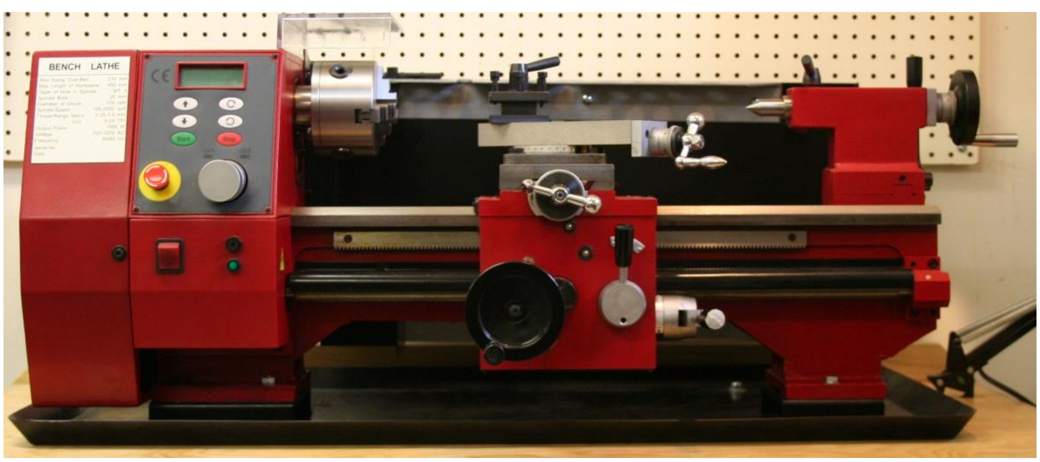
Supplement to SIEG SC4 Instruction Manual (Revised)

With Gear Change Charts, Recommended Cutting Speeds and Other Helpful Information

Robert Ackert

5/21/2014



Supplement to the instruction manual supplied with the SIEG SC4 bench lathe with information on controlling carriage movement, changing gears, gear charts, recommended cutting, drilling and reaming speeds and tooling, detailed instructions for cutting standard thread forms and mounting alternate chucks.

Supplement to SIEG SC4 Instruction Manual

Copyright Notice and Terms of Use

Copyright © 2009 and 2014 by Robert Ackert, Sault Ste. Marie, Ontario, Canada. All rights reserved.

This publication is protected by copyright. No part of this publication may be reproduced, stored in a retrieval system, or transmitted in any form or by any means, electronic, mechanical, photocopying, recording, or otherwise without the permission of the copyright holder, except as authorized below.

This publication may be reproduced in its entirety or in part for personal and educational purposes as long as any fees charged do not exceed the actual cost of reproduction, and as long as this copyright notice is included in any such reproduction. Any such reproduction does not imply the transference of any other form of copyright licence.

Disclaimer/Preface

This supplement to the Instruction Manual supplied with the SIEG SC4 Bench Lathe was prepared as part of the learning experience with a newly purchased SIEG SC4 Bench Lathe. The information and data was compiled by means of direct measurements, calculations and trial and error and, where possible, verified using published information. While a lot of care was taken to provide accurate information, the author cannot be held responsible for material contained herein which may be inaccurate or out of date in regard to any other lathe than his own. This supplement is provided to help other hobbyists and non-professionals to learn how to use and enjoy their SIEG SC4 lathes. No representation is made that any procedure or data provided here-in represents "best practice". The use of the information is entirely at the risk of the reader.

Shop practice and especially the use of power tools can be dangerous. The reader is cautioned to learn and practice all safety procedures relevant to the work at hand. Do not operate any power tool until after you have read and understand the safety procedures outlined in the instruction manual provided by the manufacture.

The author is neither an employee nor an agent of SIEG or any other company referenced in this document and he does not sell or represent any product discussed. Any opinions expressed are his own.

The SIEG CS4 is a robust lathe with a 1000 watt variable speed motor that provides plenty of power for a small lathe. The controls are simple and intuitive (once you have read SIEG's manual and this supplement). Dynamic motor braking and auto reversing are appreciated features, especially when cutting screw threads, as is the power cross feed. The machine lacks a quick change gear box. However, the two feed rates possible using the half nut and power transfer levers as outlined in section 5 largely make up for this lack and changing the gears in the SC4 for thread cutting is simple and straight forward. The SC4 also lacks a reversing lever for the leadscrew. As it is, you have to swap a gear and a spacer on the spindle drive to reverse the leadscrew rotation relative to the spindle, as shown in section 6.1. This is a small compromise and the gear change is quite easy and not frequently required.

Robert Ackert
October, 2012

Revision February 9, 2014

The main revision is to the gear change chart for metric threading Section 6.4 and to the text in Section 6. The previous version included some gear combinations that would not fit onto the lathe due to gears interfering with each other and other parts of the machine. A new "out of the box" modification has been added to Section 4 (replacing the cap screw holding the gear cover closed with a thumb screw). Some minor text revisions were also made for clarity and a few typing errors were corrected.

Robert Ackert
February, 2014

Revision May 21, 2014

This revision is not so much a revision as an addition in that the only change is the addition of a fifth recommended "out-of-the-box" modification on page 8.

Robert Ackert
May, 2014

Supplement to SIEG SC4 Instruction Manual

Contents

Supplement to SIEG SC4 Instruction Manual

By

Robert Ackert

1. Introduction

This document is a supplement to the instruction manual supplied with the SIEG SC4 lathe. It does not replace the manual provided by SIEG. Please read it thoroughly to learn the safety precautions and basic machine operating functions prior to using your lathe. However, the supplied manual lacks some important information which this document endeavors to supply. I recommend that you read both documents prior to using your lathe.

2. Cleaning Lathe Prior to First Use and Break-in Procedure

WD40 and paper towels work well to clean off the protective shipping grease. The 311 g spray can and one roll of paper towels is all you need. WD40 is safe to use following the instructions on the can and it leaves a protective lubricating film on metal surfaces. However, it does not replace machine oil as a lubricant when operating the lathe. After cleaning the external surfaces, follow the lubrication instructions given in section 3.

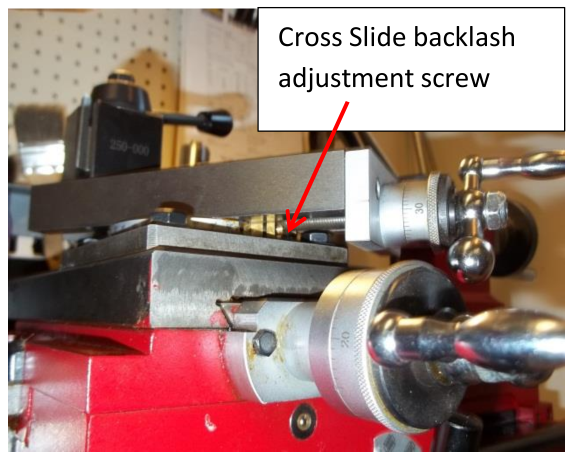
The break in procedure is very simple. Put a few drops of machine oil through the port in the rubber mat on top of the headstock, move the carriage well away from the chuck and then run the lathe for about 10 minutes while gradually varying the speed from 100 to 2000 rpm. Then unplug the machine and remove the headstock cover and thoroughly clean out the gritty run-in grease on the two gears and gear shift mechanism. Make sure all the grit is cleaned out paying special attention to the gear shift mechanism and the shaft on which the gears slide. Work the gears back and forth using the large silver-coloured knob on the control panel to make sure it moves freely (These gears engage/disengage the spindle and are used when the milling machine option is included). When clean, lubricate the gears and shaft with light grease. Spray motorcycle chain grease works well for this. Replace the headstock cover.

3. Lubrication

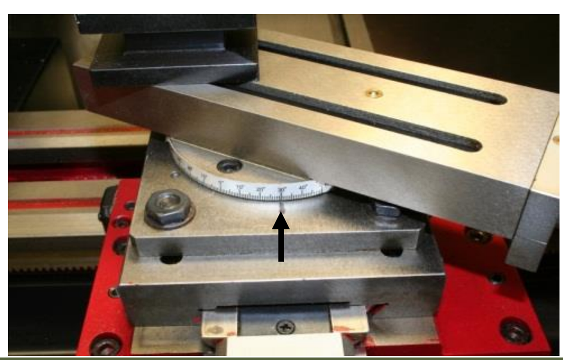
A pump-style oil can containing machine oil is an important part of the turner's tool kit. Locate all the oil ports on the machine, including the ones on the ends of the gear posts. Pump a small amount of oil into each port at the beginning of each turning session. Also lightly oil all exposed sliding surfaces. Lubricate the change gears regularly with a small amount of grease. Any modern gear grease will do but do not over do it because the excess grease will fling of and contaminate the drive belt. Motorcycle chain grease such as Permatex Chain Lub or NAPA Open Gear Lube can also be used to lubricate the gears. They comes in a spray can, lubricate both the gears and their bearings and overall are less messy that paste-type greases in the open gear case. Just be careful to keep the spray away from the drive belt. Chain lube can also be used to lubricate sliding surfaces and has the advantage that it does not hold dirt. Only a little bit is required.

4. Recommended Out-of-the-Box Modifications

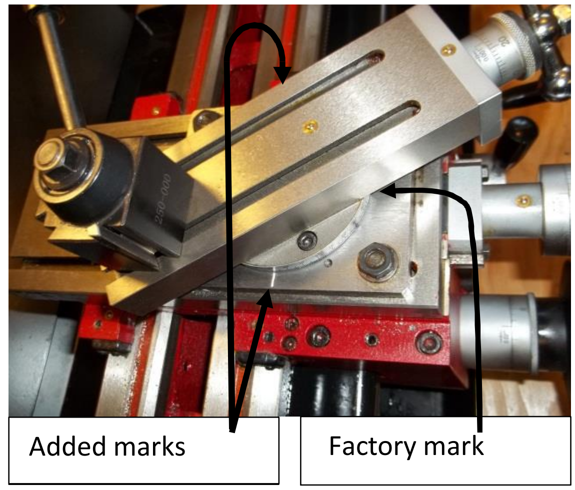
There are a few out-of-the-box modifications recommended for the lathe before it is put to into use. They are minor and each takes only a few minute to do. However, doing them right off the bat will save annoyance latter.



The first is to shorten the backlash adjustment screw in the brass feed nut for the compound slide. Having this adjustment is a nice feature but the screw may hit the nuts holding the compound assembly to the carriage when the angle of the compound is changed. Loosen the cap screws holding the compound and rotate it 360^{o}. If the screw does not hit any of the nuts, skip this modification and go on the next. If it does hit, take the cap screws out completely and remove the compound assembly by lifting it straight upwards. Then take out the two cap screws holding the end plate on and back out the main cross feed screw. Remove the small adjustment screw and shorten it just enough so that it will clear the mounting nuts when reassembled. It is best to cut off the back end and cut a new slot in it using a thin Dermel tool abrasive disk so that the lead end threads are not damaged. Reassembly is in the reverse order of the disassembly.



The second modification is even easier but it takes a bit more explanation as to why it is necessary. When the protractor scale attached to the base of the compound is lined up with the factory installed reference mark shown in the picture to the right, it measures the angle of the compound slide in relation to the longitudinal axis of the workpiece. This is the angle required when using the compound to cut a taper on the workpiece. However, when cutting threads the required angle is measured in relation to the transverse axis of the work piece. If the protractor scale covered a full 360^{o}, you would set the angle of the compound at the complement to the desired thread angle (NOTE: Two angles are complimentary to each other when they add up to 90^{o}). However, the scale only goes to 50^{o} in either direction. Since thread angles are generally 30^{o} the required complimentary angle (60^{o}) cannot be set on the SC4's protractor scale. The simple solution is to add two more reference marks at $+90^{o}$ and -90^{o} to the factory reference mark. To do this, place the 45^{o} point on the right side of the scale at the factory reference mark, secure the compound with the cap screws and then carefully stamp a mark at the left 45^{o} point on the scale. On the other side, the 50^{o} and 40^{o} points have to be used. A ¼" x ¼" lathe bit sharpened as shown can be used to make the required marks (round the back end and wear safety glasses.)



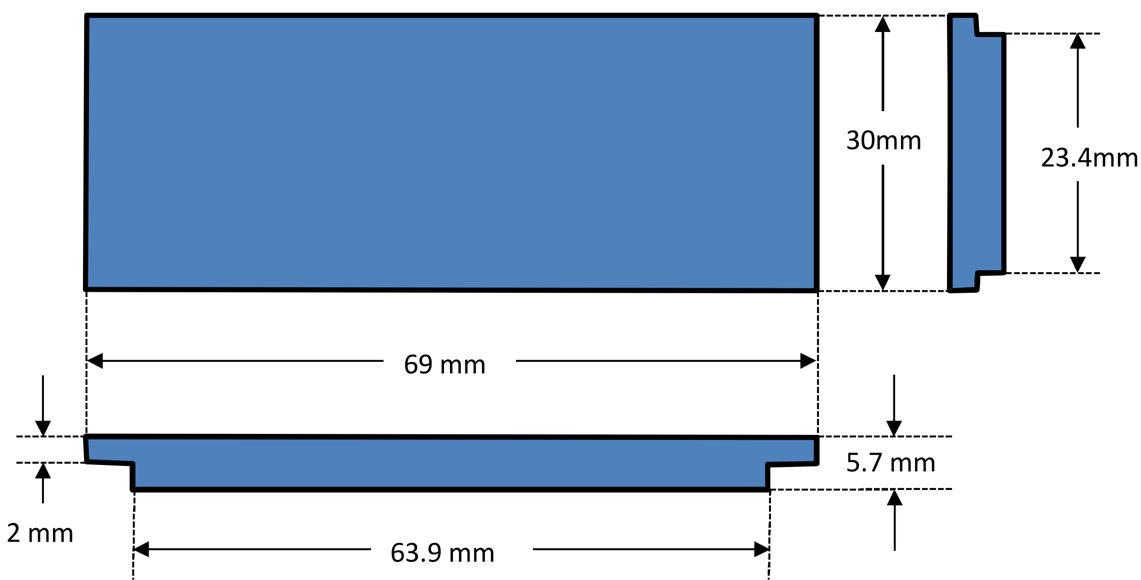
Note that if you are referencing this document for other lathe models you may find that there is not enough room on the carriage assembly to place the additional marks at the + and – 90^{o} positions. In these cases you can place the additional marks at the + and – 30^{o} positions and use them as your reference when threading.

The third modification is to replace the cap screw holding the gear cover closed with a thumb screw and washer and cutting the hole out as shown to the right so that you only have to undo the thumb screw a turn or two to open the cover.

The fourth modification is to convert the chuck so that it can be mounted with studs rather than with the supplied bolts, as outlined in Section 10.

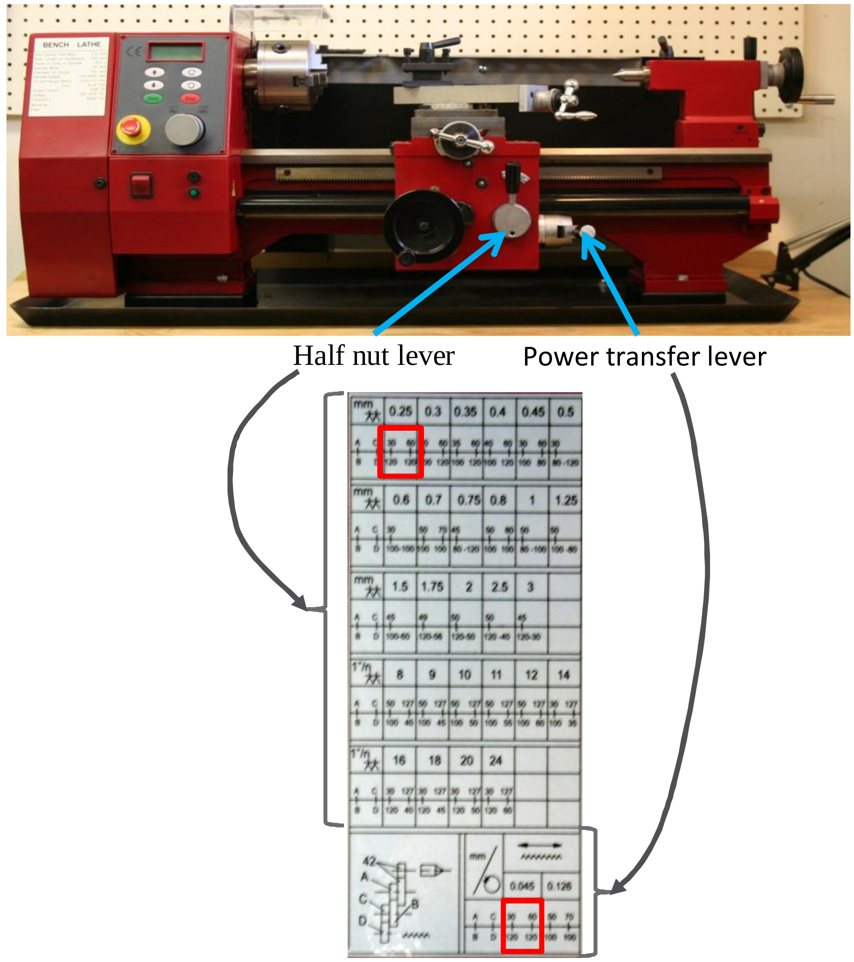
The fifth modification is not really a modification but an addition. I managed to knock the plastic lens protecting the LCD screen inward out of its mount on the inside of the control panel a couple of times. This lens is held in place only by friction and any pressure on it will dislodge it. To prevent this from happening, I made a protective Lucite lens dimensioned as shown below. Friction holds this lens in place in the LCD recess on the outside of the control panel so that any pressure on it tends to hold it in place more firmly. I left the original lens in place on the inside of the panel just because it was in place when I made the new one. I used the 5.7 mm thick Lucite because I just happened to have it and it worked out perfectly. I cut the Lucite to size using a saw and then cut the recessed edge with a very sharp carbide end mill on the milling machine. I used a moderate cutter speed to prevent melting the Lucite.

Protective Lucite Lens for SC4 LCD Screen



5. Control of Carriage movement

The carriage movement is controlled with two levers.



The ***half nut engagement lever*** is used when threading and can be used for rough cutting. It is engaged when in the down position and disengaged when in the up position.

The ***power transfer lever*** is engaged by pushing it towards the headstock (to the left) and then either up or down. In the up position, it engages the cross feed. In the down position, it engages the carriage travel at a slower pace than when the half nut lever is used (the carriage moves 0.18 times as fast). The power transfer lever is used for finishing cuts requiring a fine feed. The feed is disengaged when this lever is in the center position.

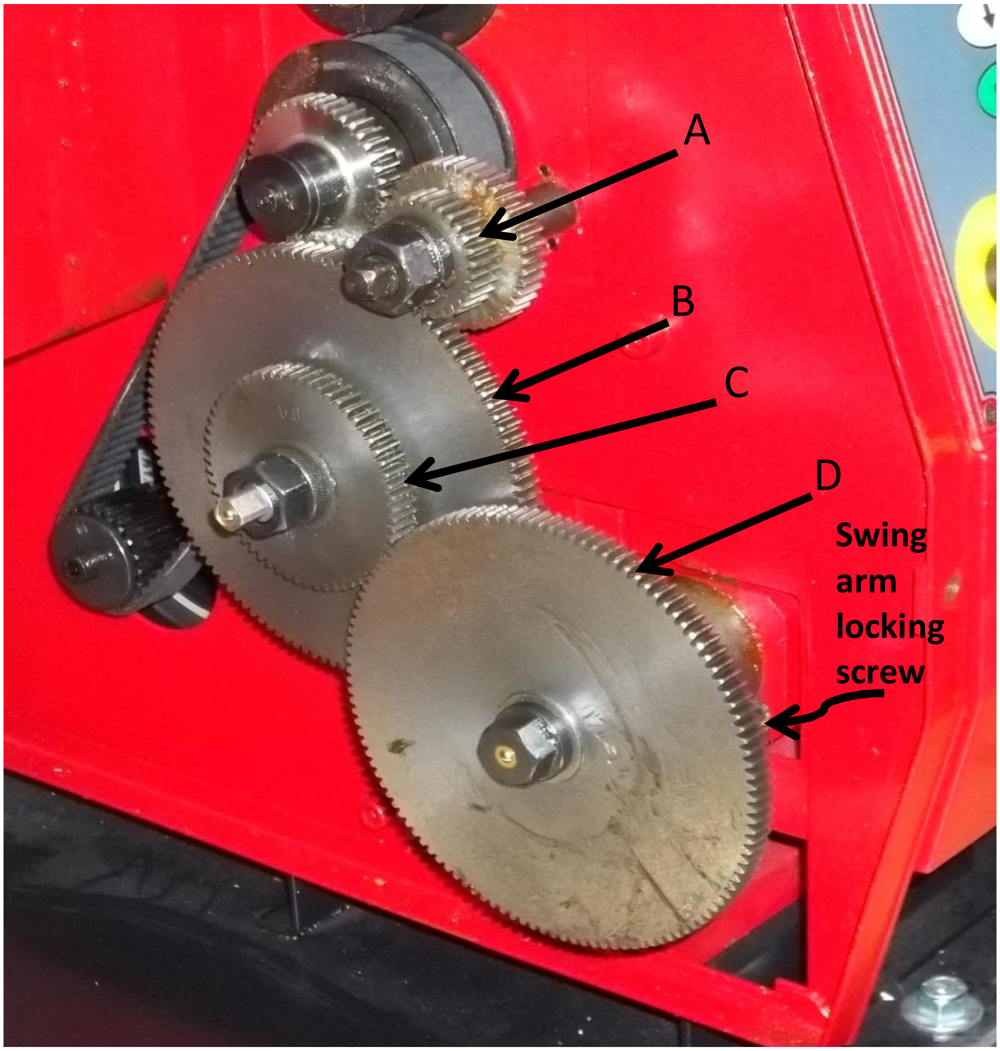
The half nut lever and the power transfer lever cannot both be engaged at the same time.

Example: When the A/B/C/D gear selection 30/120/60/120 is made as highlighted in the picture of the gear chart at the right, the half nut lever will give a feed rated of 0.25 mm (0.0098 in.) per revolution of the spindle while the power transfer lever will give a feed rate of 0.045 mm (0.00177 in.) per spindle revolution. This is a good gear combination to use for general turning because the 0.25 mm feed rate is good for rough turning and the 0.045 mm rate is good for finishing.

6. Gear Changing

The SC4 is a metric lathe with a leadscrew pitch of 2mm and metric module #1 change gears. The gear combinations shown in the metric chart in Section 6.4 give exact metric thread pitches except for the 5mm size. A 127 tooth gear is included in the change gear set so that Imperial (inch) thread pitches can be cut. All gear combinations shown in the inch chart in Section 6.3 give the exact number of threads per inch (TPI) shown except for 13 TPI which does not have the 127 tooth gear in it. It is included in the chart because ½ inch SAE coarse threaded bolts use 13 TPI. See the page 10 for more discussion regarding 13 TPI and the 5mm pitch.

The A/B/C/D gear positions referenced in the gear change charts are shown below.



The B/C gear set is mounted on a swing arm that pivots around the leadscrew behind gear D. The stud for this gear set screws into a T-nut on the back side of the arm. Proper meshing is achieved by sliding the B/C set up and down the arm to mesh C with D, locking them in place with the T-nut and then pivoting the arm to achieve proper mesh between B and A. The swing arm is locked in place by use of the socket screw on the arm behind D. Do not mesh the gears too tightly. A small amount of lash (play) is necessary to maintain proper lubrication between the gears. If the gears bind or if there is excessive drag when the chuck is rotated by hand, loosen them a bit. If they are too noisy when running under power, tighten them just a bit. Experience is the best teacher here.

My SC4 lathe was supplied with the following gear set in addition to the two 42-tooth gears permanently installed on the spindle and the position in back of A.

Tooth Count	Number In Set	Tooth Count	Number In Set	Tooth Count	Number In Set
30	1	50	2	80	1
35	1	55	1	100	2
40	1	56	1	120	2
45	1	60	1	127	1
49	1	70	1		

Some versions of the SIEG SC4 are supplied with a gear set that includes 87, 90 and 94 tooth gears but these are not required to cut any of the thread sizes shown in the inch chart in Section 6.3. They are required for few of the metric pitches shown in section 6.4. The 87, 90 and 94 tooth gears can be purchased as a set from Littlemachineshop.com for $29.95 plus tax and shipping. The link http://lmscnc.com/4214 will take you directly to it. They also have a free gear change calculator at http://littlemachineshop.com/Reference/change_gears.php that is a real time saver when determining gear combinations for thread pitches not included in the charts. This calculator gives only combinations that will physically fit the lathe. When using the calculator for the SC4, choose LMS' 8.5x16 HiTorque Bench Lathe which is a version of the SIEG SC4. The 87/90/94 gear set is worth the investment when cutting threads not included in the charts.

All the thread calculations are exact except for 13 TPI and the 5mm pitch size. The 49/Any Gear/50 gear combination for 13 TPI actually produces 12.959 TPI. Experience has shown this to be close enough for the common nut and bolt type applications for which 13 TPI is used. The 50/94/90/49 combination is included for those who have the 90 and 94 tooth gears. It gives 12.999 TPI which gives a better fit than required for 13 TPI work.

Gear combinations and feed rates other than those shown in the charts are possible. ***However, other gear combinations may not physically fit onto the lathe due to gears interfering with each other and other parts of the machine***. For example, a 60 tooth gear in position A has only about a 0.5 mm clearance to the stud on the end of the drive spindle. Larger gears will not fit at this location. The gear sizes are calculated using the equation:

$$Feed\ rate = \frac{A}{B}\ X\ \frac{C}{D}\ X\ (Pitch\ of\ leadscrew)$$

$$= \frac{A}{B}\ X \frac{C}{D}\ X\ 2\ mm$$

Where:

Feed rate is the carriage movement per one revolution of the spindle achieved using the half nut lever.

A, B, C and D are the tooth counts for gears at position A, B, C and D.

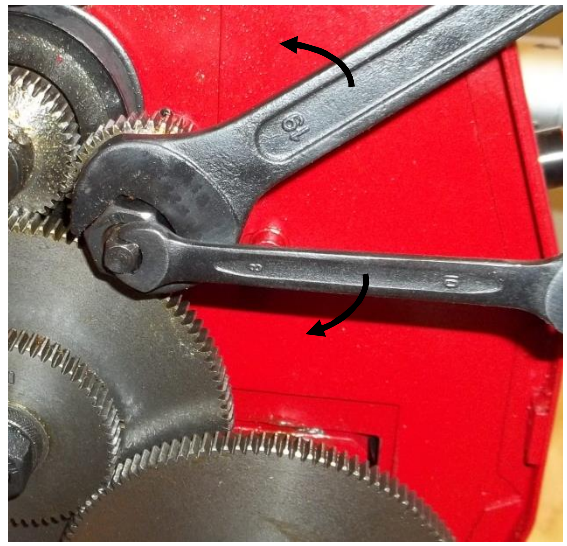
When B = C in the above calculation, use the “any gear” combination explained in Section 6.2.

The feed rate for the power transfer lever is calculated by multiplying the result of the above equation by 0.18.

The same equation is used to find the fed rate in inches except the result is divided by 25.4 since there are exactly 25.4 mm in one inch.

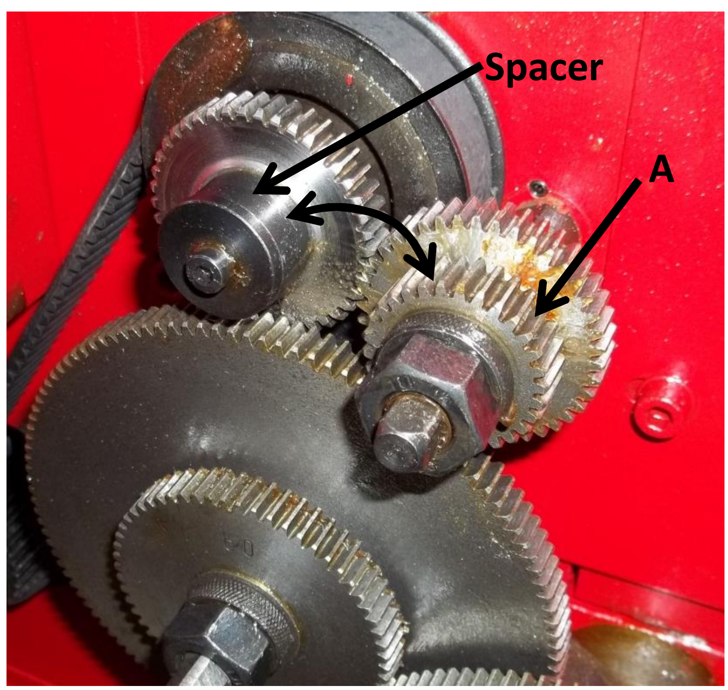
When calculating metric thread pitches, the thread pitch being cut equals the feed rate (in mm). When cutting inch threads, the threads per inch (TPI) is calculated as one divided by the feed rate (in inches).

Sometimes, two wrenches used as shown are required to remove a gear without withdrawing the gear stud on which it is mounted. When reinstalling gears, do not over tighten the retaining nuts. If the retaining nuts cannot be easily removed, remove the entire stud assembly; place the small square end securely in vise and use the larger wrench to remove the nut. Check to make sure that neither end of the bearing area of the stud has been peened over making it difficult to remove the gear. If there is any peening, mount the stud in the lathe chuck and remove the peening using a fine file. The author had to do this for the B/C stud on his lathe.



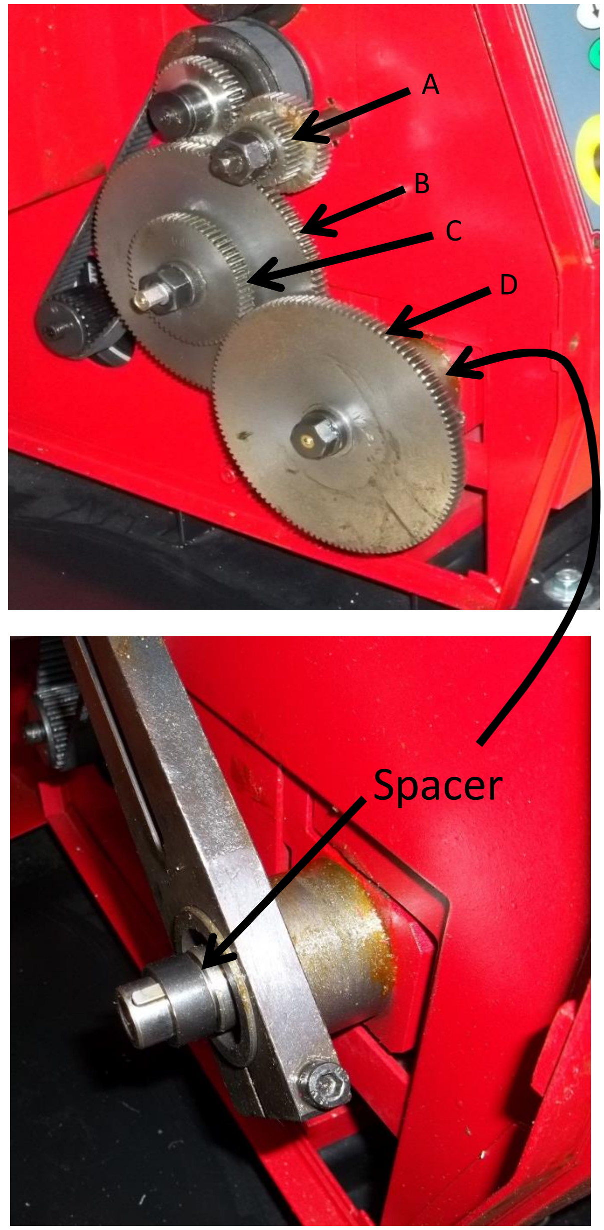
6.1 Reversing the Rotation of the Leadscrew

When cutting away from the headstock such as when left handed threads are being cut, the rotation of the leadscrew must be reversed relative to the spindle rotation. The SC4 does not have a leadscrew reverse mechanism, but it is easy to reverse the leadscrew by swapping gear A and the spacer on the spindle drive as shown to the right.



6.2 Using the Gear Change Charts

The charts in Sections 6.3 and 6.4 show the gears necessary for locations A, B, C and D required to achieve the indicated feed rates. The numbers in the A, B, C and D columns refer to the gear tooth count which is stamped on each gear. It is a good idea to copy these charts onto card stock and keep them at a location convenient to the lathe. Laminating them with clear vinyl film (available at the local dollar store) helps keep them clean.



When the charts in Sections 6.3 and 6.4 show "Any Gear" for positions B and C, remove the spacer from behind gear D, place gear D on the shaft, then position the spacer in the outboard position. For gear B, select any gear that will mesh with D without interfering with any other part of the machine. In this configuration, the gear ratio is set by gears A and D and B is just an idler gear connecting them. Gear C is a spacer to keep B in position and can be any gear that does not interfere with other parts of the machine. This situation arises when B = C in the gear equation on page 9.

If the spacer does not slide off easily, and it probably will not, place the blade of a small blade-type screwdriver into the joint between the spacer and the leadscrew (right where the arrow is pointing to in the lower picture on the right) and ***lightly*** tap the screwdriver with a small hammer. This should open up a gap so that the screw-driver can be used to wedge it off.

6.3 Gear Change Chart – Inch

The half nut columns give the threads per inch (TPI) and pitch for the indicated gear combinations when the half nuts are engaged and the PT Feed column shows the feed rate when the power transfer lever is used.

Half Nut		PT Feed	A	B	C	D
TPI	Pitch (in)					
7	0.1429	0.0257	50	100	127	35
8	0.1250	0.0225	50	100	127	40
9	0.1111	0.0200	50	100	127	45
10	0.1000	0.0180	50	100	127	50
11	0.0909	0.0164	50	100	127	55
12	0.0833	0.0150	50	100	127	60
13	0.0772	0.0139	49	Any Gear		50
13	0.0769	0.0138	50	94	90	49
14	0.0714	0.0129	30	120	127	35
15	0.0667	0.0120	40	120	127	50
16	0.0625	0.0113	30	120	127	40
18	0.0556	0.0100	30	120	127	45
20	0.0500	0.0090	30	120	127	50
24	0.0417	0.0075	30	120	127	60

Any Gear	See page 12 for explanation.
13	The 50/94/90/49 combination is the better solution for gear sets containing the 90 and 94 tooth gears. See text on page 10 for explanation.

6.4 Gear Change Chart – Metric

The half nut columns give the threads pitch in mm and inches for the indicated gear combinations when the half nuts are engaged and the PT Feed columns show the feed rate in mm and inches when the power transfer lever is used.

Half Nut		PT Feed		A	B	C	D
Pitch (mm)	Pitch (in)	(mm)	(in)				
0.15	0.0057	0.0263	0.0010	30	120	35	120
0.20	0.0079	0.0360	0.0014	30	120	40	100
0.25	0.0098	0.0450	0.0018	30	120	60	120
0.30	0.0118	0.0540	0.0021	30	100	60	120
0.35	0.0138	0.0630	0.0025	35	100	60	120
0.40	0.0157	0.0720	0.0028	40	100	60	120
0.45	0.0177	0.0810	0.0032	30	100	60	80
0.50	0.0197	0.0900	0.0035	30	Any Gear		120
0.60	0.0236	0.1080	0.0043	30	Any Gear		100
0.70	0.0276	0.1260	0.0050	50	100	70	100
0.75	0.0295	0.1350	0.0053	45	Any Gear		120
0.80	0.0315	0.1440	0.0057	50	100	80	100
1.00	0.0394	0.1800	0.0071	50	Any Gear		100
1.25	0.0492	0.2250	0.0089	50	Any Gear		80
1.50	0.0591	0.2700	0.0106	45	Any Gear		60
1.75	0.0689	0.3150	0.0124	35	Any Gear		40
2.00	0.0787	0.3600	0.0142	50	Any Gear		50
3.00	0.1181	0.5400	0.0213	45	90	120	40
4.00	0.1575	0.7200	0.0283	60	Any Gear		30
5	0.2000	0.9144	0.0360	60	100	127	30

60	There is very little clearance between the 60 tooth gear and the stud holding the 42 tooth gear on the spindle.
Any Gear	See text on page 12 for explanation.
5	There is a small error in this calculation. The actual value is 5.08mm.

7. Suggested Cutting, Drilling and Reaming Speeds

7.1 Working in Inches

Workpiece Material	Turning Speeds for HSS Cutting Tools (sfm*)		Drilling (sfm*)	Reaming (sfm*)
	Rough Cuts	Finishing Cuts		
Aluminum	300	400	240	200
Brass and Bronze	150	400	120	110
Copper	100	250	100	75
Cast Iron – soft	80	250	100	65
Cast Iron – Hard	50	150	80	60
Steel – Mild	90	250	100	55
Steel – Alloy (Hard)	40	150	30	15
Steel – Tool (not hardened)	50	150	45	30
Steel Stainless	60	180	50	30
For carbide cutting tools, double the cutting speeds given above when possible. **These values are a guide only.**			Start with a smaller drill and work up to a maximum of ½". Use of a boring tool is recommended for bores larger than ½".	
* sfm = surface feet per minute				

Calculating Spindle Speeds

$$\boldsymbol{Spindle\ Speed\ (rpm) = \frac{Cutting\ Speed\ (sfm)\ X\ 3.8}{Diameter\ of\ Workpiece(in)}}$$

Example:

Material of Workpiece = mild steel

Diameter of workpiece = ½ in.

$$\boldsymbol{Spindle\ speed = \frac{90\ X\ 3.8}{0.5} = 684\ rpm}$$

7.2 Working in Metric Units

Workpiece Material	Turning Speeds for HSS Cutting Tools (MPM*)		Drilling (MPM*)	Reaming (MPM*)
	Rough Cuts	Finishing Cuts		
Aluminum	92	122	73	61
Brass and Bronze	46	122	37	34
Copper	31	76	31	23
Cast Iron – soft	24	76	31	20
Cast Iron – Hard	15	46	24	18
Steel – Mild	27	76	31	17
Steel – Alloy (Hard)	12	46	9	5
Steel – Tool (not hardened)	15	46	14	9
Steel Stainless	18	55	15	9
For carbide cutting tools, double the cutting speeds given above when possible. **These values are a guide only.**			Start with a smaller drill and work up to a maximum of 12.5 mm. Use of a boring tool is recommended for bores larger than 12.5 mm.	
* MPM = meters per minute				

Calculating Spindle Speeds:

$$\boldsymbol{Spindle\ Speed\ (rpm) = \frac{Cutting\ Speed\ (MPM)\ X\ 318}{Diameter\ of\ Workpiece\ (mm)}}$$

Example:

Material of Workpiece = mild steel

Diameter of workpiece = 12 mm

$$\boldsymbol{Spindle\ speed = \frac{27\ X\ 318}{12} = 716\ rpm}$$

The turning speeds given above are suggested starting points. The experienced user may find that higher speeds are possible but with increased tool wear.

When drilling on a lathe, the holes are generally deeper than when using a drill press. It is essential that that the drill be extracted from the hole frequently to remove chips to prevent binding the drill. A cutting lubricant is recommended. The speeds given above give good results and exceeding them is not recommended.

Reaming and boring are the only methods that produce truly round holes. For boring, use the recommended turning speed calculated on the diameter of the bored hole. For reaming, the use of a good cutting lubricant is considered essential. Exceeding the indicated cutting speeds is strongly discouraged when using HSS reamers.

Appendix 2 presents the data from this section in tabular form for those who prefer to work from tables rather than formulas and equations.

7.3 HSS versus Carbide Tooling

While properly shaped and sharpened high speed steel (HSS) cutting bits generally give the finest surface finish, carbide tooling can also produce good results. The difficulty with carbide is that its best results are achieved at more aggressive feed rates, depths of cut and higher cutting speeds than are used with HSS.   This can be a problem with many smaller lathes. However, the SC4's robust construction and its relatively powerful 1000 watt motor facilitate the use of carbide tools very nicely. TCMT 21.51 carbide inserts in 3/8" x 3/8" tool holders work well for general purpose turning, especially for rough turning and semi finishing cuts.   They like a feed rate in the range of 0.10 to 0.4 mm (0.004 to 0.016") and a cutting depth preferably greater than 0.25 mm (0.010") at the cutting speeds given in the charts above. However, lower feeds and cutting depths can be used to produce a surface that can be quickly dressed with some emery or a fine file.

Using the gear combination shown in the example of page 4, the half nut feed produces a feed rate of 0.25 mm (0.0098") which is right in the middle of the range for the TCMT carbide inserts and works well for roughing cuts. Switching to the power transfer lever for a feed rate of 0.045 mm (0.00177") at a cutting depth of about 0.10 mm (0.004") produces a good surface using the same insert. At least this has been the author's experience when turning soft SAE 1018 cold rolled steel which is especially hard to get a good surface on.

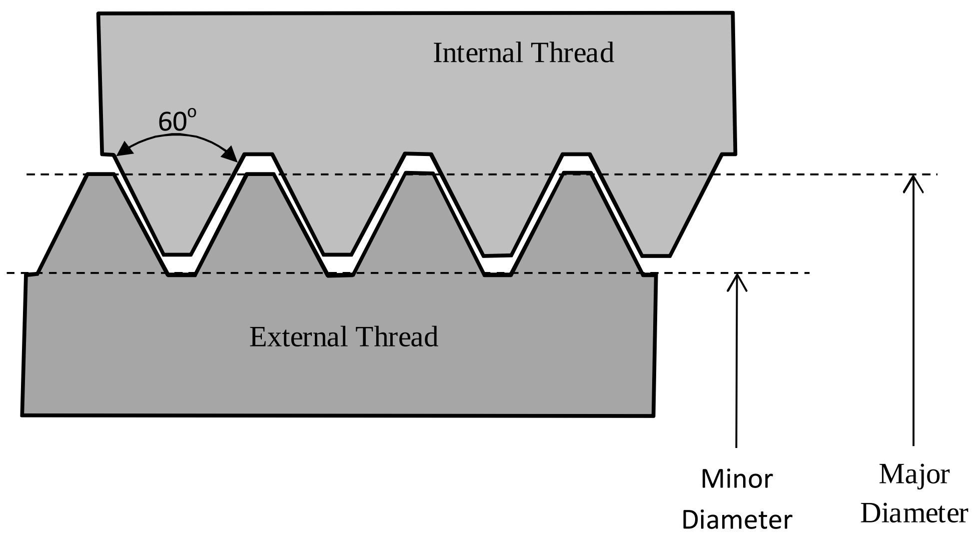
8. Threading on the SIEG SC4 Lathe

The SC4 lathe does not have a threading dial because it does not need one. Some lathes require a threading dial in order to reengage the leadscrew in synchronization with the spindle rotation after the carriage has been disconnected by opening the half nuts and manually moved back to start the second and subsequent cuts. In modern lathes like the SC4 in which the rotation of spindle and leadscrew can be stopped and reversed quickly, there is no need to disengage the leadscrew until all cuts are complete. This process is much easier and less prone to error than the use of a threading dial.

8.1 An Introduction to Thread Cutting

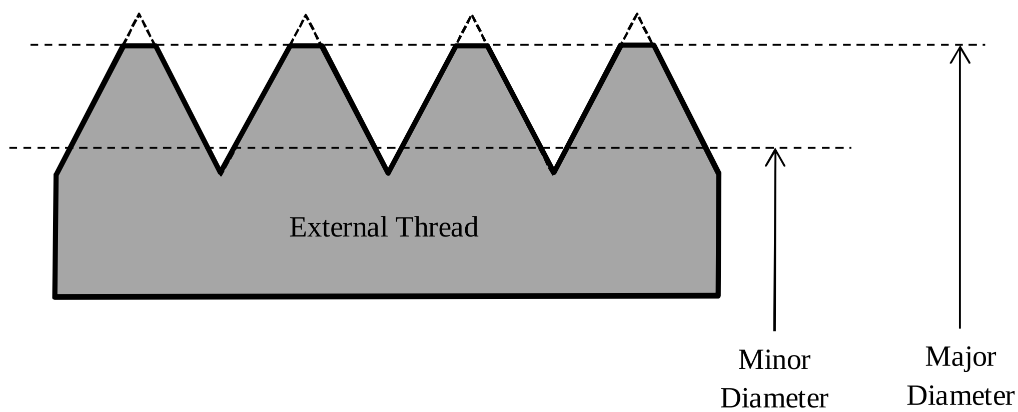
Justice cannot be given to the subject of threading in just a few paragraphs. Machinery's Handbook devotes well over 100 pages to the subject. However, there are only a few very basic things to know before starting to cut threads on a lathe.

Commercially formed threads have a form something like the cross-section shown below in which the thread crests and roots have been truncated. They are formed this way because a sharp crest has too little material present to make any practical addition to the fastener strength and the truncated root provides better fatigue resistance in highly stressed fasteners subjected to high cyclical loads.

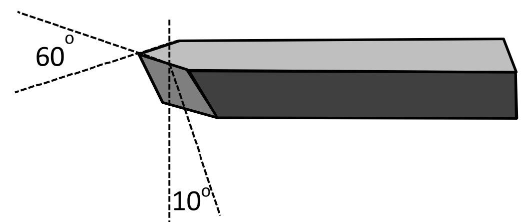


The angle of the thread is 60° for all modern standard metric and SAE screw threads. The major diameter is usually just a tiny bit below the nominal diameter of the fastener with a tolerance for being undersized. The major and minor dimensions and tolerances can be found on the internet using the search string "thread dimensions".
http://www1.mscdirect.com/PDF/FASTENERS/ThreadsAndMaterials.pdf is a good source for both metric and inch fasteners. The form of the truncation may be flat with rounded corners or the crest and root may be a complete radius. In theory the exact shape changes for every different thread pitch. Therefore a different forming tool is required for each thread type and pitch in order to produce a theoretically perfect thread. While this is practical for commercial producers it is impractical for the hobbyist and non-professional and even for many professional turners making one-offs or a very limited run of parts.

Fortunately compromises to theory can produce very acceptable threaded parts for most applications. Standards allow for +/- tolerances on all measurements. If manufacturers keep within these tolerances their parts are compatible with those of other manufacturers and the degree of thread engagement will generally be in the range of 55 to 75%. With 55% thread engagement, the holding power of a threaded hole will exceed the strength of the fastener when the length of the threaded portion of the hole is $^2/_3$ of the diameter of the fastener. In other words, a nut with a thickness equal to $^2/_3$ the diameter of the bolt will have a holding power equal to or greater than the strength of the bolt when only 55% of the available thread height is engaged between the two parts (assuming both parts are made from similar material). We can take advantage of these tolerances to cut acceptable threads on a lathe over a wide range of pitches using one single point cutting tool. External threads cut with a single point tool bit on a lathe look like this.



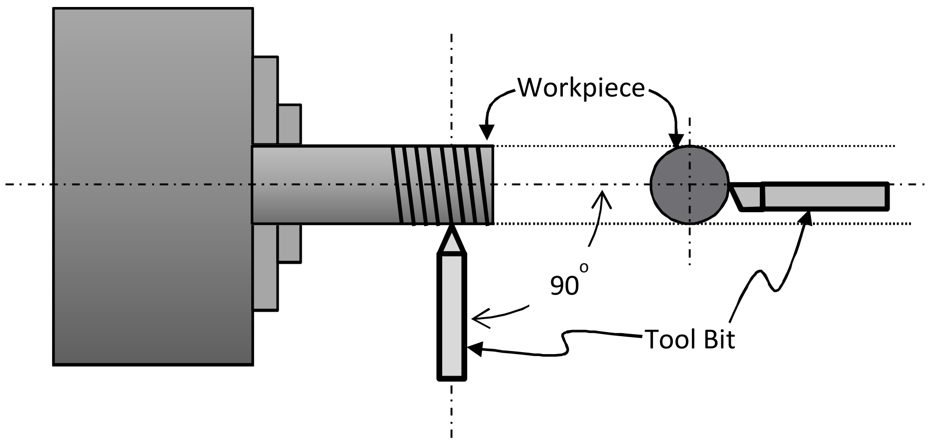
The crest should be left slightly flat. The root is sharp or it can be formed as a radius by putting a radius on the tool bit. In the majority of cases the tip of the tool bit needs only be slightly rounded with a diamond hone. The error introduced by producing a true minimum diameter a bit smaller than the listed minor diameter is inconsequential in most cases. However, in critical highly stressed applications the thread cutting tool should be given a more generous radius. This is critical when the fastener is to be heat treated after machining.



There are many types and shapes for thread cutting tool bits, but the most practical and by far the most commonly used in small lathes is a square tool bit ground as shown to the right. They can be purchased pre-formed in HSS or carbide. However, most turners grind their own using a square HSS tool bit blank and a small bench grinder. A small container of water is kept close at hand to cool the bit when it becomes too hot to hand hold. A measuring gauge such as either of the two shown here helps to get the 60° angle just right. The 10° relief angle ground on both sides of the bit is not critical at the slow turning speeds used for thread cutting on a manual lathe but it should be high enough to

keep the face of the tool from rubbing on either side of the spiral thread groove being produced.

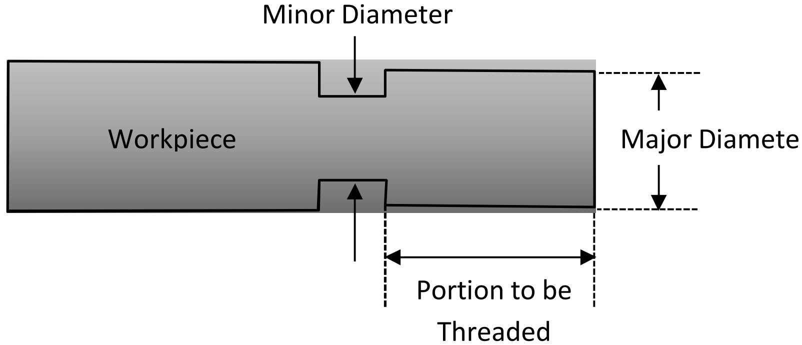
In use, the cutting tool is set a 90^{o} to the long axis of the workpiece and level with the centerline, as shown below. Either of the gauges shown above can be used to help achieve the 90^{o} approach to the workpiece.

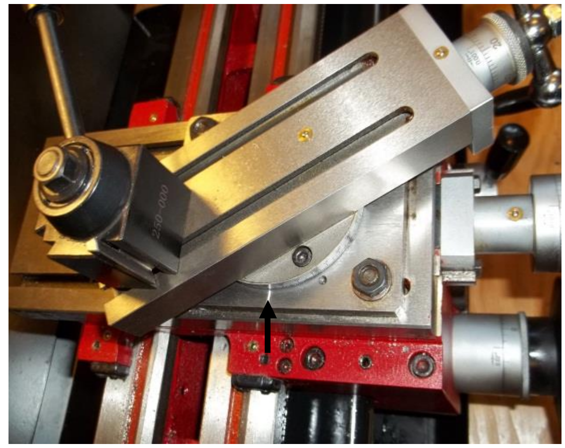


8.2 Cutting External Right Hand Threads on the SC4

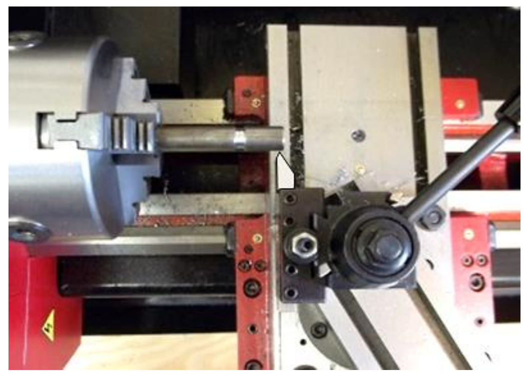
There are different methods for cutting threads. The following steps are considered good practice. If you are a beginner at thread cutting, these steps will give you good results. If you have used another approach on other lathes and you are happy with it go ahead and use it for the SC4 modifying it only in respect to leaving the half nuts engaged and using the power feed to reverse the carriage between cuts. The following steps are for turning right hand threads. Left hand threads are covered later.

1. Mount the workpiece in the chuck with as short of an extension as practical. Excessive extension out of the chuck will cause the workpiece to deflect. Tighten the chuck firmly since any slippage will disrupt the synchronization between the workpiece and the leadscrew.
2. Turn the portion of the workpiece to be threaded down to the major diameter making sure not to go below the listed minimum diameter for the fastener size you are aiming for.
3. Turn a short section of the workpiece just to the left of the portion to be threaded down to the minor diameter.

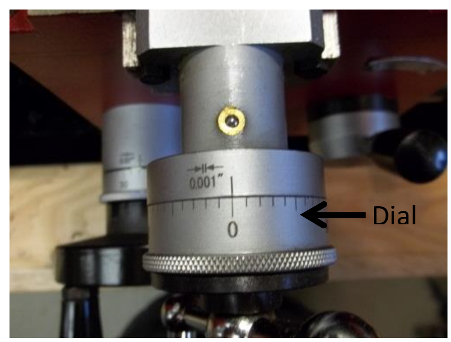




4. Change the gears to the set required to achieve the desired thread pitch.
5. Set the compound at the 29 ½° position the using the left reference mark as shown in the picture to the right. See Section 4 which shows how to add this reference mark if you have not already done so.
6. Set the tool bit on center and 90° to the workpiece. Make sure that it is properly secured in place.
7. Set the tool to just touch the workpiece using the combined motion of cross slide and compound slide. Make sure neither is near the end of their travel.



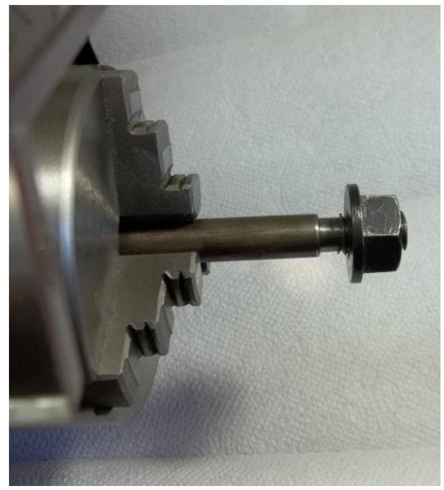
8. Set the dials on both the cross slide and the compound slide to their 0 marks (hold the crank handle and rotate the knurled dial collar to the 0 position).
9. Move the carriage to the right far enough to clear the end of the workpiece by 6mm (1/4") or more.
10. Move the **compound slide** in by 0.002 to 0.005".



11. Start the lathe in forward and keep the speed at 100 rpm.
12. While keeping the left thumb on the red stop button, engage the half nut lever.
13. When the tool bit gets to the start of the minor diameter section, immediately push the stop button. This will stop the spindle rotation almost instantly. **Do not touch the half nut lever**. The half nuts must remain engaged until the threading operation is complete.
14. Withdraw the **cross slide** enough so that the tool bit clears the crest of the threads being cut by at least one full revolution of the crank.
15. Push the **reverse** button and then restart the lathe. This will cause the carriage to move to the right. Let it move until the tool bit clears the end of the workpiece by about 6 mm or more.
16. Move the **cross slide** back in to its original 0 setting on the dial.
17. Move the compound slide in by another 0.002 to 0.005".
18. Push the forward button and then restart the lathe.
19. Repeat steps 13 to 18 until the tip of the tool bit reaches the minor diameter. With each repetition decrease the amount the compound slide is moved inwards.
20. When the tip of the tool bit reaches the minor diameter, withdraw the cutting bit using the **cross slide** and use the reverse drive to move the tool bit well away from the workpiece, then switch to forward drive and run a fine file over the thread crests. The leadscrew is still engaged so the carriage will move while you are doing this but you will have time for the file and then

stop the lathe before the tool bit gets back to the workpiece. The threads should still be a bit oversized but run a trial fit with a nut or, if you have one, a go-no-go gauge (alternately, a thread micrometer or a standard micrometer and a special set of measuring wires can be used). Most turners use a nut.

21. Continue to repeat steps 13 to 18 until a proper fit is achieved. At this point, the tool bit is probably cutting into the minor diameter section a little bit. You can see why if you refer back to the top drawing on page 19.



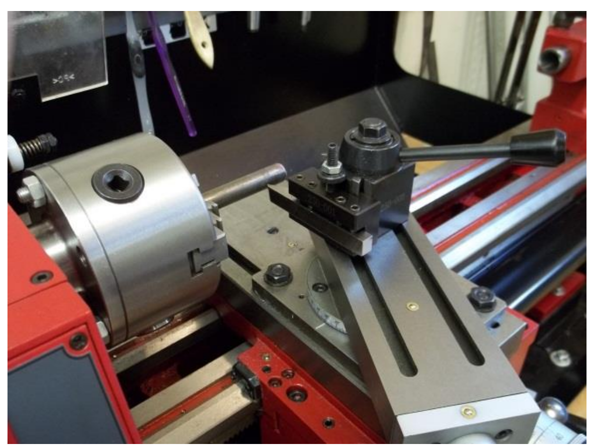
22. Disengage the half nuts and move the carriage out of the way. Run the lathe up to a moderate speed and run a fine file over the threads, then use a stiff wire brush to clean up the threads.
23. You are done.

The description above seems like a lot of work with many individual steps, but it comes quickly with practice and practice with sacrificial material is recommended before committing to serious work. It is important to stay focused and not allow any distractions.

8.3 Cutting External Left Hand Threads on the SC4

Cutting left hand threads is essential the same as right hand threads with following differences.

1. The compound slide is rotated to the 29½° position as measured at the **left side** reference mark added in Section 3, as shown here.

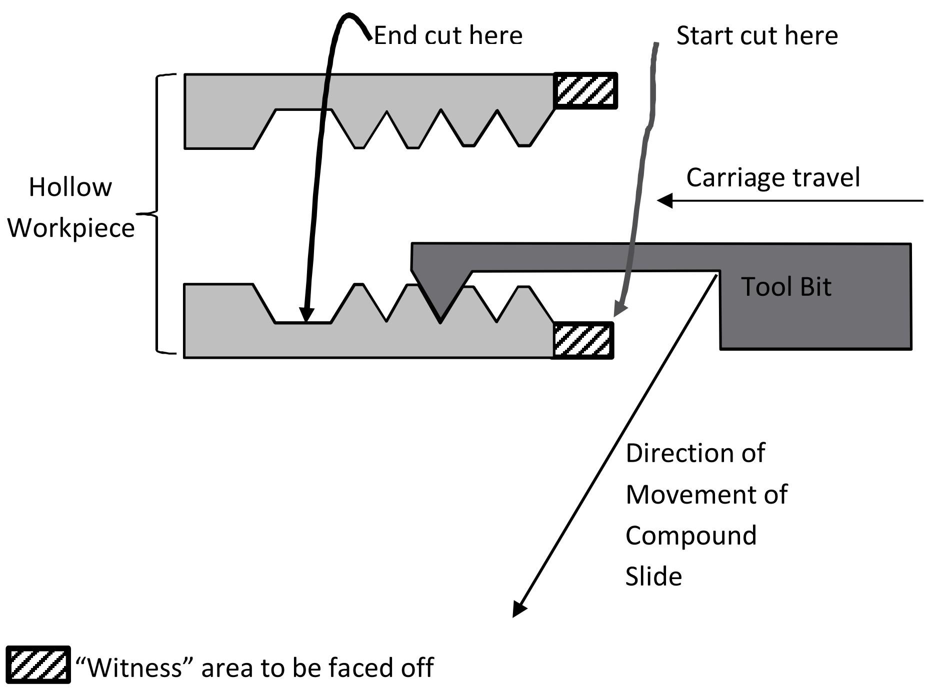


2. The leadscrew rotation is reversed (see Section 6.1).
3. The cut is started with the lathe in forward but it is started at the left end of the portion to be threaded. Since the rotation of the leadscrew has been reversed, the tool bit will travel to the right, away from the chuck.

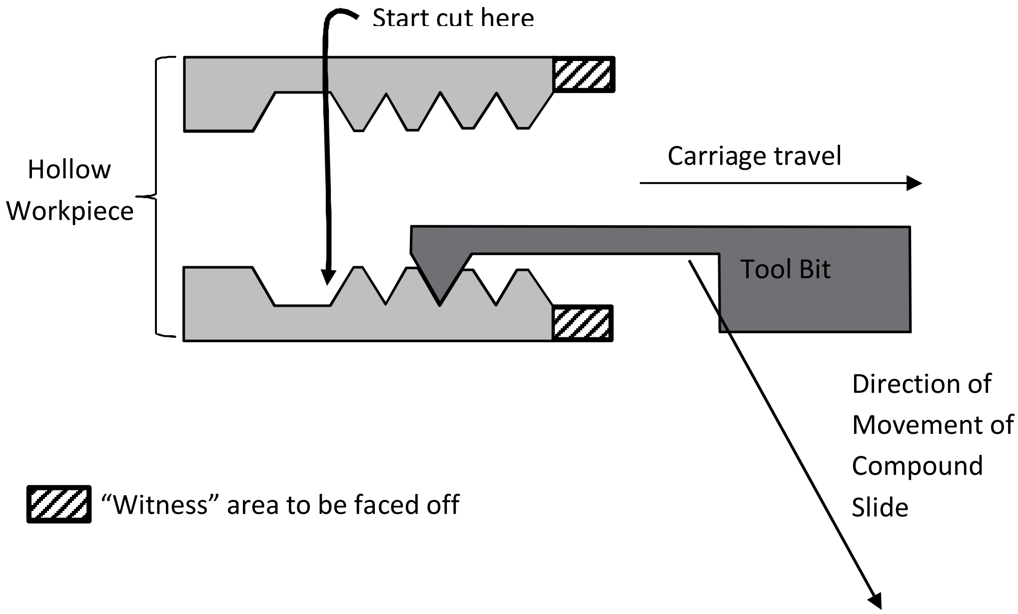
8.4 Cutting Internal Threads on the SC4

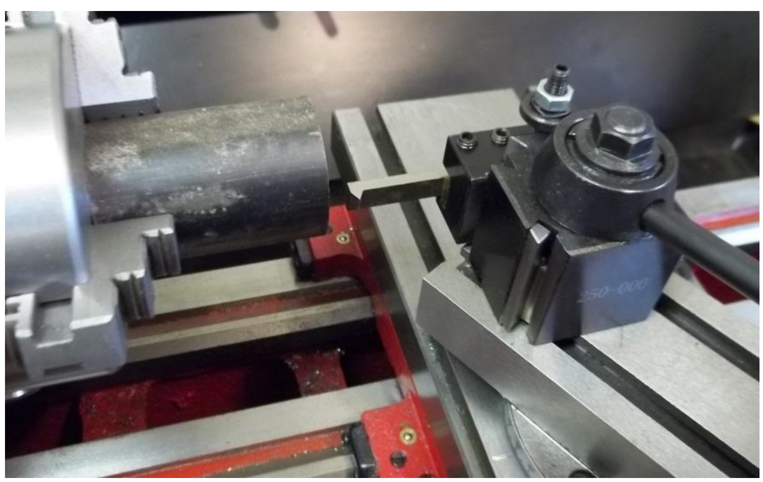
Cutting internal threads is essentially the same as external threads except that a different tool bit is used and the compound slide is rotated to the opposite direction as for the same handed external thread. The section to be threaded is drilled or bored to the minor diameter and a section at the internal end is bored to the major diameter if the threads are not going to extend right through the workpiece. When possible, it is also helpful to bore a short section at the external end out to the major diameter to act as "witness" area used to judge the progress of the threading operation. This area can be faced off when threading is complete.

Cutting Internal Right Hand Threads



Cutting Internal Left Hand Threads





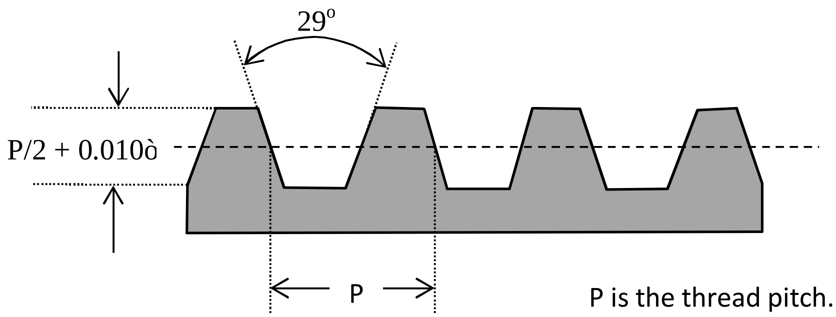
8.5 Cutting Obsolete Threads Forms

Prior to the 60^{o} thread form becoming the standard for all modern metric and inch screw threads, other thread forms were used. The most likely to be encountered is the British Standard Whitworth (BSW) 55^{o} thread form used in older British equipment. Cutting these obsolete thread forms is the same as cutting modern standard threads but making the following adjustments:

1. Grind the tool bit to conform to the thread form.
2. Set the compound angle at ½ the angle of the thread form minus ½^{o}.

8.6 Cutting Acme Threads

Acme threads have more of a trapezoid shape than standard screw threads, as illustrated below.

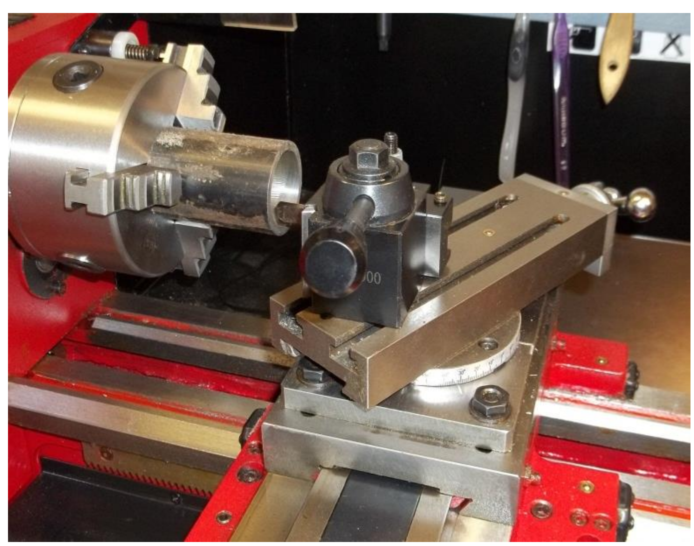


The metric trapezoid thread is very similar to the Acme thread except the angle is 30°. These thread forms are commonly used for linear actuators such as the leadscrew on metal lathes and in power lifting and holding applications such as screw jacks and vise screws. While cutting the trapezoidal thread forms is possible by making the adjustments given in Section 8.5, threading long screws on a lathe is a lot harder than making a good short one because of the deflection of the bar being cut. This is function of the material being cut and geometry of the part and is not a reflection on the lathe. When long screws with trapezoidal threads are required, purchasing them rather than making your own is recommended. An internet search will find suppliers for Acme threaded rod and their metric counterparts. If you application involves linear measurement such as the X-Y screws for a milling machine or the leadscrew for a lathe, purchase precision threaded rod. Make sure whether you need right or left hand thread before purchasing because some machinery components use left hand threads.

There are other considerations to cutting Acme threads but since a fellow that goes by the moniker of "tubalcain" on youtube.com has two good videos covering the subject they are not covered here. To find the videos, enter the search string "tubalcain machine shop tips acme" into a Google search.

8.7 Other Threading Setup Considerations and Situations

When threading, it is best to keep the workpiece as short as possible out of the chuck without running into a conflict between the cross slide, compound or your knuckles and the chuck. In this respect, the SC4 has a very good layout and the very long compound slide helps keep knuckles safely away from the chuck. However, situations may occur where reversing the compound and operating it from the rear as shown is required to safely cut internal threads, especially in very short workpieces.

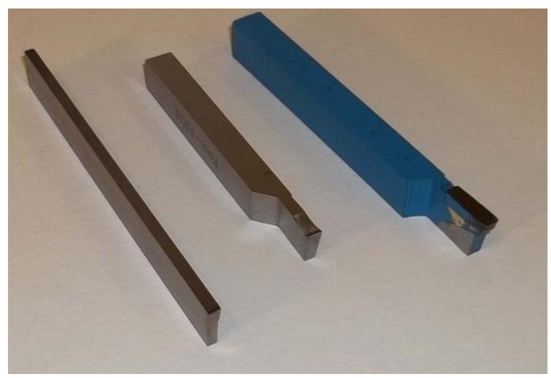


External threads on workpieces that cannot be held short in the chuck require threading between the chuck and a center in the tailstock. In these cases, allow enough sacrificial stock to begin and end the cut. About ¼” (6mm) is required to make sure all lash is out of the leadscrew and gear chain before the cutter enters the good portion of workpiece.

9. Parting Off on the SIEG SC4 Lathe

Parting off on a lathe refers to cutting a piece off by turning a groove all the way through the workpiece using a special tool bit, variously referred to by such names as a parting blade, a cut-off blade or a parting bit. Excessive chatter is a real problem when parting off on many small lathes and even on some not-so-small lathes. Many turners become so frustrated with it that they give up and use a hacksaw. If you ever do this with the workpiece still in the chuck, please put a piece of wood over the lathe ways to protect them from a nasty cut from a slip with the saw.

Parting tools come in various designs. “Tubalcain” has a series of three very good videos on youtube.com that describes parting off and the tools used. You can find a listing of his videos using the search string “tubalcain Machine Shop Tips” on Google, and then scroll down to #35, #36 and #37. The author’s own preference is to use P-type cut-off blade which has a T-shaped cross section with a slightly concave top surface.

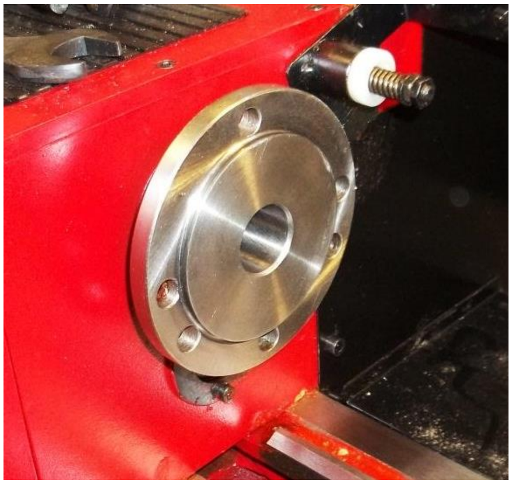


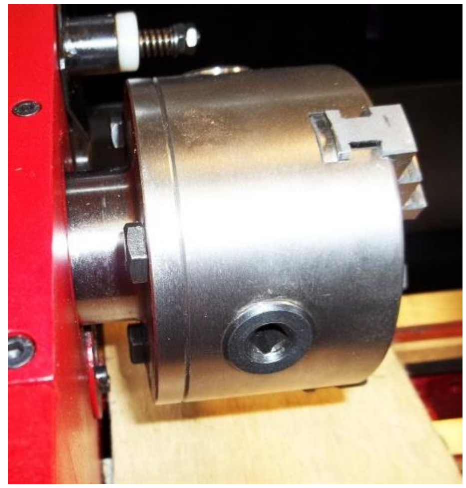
For testing purposes, the author tried three blades – a 3 mm x 12 mm T-shaped blade, a 4 mm wide HSS blade ground with a back rake and a 4.6 mm wide brazed carbide blade. All three blades were set straight on center and oriented at 90^{o} to the workpiece which was ½” cold rolled bar stock. He set the lathe at 550 rpm, and hand fed the cross slide while applying cutting oil. At first there was a bit of chatter which disappeared when he fed the tool into the workpiece more aggressively. The author liked the feel and cut quality of the 3 mm T-shaped blade the best. Thinner T-shaped blades are available and would require less power, but The SC4 has the power required to use it and the 3mm blade would be less prone to breaking.

The blade holder used for the T-type blade in this test holds the blade horizontally, as shown to the right (the tissue was placed behind the blade so that it will stand out in the photograph). An alternate style holder holds the blade at about an angle which gives the tool and effective back rake of about 5^{o}. Both setups work well. The horizontal orientation has the advantage that the tool height does not have to be readjusted when you change the blade extension out of the holder.

10. Chuck Mount and Changing Chucks

Most lathes require an adapter plate for mounting after-market chucks. SIEG lathes have the equivalent of an adapter plate as a permanent part of the spindle. The three-jaw chuck that comes standard on the SC4 is secured to this plate by three hex-head bolts coming through the back of the plate. The six holes in the mounting plate are spaced to accommodate both 3-jaw and 4-jaw chucks.

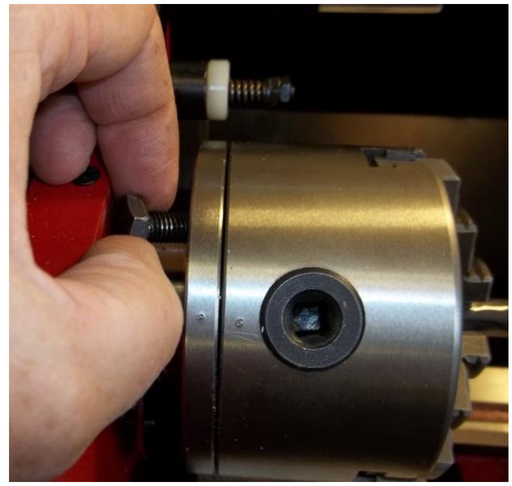


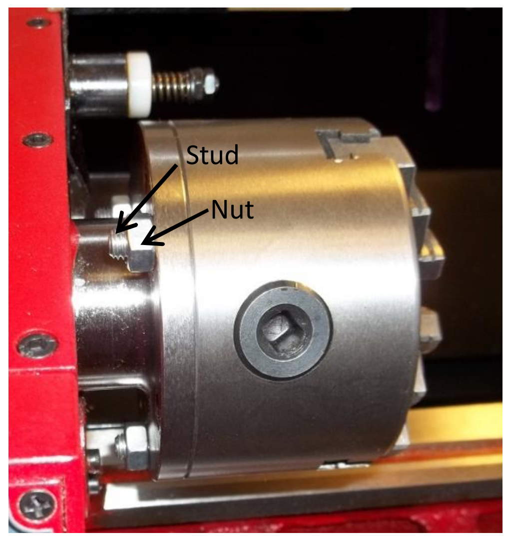


The advantage of this arrangement is that a separate adapter plate is not required when an alternate chuck is purchased as long as the alternate chuck is 100 mm in diameter. Note that almost all new 4" plane-back chucks sold today are actually 100 mm and will fit the SC4 but older 4" chucks made in North America may not.

Dismounting the chuck for the first time can be problematic since there is no way to lock the drive spindle. If the mounting bolts are torqued too tightly at the factory you may not be able to apply enough counter torque by using the chuck key in its socket in the chuck as a grip. Using a bar crosswise in the chuck jaws to provide sufficient counter torque is not recommend for fear of breaking or bending something.   A better solution is to use a strap wrench around the chuck or a socket extension bar or a bolt mounted tightly in the chuck and a socket ratchet handle and the supplied 14 mm wrench to remove the bolts, as shown to the right. In an extreme case, you may have to grind flats on the bolt to prevent its slipping in the chuck.

Before dismounting the chuck, make two adjacent punch marks, one on the chuck and one on the mounting plate as witness marks for remounting the chuck. It helps when realigning the bolt holes. A wooden cradle placed below the chuck as shown above prevents dropping the chuck onto the ways or your fingers. If you do not make a cradle, at least put a piece of wood over the ways. Your fingers heal but the chuck is heavy enough to make a permanent ding in the ways.





Now to the one feature on the SC4 that the present author does not like. It is very difficult to line up the proper bolt holes in the mounting plate and chuck and feed the bolts through. It does not help that the bolts are longer than the space between the plate and the headstock. To correct this problem, make some 25 mm long studs for the chucks by cutting the heads off some 30 mm M8 x 1.25 mm bolts. Wrap the ends of the studs going into the chucks with about 13 wraps of Teflon joint sealing tape. The Teflon prevents the studs from screwing back into the chuck when the nuts are being applied. You could use Loctite to glue the studs in place but the Teflon provides a tight enough fit and the studs can still be removed easily if the need ever arises. If you do this, make sure to trim away any excess Teflon so that stray pieces will not work their way between the chuck and mounting plate. Now it is easy to change from the 3-jaw chuck to the 4-jaw chuck. A wrench on the nuts and the chuck key in its socket provide enough counter torque to secure the chuck in place.

11. Things You May Want to Buy (and Where to Buy Them)

The items given below are recommended purchases. While the items listed can be bought from other reliable providers, the individual suppliers listed have given the author a combination of reasonable quality, price and service. The prices of course are current only on the date this was written (October, 2012) and will probably vary over time and they do not include shipping or applicable taxes.

1. **Surge protector** rated at 15 amps to protect the electronics from a current surge. This should be your first purchase even before you plug the lathe in. The author bought a 6 outlet Belkin direct plug-in surge protector rated at 1800 watts at Home Depot for $10. This is cheap insurance and while hoped never needed, Belkin protectors include a damaged equipment warranty (read the warranty and keep all the paper work). All surge protectors are not created equal and it is not worth the gamble to buy an off or no-brand unit.
2. A set of **T-handle metric Allen keys**. They are available at most good hardware stores or from grizzlytools.com (part no. H0487), busybeetools.com (part no. B761), from other on-line suppliers and ebay.com. Price with shipping should be under $20.

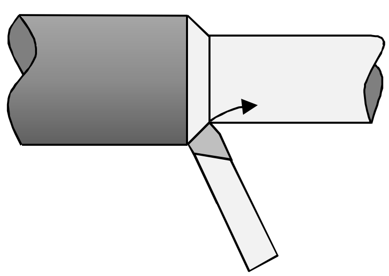
3. Permatex **Chain Lube** – amazon.com for 4.55 or at local motorcycle or auto supply shop for about $10.
4. **Quick change tool post set**, Tormach OXA and tool post mount for SC4 at littlemachineshop.com – part numbers 3112 ($129.95) and 4117 ($9.95). Additional tool holders (part no. 3114) available for 13.95 each. The stock 4-way tool post supplied with the SC4 is serviceable and sufficiently robust to do the job required of it. However, like all other such tool posts, you have to fiddle with shims to get the tool height set correctly. Quick change tool posts are much more convenient to use.
5. **5-piece 3/8" Indexable carbide tool bit set** with C6 inserts – available from grizzlytools.com, cdcotools.com, littlemachineshop.com and eBay.com for less than $50. Replacement inserts can be bought at carbidedepot.com. They sell the Carbi- Universal TCMT 21.51 insert for $5.25 each in packs of ten. Inserts can also be commonly found on eBay.com but the prices are generally not all that good and the providers do not usually tell you what the inserts are designed to cut (there are some significant differences). The Carbi-Universal inserts are for general purpose machining on all workpiece materials. For details, see http://www.carbidedepot.com/detail.aspx?ID=149188.
6. **8 pc. HSS 5/16" pre-ground tool bit set** - grizzlytools.com part no. H5871 – $41.95. Similar items are also available from other suppliers and on ebay.com.
7. **P-type parting blade** (cut-off blade) – make sure that the blade you buy is compatible with your parting blade holder. The author uses and likes his 3 mm wide x 12 mm Empire blade in his Tormach OXA parting blade holder but he got lucky to find the Empire blade on ebay.com. These blades can be hard to find. However, shars.com lists their $^{1}/_{8}$" x ½" P3S "P" Type HSS Cut-off Blade (part no. 404-1666) for $5.50.
8. **Live center – MT2** – these are available from most machinery supply houses (grizzlytools.com, busybeetools.com, cdcotools.com, shars.com, littlemachineshop.com) for about $25 to $30 and sometimes as little as $12 new on ebay.com. Do not buy a used live center on eBay.
9. The 87, 90 and 94 tooth change gears can be purchased as a set from Littlemachineshop.com for $29.95 plus tax and shipping. The link http://lmscnc.com/4214 will take you directly to it. The 87/90/94 gear set is worth the investment when cutting threads not included in the charts.

Appendix 1 – Chatter

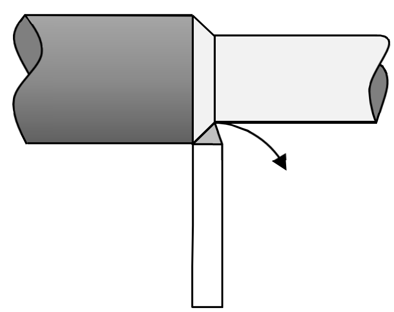
Sooner or later, every turner experiences lathe chatter. It occurs when the cutting action of the tool bit fluctuates in harmony with the natural vibration period of any part of the tool, tool holder or workpiece. It may ruin a part and, in its extreme, be damaging to the lathe. At minimum, it can be alarming to the inexperienced person and frustrating to the experienced operator. Some materials are more problematic than others. Changing the material grade or even the heat treatment within a grade can make a difference. For example, low carbon cold roll steel chatters much more than SAE 12L14, a steel grade specifically designed for machining. Some grades of brass are particularly problematic but grade 360 brass machines well. Likewise, some grades of aluminum are difficult to machine but the most common grade 6061 machines well. Long slender parts chatter more than short, stubby ones.

The usual steps for curing chatter are (more or less in the order that you should try):

1. Make sure that your cutting bit is set correctly – height of cutting tip right on the centerline of the workpiece and oriented correctly, as illustrated below.



Incorrect. Pressure on the tool tends to make it dig into the workpiece.

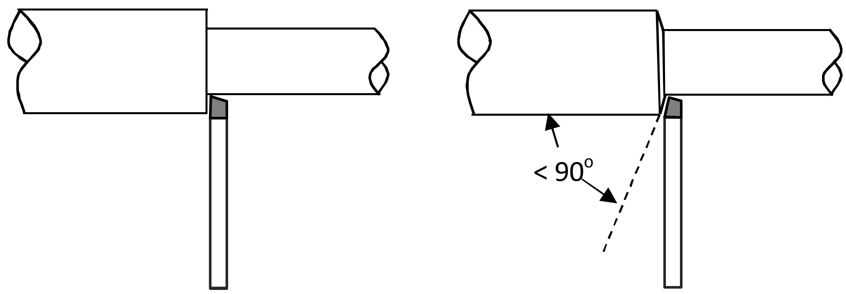


Correct. Pressure on the tool tends to make it swing away from workpiece which eliminates hogging

2. Sharpen the tool bit if it is dull.
3. Make sure your tool bit is shaped correctly with proper rake and clearance angles for the material being machined. For light lathes, keep the radius of the cutting tip small – almost pointed.
4. Reduce all overhangs as much as possible:
    a. Minimize unsupported material length out of the chuck.
    b. Retract the tool bit into it holder as far as possible.
    c. Back the tool holder into the tool post as far as the design allows.
    d. Move the compound slide back as far as possible and bring the cross slide as close as possible to the workpiece.

e. Use the tail stock center and make sure it is properly snugged up to the workpiece and locked into place. If you are using a dead center, grease it properly (just a bit of grease is required, but make sure you do not squeeze it all out by over tightening the center).
f. Use a follow rest, if available.

5. Change the tool angle in relation to the workpiece. A radial cutting edge (90^o to the long axis of the work piece) will chatter much more than one than one at a lower angle. Even a 10^o shift can make a difference.



6. Decrease the depth of cut and increase the rate of feed. (If the power of the lathe allows, you may try increasing both the depth of cut and the feed rate.) Just decreasing the depth of the cut will often eliminate chatter.
7. Change the cutting speed, up or down.
8. Wrapping a piece of soft lead sheet stock around the workpiece or clamping a weight to the workpiece will change its natural harmonic which may reduce chatter.
9. Change the cutting direction to cut away from the headstock rather than towards it. In this case, it is best to use a ball bearing live center in the tailstock if available. If not, make sure to grease the dead center well.
10. If you are using a live center in the tail stock and cutting towards the headstock, change to a dead center (grease it well).
11. Check to make sure the gibs on the cross slide, compound slide and carriage are properly adjusted.
12. Check the spindle bearings for looseness and adjust if required. (Note, end play does not usually present a problem unless your feed rate is far too small).
13. Small parts are better held in collet rather than with a jawed chuck.

When using a small lathe, you may occasionally find a piece of material which just will not turn without chattering. Perhaps more time spent on establishing the correct bit geometry would lead to success, but patience wears out before success and you put the material into the scrap bin and restart with something else. This happens most frequently when working a low machinability grade such as cold rolled SAE 1018 steel bar. It is worth the price to buy machining grade materials for some applications. SAE 12L14 steel, 360 brass and 6061 aluminum round bar are excellent for turning and can usually be purchased from local suppliers or on ebay.com.

Appendix 2 – Cutting Speed Tables

The following tables present the data from Section 7 in chart form for those who prefer to work from charts rather than equations.

The speed range for the SIEG SC4 is 100 to 2000 rpm. In some cases, the charts show speeds above or below this range. When higher speeds are shown, use 2000 rpm. Where lower speeds are indicated, use 100 rpm taking precautions such as lighter cuts and less aggressive feed rates.

As mentioned in section 6.2 in respect to the gear change charts, it is a good idea to copy these tables onto card stock and keep them at a location convenient to the lathe. Laminating them with clear vinyl film (available at the local dollar store) helps keep them clean.

Index of Tables

A Note about Cutting Speeds

Caution is advised when referring to published cutting speeds because they are often developed for very powerful and massively stiff commercial computer controlled lathes using flood coolant and very expensive cutting tools beyond what you would expect to find in the amateur`s shop. The cutting speed data given in this document was developed around smaller lathes with the type of cutting tools commonly found in the amateur`s shop and should be considered a general guide to getting started. With experience, the user may find that higher or lower speeds are possible and preferred.

			RPM for Turning With HSS Tooling										
Material	Operation	Cutting Speed (sfm)	Workpiece Diameter (in.)										
			0.125	0.25	0.375	0.5	0.75	1	1.5	2	2.5	3	4
Aluminum	Roughing	300	9120	4560	3040	2280	1520	1140	760	570	456	380	285
	Finishing	400	12160	6080	4053	3040	2027	1520	1013	760	608	507	380
Brass and Bronze	Roughing	150	4560	2280	1520	1140	760	570	380	285	228	190	143
	Finishing	400	12160	6080	4053	3040	2027	1520	1013	760	608	507	380
Copper	Roughing	100	3040	1520	1013	760	507	380	253	190	152	127	95
	Finishing	250	7600	3800	2533	1900	1267	950	633	475	380	317	238
Cast Iron - Soft	Roughing	80	2432	1216	811	608	405	304	203	152	122	101	76
	Finishing	250	7600	3800	2533	1900	1267	950	633	475	380	317	238
Cast Iron - Hard	Roughing	50	1520	760	507	380	253	190	127	95	76	63	48
	Finishing	150	4560	2280	1520	1140	760	570	380	285	228	190	143
Steel - Mild	Roughing	90	2736	1368	912	684	456	342	228	171	137	114	86
	Finishing	250	7600	3800	2533	1900	1267	950	633	475	380	317	238
Steel - Alloy	Roughing	40	1216	608	405	304	203	152	101	76	61	51	38
	Finishing	150	4560	2280	1520	1140	760	570	380	285	228	190	143
Steel - Tool (not hardened)	Roughing	50	1520	760	507	380	253	190	127	95	76	63	48
	Finishing	150	4560	2280	1520	1140	760	570	380	285	228	190	143
Steel - Stainless	Roughing	60	1824	912	608	456	304	228	152	114	91	76	57
	Finishing	180	5472	2736	1824	1368	912	684	456	342	274	228	171

Material	Operation	Cutting Speed (sfm)	RPM for Turning With Carbide Tooling — Workpiece Diameter (in.) 0.125	0.25	0.375	0.5	0.75	1	1.5	2	2.5	3	4
Aluminum	Roughing	600	18240	9120	6080	4560	3040	2280	1520	1140	912	760	570
	Finishing	800	24320	12160	8107	6080	4053	3040	2027	1520	1216	1013	760
Brass and Bronze	Roughing	300	9120	4560	3040	2280	1520	1140	760	570	456	380	285
	Finishing	800	24320	12160	8107	6080	4053	3040	2027	1520	1216	1013	760
Copper	Roughing	100	3040	1520	1013	760	507	380	253	190	152	127	95
	Finishing	500	15200	7600	5067	3800	2533	1900	1267	950	760	633	475
Cast Iron - Soft	Roughing	160	4864	2432	1621	1216	811	608	405	304	243	203	152
	Finishing	500	15200	7600	5067	3800	2533	1900	1267	950	760	633	475
Cast Iron - Hard	Roughing	100	3040	1520	1013	760	507	380	253	190	152	127	95
	Finishing	300	9120	4560	3040	2280	1520	1140	760	570	456	380	285
Steel - Mild	Roughing	180	5472	2736	1824	1368	912	684	456	342	274	228	171
	Finishing	500	15200	7600	5067	3800	2533	1900	1267	950	760	633	475
Steel - Alloy	Roughing	80	2432	1216	811	608	405	304	203	152	122	101	76
	Finishing	300	9120	4560	3040	2280	1520	1140	760	570	456	380	285
Steel - Tool (not hardened)	Roughing	100	3040	1520	1013	760	507	380	253	190	152	127	95
	Finishing	300	9120	4560	3040	2280	1520	1140	760	570	456	380	285
Steel - Stainless	Roughing	120	3648	1824	1216	912	608	456	304	228	182	152	114
	Finishing	360	10944	5472	3648	2736	1824	1368	912	684	547	456	342

			RPM for Turning With HSS Tooling										
Material	Operation	Cutting Speed (MPM)	Workpiece Diameter (mm)										
			3	4	5	10	20	25	40	50	60	70	100
Aluminum	Roughing	**92**	9752	7314	5851	2926	1463	1170	731	585	488	418	293
	Finishing	**122**	12932	9699	7759	3880	1940	1552	970	776	647	554	388
Brass and Bronze	Roughing	**46**	4876	3657	2926	1463	731	585	366	293	244	209	146
	Finishing	**122**	12932	9699	7759	3880	1940	1552	970	776	647	554	388
Copper	Roughing	**31**	3286	2465	1972	986	493	394	246	197	164	141	99
	Finishing	**76**	8056	6042	4834	2417	1208	967	604	483	403	345	242
Cast Iron - Soft	Roughing	**24**	2544	1908	1526	763	382	305	191	153	127	109	76
	Finishing	**76**	8056	6042	4834	2417	1208	967	604	483	403	345	242
Cast Iron - Hard	Roughing	**15**	1590	1193	954	477	239	191	119	95	80	68	48
	Finishing	**46**	4876	3657	2926	1463	731	585	366	293	244	209	146
Steel - Mild	Roughing	**27**	2862	2147	1717	859	429	343	215	172	143	123	86
	Finishing	**76**	8056	6042	4834	2417	1208	967	604	483	403	345	242
Steel - Alloy	Roughing	**12**	1272	954	763	382	191	153	95	76	64	55	38
	Finishing	**46**	4876	3657	2926	1463	731	585	366	293	244	209	146
Steel - Tool (not hardened)	Roughing	**15**	1590	1193	954	477	239	191	119	95	80	68	48
	Finishing	**46**	4876	3657	2926	1463	731	585	366	293	244	209	146
Steel - Stainless	Roughing	**18**	1908	1431	1145	572	286	229	143	114	95	82	57
	Finishing	**55**	5830	4373	3498	1749	875	700	437	350	292	250	175

			RPM for Turning With Carbide Tooling										
Material	Operation	Cutting Speed (MPM)	Workpiece Diameter (mm)										
			3	4	5	10	20	25	40	50	60	70	100
Aluminum	Roughing	**184**	19504	14628	11702	5851	2926	2340	1463	1170	975	836	585
	Finishing	**244**	25864	19398	15518	7759	3880	3104	1940	1552	1293	1108	776
Brass and Bronze	Roughing	**92**	9752	7314	5851	2926	1463	1170	731	585	488	418	293
	Finishing	**244**	25864	19398	15518	7759	3880	3104	1940	1552	1293	1108	776
Copper	Roughing	**62**	6572	4929	3943	1972	986	789	493	394	329	282	197
	Finishing	**152**	16112	12084	9667	4834	2417	1933	1208	967	806	691	483
Cast Iron - Soft	Roughing	**48**	5088	3816	3053	1526	763	611	382	305	254	218	153
	Finishing	**152**	16112	12084	9667	4834	2417	1933	1208	967	806	691	483
Cast Iron - Hard	Roughing	**30**	3180	2385	1908	954	477	382	239	191	159	136	95
	Finishing	**92**	9752	7314	5851	2926	1463	1170	731	585	488	418	293
Steel - Mild	Roughing	**54**	5724	4293	3434	1717	859	687	429	343	286	245	172
	Finishing	**152**	16112	12084	9667	4834	2417	1933	1208	967	806	691	483
Steel - Alloy	Roughing	**24**	2544	1908	1526	763	382	305	191	153	127	109	76
	Finishing	**92**	9752	7314	5851	2926	1463	1170	731	585	488	418	293
Steel - Tool (not hardened)	Roughing	**30**	3180	2385	1908	954	477	382	239	191	159	136	95
	Finishing	**92**	9752	7314	5851	2926	1463	1170	731	585	488	418	293
Steel - Stainless	Roughing	**36**	3816	2862	2290	1145	572	458	286	229	191	164	114
	Finishing	**110**	11660	8745	6996	3498	1749	1399	875	700	583	500	350

RPM for Drilling and Reaming With HSS Tooling

Material	Operation	Cutting Speed (sfm)	Tool Diameter (in.)							
			0.0625	0.125	0.1875	0.25	0.3125	0.375	0.4375	0.5
Aluminum	Drilling	240	14592	7296	4864	3648	2918	2432	2085	1824
	Reaming	200	12160	6080	4053	3040	2432	2027	1737	1520
Brass and Bronze	Drilling	120	7296	3648	2432	1824	1459	1216	1042	912
	Reaming	110	6688	3344	2229	1672	1338	1115	955	836
Copper	Drilling	100	6080	3040	2027	1520	1216	1013	869	760
	Reaming	75	4560	2280	1520	1140	912	760	651	570
Cast Iron - Soft	Drilling	100	6080	3040	2027	1520	1216	1013	869	760
	Reaming	65	3952	1976	1317	988	790	659	565	494
Cast Iron - Hard	Drilling	80	4864	2432	1621	1216	973	811	695	608
	Reaming	60	3648	1824	1216	912	730	608	521	456
Steel - Mild	Drilling	100	6080	3040	2027	1520	1216	1013	869	760
	Reaming	55	3344	1672	1115	836	669	557	478	418
Steel - Alloy	Drilling	30	1824	912	608	456	365	304	261	228
	Reaming	15	912	456	304	228	182	152	130	114
Steel - Tool (not hardened)	Drilling	45	2736	1368	912	684	547	456	391	342
	Reaming	30	1824	912	608	456	365	304	261	228
Steel - Stainless	Drilling	50	3040	1520	1013	760	608	507	434	380
	Reaming	30	1824	912	608	456	365	304	261	228

Use of a boring tool is recommended for bores larger than ½".

RPM for Drilling and Reaming With HSS Tooling

Material	Operation	Cutting Speed (MPM)	Tool Diameter (mm) 1	2	3	4	5	6	7	8	9	10	12.5
Aluminum	Drilling	73	23214	11607	7738	5804	4643	3869	3316	2902	2579	2321	1857
	Reaming	61	19398	9699	6466	4850	3880	3233	2771	2425	2155	1940	1552
Brass and Bronze	Drilling	37	11766	5883	3922	2942	2353	1961	1681	1471	1307	1177	941
	Reaming	34	10812	5406	3604	2703	2162	1802	1545	1352	1201	1081	865
Copper	Drilling	31	9858	4929	3286	2465	1972	1643	1408	1232	1095	986	789
	Reaming	23	7314	3657	2438	1829	1463	1219	1045	914	813	731	585
Cast Iron - Soft	Drilling	31	9858	4929	3286	2465	1972	1643	1408	1232	1095	986	789
	Reaming	20	6360	3180	2120	1590	1272	1060	909	795	707	636	509
Cast Iron - Hard	Drilling	24	7632	3816	2544	1908	1526	1272	1090	954	848	763	611
	Reaming	18	5724	2862	1908	1431	1145	954	818	716	636	572	458
Steel - Mild	Drilling	31	9858	4929	3286	2465	1972	1643	1408	1232	1095	986	789
	Reaming	17	5406	2703	1802	1352	1081	901	772	676	601	541	432
Steel - Alloy	Drilling	9	2862	1431	954	716	572	477	409	358	318	286	229
	Reaming	5	1590	795	530	398	318	265	227	199	177	159	127
Steel - Tool (not hardened)	Drilling	14	4452	2226	1484	1113	890	742	636	557	495	445	356
	Reaming	9	2862	1431	954	716	572	477	409	358	318	286	229
Steel - Stainless	Drilling	15	4770	2385	1590	1193	954	795	681	596	530	477	382
	Reaming	9	2862	1431	954	716	572	477	409	358	318	286	229

Use of a boring tool is recommended for bores larger than 12.5 mm.