewhere in the number.
- There are two kinds of numbers: integers and decimals. An integer does not have a decimal point in it; a decimal does.
- Numbers may have any number of digits, subject to the limitation on line length. Only about seventeen significant figures will be retained, however (enough for all known applications).
- A non-zero number with no sign as the first character is assumed to be positive.

Notice that initial (before the decimal point and the first non-zero digit) and trailing (after the decimal point and the last non-zero digit) zeros are allowed but not required. A number written with initial or trailing zeros will have the same value when it is read as if the extra zeros were not there.

Numbers used for specific purposes by Mach3 are often restricted to some finite set of values or some to some range of values. In many uses, decimal numbers must be close to integers; this includes the values of indexes (for parameters and carousel slot numbers, for example), M codes, and G codes multiplied by ten. A decimal number which is supposed be close to an integer is considered close enough if it is within 0.0001 of an integer.

Letter	Meaning
A	A-axis of machine
B	B-axis of machine
C	C-axis of machine
D	tool radius compensation number
F	feedrate
G	general function (see Table 5)
H	tool length offset index
I	X-axis offset for arcs X offset in G87 canned cycle
J	Y-axis offset for arcs Y offset in G87 canned cycle
K	Z-axis offset for arcs Z offset in G87 canned cycle
L	number of repetitions in canned cycles/subroutines key used with G10
M	miscellaneous function (see Table 7)
N	line number
O	Subroutine label number
P	dwel time in canned cycles dwel time with G4 key used with G10
Q	feed increment in G83 canned cycle repetitions of subroutine call
R	arc radius canned cycle retract level
S	spindle speed
T	tool selection
U	Synonymous with A
V	Synonymous with B
W	Synonymous with C
X	X-axis of machine
Y	Y-axis of machine (not relevant in Turn)
Z	Z-axis of machine

Figure 9.1 - Word initial letters

10.5.3.2 Parameter Value

A parameter value is the hash character # followed by a real value. The real value must evaluate to an integer between 1 and 10320. The integer is a parameter number, and the value of the parameter value is whatever number is stored in the numbered parameter.

The # character takes precedence over other operations, so that, for example, #1+2 means the number found by adding 2 to the value of parameter 1, not the value found in parameter 3. Of course, #[1+2] does mean the value found in parameter 3. The # character may be repeated;

for example ##2 means the value of the parameter whose index is the (integer) value of parameter 2.

10.5.3.3 Expressions and Binary Operations

An expression is a set of characters starting with a left bracket [and ending with a balancing right bracket]. In between the brackets are numbers, parameter values, mathematical operations, and other expressions. An expression may be evaluated to produce a number. The expressions on a line are evaluated when the line is read, before anything on the line is executed. An example of an expression is:

```
[1+acos[0]-[#3**[4.0/2]]]
```

Binary operations appear only inside expressions. Nine binary operations are defined. There are four basic mathematical operations: addition (+), subtraction (-), multiplication (*), and division (/). There are three logical operations: non-exclusive or (OR), exclusive or (XOR), and logical and (AND). The eighth operation is the modulus operation (MOD). The ninth operation is the "power" operation (**) of raising the number on the left of the operation to the power on the right.

The binary operations are divided into three groups. The first group is: power. The second group is: multiplication, division, and modulus. The third group is: addition, subtraction, logical non-exclusive or, logical exclusive or, and logical and. If operations are strung together (for example in the expression [2.0/3*1.5-5.5/11.0]), operations in the first group are to be performed before operations in the second group and operations in the second group before operations in the third group. If an expression contains more than one operation from the same group (such as the first / and * in the example), the operation on the left is performed first. Thus, the example is equivalent to: [((2.0/3) * 1.5) - (5.5/11.0)] which simplifies to [1.0-0.5] which is 0.5.

The logical operations and modulus are to be performed on any real numbers, not just on integers. The number zero is equivalent to logical false, and any non-zero number is equivalent to logical true.

10.5.3.4 Unary Operation Value

A unary operation value is either "ATAN" followed by one expression divided by another expression (for example ATAN[2] / [1+3]) or any other unary operation name followed by an expression (for example SIN[90]). The unary operations are: ABS (absolute value), ACOS (arc cosine), ASIN (arc sine), ATAN (arc tangent), COS (cosine), EXP (e raised to the given power), FIX (round down), FUP (round up), LN (natural logarithm), ROUND (round to the nearest whole number), SIN (sine), SQRT (square root), and TAN (tangent). Arguments to unary operations which take angle measures (COS, SIN, and TAN) are in degrees. Values returned by unary operations which return angle measures (ACOS, ASIN, and ATAN) are also in degrees.

The FIX operation rounds towards the left (less positive or more negative) on a number line, so that FIX[2.8]=2 and FIX[-2.8]=-3, for example. The FUP operation rounds towards the right (more positive or less negative) on a number line; FUP[2.8]=3 and FUP[-2.8]=-2, for example.

10.5.4 Parameter Setting

Note: Use of parameters is an advanced feature that will not be employed by many users.

A parameter setting is the following four items one after the other:

- a pound character #
- a real value which evaluates to an integer between 1 and 10320,
- an equal sign =, and

- a real value. For example "#3 = 15" is a parameter setting meaning "set parameter 3 to 15."

A parameter setting does not take effect until after all parameter values on the same line have been found. For example, if parameter 3 has been previously set to 15 and the line #3=6 G1 x#3 is interpreted, a straight move to a point where x equals 15 will occur and the value of parameter 3 will be 6.

10.5.5 *Comments and Messages*

A line that starts with the percent character, %, is treated as a comment and not interpreted in any way. Printable characters and white space inside parentheses is a comment. A left parenthesis always starts a comment. The comment ends at the first right parenthesis found thereafter. Once a left parenthesis is placed on a line, a matching right parenthesis must appear before the end of the line. Comments may not be nested; it is an error if a left parenthesis is found after the start of a comment and before the end of the comment. Here is an example of a line containing a comment: G80 M5 (stop motion)

Comments do not cause the machining system to do anything.

A comment contains a message if MSG , appears after the left parenthesis and before any other printing characters. Variants of MSG , which include white space and lower case characters are allowed. Note the comma which is required. The rest of the characters before the right parenthesis are considered to be a message to the operator. Messages are displayed on screen in the "Error" intelligent label.

10.5.6 *Item Repeats*

A line may have any number of G words, but two G words from the same modal group may not appear on the same line.

A line may have zero to four M words. Two M words from the same modal group may not appear on the same line.

For all other legal letters, a line may have only one word beginning with that letter.

If a parameter setting of the same parameter is repeated on a line, #3=15 #3=6, for example, only the last setting will take effect. It is silly, but not illegal, to set the same parameter twice on the same line.

If more than one comment appears on a line, only the last one will be used; each of the other comments will be read and its format will be checked, but it will be ignored thereafter. It is expected that putting more than one comment on a line will be very rare.

10.5.7 *Item order*

The three types of item whose order may vary on a line (as given at the beginning of this section) are word, parameter setting, and comment. Imagine that these three types of item are divided into three groups by type.

The first group (the words) may be reordered in any way without changing the meaning of the line.

If the second group (the parameter settings) is reordered, there will be no change in the meaning of the line unless the same parameter is set more than once. In this case, only the last setting of the parameter will take effect. For example, after the line #3=15 #3=6 has been interpreted, the value of parameter 3 will be 6. If the order is reversed to #3=6 #3=15 and the line is interpreted, the value of parameter 3 will be 15.

If the third group (the comments) contains more than one comment and is reordered, only the last comment will be used.

If each group is kept in order or reordered without changing the meaning of the line, then the three groups may be interleaved in any way without changing the meaning of the line. For example, the line `G40 G1 #3=15 (so there!) #4=-7.0` has five items and means exactly the same thing in any of the 120 possible orders - such as `#4=-7.0 G1 #3=15 G40 (so there!)` - for the five items.

10.5.8 Commands and Machine Modes

The control software has many commands which cause a machining system to change from one mode to another, and the mode stays active until some other command changes it implicitly or explicitly. Such commands are called "modal". For example, if coolant is turned on, it stays on until it is explicitly turned off. The G codes for motion are also modal. If a G01 (straight move) command is given on one line, for example, it will be executed again on the next line if one or more axis words is available on the line, unless an explicit command is given on that next line using the axis words or cancelling motion.

"Non-modal" codes have effect only on the lines on which they occur. For example, G04 (dwell) is non-modal.

<p>The modal Groups for G codes are</p> <ul style="list-style-type: none"> • group 1 = {G00, G01, G02, G03, G38.2, G80, G81, G82, G84, G85, G86, G87, G88, G89} motion • group 2 = {G17, G18, G19} plane selection • group 3 = {G90, G91} distance mode • group 5 = {G93, G94} feed rate mode • group 6 = {G20, G21} units • group 7 = {G40, G41, G42} cutter radius compensation • group 8 = {G43, G49} tool length offset • group 10 = {G98, G99} return mode in canned cycles • group 12 = {G54, G55, G56, G57, G58, G59, G59.xxx} coordinate system selection • group 13 = {G61, G61.1, G64} path control mode
<p>The modal groups for M codes are:</p> <ul style="list-style-type: none"> ◆ group 4 = {M00, M01, M02, M30} stopping ◆ group 6 = {M06} tool change ◆ group 7 = {M03, M04, M05} spindle turning ◆ group 8 = {M07, M08, M09} coolant (special case: M07 and M08 may be active at the same time) ◆ group 9 = {M48, M49} enable/disable feed and speed override controls
<p>In addition to the above modal groups, there is a group for non-modal G codes:</p> <ul style="list-style-type: none"> ◆ group 0 = {G04, G10, G28, G30, G53, G92, G92.1, G92.2, G92.3}

Figure 9.2 – Modal Groups

10.6 Modal Groups

Modal commands are arranged in sets called "modal groups", and only one member of a modal group may be in force at any given time. In general, a modal group contains commands for which it is logically impossible for two members to be in effect at the same time - like measure in inches vs. measure in millimetres. A machining system may be in many modes at the same time, with one mode from each modal group being in effect. The modal groups are shown in figure 9.2.

For several modal groups, when a machining system is ready to accept commands, one member of the group must be in effect. There are default settings for these modal groups. When the machining system is turned on or otherwise re-initialized, the default values are automatically in effect.

Group 1, the first group on the table, is a group of G codes for motion. One of these is always in effect. That one is called the current motion mode.

It is an error to put a G-code from group 1 and a G-code from group 0 on the same line if both of them use axis words. If an axis word-using G-code from group 1 is implicitly in effect on a line (by having been activated on an earlier line), and a group 0 G-code that uses axis words appears on the line, the activity of the group 1 G-code is suspended for that line. The axis word-using G-codes from group 0 are G10, G28, G30, and G92.

10.7 G Codes

G codes of the Duality Lathe input language are shown in figure 9.3 and are described in detail.

The descriptions contain command prototypes, set in `courier` type.

In the command prototypes, the tilde (~) stand for a real value. As described earlier, a real value may be (a) an explicit number, 4.4, for example, (b) an expression, [2+2.4], for example, (c) a parameter value, #88, for example, or (d) a unary function value, acos[0], for example.

In most cases, if axis words (any or all of X~, Z~) are given, they specify a destination point. Axis numbers relate to the currently active coordinate system, unless explicitly described as being in the absolute coordinate system. Where axis words are optional, any omitted axes will have their current value. Any items in the command prototypes not explicitly described as optional are required. It is an error if a required item is omitted.

In the prototypes, the values following letters are often given as explicit numbers. Unless stated otherwise, the explicit numbers can be real values. For example, G10 L2 could equally well be written G[2*5] L[1+1]. If the value of parameter 100 were 2, G10 L#100 would also mean the same. Using real values which are not explicit numbers as just shown in the examples is rarely useful.

If L~ is written in a prototype the "~" will often be referred to as the "L number". Similarly the "~" in H~ may be called the "H number", and so on for any other letter.

If a scale factor is applied to any axis then it will be applied to the value of the corresponding X, Z word and to the relevant I, K or R words when they are used.

Summary of G-codes	
G00	Rapid positioning
G01	Linear interpolation
G02	Clockwise circular/helical interpolation
G03	Counterclockwise circular/Helical interpolation
G04	Dwell
G10	Coordinate system origin setting
G17	XY Plane select
G18	XZ plane select
G19	YZ plane select
G20/G21	Inch/Millimetre units
G28	Return home (use with great care)
G28.1	Reference axes
G30	Return home (use with great care)
G32	Threading
G40	Cancel nose radius compensation
G41/G42	Start nose radius compensation left/right
G50	Reset all scale factors to 1.0
G51	Set axis data input scale factors
G52	Temporary coordinate system offsets
G53	Move in absolute machine coordinate system
G54	Use fixture offset 1
G55	Use fixture offset 2
G56	Use fixture offset 3
G57	Use fixture offset 4
G58	Use fixture offset 5
G59	Use fixture offset 6 / use general fixture number
G61/G64	Exact stop/Constant Velocity mode
G76	Canned cycle – screw cutting
G77	Canned cycle
G78	
G80	Cancel motion mode (including canned cycles)
G81	Canned cycle - drilling
G82	Canned cycle - drilling with dwell
G83	Canned cycle - peck drilling
G90	Absolute distance mode
G91	Incremental distance mode
G92	Offset coordinates and set parameters
G92.x	Cancel G92 etc.
G94	Feed per minute mode
G95	Feed per rev mode
G96	Constant surface speed cutting
G97	Constant RPM cutting
G98	Initial level return after canned cycles
G99	R-point level return after canned cycles

Figure 9.3 - Table of G codes

10.7.1 Rapid Linear Motion – G00

(a) For rapid linear motion, program G00 X~ Z~, where the axis words are optional, except that at least one must be used. The G00 is optional if the current motion mode is G00. This will produce co-ordinated linear motion to the destination point at the current traverse rate (or

slower if the machine will not go that fast). It is expected that cutting will not take place when a G00 command is executing.

It is an error if:

- all axis words are omitted.

If cutter radius compensation is active, the motion will differ from the above; see Cutter Compensation. If G53 is programmed on the same line, the motion will also differ; see Absolute Coordinates.

10.7.2 Linear Motion at Feed Rate – G01

For linear motion at feed rate (for cutting or not), program G1 X~ Z~ where the axis words are optional, except that at least one must be used. The G01 is optional if the current motion mode is G01. This will produce co-ordinated linear motion to the destination point at the current feed rate (or slower if the machine will not go that fast).

It is an error if:

- all axis words are omitted.

If cutter radius compensation is active, the motion will differ from the above; see Cutter Compensation. If G53 is programmed on the same line, the motion will also differ; see Absolute Coordinates.

10.7.3 Arc at Feed Rate – G02 and G03

A circular arc is specified using either G02 (clockwise arc) or G03 (counterclockwise arc). The axis of the circle or helix must be parallel to the Y-axis of the machine coordinate system. The axis (or, equivalently, the plane perpendicular to the axis) is selected with G18 (Y-axis, XZ-plane).

If cutter radius compensation is active, the motion will differ from the above; see Cutter Compensation.

Two formats are allowed for specifying an arc. We will call these the center format and the radius format. In both formats the G02 or G03 is optional if it is the current motion mode.

10.7.3.1 Radius Format Arc

In the radius format, the coordinates of the end point of the arc in the selected plane are specified along with the radius of the arc. Program G02 X~ Z R~ (or use G03 instead of G02). R is the radius. The axis words are all optional except that at least one must be used. The R number is the radius. A positive radius indicates that the arc turns through 180 degrees or less, while a negative radius indicates a turn of 180 degrees to 359.999 degrees.

It is an error if:

- both of the axis words for the axes of the selected plane are omitted,
- the end point of the arc is the same as the current point.

It is not good practice to program radius format arcs that are nearly full circles or are semicircles (or nearly semicircles) because a small change in the location of the end point will produce a much larger change in the location of the center of the circle (and, hence, the middle of the arc). The magnification effect is large enough that rounding error in a number can produce out-of-tolerance cuts. Nearly full circles are outrageously bad, semicircles (and nearly so) are only very bad. Other size arcs (in the range tiny to 165 degrees or 195 to 345 degrees) are OK.

10.7.3.2 Center Format Arc

In the center format, the coordinates of the end point of the arc in the selected plane are specified along with the offsets of the center of the arc from the current location. In this format, it is OK if the end point of the arc is the same as the current point. It is an error if:

- when the arc is projected on the selected plane, the distance from the current point to the center differs from the distance from the end point to the center by more than 0.0002 inch (if inches are being used) or 0.002 millimetre (if millimetres are being used).

The center is specified using the I and K words. There are two ways of interpreting them. The usual way is that I and K are the center relative to the current point at the start of the arc. This is sometimes called *Incremental IJ mode*. The second way is that I and K specify the center as actual coordinates in the current system. This is rather misleadingly called *Absolute IJ mode*. The IJ mode is set using the Configure>State... menu when Mach3 is set up. The choice of modes are to provide compatibility with commercial controllers. You will probably find Incremental to be best. In Absolute it will, of course usually be necessary to use both I and K words unless by chance the arc's centre is at the origin.

Note: If arcs come out at silly sizes on the toolpath display or the lathe then your part program is probably not compatible with the IJ Mode setting in the control software.

Program G02 X~ Z~ I~ K~ (or use G03 instead of G02). The axis words are all optional except that at least one of X and Z must be used. I and K are the offsets from the current location or coordinates - depending on IJ mode (X and Z directions, respectively) of the center of the circle. I and K are optional except that at least one of the two must be used. It is an error if:

- X and Z are both omitted,
- I and K are both omitted.

10.7.4 Dwell – G04

For a dwell, program G04 P~. This will keep the axes unmoving for the period of time in seconds specified by the P number. For example, with units set to Seconds, G4 P0.5 will dwell for half a second. It is an error if:

- the P number is negative.

10.7.5 Set Coordinate System Data Tool and work offset tables - G10

See details of tool and work offsets for further information on coordinate systems

To set the offset values of a tool, program

G10 L1 P~ X~ Z~ A~, where the P number must evaluate to an integer in the range 0 to 255 - the tool number - Offsets of the tool specified by the P number are reset to the given. The A number will reset the tool tip radius. Only those values for which an axis word is included on the line will be reset.

To set the coordinate values for the origin of a fixture coordinate system, program

G10 L2 P~ X~ Z~, where the P number must evaluate to an integer in the range 1 to 255 - the fixture number - (Values 1 to 6 corresponding to G54 to G59) and all axis words are optional. The coordinates of the origin of the coordinate system specified by the P number are reset to the coordinate values given (in terms of the absolute coordinate system). Only those coordinates for which an axis word is included on the line will be reset.

Note that the X coordinate is used in the G54 and G55 systems to handle front and rear tool post relative positions.

It is an error if:

- the P number does not evaluate to an integer in the range 0 to 255.

If origin offsets (made by G92 or G92.3) were in effect before G10 is used, they will continue to be in effect afterwards.

The coordinate system whose origin is set by a G10 command may be active or inactive at the time the G10 is executed.

The values set will not be persistent unless the tool or fixture tables are saved using the buttons on Tables screen.

Example: G10 L2 P1 x3.5 Z17.2 sets the origin of the first coordinate system (the one selected by G54) to a point where X is 3.5 and Z is 17.2 (in absolute coordinates).

10.7.6 Plane Selection - G17, G18, and G19

Program G18 to select the XZ-plane which is the only relevant plane for Turn.

10.7.7 Length Units - G20 and G21

Program G20 to use inches for length units. Program G21 to use millimetres.

It is usually a good idea to program either G20 or G21 near the beginning of a program before any motion occurs, and not to use either one anywhere else in the program. It is the responsibility of the user to be sure all numbers are appropriate for use with the current length units.

10.7.8 Return to Home - G28 and G30

You are advised not to use these functions.

A home position is defined by the software. The values are in terms of the absolute coordinate system and so are only valid if the machine has been referenced. They are in setup length units.

To return to home position by way of the programmed position, program G28 X~ Z~ (or use G30). All axis words are optional. The path is made by a traverse move from the current position to the programmed position, followed by a traverse move to the home position. If no axis words are programmed, the intermediate point is the current point, so only one move is made.

10.7.9 Reference axes - G28.1

Program G28.1 X~ Z~ to reference the given axes. The axes will move at the current feed rate towards the home switch(es), as defined by the Configuration. Provided the current absolute position is approximately correct, then this will give a soft stop onto the reference switch(es).

10.7.10 Threading - G32

To cut a single pass of a thread program G32 X~ Z~ F~

Before the command the X and Z position of the controlled point should be set to give Z clearance at the start of the thread (two or three pitches should suffice with a reasonably fast Z axis). The F word specifies the feed per revolution (i.e. the pitch of the thread to be cut).

The Z word gives the finishing position of the thread and the X word (if present) gives the X value at the end of the thread.. If X is omitted then it remains unchanged through the cut (i.e. a parallel thread is cut).

For example (Diameter and G21 modes): Suppose a piece of stock 12 mm in diameter is chucked with its faced end as Z=0.0 The preparatory moves could be:

```
G00 Z4
G00 X11.9 (position at cut depth with Z clearance)
```

A 1.5 mm pitch thread 0.05 mm deep 25 mm along the bar will be cut by

G32 Z-25 F1.5

And the tool is retracted to a clear X position by

G00 X14

And returned to the starting clearance.

The control software will wait until it knows the spindle speed and position (using the spindle sensor signal) and then accelerate the Z axis in the air so that it is up to the specified "feed" when it cuts. If a change in X is specified then this will be executed as motion coordinated with Z giving a tapered thread (air and cut) from the starting position.

The next pass would require a calculation of the new Z start position, if an angled in-feed is required. The X movement needs to be calculated to give the desired chip size. Typically the cut depth will be reduced as the width of the chip rises as the thread is formed.

Multiple start threads are produced by adding an offset (e.g. half, one third etc. of a pitch) to the starting Z positions.

For imperial threads which are usually specified as "threads per inch" the feed is the reciprocal of the TPI) e.g. 16 TPI is a pitch of $1 / 16 = 0.0625$ inch.

G32 is ideal for the code output by a CAD/CAM post-processor, as the writer has total control of the thread. It is, however, very inconvenient for hand coding on account of the amount of calculation required to set X and Z correctly. The G76 canned threading cycle is provided to overcome this difficulty.

10.7.11 **Nose Radius Compensation - G40, G41, and G42**

To turn cutter radius compensation off, program G40. It is OK to turn compensation off when it is already off.

To turn nose radius compensation on left (i.e., the cutter stays to the left of the programmed path when the tool radius is positive), program G41 D~ To turn cutter radius compensation on right (i.e., the cutter stays to the right of the programmed path when the tool radius is positive), program G42 D~ The D word is optional; if there is no D word, the radius of the tool currently in the spindle will be used. If used, the D number should normally be the slot number of the tool in the spindle, although this is not required. It is OK for the D number to be zero; a radius value of zero will be used.

It is an error if:

- the D number is not an integer, is negative or is larger than the number of carousel slots,
- cutter radius compensation is commanded to turn on when it is already on.

The behavior of the machining system when cutter radius compensation is ON is described in the chapter xx. Notice the importance of programming adequate clearance at the start and end of moves and at changes of direction.

10.7.12 **Scale factors G50 and G51**

To define a scale factor which will be applied to an X, Z, A, B, C, I & J word before it is used program G51 X~ Z~ A~ B~ C~ where the X, Z etc. words are the scale factors for the given axes. These values are, of course, never themselves scaled.

It is not permitted to use unequal scale factors to produce elliptical arcs with G2 or G3.

To reset the scale factors of all axes to 1.0 program G50

10.7.13 **Temporary Coordinate system offset – G52**

To offset the current point by a given positive or negative distance (without motion), program G52 X~ Z , where the axis words contain the offsets you want to provide. The axis words are optional, except that at least one must be used. If an axis word is not used for a given axis, the coordinate on that axis of the current point is not changed. It is an error if:

- all axis words are omitted.

G52 and G92 use common internal mechanisms in Mach3 and may not be used together.

When G52 is executed, the origin of the currently active coordinate system moves by the values given.

The effect of G52 is cancelled by programming G52 X0 Z0 etc.

Here is an example. Suppose the current point is at X = 4 in the currently specified coordinate system, then G52 X7 sets the X-axis offset to 7, and so causes the X-coordinate of the current point to be -3.

The axis offsets are always used when motion is specified in absolute distance mode using any of the fixture coordinate systems. Thus all fixture coordinate systems are affected by G52.

10.7.14 **Move in Absolute Coordinates - G53**

For linear motion to a point expressed in absolute coordinates, program G01 G53 X~ Z~ (or similarly with G00 instead of G01), where all the axis words are optional, except that at least one must be used. The G00 or G01 is optional if it is in the current motion mode. G53 is not modal and must be programmed on each line on which it is intended to be active. X will always be a radial value irrespective of the mode (Radius or Diameter) for which Mach3 is configured. This will produce co-ordinated linear motion to the programmed point. If G01 is active, the speed of motion is the current feed rate (or slower if the machine will not go that fast). If G00 is active, the speed of motion is the current traverse rate (or slower if the machine will not go that fast).

It is an error if:

- G53 is used without G0 or G1 being active,
- G53 is used while cutter radius compensation is on.

Note: that a G53 move with an un-referenced machine is likely to cause a crash as the machine coordinate system will contain undefined values. See chapter 7 for an overview of coordinate systems.

10.7.15 **Select Work Offset Coordinate System - G54 to G59 & G59 P~**

In turning it is unusual to use multiple work offsets. If you wish to do so please note that you should avoid G55 as it is used internally by the control software.

To select work offset #1, program G54, and similarly for the first six offsets. The system-number-G-code pairs are: (1-G54), (2-G55), (3-G56), (4-G57), (5-G58), (6-G59)

To access any of the 254 work offsets (1 - 254) program G59 P~ where the P word gives the required offset number. Thus G59 P5 is identical in effect to G58.

It is an error if:

- one of these G-codes is used while cutter radius compensation is on.

10.7.16 **Set Path Control Mode - G61, and G64**

Program G61 to put the machine into exact stop mode, or G64 for constant velocity mode. It is OK to program for the mode that is already active. These modes are described in detail above.

10.7.17 Canned Cycle – High Speed Peck Drill G73

The G73 cycle is intended for deep drilling or milling with chip breaking. See also G83. The retracts in this cycle break the chip but do not totally retract the drill from the hole. It is suitable for tools with long flutes which will clear the broken chips from the hole. This cycle takes a Q number which represents a "delta" increment along the Z-axis. Program

```
G73 X~ Z~ A~ B~ C~ R~ L~ Q~
```

- Preliminary motion, as described in G81 to 89 canned cycles.
- Move the Z-axis only at the current feed rate downward by delta or to the Z position, whichever is less deep.
- Rapid back out by the distance defined in the *G73 Pullback* DRO on the Settings screen.
- Rapid back down to the current hole bottom, backed off a bit.
- Repeat steps 1, 2, and 3 until the Z position is reached at step 1.
- Retract the Z-axis at traverse rate to clear Z.

It is an error if:

- the Q number is negative or zero.

10.7.18 Canned Cycle – Threading G76

Program G76 X~ Z~ Q~ P~ H~ I~ R~ K~ L~ C~ B~ T~ J~ to cut a complete thread.

```
X - XEnd
Z - ZEnd
Q - Spring Passes. (optional)
P - Pitch
H - Depth of first pass
I - Infeed angle
R - XStart (optional)
K - Z Start (optional)
L - Chamfer (optional)
C - X Clearance
B - Depth Last Pass (optional)
T - Taper (optional)
J - Minimum depth per pass (optional)
```

The controlled point should give a suitable clearance for air cutting on the Z axis and have X equal to the crest diameter (radius). This is not necessary if R (the starting X value) and/or K (the starting Z value) are specified.

The length and depth of the thread are given by X and Z (in relation to the start values). If the "depth" is negative then the thread is taken as internal.

The C word gives the X clearance for the rapid return motion. It is always specified as positive and Mach3 handles the reversed direction for an internal thread

The P word gives the pitch of the thread (use 1 / (TPI) for imperial threads)

The H word determines not only the first pass but, because the G76 ensures that subsequent passes cut the same volume of material per revolution determine the cuts for the entire thread and hence the number of passes.

The B word determines the depth of the last pass. If this is set too large then it can interact in an unexpected way with the constant volume calculations and the number of actual passes used.

The Q word gives the number of spring passes (i.e. cuts at the X end position).

The I word gives the infeed angle in degrees. This is Zero for radial infeed and typically 29 or 29.5 degrees for a unified or ISO metric thread or 27 degrees for 55 degree angle forms like BSW or BSF.

The values of words which are omitted are taken from the values set on the Turn Options tab of Config>Ports & Pins.

Note: This cycle is actually implemented by the macro M1076.M1S which can be inspected for implementation details. Note that one instance of this macro (usually in C:\PCNC3\Macros) is used for all profiles.

10.7.19 *Canned cycle – Turning G77*

Note: This cycle is implemented by the macro M1077.M1S which can be inspected for implementation details. There is a separate instance of this macro for the profile that you are using

10.7.20 *Canned cycle – Facing G78*

Note: This cycle is implemented by the macro M1078.M1S which can be inspected for implementation details. There is a separate instance of this macro for the profile that you are using

10.7.21 *Cancel Modal Motion - G80*

Program G80 to ensure no axis motion will occur. It is an error if:

- Axis words are programmed when G80 is active, unless a modal group 0 G code is programmed which uses axis words.

10.7.22 *Canned Cycles - G81 to G89*

The canned cycles G81 through G89 have been implemented as described in this section. Two examples are given with the description of G81 below.

All canned cycles are performed with respect to the currently selected plane. In Turn XY should be selected.

All canned cycles use X, R, and Z numbers in the NC code. These numbers are used to determine X, R, and Z positions. The R (usually meaning retract) position is along the axis perpendicular to the currently selected plane (Z-axis for XY-plane). Some canned cycles use additional arguments.

Notice that non-zero X values are only valid with a twist drill if the toolholder is live (i.e. turns the drill). If the spindle turns the work then a boring bar is being used.

For canned cycles, we will call a number "sticky" if, when the same cycle is used on several lines of code in a row, the number must be used the first time, but is optional on the rest of the lines. Sticky numbers keep their value on the rest of the lines if they are not explicitly programmed to be different. The R number is always sticky.

In incremental distance mode: when the XY-plane is selected, X, and R numbers are treated as increments to the current position and Z as an increment from the Z-axis position before the move involving Z takes place. In absolute distance mode, the X, R, and Z numbers are absolute positions in the current coordinate system.

The L number is optional and represents the number of repeats. L=0 is not allowed. If the repeat feature is used, it is normally used in incremental distance mode, so that the same sequence of motions is repeated in several equally spaced places along a straight line. In absolute distance mode, L > 1 means "do the same cycle in the same place several times," Omitting the L word is equivalent to specifying L=1. The L number is not sticky.

When L>1 in incremental mode with the XY-plane selected, the X position is determined by adding the given X number either to the current X position (on the first go-around) or to the X position at the end of the previous go-around (on the repetitions). The R and Z positions do not change during the repeats.

The height of the retract move at the end of each repeat (called "clear Z" in the descriptions below) is determined by the setting of the retract mode: either to the original Z position (if that is above the R position and the retract mode is G98), or otherwise to the R position.

It is an error if:

- X, and Z words are both missing during a canned cycle,
- a P number is required and a negative P number is used,
- an L number is used that does not evaluate to a positive integer,
- rotational axis motion is used during a canned cycle,
- inverse time feed rate is active during a canned cycle,
- cutter radius compensation is active during a canned cycle.

The Z number is sticky, and it is an error if:

- the Z number is missing and the same canned cycle was not already active,
- the R number is less than the Z number.

10.7.22.1 Preliminary and In-Between Motion

At the very beginning of the execution of any of the canned cycles, with the XY-plane selected, if the current Z position is below the R position, the Z-axis is traversed to the R position. This happens only once, regardless of the value of L.

In addition, at the beginning of the first cycle and each repeat, the following move is made:

- a straight traverse parallel to the XY-plane to the given X-position at the same time as a straight traverse of the Z-axis only to the R position, if it is not already at the R position.
- Beware of the dual axis nature of the move if the starting or finishing Z lies "within" the part as a crash of X could occur.

10.7.22.2 G81 Cycle

The G81 cycle is intended for drilling. Program G81 X~ Z~ R~ L~

- Preliminary motion, as described above.
- Move the Z-axis only at the current feed rate to the Z position.
- Retract the Z-axis at traverse rate to clear Z.

Example 1. Suppose the current position is (1, 3) and the following line of NC code is interpreted.

```
G90 G81 G98 X4 Z1.5 R2.8
```

This calls for absolute distance mode (G90), old "Z" retract mode (G98) and calls for the G81 drilling cycle to be performed once. The X number and X position are 4. The Z number and Z position are 1.5. The R number and clear Z are 2.8. The following moves take place.

- a traverse parallel to the XY-plane to (X=4, Z=3)
- a traverse parallel to the Z-axis to (X=4, Z=2.8)
- a feed parallel to the Z-axis to (X=4, Z=1.5)
- a traverse parallel to the Z-axis to (X=4, Z=3)

Example 2. Suppose the current position is (X=1, Z=3) and the following line of NC code is interpreted.

```
G91 G81 G98 X4 Z-0.6 R1.8 L3
```

This calls for incremental distance mode (G91), old "Z" retract mode and calls for the G81 drilling cycle to be repeated three times. The X number is 4, the Z number is -0.6 and the R number is 1.8. The initial X position is 5 (=1+4), the clear Z position is 4.8 (=1.8+3), and the Z position is 4.2 (=4.8-0.6). Old Z is 3.0

The first move is a traverse along the Z-axis to (X=1, Z=4.8), since old Z < clear Z.

The first repeat consists of 3 moves.

- a traverse parallel to the XY-plane to (X=5, Z=4.8)
- a feed parallel to the Z-axis to (X=5, Z=4.2)
- a traverse parallel to the Z-axis to (X=5, Z=4.8)
- The second repeat consists of 3 moves. The X position is reset to 9 (=5+4).
- a traverse parallel to the XY-plane to (X=9, Z=4.8)
- a feed parallel to the Z-axis to (X=9, Z=4.2)
- a traverse parallel to the Z-axis to (X=9, Z=4.8)

The third repeat consists of 3 moves. The X position is reset to 13 (=9+4).

- a traverse parallel to the XY-plane to (X=13, Z=4.8)
- a feed parallel to the Z-axis to (X=13, Z=4.2)
- a traverse parallel to the Z-axis to (X=13, Z=4.8)

10.7.22.3 G82 Cycle

The G82 cycle is intended for drilling. Program

```
G82 X~ Z~ R~ L~ P~
```

- Preliminary motion, as described above.
- Move the Z-axis only at the current feed rate to the Z position.
- Dwell for the P number of seconds.
- Retract the Z-axis at traverse rate to clear Z.

10.7.22.4 G83 and G83.1 Cycles

The G83 cycle (often called peck drilling) and G83.1 (rapid peck drilling) is intended for deep drilling or milling with chip breaking. See also G73. The retracts in this cycle clear the hole of chips and cut off any long stringers (which are common when drilling in aluminum). This cycle takes a Q number which distance used by each peck. Program

```
G83 X~ Z~ R~ L~ Q~
```

- Preliminary motion, as described above.
- Move the Z-axis only at the current feed rate downward by delta or to the Z position, whichever is less deep.
- For G83 rapid back out to the clear Z. For G83.1 rapid out by the peck distance (Q)
- Rapid back down to the current hole bottom, backed off a bit.
- Repeat steps 1, 2, and 3 until the Z position is reached at step 1.
- Retract the Z-axis at traverse rate to clear Z.

It is an error if:

- the Q number is negative or zero.

Note: This cycle is implemented by the macro M1083.M1S which can be inspected for implementation details. There is a separate instance of this macro for the profile that you are using.

10.7.23 **Set Distance Mode - G90 and G91**

Interpretation of Mach3 code can be in one of two distance modes: absolute or incremental.

To go into absolute distance mode, program G90. In absolute distance mode, axis numbers (X, Z) usually represent positions in terms of the currently active coordinate system. Any exceptions to that rule are described explicitly in this section describing G-codes.

To go into incremental distance mode, program G91. In incremental distance mode, axis numbers (X, Z) usually represent increments from the current values of the numbers.

I and K numbers always represent increments or absolute distances depending on IJ Mode setting

10.7.24 **G92 Offsets - G92, G92.1, G92.2, G92.3**

You are strongly advised **not** to use this legacy feature on any axis where there is another offset applied. G52 is much easier to use.

To make the current point have the coordinates you want (without motion), program G92 X~ Z~, where the axis words contain the axis numbers you want. The axis words are optional, except that at least one must be used. If an axis word is not used for a given axis, the coordinate on that axis of the current point is not changed. It is an error if:

- all axis words are omitted.

G52 and G92 use common internal mechanisms in Mach3 and may not be used together.

When G92 is executed, the origin of the currently active coordinate system moves. To do this, origin offsets are calculated so that the coordinates of the current point with respect to the moved origin are as specified on the line containing the G92. In addition, parameters 5211 to 5216 are set to the X, Z, A, B, and C-axis offsets. The offset for an axis is the amount the origin must be moved so that the coordinate of the controlled point on the axis has the specified value.

Here is an example. Suppose the current point is at X=4 in the currently specified coordinate system and the current X-axis offset is zero, then G92 X7 sets the X-axis offset to -3, sets parameter 5211 to -3, and causes the X-coordinate of the current point to be 7.

The axis offsets are always used when motion is specified in absolute distance mode using any of the fixture coordinate systems. Thus all fixture coordinate systems are affected by G92.

Being in incremental distance mode has no effect on the action of G92.

Non-zero offsets may be already be in effect when the G92 is called. They are in effect discarded before the new value is applied. Mathematically the new value of each offset is A+B, where A is what the offset would be if the old offset were zero, and B is the old offset. For example, after the previous example, the X-value of the current point is 7. If G92 X9 is then programmed, the new X-axis offset is -5, which is calculated by $[[7-9] + -3]$. Put another way the G92 X9 produces the same offset whatever G92 offset was already in place.

To reset axis offsets to zero, program G92 . 1 or G92 . 2 G92.1 sets parameters 5211 to 5216 to zero, whereas G92.2 leaves their current values alone.

To set the axis offset values to the values given in parameters 5211 to 5216, program G92 . 3

You can set axis offsets in one program and use the same offsets in another program. Program G92 in the first program. This will set parameters 5211 to 5216. Do not use G92.1 in the remainder of the first program. The parameter values will be saved when the first program exits

and restored when the second one starts up. Use G92.3 near the beginning of the second program. That will restore the offsets saved in the first program.

10.7.25 ***Set Feed Rate Mode - G94 and G95***

Two feed rate modes are recognized: units per minute and units per revolution of spindle. Program G94 to start the units per minute mode. Program G95 to start the units per rev mode.

In units per minute feed rate mode, an F word on the line is interpreted to mean the controlled point should move at a certain number of inches per minute, millimetres per minute, or degrees per minute, depending upon what length units are being used and which axis or axes are moving.

In units per rev feed rate mode, an F word on the line is interpreted to mean the controlled point should move at a certain number of inches per spindle revolution, millimetres per spindle revolution, or degrees per spindle revolution, depending upon what length units are being used and which axis or axes are moving.

10.7.26 ***Set spindle speed mode – G96 and G97***

This modes are not applicable to the Duality Lathe as the control software does not have control of the spindle. They are documented for completeness.

Program G96 S~ to enter Constant Surface Speed mode. The S word is the desired surface speed of the workpiece. It is in Feet per minute in G20 mode or meters per minute in G21 mode. The actual commanded spindle speed will be calculated from the X diameter of the controlled point (i.e. the feature takes account of tool and work offsets)

Program G97 S~ to enter Constant RPM mode. The S word is the desired spindle speed in RPM.

10.7.27 ***Set Canned Cycle Return Level - G98 and G99***

When the spindle retracts during canned cycles, there is a choice of how far it retracts:

1. retract perpendicular to the selected plane to the position indicated by the R word, or
2. retract perpendicular to the selected plane to the position that axis was in just before the canned cycle started (unless that position is lower than the position indicated by the R word, in which case use the R word position).

To use option (1), program G99 To use option (2), program G98 Remember that the R word has different meanings in absolute distance mode and incremental distance mode.

10.8 **Built-in M Codes**

M codes interpreted directly by Mach2 are shown in figure 9.4.

10.8.1 ***Program Stopping and Ending – M00, M01, M02, M30***

To stop a running program temporarily (regardless of the setting of the optional stop switch), program M00.

To stop a running program temporarily (but only if the optional stop switch is on), program M1.

It is OK to program M00 and M01 in MDI mode, but the effect will probably not be noticeable, because normal behavior in MDI mode is to stop after each line of input, anyway.

If a program is stopped by an M00, M01, pressing the cycle start button will restart the program at the following line.

M-code	Meaning
M00	Program stop
M01	Optional program stop
M02	Program end
M03/04	Rotate spindle clockwise/counterclockwise
M05	Stop spindle rotation
M06	Tool change (by two macros)
M07	Mist coolant on
M08	Flood coolant on
M09	All coolant off
M30	Program end and Rewind
M47	Repeat program from first line
M48	Enable speed and feed override
M49	Disable speed and feed override
M98	Call subroutine
M99	Return from subroutine/repeat

Figure 9.4 - Built in M-codes

To end a program, program M02 or M30. M02 leaves the next line to be executed as the M02 line. M30 "rewinds" the G-code file. These commands can have the following effects depending on the options chosen on the Configure>Logic dialog:

- Axis offsets are set to zero (like G92.2) and origin offsets are set to the default (like G54).
- Selected plane is set to XZ (like G18).
- Distance mode is set to absolute (like G90).
- Feed rate mode is set to Units per minute mode (like G94).
- Feed and speed overrides are set to ON (like M48).
- Cutter compensation is turned off (like G40).
- The spindle is stopped (like M05).
- The current motion mode is set to G01 (like G01).
- Coolant is turned off (like M09).

No more lines of code in the file will be executed after the M02 or M30 command is executed. Pressing cycle start will resume the program (M02) or start the program back at the beginning of the file (M30).

10.8.2 Spindle Control – M03, M04, M05

These codes are not applicable to the Duality Lathe as the spindle is manually controlled by the operator but are included for completeness

To start the spindle turning clockwise at the currently programmed speed, program M03.

To start the spindle turning counterclockwise (CCW) at the currently programmed speed, program M04. Some lathes have chucks mounted on a screwed spindle nose. Use CCW rotation with the greatest care if it is available as the chuck will tend to come unscrewed.

For a PWM or Step/Dir spindle the speed is programmed by the S word. For an on/off spindle control it will be set by the gearing/pulleys on the machine.

To stop the spindle from turning, program M05.

It is OK to use M03 or M04 if the spindle speed is set to zero. If this is done (or if the speed override switch is enabled and set to zero), the spindle will not start turning. If, later, the spindle speed is set above zero (or the override switch is turned up), the spindle will start turning. It is permitted to use M03 or M04 when the spindle is already turning or to use M05 when the spindle is already stopped but see the discussion on safety interlocks in configuration for the implications of a sequence which would reverse an already running spindle.

10.8.3 Tool change – M06

M06 is not needed as tool changing is actually performed when the T word is used.

10.8.4 Coolant Control – M07, M08, M09

Coolant is not to be used with the Duality Lathe but the meaning of these codes is included for completeness.

To turn flood coolant on, program M07.

To turn mist coolant on, program M08.

To turn all coolant off, program M09.

It is always OK to use any of these commands, regardless of what coolant is on or off.

10.8.5 Re-run from first line - M47

On encountering an M47 the part program will continue running from its first line. It is an error if:

- M47 is executed in a subroutine

The run can be stopped by the Pause or Stop buttons

See also the use of M99 outside a subroutine to achieve the same effect.

10.8.6 Override Control - M48 and M49

To enable the speed and feed override, program M48. To disable both overrides, program M49. It is OK to enable or disable the switches when they are already enabled or disabled.

10.8.7 Call subroutine - M98

This has two formats:

(a) To call a subroutine program within the current part program file code M98 P~ L~ or M98 ~P ~Q The program must contain an O line with the number given by the P word of the Call. This O line is a sort of "label" which indicates the start of the subroutine. The O line may not have a line number (N word) on it. It, and the following code, will normally be written with other subroutines and follow either an M2, M30 or M99 so it is not reached directly by the flow of the program.

(b) To call a subroutine which is in a separate file code M98 (filename) L~

for example M98 (test.tap)

For both formats:

The L word (or optionally the Q word) gives the number of times that the subroutine is to be called before continuing with the line following the M98. If the L (Q) word is omitted then its value defaults to 1.

By using parameters values or incremental moves a repeated subroutine can make several roughing cuts around a complex path or cut several identical objects from one piece of material.

Subroutine calls may be nested. That is to say a subroutine may contain a M98 call to another subroutine. As no conditional branching is permitted it is not meaningful for subroutines to call themselves recursively.

10.8.8 ***Return from subroutine***

To return from a subroutine program M99 Execution will continue after the M98 which called the subroutine.

If M99 is written in the main program, i.e. not in a subroutine, then the program will start execution from the first line again. See also M47 to achieve the same effect.

10.9 **Macro M-codes**

10.9.1 ***Macro overview***

If any M-code is used which is not in the above list of built-in codes then Mach2 will attempt to find a file named "Mxx.M1S" in the Macros folder. If it finds the file then it will execute the VB script program it finds within it.

New macros can be written using an external editor program like Notepad and saved in the Macros folder.

10.10 **Other Input Codes**

10.10.1 ***Set Feed Rate - F***

To set the feed rate, program F~

Depending on the setting of the Feed Mode toggle the rate may be in units-per-minute or units-per-rev of the spindle.

The units are those defined by the G20/G21 (G70/G71) mode.

A revolution of the spindle is defined as a pulse appearing on the Index input.

The feed rate may sometimes be overridden as described in M48 and M49 above.

10.10.2 ***Set Spindle Speed - S***

To set the desired speed in revolutions per minute (rpm) of the spindle, program S~ The spindle is actually manually controlled but an on-screen br-graph will display . It is OK to program an S word whether the spindle is turning or not. If the speed override switch is enabled and not set at 100%, the speed will be different from what is programmed. It is OK to program S0; the spindle will not turn if that is done. It is an error if:

- the S number is negative.

10.10.3 ***Select Tool – T***

To select a tool, program T~ where the T number is entry number in the tool table for the tool to be used and the entry number for the offsets to be applied.

For example T0202 (or equivalently T202) selects tool 2 with its own offsets and T0207 selects tool 2 with the offsets for tool 7.

T02 is treated as equivalent to T0202 and similarly for any other value less than or equal to 99.

When the command is encountered the Control Software will then wait for Cycle Start to be pressed and continue running the part program.

It is OK to program T0; no tool will be selected. This is useful if you want the spindle to be empty after a tool change. It is an error if:

- a negative T number is used, or a T number larger than 9999 is used.

10.11 Error Handling

This section describes error handling in the control software.

It attempts to report syntax errors in part programs. If, however, a command does not work as expected or does not do anything then check that you have typed it correctly. Common mistakes are G0, instead of G0 i.e. letter O instead of zero) and too many decimal points in numbers. The software does not check for excessively high feeds or speeds. Nor does it does not detect situations where a legal command does something unfortunate, such as machining the chuck.

10.12 Order of Execution

The order of execution of items on a line is critical to safe and effective machine operation. Items are executed in the order shown in figure 9.5 (on next page) if they occur on the same line. If you are in any doubt code commands on separate lines. There is no cost in terms of part-program performance.

Order	Item
1	Comment (including message)
2	Set feed rate mode (G93, G94, G95)
3	Set feed rate (F)
4	Set spindle speed (S)
5	Select tool
6	Tool change (M6) and Execute M-code macros
7	Spindle On/Off (M3, M4, M5)
8	Coolant On/Off (M7, M8, M9)
9	Enable/disable overrides (M48, M49)
10	Dwell (G4)
11	Set active plane (G17, G18, G19)
12	Set length units (G20, G21)
13	Cutter radius compensation On/Off (G40, G41, G42)
14	Tool table offset On/Off (G43, G49)
15	Fixture table select (G54 - G58 & G59 P~)
16	Set path control mode (G61, G61.1, G64)
17	Set distance mode (G90, G91)
18	Set canned cycle return level mode (G98, G99)
19	Home, or change coordinate system data (G10), or set offsets (G92, G94)
20	Perform motion (G0 to G3, G12, G13, G80 to G89 as modified by G53)
21	Stop or repeat (M0, M1, M2, M30, M47, M99)

Figure 9.5 - Order of execution on a line

11. Tool Tip Radius Compensation

Tool tip radius compensation is a feature of the control software which currently under development. Most CAD/CAM programs can be told the profile of your tool and will output part programs which cut the part profile which you have drawn by allowing for the tool tip shape. Because the CAD/CAM software has a better overall view of the shapes being cut it will generally do a better job than the control software can if you have a complicated profile.

11.1 Introduction to Compensation

As we have seen the control software controls the movement of the Controlled Point. Figure 10.1 shows a "sharp pointed" tool cutting a diameter, a taper and facing.

Now if we have a radiused tip (say 3 mm radius like a 6 mm button tool) centred where the point was it will cut the diameter too small as shown at A in figure 10.2. This can be corrected by adding 3 mm to the X coordinate given by the program (this is of course 6 mm in Diameter mode). The facing cut would also be wrong by 3 mm, so 3 mm needs adding to the Z coordinate too. This can of course be done by putting 3.0 in the *X Offset* and *Z Offset* columns of the tool table for the current tool.

This will mean that the tool cuts as in positions B and D in figure 9.2. There is however a problem, at position C, on the chamfer. The centre of the tool is 4.24 mm away from the correct line of cut (i.e. the hypotenuse of a right-angled triangle with sides 3 mm). It should of course only be 3 mm away.

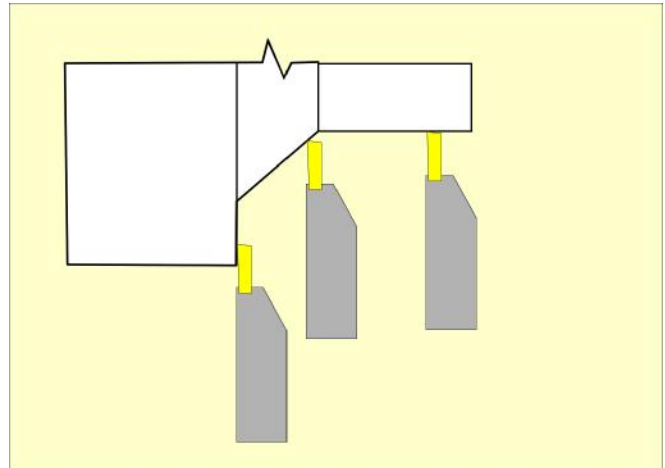


Figure 10.1 – Turning with a "sharp" tool

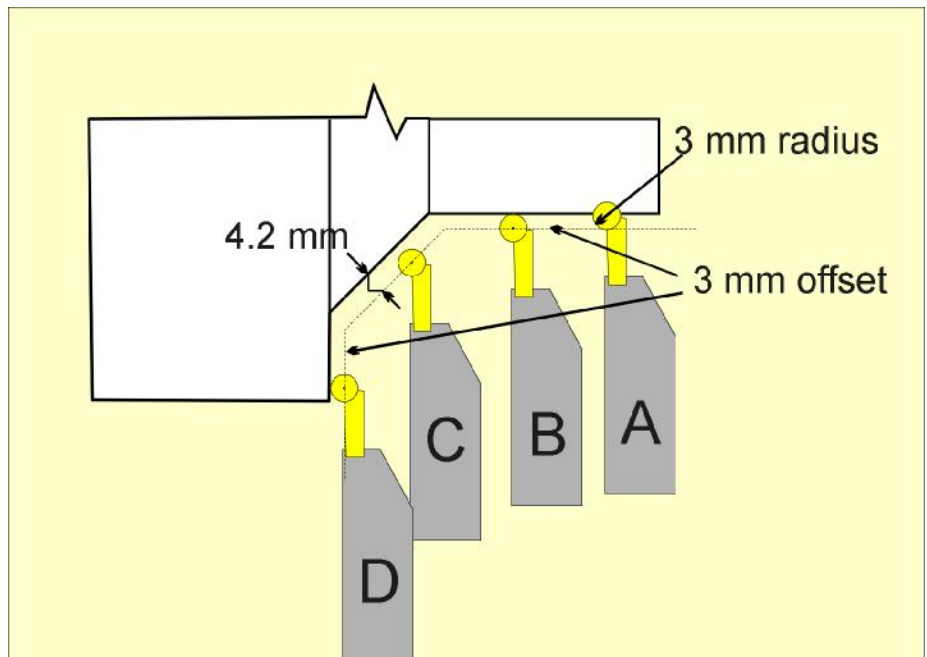


Figure 10.2 – The offset toolpath for a radiused tool tip

Tool Tip Radius Compensation

If the control software knows the angle of a chamfer then it is quite easy to calculate the sides of the triangle to get the hypotenuse 3 mm away from the work. Figure 10.3 shows a 60° chamfer and some more intricate profiling. The tool at position A is compensated by an offset of 2.6 mm in Z ($3 \times \sin(60)$) and 1.5 mm in X ($3 \times \cos(60)$) giving a hypotenuse of 3 mm.

There are, however some difficulties when the tool is near a corner.

What the control software needs to know is the angle of the tangential contact line. This is not hard where two straight lines meet but is non-trivial when near curves like positions B and C. These curves may actually be made up of moves in many blocks of G-code and the software can only know where the tool should touch the profile by looking ahead.

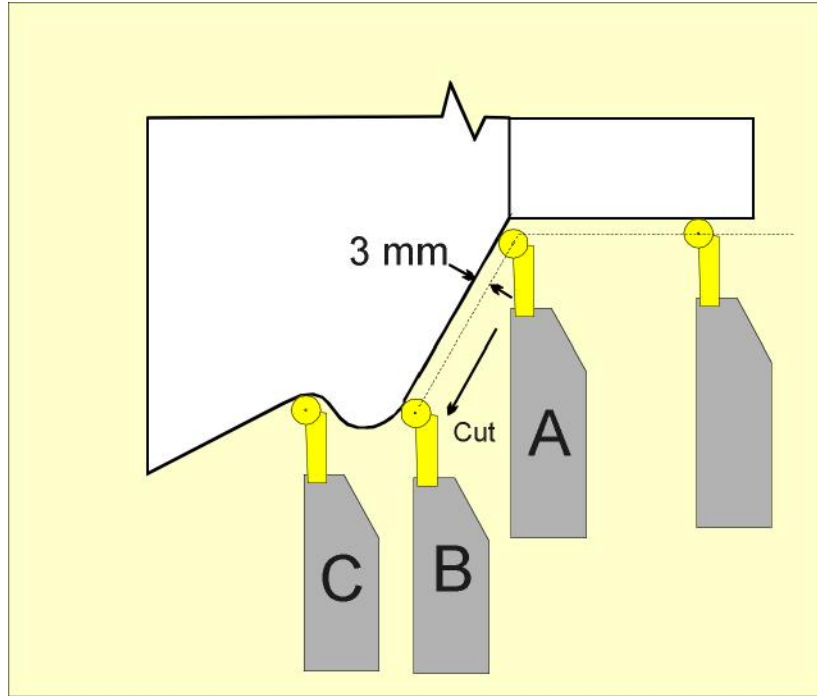


Figure 10.3 – Tools contacting at different angles

You are very strongly advised to allow your CAD/CAM software to take account of this. This software has an overall view of the shapes. The control software only sees a local view.

If you are hand coding or using a Wizard it is useful for the control software to do the best with what it **does** know.

11.2 How compensation is specified

When you set up the X and Z offsets for a tool you are defining the effective cutting point. In

ToolTable								
Tool	Description	Tip Dir...	Tip Ra...	X Offset	Z Offset	X Wear	Z Wear	Turret...
0	Ref. Tool	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000
1	Roughing	0.0000	0.4000	0.0000	0.0000	0.0000	0.0000	0.0000
2	Thread	0.0000	0.0000	2.5600	6.540...	0.2000	0.0000	0.0000
3	Part	0.0000	0.0000	-4.6000	1.5000	0.3000	0.0000	0.0000
4	Profile 2	0.0000	6.0000	3.15	-3.4	0.0000	0.0000	0.0000
5	Finishing	0.0000	0.1000	2.4400	-2.1000	0.0000	0.0000	0.0000

Figure 10.4 – Tool direction entry in table

Tool Tip Radius Compensation

order to do tool tip radius correction the control software needs to know the radius involved and where the centre of this radius is. The centre can, of course, easily be worked by using the radius and effective cutting point provided the software knows the direction in which the tool was pointing when the offsets were setup. This direction is given in the Tip Direction column of the tool table – see figure 10.4.

In the example (figure 10.3) we would offset to the left when cutting from tailstock to chuck with a tool in the front toolpost as indicated by the arrow.

Figure 10.5 shows the numbering of tools in the front toolpost as used in the Duality

Compensation offsetting is switched on by G41 or G42. G41 offsets to the left and G42 offsets to the right. G40 switches tip radius compensation off.

G41 and G42 use the tool nose radius and direction of the current tool unless a D word is specified. If D is specified then its value is the index number in the tool table which specifies the radius and direction.

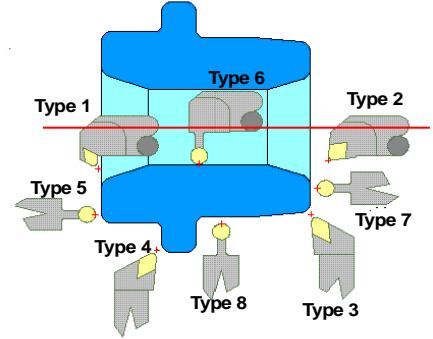


Figure 10.5 – Front tool post directions

11.3 Potential difficulties

Obviously the tool will move into its new position in the move after you switch compensation on with G41 or G42. You should make sure that it is not near the stock when you turn compensation on. A value of twice the tip radius in the Z direction and four times on the diameter is always safe.

Sometimes when you change the direction of a cut at a corner you will need to change from right to left hand compensation or vice-versa. You must be sure that the tool will not crash into the stock when this change is taking place.

12. Warranty, Specification and Maintenance

12.1 Intended Use Statement

The Duality lathe is intended for use as a general purpose CNC lathe with the option of being used manually. The intended use includes cutting conventional (non-abrasive) materials such as unhardened steel, aluminum, plastics, wood and similar materials. In CNC mode, the Duality lathe is intended to be used in conjunction with a PCNC 1100 milling machine with the software configuration files provided by Tormach.

12.2 Support

Tormach provides free technical support through multiple channels. The methods are listed below, in order of preference. The quickest way to get the answers you need is normally checking in order of preference:

- **This manual – ALWAYS the first place to check!!**
- **Related documents found at:** <http://www.tormach.com/documents.htm>
- **Our website at:** www.tormach.com
- **Email to:** info@tormach.com
- **Telephone Tormach at:** 608-849-8381
- **Fax Tormach at:** 209-885-4534

12.3 Outside of the Scope of Intended Use

Applications for the equipment or modifications of the equipment outside of the Intended Use Statement are supported through consulting engineering, not through our free support policy. There are no limits to the applications that Tormach products can be used for or to the modifications that can be applied to the Tormach machinery. Tormach designs use standard industrial components and incorporate the principles of Open Architecture specifically to allow and promote these variations. With Open Architecture controls, industrial engineers will find Tormach products cost effective to incorporate into larger manufacturing systems, or with easily separable base, column and head sections manufacturing engineers looking to design specialized in-house equipment will find they can use the base as a low cost motorized XY table. Some machinists may want to convert a stepper mill to servos or some software engineers may want to replace the Tormach approved software with something of their own creation.

All of the technical information and insight required to support these variations from the intended use cannot possibly be foreseen. If the extensive documentation provided does not supply all the information you need, we can provide additional information and engineering support required for your project on a consulting engineering basis. We are actually very good at this. If you have your questions well organized we can normally provide all the information you need in short order. Consulting engineering is done by electrical and mechanical engineers and billed at current hourly rates.

As you might expect, all warranties for Tormach equipment are voided through modification to the equipment or use outside of the Intended Use. Individuals or companies involved with modifying the equipment or applying the products assume all consequent liability.

12.4 Specification

Swing Over Bed: 7" (180mm)

Swing Over Carriage: 4.3" (110mm)

Distance Between Centers: 11.8" (300mm)

Spindle Hole: 0.79" (20mm)

Range of manual Threads: 4-100 TPI (0.25-5mm)

Spindle Taper: #3 Morse Taper

Tailstock Taper: #2 Morse Taper

Low Range Spindle Speed: 50-1100 RPM

High Range Spindle Speed: 100-2500 RPM

Spindle Motor power input 350 watts

Voltage: 115 VAC (+/- 10%)

Weight: 110 lbs (50 kgs)

Shipping Dimensions: 32" x 14" x 18"

12.5 Maintenance

12.5.1 CNC mode

When used in CNC mode the Duality lathe has few moving parts but the spindle drive belt needs to be aligned correctly or it will wear on one side or the other.

To adjust the drive belt:

1. Unplug the power cord.
2. Remove the motor cover from the back of the lathe.
3. Remove the change gear cover.
4. Remove the control box from the front of the lathe, but do not disconnect any wires.
5. Loosen all the lock nuts on the front of the lathe.
6. Back off the top and bottom set screws a couple turns.
7. Make sure that nothing will get into the change gear drive.
8. Make sure the control box is in a safe but accessible position.
9. Plug in the power cord.
10. Reach around to the back of the lathe and hold the motor.
11. Turn the motor on at slow speed.
12. While the motor is running, move the motor to a position where the belt has sufficient tension and does not rub either side of the pulley.
13. Tighten the nuts on the two studs that are aligned horizontally.
14. Snug the setscrews that are aligned vertically.
15. Turn off the motor.
16. Tighten the lock nuts.
17. Replace the control box.

18. Replace the change gear cover.
19. Replace the motor cover.

12.5.2 Manual mode

12.5.2.1 Adjustments

Keeping your lathe in adjustment is an ongoing process. You should check all the following adjustments when you set up your lathe and then periodically as you use your lathe.

Carriage

The carriage is held on the ways by two adjustable retaining plates that are bolted to the bottom of the carriage.

There are several fasteners in the carriage retainers. The socket head cap screws are used to adjust the position of the retainers. The setscrews and lock nuts lock the adjustments in place.

To adjust the carriage retainers:

1. Remove the right lead screw mounting bracket.
2. Disconnect the apron by removing the two socket head cap screws through the front of the carriage.
3. Slide the apron to the right and off the lead screw.
4. Loosen all the fasteners on both retainers.
5. Snug the socket head cap screws so the carriage can move, but without play.
6. Snug the setscrews. Do not over tighten or you might break the retainers.
7. While holding the setscrews from turning, tighten the lock nuts.
8. Replace the apron.
9. Replace the right lead screw mounting bracket.

Cross Slide Gibs

A gib is a strip of metal placed between the bearing surface of two machine parts to ensure a precision fit and provide adjustment for wear. The Duality lathe has gibs in several places, including the cross slide.

To adjust the cross slide gibs:

1. Loosen the three lock nuts on the side of the cross slide.
2. Slightly loosen all three setscrews on the side of the cross slide.
3. Snug each setscrew equally. This will lock the cross slide in position.
4. Loosen each setscrew 1/8 turn to allow the cross slide to move.
5. While holding the setscrews from turning, tighten the lock nuts.
6. Test by turning the handle. Loosen or tighten all the setscrews the same amount until the cross slide moves freely, but without play in the dovetail.

Cross Slide Nut

The cross slide nut is adjustable to remove free play from the cross slide feed handle.

The three screws in the top of the cross slide adjust the cross slide nut.

The two outer screws tip the nut off horizontal to reduce the endplay in the threads. The center screw locks the adjustment in place.

To adjust the cross slide nut:

1. Loosen all three screws.
2. Back the cross slide all the way away from the center of the lathe, until it just contacts the dial.
3. Tighten the center screw until the feed handle becomes hard to turn.
4. Loosen the center screw until the feed handle just turns freely.
5. Tighten the near screw until the feed handle becomes hard to turn.
6. Loosen the near screw until the feed handle just turns freely.
7. Snug, but do not tighten the far screw.

Compound Rest Gibs

The compound rest also incorporates a gib for adjustment.

To adjust the compound rest gibs:

1. Loosen the three lock nuts on the side of the compound rest.
2. Slightly loosen all three setscrews on the side of the compound rest.
3. Snug each setscrew equally. This will lock the compound rest in position.
4. Loosen each setscrew 1/8 turn to allow the compound rest to move.
5. While holding the setscrews from turning, tighten the lock nuts.
6. Test by turning the handle. Loosen or tighten all the setscrews the same amount until the compound rest moves freely, but without play in the dovetail.

Apron Position

The apron is adjustable to center the half nuts horizontally on the lead screw.

To adjust the apron position:

1. Loosen the two socket head cap screws that secure the apron to the carriage. They are at the front edge of the carriage.
2. Engage the half nuts on the lead screw.
3. Tighten the two socket head cap screws.

Half Nuts

There are two adjustments for the half nuts. The half nut gibs take the play out of the half nuts. The half nut closing limit stops the half nuts from closing too tightly on the lead screw.

To adjust the half nut gibs: Tighten the three setscrews in the back edge of the apron to remove play from the half nuts.

To adjust the half nut limit:

1. Loosen the lock nut on the bottom of the half nuts.
2. Adjust the setscrew until the half nuts close without binding on the lead screw.
3. While holding the setscrew from turning, tighten the lock nut.

Lead Screw Mounting

The brackets that mount the lead screw can move slightly to ensure that the lead screw does not bind in the half nuts.

To adjust the right lead screw mounting bracket:

1. Remove the tailstock by sliding it off the end of the ways.
2. Loosen the two mounting socket head cap screws on the right bracket.

3. Move the carriage as far to the right as possible.
4. Engage the half nuts on the lead screw.
5. Tighten the bracket mounting socket head cap screws.
6. Replace the tailstock.

To adjust the left lead screw mounting bracket:

1. Remove the change gear cover.
2. Loosen the locking nut on the change gear adjuster.
3. Loosen the two mounting socket head cap screws on the left lead screw bracket.
4. Move the carriage as far to the left as possible.
5. Engage the half nuts on the lead screw.
6. Tighten the bracket mounting socket head cap screws.
7. Tighten the locking nut on the change gear adjuster.
8. Replace the change gear cover.

12.5.2.2 Lubrication

We recommend the use of two lubricants on your lathe.

- Where oil is required, we recommend Mobil 1 synthetic motor oil. Mobil 1 far exceeds the lubrication needs of the Duality lathe, and maintains a good surface film between applications.
- Where grease is required, we recommend Lubriplate 630-AA lithium (white) grease. Lithium grease is a plastic-friendly grease that is easy to find and easy to use.

The following points on your lathe require lubrication.

Location	Lubricant	Frequency	Notes
Lathe ways	Oil	Daily	Apply oil to both the front and back ways on both sides of the carriage. Move the carriage back and forth to spread the oil.
Lead screw threads	Oil	Daily	Clean swarf (chips, shavings, and debris) daily.
Compound rest dovetail	Oil	Daily	Advance the compound rest to the extent of its normal travel. Apply oil to the end of the gib and the ends of the dovetails. Retract the compound rest.
Cross slide dovetail	Oil	Daily	Advance the cross slide to the extent of its travel. Apply oil to the end of the gib and the ends of the dovetails. Retract the cross slide

Warranty, Specification and Maintenance

Lead screw bushings	Oil	Weekly	There is an oil fitting on the top of each one. Remove the change gear cover to lubricate the left bushing.
Other machined surfaces	Oil	Weekly	Oil lubricates and prevents corrosion.
Chuck	Oil	Monthly	Disassemble, clean and lubricate. Wrap with a paper towel, secure with an elastic band, and run lathe to sling out excess oil.
Cross slide feed screw	Grease	Yearly	
Compound rest feed screw	Grease	Yearly	
Lead screw drive gears and bushings	Grease	Yearly	Also lube change gears as you use them
Transmission gears	Grease	Yearly	See procedure below for lubricating the transmission gears without removing the headstock
Carriage hand wheel drive gears	Grease	Yearly	
Tailstock quill and screw	Grease	Yearly	

The spindle and countershaft bearings are deep groove ball bearings that are shielded and do not require additional lubrication.

Lubricating the Transmission Gears

You can lubricate the transmission gears without removing the headstock by using a spray can of lithium grease.

To lubricate the transmission gears:

1. Unplug the power cord.
2. Remove the control box from the front of the lathe, but do not disconnect any wires.
3. Remove the screw or screws holding the ground wires to the front of the headstock.
4. Insert the lithium grease can's spray tube into one of the open holes.
5. Spray the grease while rotating the chuck by hand.
6. Shift the high/low speed shifter to the opposite position.
7. Spray the grease while rotating the chuck by hand.
8. Reconnect the ground wires.
9. Replace the control box.
10. Plug in the power cord.

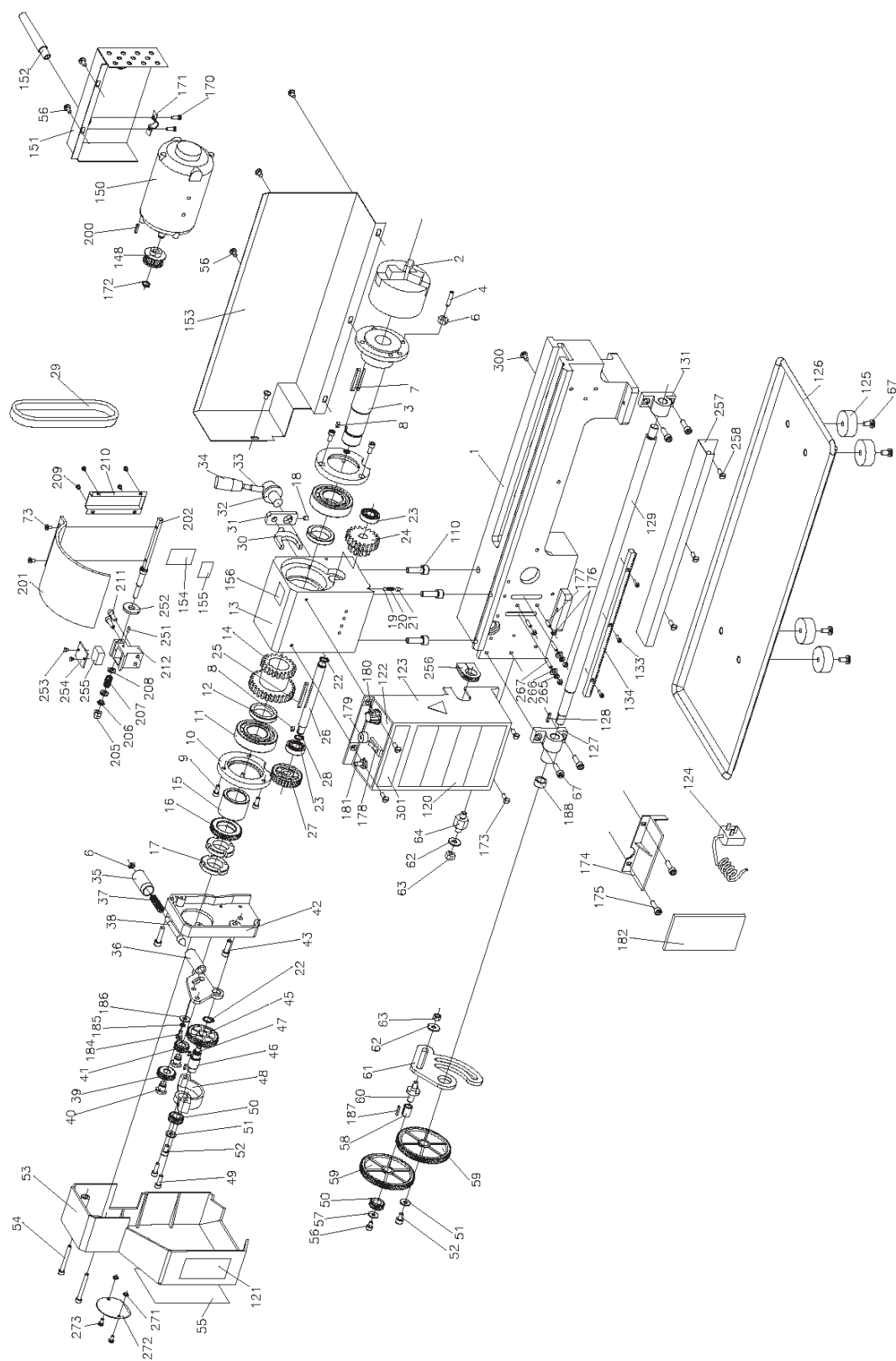
12.5.3 Acknowledgements

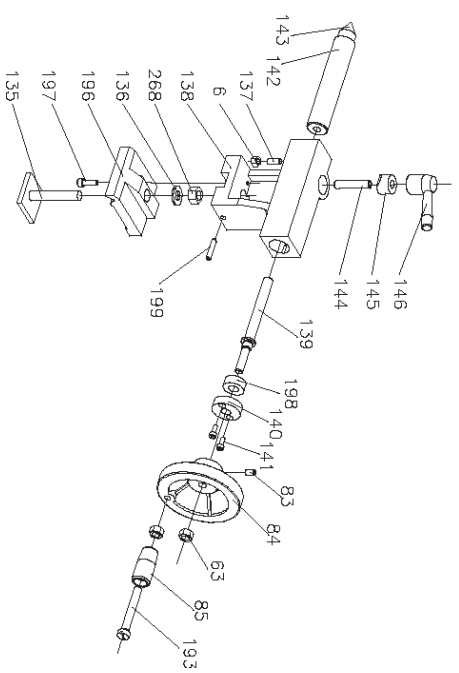
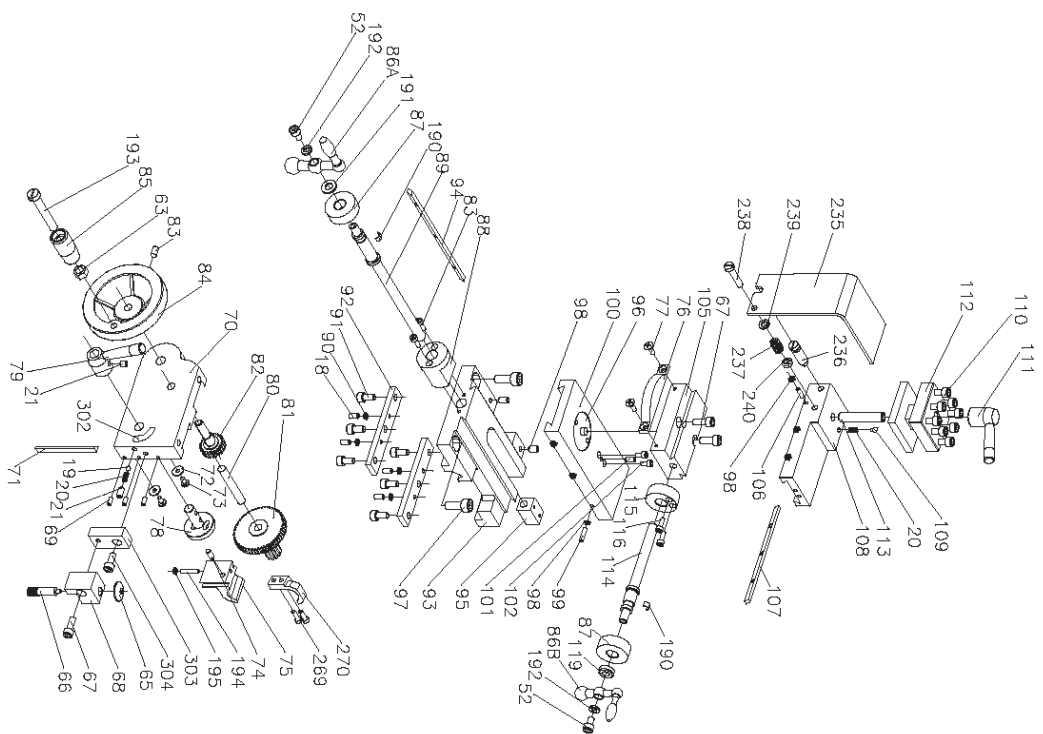
Thanks are due to LittleMachineShop.com for the material in this maintenance section.

13. Appendices

13.1 Appendix 1 – Exploded Parts Views

To follow





PN	Description	ID	PN	Description	ID
30791	Bed way	1	30842	Screw M5x45	54
30792	3 jaw chuck	2	30843	thread cutting chart	55
30793	Spindle	3	30844	Screw M5x8	56
30794	Screw M6x25	4	30845	Washer M4	57
30795	Nut M6	6	30846	bush w/key	58
30796	Key 5x40	7	30847	Gearwheel 80T	59
30797	Key 4x8	8	30848	Shaft	60
30798	Screw M5x12	9	30849	Support plate	61
30799	Cover	10	30850	Washer 8	62
30800	Ball bearing 80206	11	30851	Nut M8	63
30801	Spacer	12	30852	Shaft	64
30802	Headstock casting	13	30853	Dial label	65
30803	H/L gear 21T/29T	14	30854	Set screw	66
30804	Spacer	15	30855	Screw M6x16	67
30805	Spur gear 45T	16	30856	Dial indicator body	68
30806	Nut M27x1.5	17	30857	Set screw M4x10	69
30807	Set screw M5x8	18	30858	Apron	70
30808	Steel ball 5	19	30859	Gib strip	71
30809	Comperssion spring	20	30860	Washer	72
30810	Set screw M6x8	21	30861	Screw M4x8	73
30811	Retaining ring 12	22	30862	Shaft	74
30812	Ball bearing 6201Z	23	30863	Half nut base	75
30813	H/L gear 12T/20T	24	30864	Augle block	76
30814	Parallel key 4x45	25	30865	Screw M4x10	77
30815	H/L gear shaft	26	30866	Groove cam	78
30816	Pulley	27	30867	Handle	79
30817	Retaining ring 10	28	30868	Shaft	80
30818	Timing belt L136	29	30869	Feeding gear 11T/54T	81
30819	Shifting fork	30	30870	Feeding gear 24T	82
30820	Shifting arm	31	30871	Screw M6x10	83
30821	Shifting knob	32	30872	Wheel	84
30822	Shifting lever	33	30873	Knob	85
30823	Shifting grip	34	30874	Handle big	86A
30824	Handle	35	30875	Handle small	86B
30825	Handle mount	36	30876	Dial	87
30826	Spring	37	30877	Bracket	88
30827	Indicator	38	30878	Feeding screw	89
30828	Pinion 25T	39	30879	Nut M5	90
30829	Support screw	40	30880	Screw M6x12	91
30830	Pinion 20T	41	30881	Slide plate	92
30831	Fixed cover	42	30882	Saddle	93
30832	Screw M6x20	43	30883	Gib strip	94
30833	Gear 45T	45	30884	Feeding nut imperial	95
30834	Shaft	46	30885	Swivel disk	96
30835	Parallel key 3x8	47	30886	Screw M8x20	97
30836	Mount	48	30887	Nut M4	98
30837	Screw M5x18	49	30888	Screw M4x16	99
30838	Gearwheel 20T	50	30889	Cross slide	100
30839	Washer M6	51	30890	Screw M5x10	101
30840	Screw M6x8	52	30891	Screw M4 x 8	102
30841	Cover	53	30892	Compound rest(B)	105

PN	Description	ID	PN	Description	ID
30893	Screw M4x14	106	30944	Gearwheel 57T	163
30894	Gib strip	107	30945	Gearwheel 60T	164
30895	Compound rest(A)	108	30946	Gearwheel 65T	165
30896	Position pin	109	30947	External jaws(set)	166
30897	Screw M6x25	110	30948	3-jaw chuck key	167
30898	Clamping lever	111	30949	Screw M4x8	170
30899	Tool rest	112	30950	Clamp block	171
30900	Stud M10x65	113	30951	Check ring 8	172
30901	Cross feed screw	114	30952	Screw M5x10	173
30902	Bracket	115	30953	Protector	174
30903	Screw M4x12	116	30954	Screw M5x10	175
30904	Nut M18	119	30955	Nut M6	176
30905	Model lable	120	30956	Screw M6x25	177
30906	Dial indicator label	121	30957	Power switch	178
30907	Switch label	122	30958	Fuse box	179
30908	Control box	123	30959	Variable speed control knob	180
30909	Plug w/cord	124	30960	Forward/off/reverse switch	181
30910	Rubber foot	125	30961	P.C.board	182
30911	Chip tray	126	30962	Screw M5x10	184
30912	Bracket	127	30963	Spring washer 5	185
30913	Key M3x16	128	30964	Washer 5	186
30914	Lead screw	129	30965	Key 3*16	187
30915	Bracket	131	30966	Spacer	188
30916	Screw M3x10	133	30967	Spring	190
30917	Rack	134	30968	Washer 8	191
30918	Clamp plate	135	30969	Spring washer	192
30919	Washer M10	136	30970	Screw M8x55	193
30920	Screw M5x16	137	30971	Screw M4x38	194
30921	Tailstock casting	138	30972	Nut M4	195
30922	Tailstock screw	139	30973	Tailstock plate	196
30923	Bracket	140	30974	Screw M5x16	197
30924	Screw M4x10	141	30975	Flange	198
30925	Tailstock quill	142	30976	Screw M5x25	199
30926	Center	143	30977	Key 3x12	200
30927	Stud M8x40	144	30978	Chuck protect cover	201
30928	Clamp	145	30979	Hinge	202
30929	Handle	146	30980	Spring washer 6	205
30930	Pulley	148	30981	Big washer 6	206
30931	Motor	150	30982	Spring	207
30932	Cover	151	30983	Washer 6	208
30933	Cable gland	152	30984	Screw M3x4	209
30934	Rear splash guard	153	30985	Switch cover	210
30935	F/N/R ladel	154	30986	Screw M5x16	211
30936	High-low label	155	30987	Fixed cover	212
30937	top warning label	156	30988	Protective cover	235
30938	Gearwheel 30T	157	30989	Sloting screw	236
30939	Gearwheel 35T	158	30990	Compression spring	237
30940	Gearwheel 40T	159	30991	Sloting Screw M6x30	238
30941	Gearwheel 45T	160	30992	Small washer 6	239
30942	Gearwheel 50T	161	30993	Hexagon nut M6	240
30943	Gearwheel 55T	162			

PN	Description	ID
30994	Round pin	251
30995	Rotate plate	252
30996	Screw 2.9x4.5	253
30997	Cover	254
30998	Micro switch	255
30999	Dustproof sleeve	256
31000	Protective cover for leadscrew	257
31001	Screw M5x8	258
31002	Spring washer 6	265
31003	Big washer 6	266
31004	Screw M6x25	267
31005	Nut M10	268
31006	Screw M5x14	269
31007	Leadscrew support	270
31008	Nut M4	271
31009	Protective cover	272
31010	Screw M4x6	273
31011	Screw	300
31012	Label	301
31013	Label	302
31014	Plate	303
31015	Screw M6x12	304
31016	Oil can	310
31017	L hex wrench set S: 3,4,5,6	311
31018	Double end wrench 8—10	312
31019	Double end wrench 14-17	313
31020	Fuse	314
31021	Package box	315
31022	Foamed plastics box	316
31023	Instruction Manual	317
31024	Carbon brush set	318
31026	Hexagon Nut	not shown
31027	T-Nut	not shown
31028	Left Feet	not shown
31029	Right Feet	not shown
31030	Upper Holder	not shown
31031	Lower Holder	not shown
31032	Left Clamp	not shown
31033	Right Clamp	not shown
31034	Gasket	not shown
31035	Block	not shown

13.2 Appendix 2 – Advanced Manual Turning Operations

13.2.1 Threading

Much of the mechanism of your lathe is provided to allow you to cut threads. Your lathe can cut a broad range of thread pitches. In fact, with the standard change gears, you can cut many more thread pitches than those shown on the table on the lathe.

13.2.2 Change Gears

The series of gears that drive the lead screw are called change gears because you change them to turn different thread pitches.

There are 4 positions for the change gears, commonly called A, B, C and D.

A This is the top change gear position. When you received your lathe it had a 20 tooth metal gear in this position.

B Gear positions B and C are on the same shaft, between positions A and D. Position B is the inside gear on this shaft. When you received your lathe it had an 80 tooth plastic gear in this position.

C Gear positions B and C are on the same shaft; between positions A and C. Position C is the outside gear on this shaft. When you received your lathe it had a 20 tooth metal gear in this position.

D Position D is the end of the lead screw. When you received your lathe it had an 80 tooth plastic gear in this position.

Changing these gears varies the speed that the lead screw turns in relation to the speed that the spindle turns. This allows you to cut threads with different numbers of threads per inch. The following table shows the gears to use for various common threads per inch.

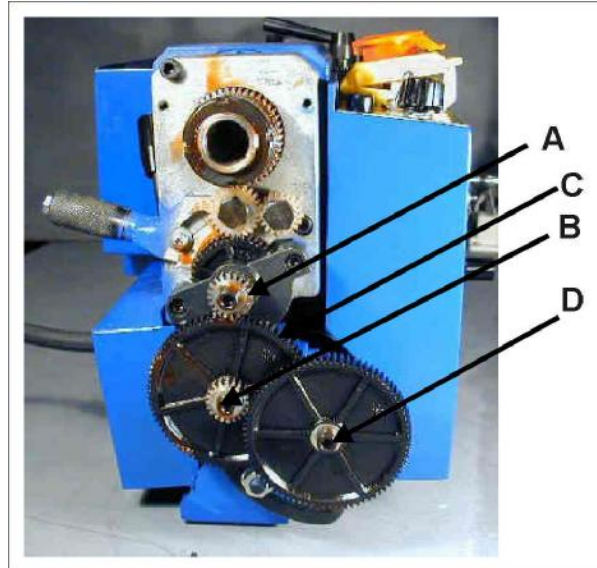


Figure 13.xx – Change gears - REPLACE

Threads per inch	A	B	C	D
12	40	Any	Any	30
13	40	65	60	30
14	40	Any	Any	35
16	40	Any	Any	40
18	40	Any	Any	45
19	40	50	60	57
20	40	Any	Any	50
22	40	Any	Any	55
24	40	Any	Any	60

Threads per inch	A	B	C	D
26	40	Any	Any	65
28	20	Any	Any	35
32	20	Any	Any	40
36	20	Any	Any	45
38	20	50	60	57
40	20	Any	Any	50
44	20	Any	Any	55
48	20	Any	Any	60
52	20	Any	Any	65

For normal turning, use the following gears.

A	B	C	D
20	80	20	80

The change gears are commonly tight on the shaft when new. You might need to use a screwdriver behind them to pry them off. Gear positions B and C are on a hollow shaft that comes off easily when the retaining socket head cap screw is removed. Then you can use an arbor press to remove the gears from the hollow shaft. Be careful that you do not lose the key.

Gear position D has a spacer behind the gear. When you only use three gears, put the spacer on outside the gear so the gear will align with the gear in position B.

In the change gear chart, many of the combinations have “Any” in column B. This means that you can use a gear with any number of teeth in position B. It is an idler and does not affect the overall gear ratio. Use a gear that makes it easy to properly engage the gears. For these combinations, you can use any gear for position C; this gear acts only as a spacer and does not engage the other gears.

13.2.3 To change the gears

1. Using a 4 mm hex wrench, remove the change gear cover.
2. Using 4 mm and 5 mm hex wrenches, remove all three retaining socket head cap screws from the ends of the shafts.
3. Use a 10 mm end wrench to loosen the nut that is on the back end of the shaft in position B-C. This allows the B-C shaft to move in the adjustment slot.
4. Use a 14 mm end wrench to loosen the nut on the arc below and behind the gear in position D. This allows the entire bracket on which the B-C shaft mounts to swing down.
5. Remove all the gears.
6. If you will be using three gears, remove the spacer behind the D position gear from the end of the lead screw. Be careful that you do not lose the key.
7. Replace the gears with the gears shown in the chart for the threads per inch that you want to cut.
8. If you will be using three gears, replace the spacer outside the D position gear on the end of the lead screw. Be sure to insert the key.
9. If you are using three gears, place any gear in position C to act as a spacer.
10. Replace the three retaining socket head cap screws from the ends of the shafts. Snug, but do not tighten, until the gear train is adjusted.
11. Move the B-C shaft until all the gears are properly engaged.
12. Use a 14 mm end wrench to tighten the nut on the arc below and behind the gear in position D.
13. Use a 10 mm end wrench to tighten the nut that is on the back end of the shaft in position B-C.
14. Using 4 mm and 5 mm hex wrenches, tighten the three retaining socket head cap screws on the ends of the shafts.
15. Using a 4 mm hex wrench, replace the change gear cover.

13.2.4 Threading Dial

When cutting screw threads on a lathe, you must make multiple cutting passes to cut the threads to full depth. The threading dial helps you align the cutting tool with the emerging thread before you start a cutting pass.

The gear on the bottom of the threading dial's shaft engages the lead screw. The dial turns when the half nuts are not engaged with the lead screw. When the half nuts are engaged, the carriage moves and the threading dial stops turning.

The gear on the threading dial has 16 teeth, and the lead screw has 16 threads per inch, so each revolution of the threading dial represents one inch of motion of the carriage. Each of the eight divisions on the dial represents 1/8" of motion.

If you are cutting 16 threads per inch, you can engage the half nuts when the threading dial is on any line. Since a line represents 1/8" of travel, it will always align with a thread groove.

If you are cutting 13 threads per inch, you must only engage the half nuts when the threading dial is at 1. Since 13 and 16 have no common factors but 1, you must only engage the half nuts at even inch increments of motion.



Figure 13.xx – Threading dial

The following table shows where you can engage the half nuts for various threads per inch.

Threads per inch	Dial divisions
12	1, 3, 5, 7
13	1
14	1, 5
16	Any
18	1, 5
19	1
20	1, 3, 5, 7
22	1, 5
24	Any
26	1, 5
28	1, 3, 5, 7

Threads per inch	Dial divisions
32	Any
36	1, 3, 5, 7
38	1, 5
40	Any
44	1, 3, 5, 7
48	Any
52	1, 3, 5, 7

13.2.5 Tool and Setup

For threading, the tool bit is ground to the profile of the thread. We recommend purchase of special threading inserts.

For most threads, this is a point with a 60° included angle. The front of the tool should have about 10° of relief. No back rake is used. The left side should have about 8° of relief, and the right side should have about 10° of relief. The tip of the tool should have a flat that is 1/8 of the thread pitch.

Set the compound rest at a 29.5° angle from a line perpendicular to the axis of the lathe. This allows you to advance the tool with the compound rest. At this angle the tool cuts only on the left side of the thread form. This helps prevent chatter that might result from cutting the entire V form of the thread at once. It is best to use an accurate protractor when setting the compound rest. The protractor on the lathe is not accurate enough. See figure 13.xx

Even though the compound rest is set at an angle to the work piece, the thread cutting tool must be set square to the work piece. A center gauge makes this setting possible. A center gauge has several V-shaped cutouts. They can be used to check the angle of the tool with respect to the work piece.



Figure 13.xx – Setting compound slide

To align the tool bit to the work:

1. Ensure that the point of the tool bit is set at the center height of the lathe.
2. Place the center gauge between the point of the tool bit and the work piece. Leave enough room so that the center gauge can be moved back and forth so you can check each side of the tool bit separately.
3. Align the tool bit to the sides of the V-shaped cutout in the side of the center gauge.
4. Secure the tool bit in position.
5. Advance the tool bit until the point just makes contact with the work piece.
6. Zero the cross slide dial. Hold the cross slide feed handle and rotate the graduated dial.



Figure 13.xx – Center Gauge

13.2.6 Threading Process

It takes several passes to cut a thread to full depth. You must follow the correct procedure during each pass to ensure the thread is cut correctly.

Use the power feed forward/neutral/reverse lever to engage the lead screw drive. The carriage should move from right to left (toward the head stock) to cut right-hand threads, or from left to right (away from the head stock) to cut left-hand threads.

For each pass in cutting threads:

1. Move the carriage to the beginning of the cut.
2. Advance the cross slide to the initial position. For the first pass, you are already there. For additional passes, advance it 2 complete turns to the 0 mark.
3. Advance the compound rest to move the tool bit into the work. For the first pass, this should be only 0.001". For additional passes, it should be 0.005 to 0.010".
4. Start the lathe. Run it at the lowest speed that develops sufficient torque to make the cut.
5. When the threading dial reaches an appropriate mark, engage the half nuts. Note that you must be right on the mark. The half nuts will also engage half way between each mark, but this will ruin your thread.
6. When the tool reaches the end of the thread, disengage the half nuts.
7. Back off the cross slide exactly 2 turns.

After you have made the first pass, which should leave just a spiral mark on the work piece, use a thread gage to check that you are cutting the correct number of threads per inch.

Use a nut or the matching part to tell when you are done cutting the thread.

13.2.7 Acknowledgements

Thanks are due to LittleMachineShop.com for the material in this appendix.

13.3 Revision history

Rev A1-2	2 July 2008	Index added. Various detailed clarifications
Rev A1-1	26 June 2008	First draft of manual

14. Index

Hint: Where there is a choice, most index entries are made using the name of a thing (e.g., tool offset) rather than an action (e.g., measuring) so you will get better results thinking about the part of the machine on which you want information. Thus looking for “Tool offset – measuring” will give better results than looking for “Measuring – tool offset.” For important information both entries will probably appear.

If you have difficulty because you tried to look something up and the index entry was missing, please take a moment to e-mail tech@tormach.com with a note of (a) the words you were looking up and (b) where in the manual you found the information you wanted – assuming you did!

A

Absolute distance mode	
G90	93
Absolute IJ mode	85
Absolute machine coordinates	
G53 - move in	88
Adjustments and maintenance	104
Alibre CAD	
with SprutCAM	48
Aligning lathe bed	15
Arc - center format	85
Arc - radius format	84
Arc at feed rate	
G02/G03 defined	84
Arc motion	
defined	74
Arcs look wrong in tolpath	58
Axis position DROs	52

B

Back gear	21
Binary operations	
defined	79
Block	
format of code	76
Block Delete	
action of	75
Block Delete operation	52

C

CAD/CAM	
ideas into a part	39
CAM	
ideas into a part	39
Cancel modal motion	
G80 explained	90
Canned cycle return level	
G98/G99	94
Canned cycles	90
in-between motion	91
preliminary motion	91

repeats by L word	90
retract defined by R word	91
sticky numbers	90
Carriage	19
used in manual turning	23
Center format arc	85
Center height	
adjusting for CNC	59
Change gears	
description	19
Chuck	20
CNC operation	
configuring parts for	24
Code definition syntax explained	82
Comments	
defined	80
Compound slide	
in manual turning	24
Computer aided manufacture	
ideas into a part	39
Constant velocity mode	
G64 - setting	88
purpose of explained	75
Control software	
installing	18
Controlled point	
defined	73
Coolant	
control of	74
M07 - mist on	96
M08 - flood on	96
M09 - all off	96
Coordinate systems	
reference definitions	76
Co-ordinated linear motion	
defined	73
Copyright statement	10
Cross slide	
description	19
Current position	
defined	74
Cutoff	
by Visicycle	42

Cutter radius compensation	
G40/G41/G42 defined	87
Cutting First Part	33

D

Diagnostics screen	60
Disclaimer of liability	10
Drilling canned cycle	
G81	91
Drilling with dwell canned cycle	
G82	92
DROs will not keep new value	45
Dwell	74
G04 - defined	85

E

Editing G-code	51
E-stop	
Integrated	15
Exact stop mode	
G61 - setting	88
purpose of explained	75
Execution of words	
order of	98
Expressions	
defined	79

F

F word -feed rate	97
Facing	
by Visicycle	41
Manual turning	36
Feed and speed override	
controlled by M48/M49	96
Feed rate	
defined	74
F word to set	97
inverse time - G93	94
units per minute - G94	94
units per rev - G95	94
Feed rate units per rev - G95	94
Feedhold	52
Filed subroutine call	
M98	96
First part	
making	27
Fixture coordinate select	
G54-G59 defined	88
Fixture coordinate systems - setting - G10	85

G

G00 - rapid linear motion	83
G01 - linear feed rate move	84
G02 - clockwise arc	84
G03 - counterclockwise arc	84
G04 - dwell	85
G10 - set coordinate systems	85
G17 - select XY plane	86
G18 - select XZ plane	86
G19 - select YZ plane	86
G20 - inch units - setting	86

G21 - millimetre units - setting	86
G28 - return to home	86
G28.1 - reference axes	86
G30 - return to home	86
G40 - cutter radius compensation - Off	87
G41 - cutter radius compensation - Left	87
G42 - cutter radius compensation - Right	87
G50 - clear axis scale factors	87
G51 - set axis scale factors	87
G52 offsets	88
G53 - move in absolute machine coordinates	88
G54 - select fixture 1	88
G55 - select fixture 2	88
G56 - select fixture 3	88
G57 - select fixture 4	88
G58 - select fixture 5	88
G59 - select any fixture	88
G61 - set exact stop mode	88
G64 - set constant velocity mode	88
G73	
pullback DRO	89
G73 - high speed peck drilling canned cycle	89
G80 - cancel modal motion	90
G81 - drilling canned cycle	91
G82 - drilling with dwell canned cycle	92
G83 - peck drilling canned cycle	92
G90 - absolute distance mode	93
G91 - incremental distance mode	93
G92 - workpiece offsets	
interaction with parameters	93
G92 offsets	93
G93 - feed rate inverse time	94
G94 - feed rate units per minute	94
G98 - canned cycle return level	
to old Z	94
G99 - canned cycle return level	
to R word	94
G-codes	
summary table	83

H

Headstock	
description	19
Headstock gearing	21
Home - return to G28/G30	86

I

IJ mode - "Absolute"	85
IJ mode - Increments	85
IK mode settings	58
Inch units	
G20 - setting	86
Incremental distance mode	
G91	93
Incremental IJ mode	85
Installation	
Aligning bed	15
CNC mode	14
Manual mode	14
Installing software	18
Installing tool post	17

Appendices

Integrated E-stop	15
Intellectual property statement	10
Intended use	
policy if use is out of scope	103
statement of	103
Introduction	
by making First Part	27
J	
Jerky motion with short lines	
Constant velocity mode to avoid	75
Jogging	53
L	
Lathe controls	
familiarisation	19
License statement	10
Line	
format of code	76
Line number	
format of	77
Linear axes	
defined	73
Linear feed rate move	
G01 defined	84
Loading G-code	51
Lubrication	107
M	
M00 - program stop	94
M01 - optional program stop	94
M02 - program end	95
M03 - spindle clockwise	95
M04 - spindle counterclockwise	95
M05 - stop spindle	96
M07 - mist coolant on	96
M08 - flood coolant on	96
M09 - all coolant off	96
M30 - program end	95
M48 - feed and speed override on	96
M49 - feed and speed override off	96
M98 - filed subroutine call	96
M98 - subroutine call	96
M99 - subroutine return	97
Machine coordinates	
G53 - move in	88
Machine modes	
defined	81
Macro M-codes	97
Macros	
overview on writing	97
Maintenance and adjustments	104
Manual Data Input	54
Manual turning	35
M-code	
macros	97
M-codes - built in	
summary table	94
MDI	54
Messages	
from part program, defined	80

Millimetre units	
G21 - setting	86
Milling and turning a part	
details	69
Milling and turning combined on a part	59
Mirroring parts	73
Modal groups	
defined	81
Modal motion, cancelling	
G80 explained	90
Modes	
machine - defined	81
MSG,	
string introduces an operator message	80
N	
Nomenclature	11
Number	
format of	77
O	
Offsets	
G52	88
G92	93
setting	58
Offsets display screen	57
Operators - binary	
defined	79
Operators - unary	
defined	79
Optional program stop	
M01	94
Optional Stop	
action of	75
Order of G-code items on line	80
Override	
for feed and speed - disabling	75
Override feed and speed	
controlled by M48/M49	96
Overview	7
P	
Packung list	13
Parameter	
setting value of	79
using value of	78
Parameters	
predefined	75
Part program	
repeating indefinitely - M47	96
repeating indefinitely - M99	97
Pause or Feedhold	52
Peck drilling canned cycle	
G83	92
Peck drilling canned cycle - high speed	
G73	89
Performance	
Accuracy	10
Cutting ability	9
Plane selection	
G17/G18/G19 defined	86

Power feed	
with manual turning.....	36
Preparation for use.....	13
Program	
error handling.....	98
Program end	
M02/M30.....	95
Program stop	
M00.....	94
Pullback DRO	
G73.....	89

Q

Quick Change tool post	
setting height.....	25

R

Radius format arc.....	84
Rapid motion	
G00 definrd.....	83
Reference - G28.1.....	86
Repeating part program indefinitely - M47.....	96
Repeating part program indefinitely - M99.....	97
Return level after canned cycle	
G98/G99.....	94
Running gear	
description.....	19
Running part-program.....	51

S

S word - spindle speed.....	97
Saddle.....	<i>See</i> Carriage
Safety	
Electrical.....	9
further reading.....	7
General Comments.....	7
Scale factor - on axis data - G50, G51.....	87
Scaling coordinates.....	73
Scaling parts.....	73
Selected plane	
defined.....	74
Self acting feed	
Manual - gearing for.....	22
Sensor	
spindle.....	16
Set fixture coordinate systems - G10.....	85
Settings screen.....	58
Single block operation.....	52
Software	
installing.....	18
Speed and feed override	
controlled by M48/M49.....	96
Speed control	
Spindle.....	21
Spindle	
M03 - clockwise.....	95
M04 - counterclockwise.....	95
M05 - stop.....	96
Spindle controls.....	54
Spindle indClamp	
spindle, indexing.....	22

Spindle Indexing	
spindle, clamp.....	22
Spindle indexing clamp.....	22
Spindle overload warning.....	54
Spindle sensor.....	16
Spindle speed	
S word to set.....	97
Spindle speed calculator.....	60
Spindle speed control.....	21
Subroutine call	
M98.....	96
repeating several times.....	96
Subroutine label	
format of.....	77
Subroutine return	
M99.....	97
Support	
from Tormach.....	103
Syntax - Code definition.....	82

T

T word - tool select.....	97
Tailstock	
clamping and adjusting.....	23
description.....	19
Taper turning	
by Visicycle.....	42
Manual.....	37
Threading	
by Visicycle.....	42
by Wizard.....	46
Manual - gearing for.....	22
manual, details.....	110
Tool center height	
detailed settin procedure.....	61
Tool change	
supplied M6 macros.....	98
Tool nose direction codes.....	101
Tool nose radius compensation	
introduction.....	99
Tool post	
4-way, manual.....	20
installation.....	17
Quick change, setting height.....	25
Tool select	
T word.....	97
Tool table	
editing.....	63
Toolpath display.....	53
Toolpost	
Options, manual.....	24
Trademarks.....	11
Turbo CAD/CAM.....	48
Turning	
manual.....	35
Turning and milling a part	
details.....	69
Turning and milling combined on a part.....	59
Turning manual	
with power feed.....	36
Typographical conventions.....	11

Appendices

U

Unary operators	
defined	79
Units	
inch, degree and millimetre	74
Unpacking	13

V

Visicycle	
details.....	55
Visicycles	
ideas into a part	39
using	39

W

Wizard	
for threading.....	46
Wizards	
example turning a ball	44
ideas into a part	39
Word	
format of	77
initial letters	77
Work holding	20