SECTION 1: Cutting train wheels

What follows is my text and pictorial descriptions of the modern means and method for cutting clock train wheels. First, a summary of the major tasks:

Summary of Major Tasks:

Task 1: Calculations for wheel cutting
Task 2: Material choices and blank specifications for arbor, tightening washer, backer plate and end mills.
Task 3: Machine the arbor
Task 4: Tightening washer: Buy it or machine it
Task 5: Machine backer plate
Task 6: Machine wheel blank
Task 7: Machine single point cutter
Task 8: Harden single point cutter
Task 9: Sharpen single point cutter
Task 10: Milling machine/CNC rotary table setup
Task 11: Wheel cutting

MAJOR TASKS - DETAILED:

Task 1: Calculations for Wheel Cutting

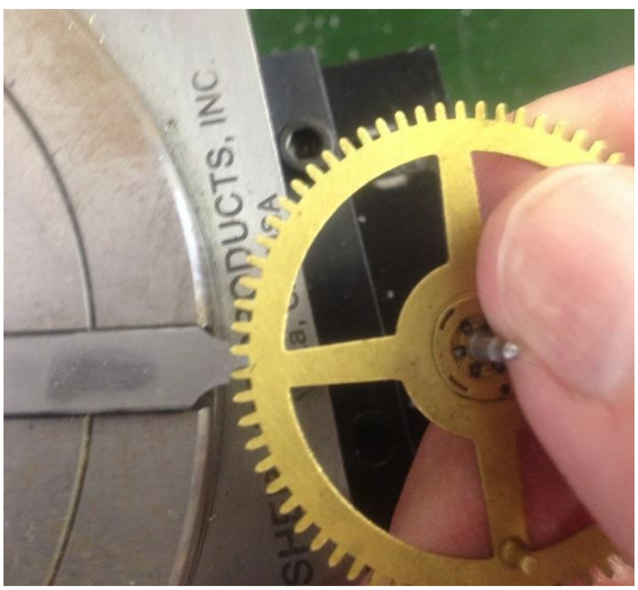
1. Total Number of Teeth to Cut:
   This is a critical number and is used in two places. It will become the number to enter into the Sherline programmable controller that is connected to the CNC Rotary Table (CNC RT) that is used for indexing the wheel and it also is used on the Wheel Cutting Worksheet (All items to be discussed in detail later) to calculate Diametrical Pitch (DP) that in turn is used to calculate the module. While neither the DP nor the module are required information in Jerry's methods. I include them for curiosity's sake and for reference.
   a. Existing wheel - simply count the number of teeth. In doing so, it is always a good idea to mark the first tooth counted, so one will know where to end the count ☺….and one should count the teeth at least twice.
   b. Design wheel – refer to the clockmaker's drawings.
   c. Formulas
      i. DP teeth ÷ Pitch Diameter (PD) in inches
      ii. Module = 25.4 ÷ DP (converts inches to metric)

2. Outside Diameter (OD) of wheel in inches:
   The wheel blank will be machined to this size. This is a critical measurement and ideally is machined exactly or + (-) .001".
   a. If one has the existing wheel with enough good opposing teeth, using the dial calipers, measure from one good tooth tip to an opposite good tooth tip. The resulting reading is the OD.
   b. Another method if one has an existing wheel and a good measurement of tooth height: Using the calipers, place one jaw tip on the bottom of a good tooth space (tooth root) and place the other jaw tip on an opposing good tooth space. The resulting reading + the tooth height = OD.
   c. Otherwise it will need to be recalculated.
      i. # Teeth (n) + 2.76 (factor for wheels > 20 teeth) ÷ Diametrical Pitch (DP)
      ii. Or stated the W.R. Smith way "OD = M_{inches} (N+2.76), where M_{inches} is the module in inches, N is the number of teeth and…the 2.76 factor is the number of imaginary teeth required to increase the size from the pitch diameter to the outside diameter of the wheel blank.[20]

3. Thickness of wheel. Measure with a micrometer the thickness of the original wheel and strive for that thickness or close to that with the wheel blank material.

4. Pitch Diameter in Inches (PD):
   This measurement is used in JK's formula/method for calculating the tooth height and is also used in my Wheel Cutting Worksheet to calculate the diametrical pitch (DP) and in turn the "module". The PD in inches can be calculated using a variety of formulas, however, JK has devised a physical method to measure PD, a measurement that some books say can't be measured. Using the dial caliper, position one of its jaw tips on the bottom (root) of a good tooth space while placing the other jaw tip on an opposing good tooth tip and with the resulting reading one has effectively measured PD. At this point, it is helpful to visualize what the PD is actually, it is a reading that is approximately at the halfway point of the tooth i.e. between its root and its tip. Using JK's method simulates this distance. JK's physical method of using the calipers to determine the OD and PD is quite valid for calculating the DP and module. For example, the PD could be off as much .025"+(-) and still not change the DP i.e. the fractional results just don't change the whole number DP until the measuring error is .025"+ (-) and, if being careful, the measurement error just isn't going to be that far off.

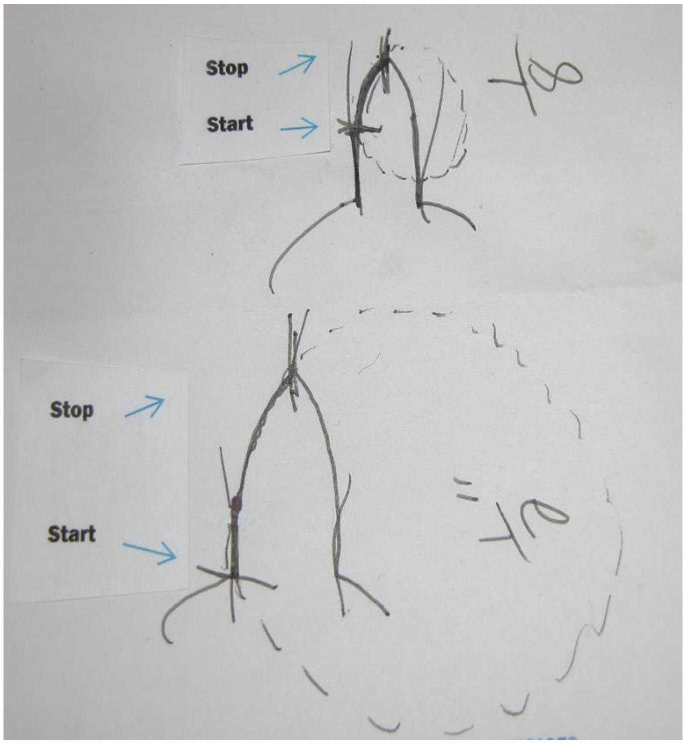
5. Tooth Height, Tooth Radius, & Tooth Space (Width). All measurements that are needed to machine the single point cutter. Any inaccuracies in these measurements will result in a cutter tip that does not represent the original.



Here, the damaged wheel being replaced is shown with the cutter that has just been machined to cut the new replacement wheel. This demonstrates the desired fit between the cutter and the tooth space. The cutter tip with its machined radius should fit into the space like a glove i.e. it should fill the tooth space fully/tightly and enclose the tooth addendum curves. This also demonstrates the quality check one needs to do as part of machining a cutter. After machining the cutter, check the fit as shown above. If you don't achieve this fit, then something is suspect.

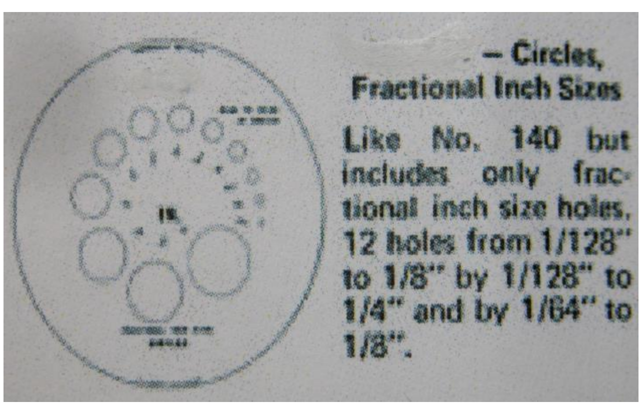
Tooth Height: (Using above measurement methods) OD – PD = Tooth Height
If after machining the cutter, the cutter tip i.e. tooth height appears too long and the fit is not complete and snug, it may be that you've judged the radius incorrectly. Using the wrong diameter end mill to machine the radius can make it appear that the cutter tip is too long. Ask me how I know☺. I should also note that it is a bit easy to have small errors (.005" + -) when measuring with the calipers and as the OD and PD are used to calculate the tooth height, being off .005" on the tooth height could be significant. The tooth space that would be cut would not be like the original and certainly the fit discussed above would not be achieved. A couple of thoughts: 1) If, as part of the quality check, the cutter tip is too long and the radius appears correct, what I've been able to do is machine off more front relief. That has worked, but is a bit tedious and one has to be careful to avoid "spoiling the job" and 2) I've added the theoretical calculation for tooth height (DP ÷ # teeth = tooth height), to the worksheet for either use or reference. In some instances it may present an alternate method. Ultimately, practice and experience will be the guide.

Tooth Radius:
For a couple of reasons, gage pins are good tools to compare to the start and stop of the tooth curve (see JK's drawing below) to determine the correct radius: 1)their circle shape a/k/a its circular arc allows one to use the comparative measuring method to judge the radius and 2) they are available in .001" increments. Typically this is an iterative process whereby one is choosing a gage pin and placing its circle shaped end within the start and stop of the tooth curve and judging if the gage pin's end matches the tooth curve and, if not, trying another gage pin. It is critical to determine the correct radius. Choosing the wrong gage pin will result in choosing the wrong end mill which will result in the wrong radius and the resulting tooth shape will not be as the original. This step does take a trained eye and for me at least, I will only get good at this with experience, trial and error if you will. The good news is if the radius is wrong, one just tries again. The W-1 material is really cheap at the 1 ½" length and it is really just ones time to redo. First, determine where the radius starts on the tooth form and where it ends. That is the area to be measured. See JK's drawings below. They depict the start and stop of the curve. The concept is for a gage pin with its circular arc (refer also to drawing and description on page 7) to fit within the start and stop area. To better assist with judging the radius, hold the gage pin behind the tooth against a white background.



Jerry's "napkin sketch" of two cycloidal tooth forms (and my start/stop annotations). Each depict the start and stop of their curve. The idea is for the gage pin with its full circle a/k/a its circle arc (refer to drawings and comments on page 7) to fit within the respective start and stop area.

Another option for judging a radius is to obtain a hand held optical a/k/a measuring comparator with reticles that have circles of various sizes from at least 1/16" to 1/4". This would take the place of gage pins. Likely one will need more than one reticle to have sufficient choices of circle diameters. Reticles having circles only are best as there is less clutter and when matching a circle image to the tooth's curve it is easier to discern. I struggled to grasp the reticle image to use and struggled again searching to buy. Thus, the below are included for easy reference. This tool quickly improved my judgment and is now my favorite tool for determining the radius.



Reticle with circles only. Purchased from Paul Gardner Co. Gardco.com.

6x hand held comparator/magnifier.
It's barrel is transparent to allow in light.
Also from Gardco.com.

Recognize there may be times that you will need to remove the wheel from its collet so as to make the comparison. The comparator needs to sit on top of the tooth and sometimes the pinion or arbor are in the way. This should be of no consequence as one is making a replacement wheel and the original wheel will have already been removed. Remember the correct circle will fit within the start and stop of the curve. While on the topic of optics. Be aware of the need for non-distortion a/k/a distortion free optics. Otherwise one's view is distorted and no amount of assists will help with your correct judgment of tooth forms. 10x magnifiers/loupes are recommended. For reference, the Optivisor headband types of magnifiers are generally not distortion free. They serve their purpose but are not distortion free. On the other hand Behr single and double loupes are distortion free optics and fit nicely onto your safety glasses or regular glasses.

The radius for most clock train wheel teeth will measure 3/32" and greater i.e. 3/32", 7/64", 1/8", 3/16". Stay mindful that the rolling contact is at the tooth's pitch circle and slightly above and below the pitch circle, the rest of the tooth's surface is for clearance. Thus, while it is good workmanship to mimic the original tooth, the curve of the tooth top is not that critical.

Tooth Space (tooth width):

Again gage pins are a good tool for measuring the tooth space. This is a progressive trial and error process whereby gage pins are placed in between the teeth i.e. tooth space and pressed down lightly until a gage pin will not bottom out i.e. it becomes stuck within the tooth sides and will not touch the bottom of the tooth space. Then choose the gage pin diameter just before it as the correct tooth space size. This is a critical measurement as it contributes to the calculation of the cutter tip, specifically the width of the cutter tip. A cutter tip that is either too narrow or too wide will, of course, not replicate the original tooth space. More importantly, it might not work. It could create excessive friction/stickiness, faults that can cause a clock train to stop. While there is some tolerance for error, JK would say a commercial module generally doesn't even come close to replicating the original tooth and it will work, it is always best to target the original tooth shape space. Note that measuring the width of the tooth space is by default measuring the width of the tooth root, it is not measuring the tooth radius. That is a separate measurement step as discussed above. Thus, expect that the tooth radius will be a different gage pin measurement than the tooth space.

6. Wheel Center Hole:
   First some background:
   Typically, wheels are mounted on a wheel collet that is affixed to the arbor. The diameters of the collet, wheel center hole and arbor are all very nicely choreographed to allow for a concentric, tight fit. In most instances, when the original wheel is removed from the collet (all to be discussed later) the concentric fit and the metal are upset to the point that the wheel collet is not reusable for mounting the newly made wheel. Consequently, this article assumes a new wheel collet will be made. Given this, one has flexibility on the final dimension of the wheel center and its collet and by extension the arbor step diameter (all to be further discussed in a bit).

   All that said, it is fine to take a reading of the original wheel center and use it as a guideline for making the center holes for the wheel blank and the backer plate. Recognize that the accuracy of the dial caliper's jaws designed to measure ID is at best only within .001"/.002". Therefore, gage pins are the recommended means of measuring and this will be referred to throughout this article. The measuring method is a progressive trial and error process whereby gage pins are passed thru the center hole until a gage pin will not go thru the center hole. Then choose the gage pin diameter just before it, as it is the correct size of the hole.

7. Enter all these measurements and calculations onto the "Wheel Cutting Worksheet" and the work point offsets will be calculated. Of course, that assumes you have constructed an Excel spreadsheet with the formulas ☺.

Task 2: Material Choices and Specifications for Arbor, Tightening Washer, Backer Plate and End Mills:

1. Arbor:
    a. 5/8” diameter 12L14 Steel. This is low carbon steel with lead added and is known variously as “leaded free cutting steel”, “free machining steel” or “cold rolled steel”. The lead improves its machinability with little effect on its mechanical properties.
        i. 5/8” diameter size is critical as this is the largest diameter that fits thru the center hole of the Sherline 3 jaw chuck.
    b. Purchase in 12” lengths; Hacksaw blanks into approximate lengths of 2 1/2” .
2. Tightening washer:
    a. Buy it: Commercial washer, if appropriate size is available. There are several choices. Below are a couple of suggestions:
        i. Fender washer with a 5/32” center hole x 7/8” OD suitable for a 10-32 Allen screw
        ii. Fender washer 1/8” center hole x 7/8” OD suitable for an #8-32 Allen screw

Variety of machined and purchased washers

    b. Machine it
        i. 5/8” diameter 12L14 Steel (leaded steel) x 1/8” thick washers can be machined (drilled and parted off) to receive the 10-32 Allen screw
        ii. ½” diameter 12L14 steel (leaded steel) x 1/8” thick is another choice. Most center holes are less than ½” due to their arbor diameters or their collets being less than ½”.
        iii. Machine these before any other wheel cutting steps are started and have them on hand. Drill a hole to receive the 10-32 Allen screw and part off at 1/8” thick. Hardening and tempering is not necessary. Note that the perfect size drill for clearance is # 11 drill (.191”) for 10-32 thru clearance; the # 18 (.169”) drill for 8-32 thru clearance; and the # 28 (.140”) drill for 6-32 thru clearance.

3. Backer Plate Material
    a. In the wheel cutting setup, the wheel requires underneath support during the cutting operation. The backer plate provides this support. Note that one 3/16" Delrin backer plate is sufficient.
    b. Material: "Delrin" Plastic. The Delrin plastic material is very rigid, yet cuts like butter which is why it is ideal for a backer plate. It is available from supply houses such as MSC.com. If the backer plate isn't the exact right size, the single point cutter cuts right through it without stressing the wheel cutting setup.
    c. Purchase in 12" squares x 3/16" thick. Cut length and width on table saw. While there is some flexibility on the actual length and width of the square blank, generally it should not exceed the wheel OD. Beyond that is just wasting material.

4. End Mills

   I discuss the specifics of end mill sizes with a bit more detail as it matters. For example, you'll need a minimum flute length, a certain length of cut (LOC) to machine the cutters and there can be a variety of LOC's for a given end mill cutting diameter. Consequently, to help sort through that, I've included the below detail.
    a. Mill collets: For efficient, effective micro machining, end mills need to be held in collets. Sherline sells a mill collet set that satisfies most needs. Their sizes are 1/8", 3/16" and ¼".
    b. End mill shank diameters: Supply houses sell end mills with 1/8", 3/16" and ¼" shanks.
    c. End mill flute length a/k/a "length of cut" or "LOC": You will need miniature end mills with flute lengths (LOC) of at least 1/8" (.125"), preferably more. Different supply houses use different labels for essentially the same size. As a rule, the longer the overall length, the longer will be the LOC. Look for the LOC label "regular", "standard" or "long lengths". Just don't choose "stub" flute lengths as this is too short.
    d. End mill overall length (OAL): Ideally, overall lengths of end mills will be 2" or less. Longer than that, takes some fiddling to avoid conflicting with the Sherline mill's drawbar while minimizing how far the end mill is extended. You want as rigid a cutting tool setup as possible. On occasion, to obtain sufficient LOC, I've purchased miniature end mills with an overall length of 2 ¼" and either ground off one of the cutter ends or cut them in half.
    e. Supply houses that have worked for me are:
        i. Performance Micro Tool at www.pmtnow.com for end mill cutting diameters up to 3/32" as they have a more than adequate flute i.e. length of cut (LOC). PMT's end mills are single end with shank diameters of 1/8" and overall lengths of 1.50".

ii. Enco at www.use-enco.com for end mill diameters greater than 3/32" as above that diameter their LOC's are usable (1/8" or greater). Enco's end mills are double end with shank diameters of 3/16" and come in several lengths.

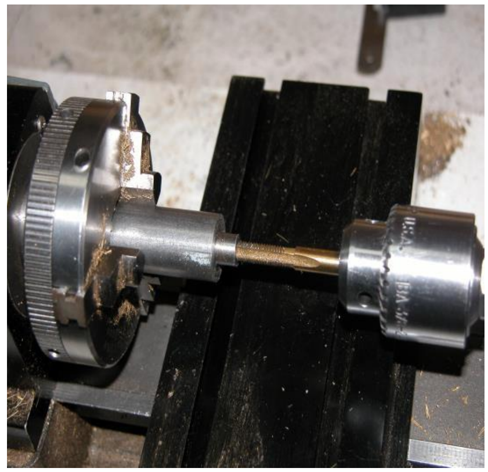
f. First cut, second cut and third cuts use a ¼" diameter stub end mill (to provide you with a rigid tool setup) with these characteristics: HSS single end, stub length end mill with two flutes and1/4" shank.

g. Fourth cut uses a variety of end mill diameters, all with these same characteristics: Miniature, two flute, single or double end, flutes (LOC) as discussed above, HSS end mill with 1/8" and 3/16" shanks as discussed above. Note that the end mill diameters are driven by the radius that has been judged above. A full set of end mills ranging in size from 1/32" – 3/16" would be advisable to have in ready preparation for most all wheel cutting jobs.

Task 3: Machine Arbor Blank

Machining the arbor blank involves machining a step on its end and thread cutting. I prefer internal threads as they present a bit more elegant result than external threads. Also, I’ve found it easier to make the internal threads.


Step machined on arbor


Lathe setup with tap. Make sure the tailstock is free to move and the Jacobs chuck taper is loose in the tailstock MT


Arbor with internal threading


Quality check: finished arbor with its Allen screw

When wheel cutting, the setup must be concentric. This is best achieved by leaving the arbor tight in the Sherline 3 jaw chuck at all times during machining – once secured in the 3 jaw chuck, don't remove arbor from chuck. This need to be concentric starts with machining the step on the arbor blank. Once it is secured into the 3 jaw chuck and the step machined, the arbor is not to be removed until all the wheel cutting steps are complete. Until you build up a variety of choices, one should expect to machine a new arbor each time a wheel is cut.

Step 1: Prepare the arbor blank:
Fit up the blank into the Sherline 3 jaw chuck. Face off both ends. One end to leave a square surface for the subsequent machining…the other end for good workmanship! Remove blank from chuck and scratch a mark ¾" or .750" from one end. This is easily done with the caliper's jaw tips. Fit up the blank back into the chuck so that the jaw front tips just touch the .750" scratch mark. Reason for this limiter mark is to aide subsequent machining steps on the CNC RT. If the blank is set deeper than .750" in the chuck the arbor body will bottom out on the RT adapter plug, rather than bottoming out on the threads of the 3 jaw chuck and thus prevent a secure fit. Finish blank size approximately 2" x 5/8". For rigidity, one wants as short an arbor as possible ergo the length of 2". Again, to maintain concentricity, leave the arbor in the chuck until all wheel cutting steps have been completed.

Step 2: Calculate step length and diameter:
With internal threads a socket head cap screw (a/k/a Allen screw) is used to tighten the backer plate, wheel blank and tightening washer against the arbor wall. This is best achieved by leaving the wheel blank and backer plate .010" proud of the step and machining a tightening washer or using a commercial washer to snug it up against the wheel blank.

a. Determine Step Length:
   Measure thickness of the wheel with micrometer. Add thickness of the wheel plus 3/16" thickness of the Delrin Backer Plate minus a "tightening factor" of .010". This will leave the step a bit short and allow the washer to press against it. (WT + BT - TF (.010") = Step Length). This is not a critical dimension i.e. does not need to be exact. There is some tolerance here. If the wheel blank stands proud a little more than .010" that is of no consequence so long as the majority of the wheel thickness rests on the step. If the wheel blank doesn't protrude enough, then face off the step to remove a bit of material.

b. Determine Step Diameter:
As a reminder, this article assumes the making of a new wheel collet thus leaving flexibility for the diameters of the collet and wheel center hole and by extension the arbor's step diameter. Such that, given the machining required to produce a step that will allow for a concentric fit (as discussed below), the arbor step will be what it will be. Said another way, the step diameter controls the diameters of both the collet and wheel center hole. That said it is fine, and probably good practice, to use the original wheel center hole as a guide.

To ensure concentricity, it is critical that the wheel center hole be a snug fit onto the step. If not snug, it becomes a do over. While that may sound excessive, I assure you it is not. If the fit is even .005" too loose, the wheel blank will sit loosely on the step and the concentricity will be lost. When wheel cutting, you'll find yourself cutting deeper on one half of the wheel and shallow on the other half. A "spoiled job" as they say. For the benefit of first timers, a snug fit would be a step diameter of .220" and a wheel center hole of .220".

Using the original wheel center hole as a guide a/k/a target diameter, proceed. It is best to measure progress with a micrometer instead of the dial calipers and to "sneak up on the target diameter". When .010" away from the aforementioned target diameter, change tactics to light material removal (.001", .002") and checking progress with the micrometer until the step is approximately.005"+ the target diameter. As discussed below, the final pass will include machining a square shoulder and the final step diameter will be what it will be.

Step 3: Machine the arbor step diameter and length:
As part of this machining is the need to leave a square shoulder for the backer plate and wheel blank to press against and, more importantly, to produce a square shoulder that will allow for a concentric fit. This is best achieved by using a lathe cutting tool known as a right hand facing tool that has been honed sharp and on the final machining pass for diameter and length, to bring the lathe cutting tool all the way into the shoulder and then bring the cutting tool back out towards the operator. It is best to ensure the step is too long than too short. If too long, one can just face it off a bit. Being too short requires one to repeat the above process to achieve enough length and a square shoulder. Not the end of the world, but less efficient. So once one is approximately .005"+ the target diameter, dial in that .005" cut, make the final diameter cut and then make the shoulder cut. It is not critical that the final step diameter be exactly at the target diameter. It is critical, however, to leave a square shoulder. So if one more pass is necessary for

that to be achieved, then go for it, as the step diameter can be what it will be. Just be aware that having just made a square shoulder, one now has lost the zero position on the diameter. Consequently, one is left with returning the lathe tool to its zero position on the diameter and a few thousandths deeper to make another pass towards the shoulder, remove more material and achieve that critical square shoulder result. Measure the final step diameter result for it now becomes the critical target diameter for the center holes of both the wheel and the backer plate.

Cutter blanks that I've ground into right hand facing tools in both ¼" and 3/16" squares. Special thanks to Werner Paul, a nearby master micro-machinist, for getting me started with tool grinding. An Internet search for South Bend Lathe Works Bulletin No. 35"How to Grind Lathe Tool Cutter Bits" will provide you with a great reference as well. Buy the cheap M2 cutter tool bits and practice grinding until you develop the ability for repeatable outcomes.

While on the topic of tool grinding, my favorite material for lathe cutter blanks is HSS T-15, preferably in a non-import brand. With the exception of French pivots, you can machine the full range of materials you would encounter as a clockmaker including 12L14 mild steel, 0-1 drill rod and blue pivot steel with using just this one type of cutter blank material. Those hardened French pivots invariably require a carbide graver to remove, but otherwise have a go with this type of material. It was a tip I received from an ole' machinist at a model engineering fair. Works great! For additional reference and to dispel an old notion, with the materials used today in cutter tool bits, you really can't ruin the cutter material or its cutting edge by getting it "too hot" from grinding to the desired shape. You'll want water nearby to dip the cutter tip into and cool it down for handling purposes, but you really can't hurt the material or the cutting edge you are grinding.

With the arbor blank and step machined it is time to drill and tap to create the internal thread hole.

Step 4: Preparation for making the internal threads

Generally, the threads will be made for a 10-32 x ½" length Allen screw. This will require a #21 drill bit and a depth of ½" plus. A good way to ensure the right depth is to put a limiter mark (masking tape works good) on both the drill and the tap to use as a point of reference. It is ok to be a bit too deep. Not OK to be short as then the Allen screw will bottom out on the threads instead of against the wheel blank and the result will not be tight. This will result in another do-over of the arbor blank if the right thickness tightening washer is not readily available.

- Fit up the Jacobs chuck into the tail stock.
- Spot center with center drill. Fit up the # 1 center drill in the tailstock and drill the starter hole.
- Using the specified drill (#21) with a limiter tape at the ½" mark, drill the hole.
- Fit up the 10-32 tap, with its ½" limiter tape to indicate finish depth, into the Jacobs drill chuck and tighten firmly.

Now here is where the trick and finesse comes into play to tap an internal thread on a lathe…the key to successful tapping on the lathe is a tailstock with complete freedom and a Jacobs chuck taper that is free to move inside the tailstock's Morse Taper…Tailstock must be loose on its bed to allow the tailstock to be immediately pulled along as the tap progresses deeper into the hole being tapped. The Jacobs chuck needs to be free in its Morse Taper (MT) so that it will freely turn if the tap sticks or catches. That said, the Jacobs chuck needs to remain in the tailstock MT , albeit loosely, to maintain straightness i.e. provide guidance and direction for the tap to start straight and stay straight as it enters and progresses into the drilled hole. Success is all the more enhanced, by turning the lathe hand wheel by hand, rather than turning on the lathe, to create the rotating motion needed to create the threads. This contends with resistance from a sticky tap and avoids stripping threads and broken taps.

JK showed how to make threads by turning the lathe on, but this will take some trial and error practice. I think for now, with the one-off threads I will do, making threads by hand turning the hand wheel makes the best sense for me.

- Loosen the tailstock so that it moves freely on the bed.

- Eject the Jacobs chuck from its secure position in the tailstock, while still maintaining it loosely in the tailstock MT housing. This provides both support and the straight on position needed for successful tapping.

Step 5: Create the threads

Place one's right hand on the Jacobs chuck and place the left hand on the lathe hand wheel. Place/feed the front end of the tap, its business end, into the drilled hole while starting to turn the hand wheel by hand. When both are aligned, apply slight horizontal pressure to the tap i.e. push the Jacobs chuck and turn the hand wheel. You will find the turning action of the hand wheel coupled with the slight horizontal pressure will pull the tap forward and create "the magical threads". Of course, while applying the slight horizontal pressure one is also holding the Jacobs chuck body to prevent it from turning. Soon a rhythm will begin and threads will be made. As one progresses likely resistance will be encountered. This is a good time to reverse the hand wheel and back out the tap. Brush off the chips, apply some cutting oil to the tap threads, reenter the tap and continue turning the hand wheel, all the while holding the Jacobs chuck firmly enough so that it doesn't turn, yet being ready to release if too much resistance is encountered. As one gets deeper, closer to the finish depth, this should be evident by the limiter tape, it is not unusual to need to apply some extra torque to the hand wheel…interpret that as elbow grease to finish up the threads.

Step 6: Quality Check

Once it appears that the finish depth is reached and threads have been made, back out the tap. Remove any obvious metal chips and debris. At this point I do a quality check by screwing in the 10-32 x ½" Allen screw. If resistance is encountered or more depth is needed, reinsert the tap and redo the threads and/or make a few more threads if not yet bottomed out and then back out the tap and again try the Allen screw. If resistance is still encountered and I believe the depth is correct, I will use the Allen wrench, place it in the socket head and then turn the screw to make that needed finish fit of screw and thread. I really want the Allen screw to be able to turn easily in and out while bottoming out at the needed depth.

Task 4: Tightening Washer: Buy It or Machine It

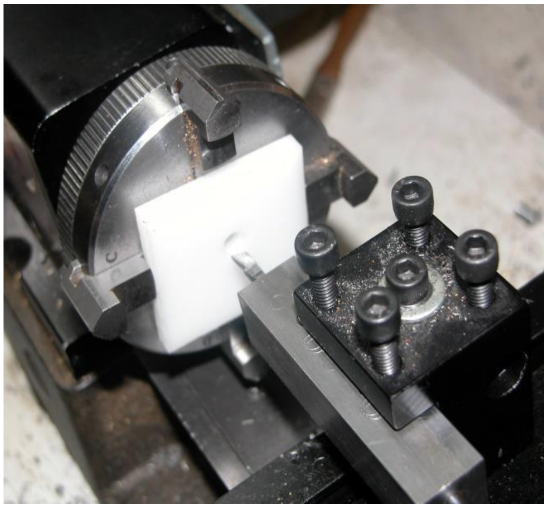
By this point, one will know whether a commercial washer is available or whether a washer needed to be made. If one needs to be made, follow the aforementioned instructions.

Task 5: Machine Backer Plate

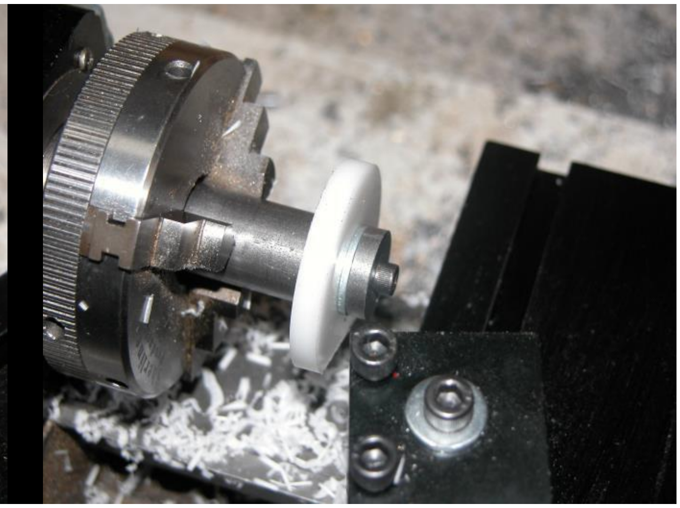
The backer plate requires two separate machining operations. 1) Its needs a center hole that will fit snugly onto the arbor step and 2) its square shape needs to be reduced to a round shape of a diameter that extends just to the tooth root. The finish size of the center hole will be achieved using a boring tool. The boring tool removes twice the material of the reading. It is like the lathe turning operation and thus one needs to plan accordingly.

Step 1: Fit up the square blank into the 4 jaw chuck, having already reversed its jaws and secure the square blank. Then fit up the 4 jaw chuck onto the lathe. Spot the center hole using the appropriate size center drill (usually at least a # 1). You could then either follow this with a drill, approximately .020" undersize the center hole target size and drill the hole, or jump right into boing the hole. If, after using a drill, remove the drill from the tail stock and fit up the boring bar into the tool holder, making sure it is straight. If the boring bar does not enter the hole straight, you will lose control of the cut as both the boring bar's cutting tip and its side will be making contact with the sides of the hole and removing more material than desired. Double check the tool post to ensure there is no metal shavings or debris between the boring bar tool and the inside of tool post, again to ensure it is straight. Position the tool post so that it sets square on the saddle. This is a good place to use one of the smaller engineering square. Bring it against the tool post and saddle to ensure being square. Bring the boring tool up to the side of the drilled hole and adjust the saddle so that the boring tool is just touching the side. Then begin a process of lightly removing material until the target size is reached remembering that one is removing twice the material of the reading. The center hole diameter is checked frequently with the gage pin until the target size has been reached. A reminder that the target diameter was determined earlier when machining the arbor step diameter.

Step 2: Now that the square blank has a center hole, it can be mounted onto the arbor step and machined round. The finish diameter needs to be as near to the root of the tooth as is practical. To firmly secure the backer plate onto the arbor step, note that a spacer will be needed as illustrated below. This spacer temporarily takes the place of the wheel blank and allows the tightening washer to do its job.



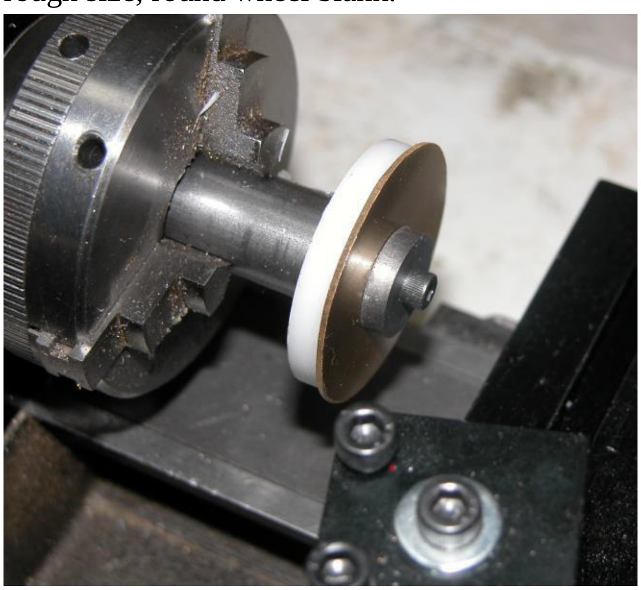
Square backer plate being (Delrin plastic) bored to size



Backer plate machined round. Note the additional washer being used as a spacer.

Task 6: Machine the wheel blank:

- The wheel blank can be arrived at by different means. Rough size wheel blanks can be purchased from clockmaker supply houses that are already spoked and machined for the center hole. They can be cut out of brass sheets or can be purchased in the rough round from certain suppliers. To date, my preference has been to buy the wheel blanks in the rough round from Ian Cobb of England. It is nice, CZ120 engraving brass to work with and I am not spending time sawing and breaking saw blades to create a round and my guess is, that usually, the purchased blanks that are already spoked with a center hole will not have the right size center hole. While this can be countered with machining a collet, etc, well you get my picture. I'd rather spend my time jumping right into machining the wheel. Anyhow, the steps to machine the wheel blank will start with assuming one has a rough size, round wheel blank.



Wheel blank setup with backer plate. Ready to machine the wheel blank's outside diameter.

- As part of machining the wheel blank is the machining of a concentric, right size center hole. In my humble opinion, this is best achieved with a boring bar. Some may consider the reamer tool as an alternative but in this instance I would disagree. A reamer serves its purpose, but a boring bar provides the ultimate in control over the size of a hole. A discussion is included below on the making and using of boring bars to assist first timers. A reminder that the boring bar tool removes twice the material of the reading thus one needs to plan accordingly. As a further reminder, the wheel center hole was determined earlier when machining the arbor step.

- Measuring progress when boring is best accomplished with gage pins as calipers are not accurate enough to provide a reliable finish measurement. On the other hand, pin gages are highly accurate and are available in sets of .001" increments. The caliper ID jaws can be used to provide one a rough idea of the size, but for absolute accuracy rely on pin gages. The measuring method is the same progressive trial and error process as described above for measuring the tooth space. Except in this instance the gage pins are passed thru the center hole until a gage pin will not go thru the center hole. Then choose the gage pin diameter just before it, as it is the correct size of the hole

A quality gage pin set: .061" - .250". I would not recommend buying gage pin sets designated as imports. I did so with my first purchase and ended up returning them as their quality was not satisfactory. Recognize also that a full set of drills, while they sometimes can be a hole measuring option as well, have limitations in that they lack every .001" measurement. Remember also that in a pinch, you can always make a gage pin. A much better alternative than trying to bore an accurate hole using calipers to measure progress.

Boring Bar Tools: Buying, Making and Using:

- Boring is an internal turning operation used to make holes bigger. Single-point cutting tools i.e. boring bars are the most commonly used tools. The boring process removes twice the material of the reading, similar to the external turning process. Micro boring bars can be purchased with 1/16" and 1/8" shanks from the usual machinist supply houses. They can be purchased singularly or in sets. I purchased a Borite HSS micro set to round out my tools from "All American Made Tools.com". As described below, I also went down the homemade route.

Borite HSS micro boring bar set from 1/16" – ¼". Solid carbide is deemed a better boring bar tool as it is more rigid than HSS but it exceeded my pocketbook ☺.

- Mastering the use of the boring bar and even making a homemade boring bar are both fruitful endeavors. One never knows when those skills will come in handy in home shop machining. For those who wish to give it a try I will share my experience. First some key points:
    - Boring Bar
        - When setup to cut, the boring bar's finished cutting tip must be at the center height of the lathe.
        - Material choices
            - Grinding the correct relief and clearances is easiest with HSS round stock. Blue pivot steel, HSS drill blank stock and worn out HSS drills all work nicely. Even worn out HSS end mills are used by some. I chose blue pivot steel for my boring bars. Diameter is whatever you need it to be. I made two boring bars. One was .125" (1/8") stock and the other was .099" stock.

        - However, using round stock does require a means of securing. You can follow JK's advice and bore a hole in a Sherline tool post or you can use steel squares for holders. I used steel square stock to experience the nuances associated with machining a boring bar holder and, as described below, to give me a grinding jig. That said, JK's method described herein is much more straightforward, and easier for that matter, as you avoid altogether making the separate holder.
        - Using ¼" square lathe cutter bits (M2, M46, etc.) for a boring bar is an option as well, but it does involve considerable effort to grind the relief and clearances. Some home shop machinists consider this much effort only for masochists ☺. Certainly not for me. Though, its advantage is that it does not require a holder. It becomes its own holder and will fit nicely in a ¼" tool holder and be at the lathe's center height. Sherline sells such a boring bar, and it worked fine for me. Though it's ¼" wide cutting tip is of no help when it comes to holes requiring a smaller diameter.

    - Boring Bar Holder
        - 12L14 mild steel squares make for suitable material for the holder.
        - Recognize that the location of the thru hole will be eccentric (see picture below). Given the need for the boring bar's cutting tip to be at the lathe's center height (this is essentially the top of the boring bar) the thru hole ends up needing to be eccentric. To resolve, I used "oversized" square stock. In the Sherline tool post that normally takes ¼", I used 7/16" squares. In the Sherline tool post that normally takes 3/8", I used 9/16" squares. In both instances, this leaves sufficient material for both a thru hole and a rigid result. Both were cut to lengths of approximately 2" , secured in the micro milling vise and their ends were squared off/faced off with the micro mill.
        - Another suggestion and option from Jerry is to use the basic Sherline ¼" tool post and to drill the thru hole on its blank side (side opposite the tool holder) and then drill and tap the top of the tool post for two set screws.

- Sequence that worked for me
    - Machine the tool holder.
        - Secure the oversized steel square in the tool post, insert the drill in the headstock of the same size as the round stock and drill a thru hole.
        - Fit up the square into your mill or drill press. Drill and tap holes for a set screw. In this instance I used two #6-40 x 1/8" socket head screws.

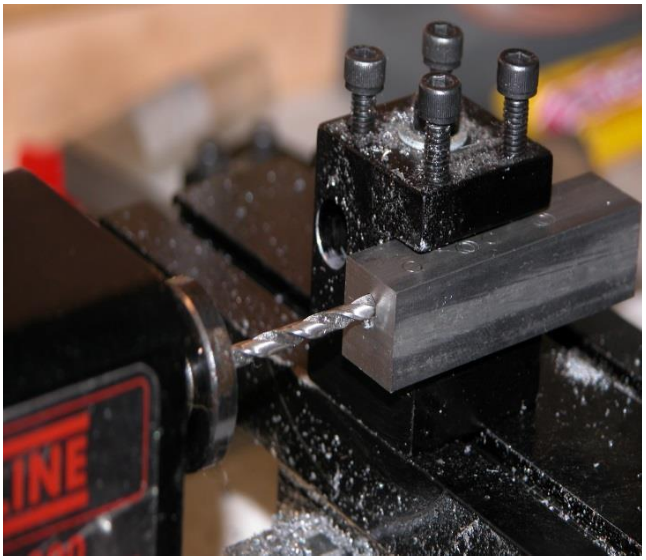
- Grind the reliefs and clearances in the round stock
  - Using the finished tool holder as a grinding jig, insert the round stock, tighten the set screws and grind the relief and clearances as described below.

  - Grind the end of the bar to a half round and insert the holder into the tool post and compare the cutting tip to a male center inserted into the Sherline headstock to assess whether it is at the lathe's center height. Repeat grinding and checking as necessary to achieve the needed result. Some final, judicious grinding may be required.

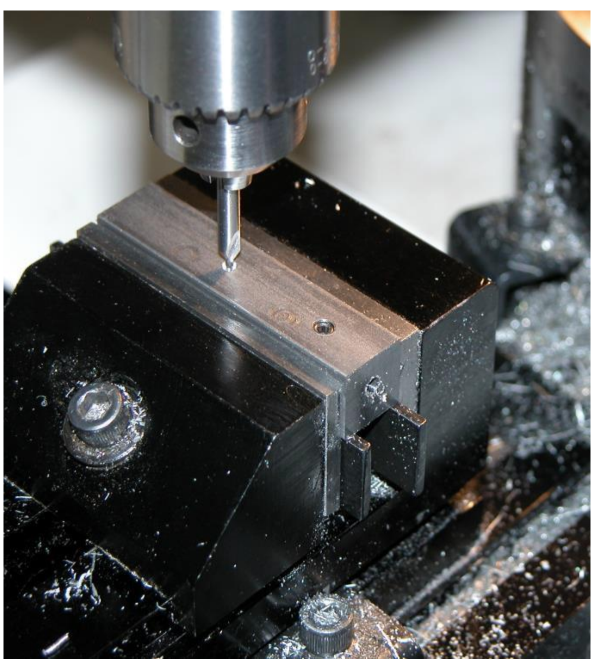
  - Once the cutting tip is ground to the lathe's center height, then grind the compound clearance and relief angle on the front of the boring bar. See JK's pictures below.
    - Note, it is at this point that one could grind either a right hand or a left hand boring bar.
      - Right hand boring bar cuts the front side of the hole i.e. that side nearest the operator. When choosing to cut the front side, the headstock, chuck and workpiece are revolving counter clockwise i.e. the normal turning position.
      - Left hand boring bar cuts the back side and requires the headstock, chuck and work piece to be revolving clockwise i.e. in reverse.
        - Some home shop machinists prefer to bore with the lathe headstock operating in reverse because it makes it easier to see what's happening inside the hole being bored. I opted for the right hand boring bar approach.
        - A caution associated with the lathe chuck operating in reverse. For a lathe chuck that is threaded on rather than bolted on, it could spin off its spindle during a machining operation. Thus a possible safety issue. So light cuts are in order. The use of collets and drawbar will mitigate this effect.

- I recommend as a final step to hone the boring bar on an oiled Norton India stone. Removes any burrs and ensures a nice smooth surface on the interior walls of the bored hole.

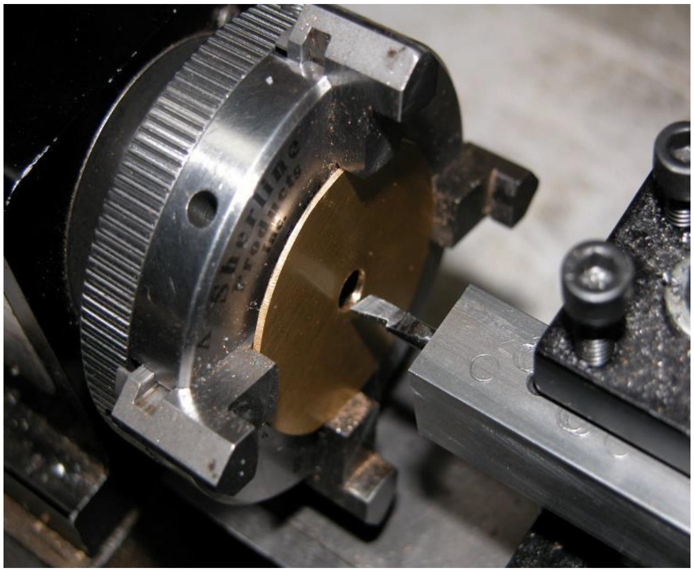
- After you think all is correct, try out the homemade boring bar and its holder on a junk wheel or wheel blank to verify and correct by grinding as necessary.
    - Ultimately, as stated above, achieving center height with the cutting tip may require some final, judicious grinding.



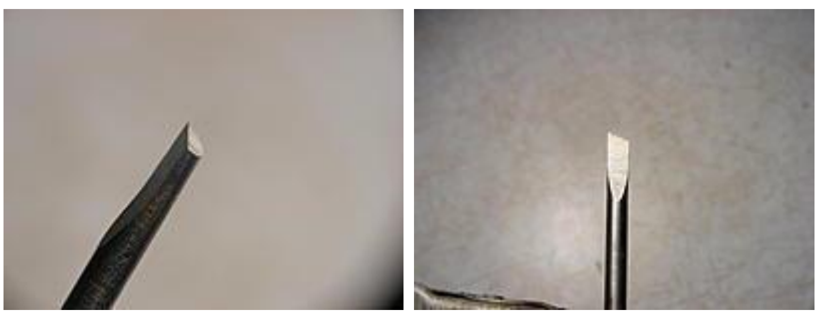
Making home made boring bar holder. Using an oversized square 12L14 material, drill the thru hole in the square at lathe center height. Notice the hole shows off center though it is at the center height of the lathe. The oversized square compensates for this off center position by leaving plenty of material to work with. The hole size should be the same size as the round rod (in this instance, .125” for 1/8” blue pivot steel stock) that will become the boring bar.



The oversized square with its thru hole drilled is then fit up into the micro machinist vice on the Sherline mill to drill and tap for the set screws. Center the drill bit using the same centering technique described below to center the cutter.



Homemade right hand boring bar and its holder at work boring the wheel blank's center hole. Note its positioning at the front side of the hole. Notice how the side requires relief to provide clearance. Notice also that Sherline's self-centering 4 jaw chucks make fine wheel chucks. Saves the search for a good watchmaker bezel chucks, plus new Sherline chucks are about 1/3 the cost. "In those instances where absolute accuracy is required, exchange the hardened jaws with Sherline's soft jaws and using a boring bar, bore a shallow pocket in the soft jaws just deep enough to hold a wheel" (JK). This securely seats the wheel blank, however, being this is an interrupted cut, a sturdy boring bar is recommended…at least ¼".



JK means and method for a homemade boring bar: "I grind a quick bar from quality drill bit shanks as follows. I first grind the shank to a half round as shown in the first photo. I then grind front width and depth relief as shown in the second photo. The round shank surface already provides ID surface relief. These bars work very well for brass, aluminum and mild steels. A #80 drill ground in this manner can be used to machine holes down to .015". They can be ground right or left". Pictures courtesy Jerry Kieffer.

- Once the center hole has been bored to the right size, the wheel blank is mounted on the arbor and machined to the finish size OD.

Machining the Wheel Blank (assumes one will be boring the center hole):
Step 1: Fit up the rough wheel blank into a bezel chuck or the 4 jaw chuck, having already reversed its jaws and secure it.

Step 2: Fit up the chuck onto the lathe and spot the center hole using an appropriate sized center drill. At this point, again refer to your notes for the target diameter of the wheel's center hole.

Step 3: Fit up a boring tool into the tool holder, making sure it is straight and the tool post is straight. Again, a good time to use an engineers square to ensure the tool post is square to the work (saddle) or find a way "by eye" to ensure it is square to the work.

Step 4: Bring the boring bar towards the wall and just touch the wall…you will hear the faint noise of the cutting sound. Turn the lathe motor off and bring the boring bar out, zero the hand wheel then turn it .005" or so. Turn the lathe motor on and make the first cut remembering that the boring tool is removing twice the .005" reading or .010". Boring is like lathe turning that way. Ok to take a first reading with the dial caliper's inside jaws. Progress and finish readings will require gage pins. Remember also, that once the boring bar is thru the hole, to turn off the lathe before backing out the boring tool. Otherwise, when backing out more material than desired will be removed and you'll lose control of the cut. Always turn the lathe off in between passes. Sneak up on the final target size by removing only .002" to .004" per pass in between checking the diameter with the gag pin. This will ensure a snug fit on the arbor and the needed concentricity. The beauty of machining with a boring bar and measuring with gage pins is you can hit your target diameter precisely.

Step 5: With the wheel blank center hole at finish size and the arbor with its machined step secured into the 3 jaw chuck:

a. Fit up the backer plate onto arbor step first. This will position it underneath the wheel.
b. Fit up the wheel blank next onto arbor step
c. Fit up tightening washer and Allen screw and tighten with Allen wrench until very secure. Try to twist the wheel blank to ensure it is securely tightened. If the wheel blank moves when trying to twist it, the step length may need to be shortened.
d. Fit up all onto lathe and turn the wheel blank to the finish OD. This is a critical size and needs to be exact or within .001". Thus, careful precision machining is again the order of the day. Leave the chuck and arbor on the lathe until the single point cutter has been machined. This will keep it nicely out of the way!

<u>To summarize the sequence:</u> machine the arbor and its step; machine the backer plate and then machine wheel blank each with their finished center holes. Tightening washers should be batch made and in stock. Assemble all onto the arbor. Try to twist the wheel blank to ensure its securely tightened. You really don't want the wheel blank moving on you while cutting the wheel as you will likely spoil the wheel.

Task 7: Machine single point cutter

The steps associated with machining a single point cutter have been handled with my creation of the “Wheel Cutting Worksheet” a/k/a “Calculating Work Point Offsets”.

- Wheel Cutting Worksheet/Calculating Work Point Offsets: After taking JK’s 2 day course and considerable practice, I came to associate Jerry’s means and method for machining the single point cutter with the work point offsets used in CNC machining, only in “manual mode” if you will. Thus, the secondary title of this worksheet. In addition, there are several measurements, calculations to be made, numbers to carry around, if you will, and one needs a place to hold them for reference. What better place than a “worksheet”! This helped me to better grasp the concept Jerry was teaching. As important, it allowed me to sophisticate my process such that I could troubleshoot and resolve the accumulation of errors that had been consistently occurring. Ultimately it helped me to achieve good repeatable outcomes. An example of a worksheet is reproduced below. You’ll note it excludes the more straightforward first and second cuts.

Wheel Cutting Worksheet/Calculating Work Point Offsets

Machining Sequence: Zero end mill @ front relief; TURN OFF MILL BETWEEN CUTS; Traverse end mill along Front Facing Side Distance A.1; Traverse end mill until contact is made with Front Facing Side to establish zero point; Zero Handwheel; Traverse end mill Distance B; Traverse end mill back along Front Facing Side Distance A.2; Traverse end mill across front relief; Repeat with Back Facing Side and traverse end mill Distances A.1, C & A.2. Repeat sequence for Distances D.1, E, F & D.2.

Base Measurements:	Example	
Tooth Space (Gage Pin)	0.048	
Tooth Radius (Gage Pin)	0.043	
Nearest End Mill Size	0.043	
Outside Diameter (Dial Calipers)	1.373	Good Tooth Tip: Good Tooth Tip
Less Pitch Diameter (Dial Calipers)	1.300	Root: Good Tooth Tip
Tooth Height	0.073	

44	Number of Teeth
34	Diametrical Pitch (# Teeth/PD)
0.75	Module (25.4/DP)

Machine Third Cuts:				Handwheel Turns Section:		
Finish Length- Fixed length .250" Distance A.1 & A.2		0.250 (Finish Length) Distance A	0.050 (Divided by Ttl Gradations)	5.00 (Length ./. Gradations) 0.050	5 (Hand wheel Turns)	(+ Gradations)
Finish Width (2 Teeth + 1 Space = 3) (Width of tooth base same size as tooth space)	3	0.048 (Tooth Space)	0.144 (Finish Width)	0 (Gradations)	Distance A.1 (Front to Back) "X" Handwheel Counterclockwise Distance A.2 (Back to Front) "X" Handwheel Clockwise	
Width Each Side	0.250 (1/4" Square)	0.144 (Minus Finish Width)	0.106 (Ttl Matl to Remove)	0.0530 (Material Removed ./. 2 from each side)	0.053 (Round Down)	
Machine Front Facing Side Distance B "Y" Handwheel Counterclockwise	**0.053** Distance B					
Machine Back Facing Side Distance C "Y" Handweel Clockwise	**0.053** Distance C		0.140 (Third Cut Reading Bumping Factor)			

Machine Fourth Cuts:

Finish Length: Tooth Height
Distance D.1 & D.2

0.073	0.050	1.46
(Tooth Height) Distance D	(Divided by Ttl Gradations)	(Height ./. Gradations)

Handwheel Turns Section:

1.46	
(Height ./. Gradations)	
1	23
Handwheel Turns	+ Gradations
0.46	
0.050	
0.023	
(Gradations)	

Distance D.1 (Front to Back)
"X" Handwheel Counterclockwise

Distance D.2 (Back to Front)
"X" Handwheel Clockwise

Theoretical Tooth Height for Reference or Use:

2.157 (factor)	./.	DP	= Tooth Height
2.157		34	0.064

Finish Width
Same size as Tooth Space

0.140	0.048	0.092	0.0460	0.002	0.0440
(Third Cut Reading)	(Minus Tooth Space)	(Ttl Matl to Remove)	(Material Removed ./. 2 from each side)	(Minus Bumping Factor)	(Net Material removed from each side)

0.144
Reference: Two teeth & one space

Machine Front Facing Side
Distance E
"Y" Handwheel Counterclockwise

0.044
Distance E

Final Cut Workspace (Optional):

	0.048	-0.048
(Final Reading)	(Tooth Space)	(Revised Distance F)

Machine Back Facing Side
Distance F
"Y" Handweel Clockwise

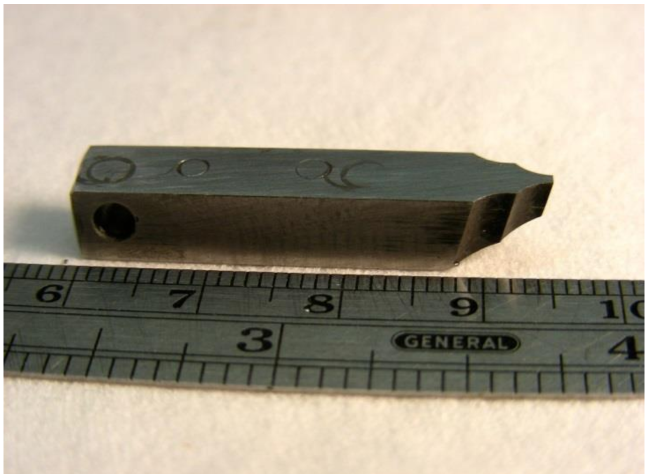
0.044
Distance F

Final Reading: Using vernier calipers measure the material remaining after the (Fourth Cut) Front Facing Side has been machined and enter in the above "Final Cut Workspace" section. Note that removing more or less than the calculated distance generally will leave the cutter tip off center. This is of no consequence as the means and method of centering the cutter tip for wheelcutting corrects for this.

Thickness: Wheel
Diameter: Wheel Center Hole

Some additional comments and instruction to provide more direction and perspective…

- The whole purpose of custom making a single point cutter is to make the perfect size cutter tip and thus be a perfect match for the original tooth. BEWARE: Arriving at the correct size cutter tip is an iterative process.



Wheel and pinion cutters do not have side relief. If they did they could not be sharpened without losing profile. Picture and observation courtesy of Jerry Keiffer.

- Ending up with the correct size cutter tip is an iterative process. The calculations to arrive at the correct size tip needs to include interim measurements. This need is driven by several factors:
  - The ¼" W-1 drill rod square stock can be + (-) .001"
  - There are four (4) bumping factors. Each of the third and fourth cuts have a bumping factor that could be anywhere from .001" to .004". Using JK's method of "bumping" the front facing and back facing sides to be at zero, while necessary to ensure one is at zero, is sometimes hard to hear, hard to detect and as a result an undesired, unknown amount material will be removed. Each time a bit of material could be removed that cumulative could account for .001" - .004".
    - Third cut: Front facing side and Back facing side
    - Fourth cut: Front facing side and Back facing side
  - Hand wheel errors.
  - Hand wheel errors while one is accounting for backlash
  - Tool deflection
  - Error measuring tooth space size. Errors here should be largely mitigated by the correct use of pin gages to measure, but until you get the hang of things, it could still be a source of error.
  - Round up or round down error in the calculations.

The accumulated errors could be as much or more than .005” which will leave a cutter tip too narrow and thus the resulting tooth space will be incorrect. One can still “save the job” as my late father in law would say…

    - The best remedy for these accumulated errors is to take interim measurements and base subsequent cuts on this new measurement. Arbitrary minus adjustments are just guesswork and just not in keeping with the precision machining mode.
        - This would be particularly good to do after the third cuts are finished. Take a reading with a micrometer, enter this new measurement onto the worksheet and recalculate the fourth work point offset cuts.
        - A final optional interim measurement can be taken after the fourth cut front facing side has been machined and the adjustment made on the back facing side. This likely will result in the cutter tip being off center but this is of no consequence as this is accounted for in the wheel cutting setup. Taking a measurement at this point may be a bit of overkill. I am thinking that this reading may not be necessary except for those very small clock wheel teeth or watch wheel teeth. Again, time and experience will be the best guide. On the other hand, it wouldn’t hurt anything to do, just a bit of extra time.

- Material Choice for cutter blanks is ¼” W-1 drill rod squares. This is a tool steel produced in the annealed condition for ease of machining. After machining, the steel is heat-treated and quenched. Purchase in 36” lengths. Cut into 12” lengths for easy storage.

- Make up a batch of cutter blanks. Cut them to length, dress them and make the first cut. It saves time to do a batch and the heavier, messy work (you’ll need liberal amounts of cutting fluid for the first cut) will already be done when one encounters a situation requiring a cutter be machined.
    - Batch them and dress them.
        - When needing to make up the cutter blanks, take a 12” length of W-1 square, measure eight 1 ½” lengths and hacksaw the eight pieces. Using the bench grinder, grind both their ends approximately square and scrub all four of their sides (S4S) on fine emery paper. This is not really necessary to machine a cutter blank, but is good workmanship, leaves one with a nicely dressed blank and makes them nicer to handle.

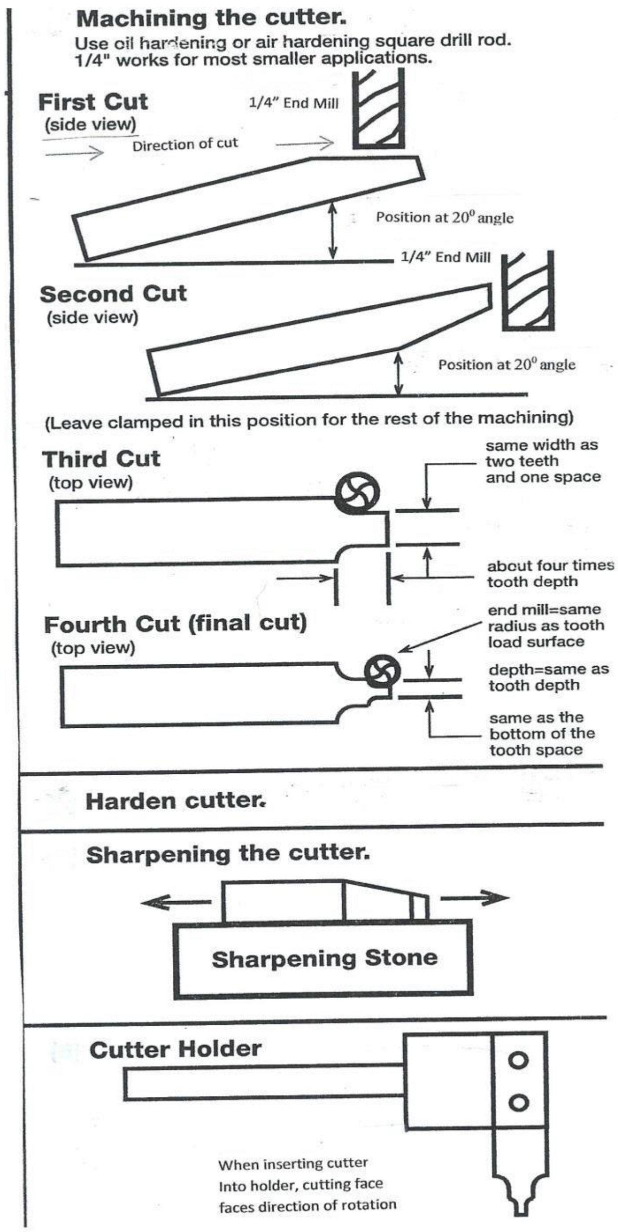
Batch them and dress them

- Important points when machining the single point cutter
  - Ensure one is at "zero" prior to start of machining. This is key for precision machine cuts.
  - On the first cut progress the table right to left, bringing the cutter blank under the end mill.
  - The second cut is a cross cut and machines the front relief. Take light cuts and make several passes to obtain the squared up result. One can observe the progress as the end mill makes its way up the front by noticing the freshly cut surfaces are shiny and the surfaces yet to be cut are dull.
    - Note there is a second front relief cut as part of the fourth cuts.
  - Take the third and fourth cuts all at once. Note that each of these cuts includes machining the front facing side and the back facing side.
  - Check and recheck measurements. This allows for precision machining.

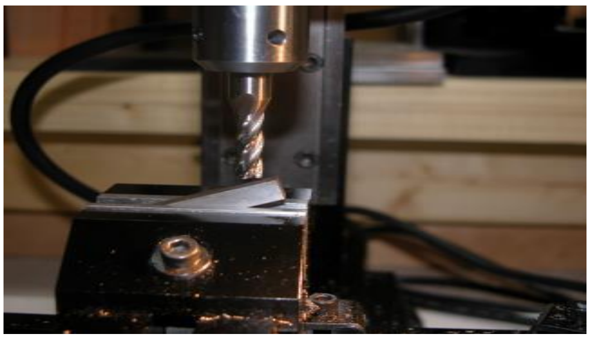
- **Always turn the milling machine off in between cuts.** Returning the cutter blank past the end mill after a cut while the end mill is still running will remove material and consequently you will lose control of the cut. More material will be removed than desired.

- **Move the hand wheel the exact calculated amount.** Moving more than this will remove more material than desired, or will move the end mill beyond the intended work point offset and lose your point of reference.

- To better demonstrate the sequence for preparing a cutter, JK's class sketch is reproduced below. It includes the sequence of cuts (First Cut thru Fourth Cuts) along with the hardening, sharpening and securing in cutter holder. Following this are my experiences.

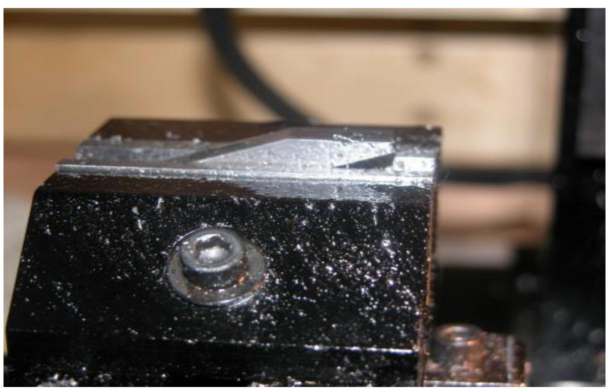


- **Machine first cut (see picture).**
  - This involves positioning the cutter blank approximately in the middle of the machinist vise (see picture) at a 20^0 angle and then tighten the vise very snugly. It is a good habit to be accurate with the 20^0 angle and can be readily achieved by using a protractor and a ruler to extend the angle as a guide. The same angle will be needed for the second, third and fourth cuts. Done in this way will allow one to remove the cutter blank from the vise for close inspection and trying in the wheel and returning the cutter blank at or nearly at the same position if further machining is needed.


Blank secured in vise and ready to make the first pass of the first cut

    - Choice of End Mill: ¼" stub end mill is the cutting tool of choice seated into a ¼" collet.
    - Tip: Positioning the end mill:
      There is a need to properly position the end mills before starting their cuts, otherwise one risks cutting into the machinist vise. One of the indications of a good machinist is one who does not ruin the tools! That said the trick to avoid making contact with the either vise or its vice jaw/jaw inserts while machining is to use the right limiter, right point of reference…In this instance, bring the end mill down via the "z" axis so that the end mill touches the tops of the jaw inserts and not the jaws themselves and then bring the "z" axis up .050". Together this will provide sufficient clearance. The tops of the jaw inserts stands proud of the jaws and the end mill needs to be able to clear the inserts when it is traversed from one work point to another. Precision machining at its finest!

- Machining direction: Progress table right to left. Once end mill is positioned over the back of the cutter blank, as pictured above, turn the mill table (x handwheel) clockwise to move the table to the left. This results in the workpiece (in this instance the cutter blank) progressing under the end mill. Always return the end mill to the back before removing more material. Then turn the Z axis down .010" and repeat the sequence.
- Machine RPM, Machining feed rate, material removal and cutting oil: Set machine RPM at approx. 1000 RPM, Remove .010" per pass, Apply lots of cutting fluid and use a moderate feed rate.
- Total material to remove: Remove approximately 2/3 or .167" of the material, leaving 1/3 or .825". Squares are .250" - .167" = .825". Progress can be measured with the dial calipers. This is not a critical size, it can be off .005" - .010", but it is good workmanship to be close.


First cut finished

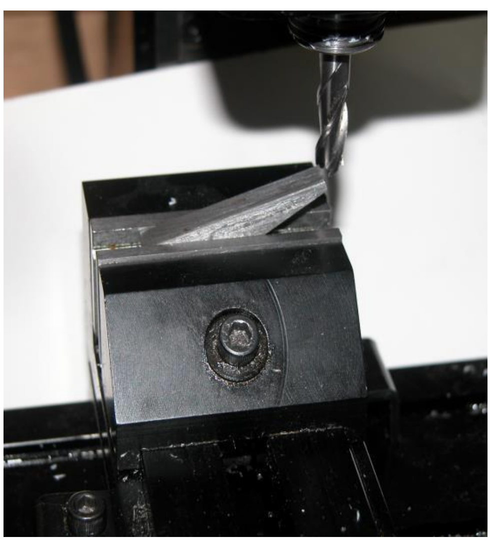
- Loosen the vise and flip the cutter blank over for the remainder of the cuts and secure again in vise at the 20º angle

In position and ready to machine the remainder of the cuts
(front relief second cut, third cut, and fourth cuts).

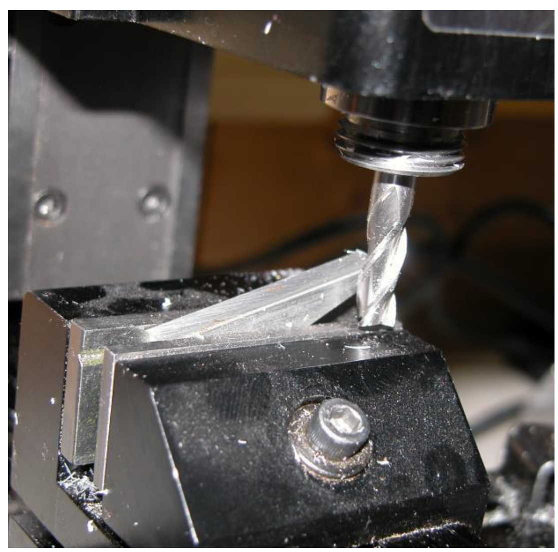
- **Machine second cut: Front relief cut**
    - ¼” stub end mill is used
    - Material removal: .005” per pass
    - Machine RPM: 1,000 RPM (approx.)
    - Feed rate: slow
    - Establish zero position for the front relief position…Zero the handwheel, then make the final pass. Once the front relief is machined square, loosen the resettable hand wheel and mark it .003” and then make the final pass to arrive at “0”.
        - JK’s method of ensuring one is at Zero is one of his keys to successful precision machining. He established machining steps to doubly ensure the end mill was at Zero in the front relief position to provide an accurate point of reference to continue with the work point offset machining required for the front facing and back facing sides.

- **Machine third cuts: Front relief, front facing side and back facing side**
  - Non-critical cut, but it is good workmanship to end on the desired calculated points.
  - After machining several cutters, instead of following JK's guidance "four times the tooth depth" for length, it became easier to standardize on a fixed length. Thus, I opted to go to a fixed length of .250" for the third cut for all cutters. The worksheet is designed with this change.
  - ¼" stub end mill is used
  - Material removal: Remove full amount of material in one pass
  - Machine RPM: 1,000 RPM (approx.)
  - Feed rate: slow
  - Bump each side to establish zero point. Follow above instructions to zero end mill.
  - Measure the remaining material after the third cut front facing and back facing sides are finished and enter this onto the worksheet. This will pick up accumulated errors. The worksheet will then calculate a new work point offset distance for the fourth and final cuts which are critical.

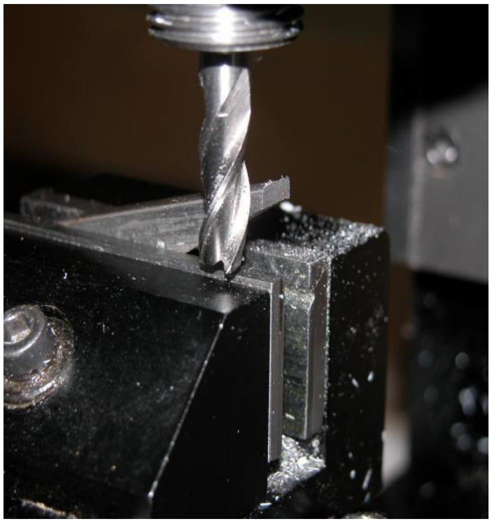
- Below pictures depict the sequence for machining the third cut. The narrative that accompanies each picture follows along with the instructions as shown on the "Wheel Cutting Worksheet/Calculating Work Point Offsets".



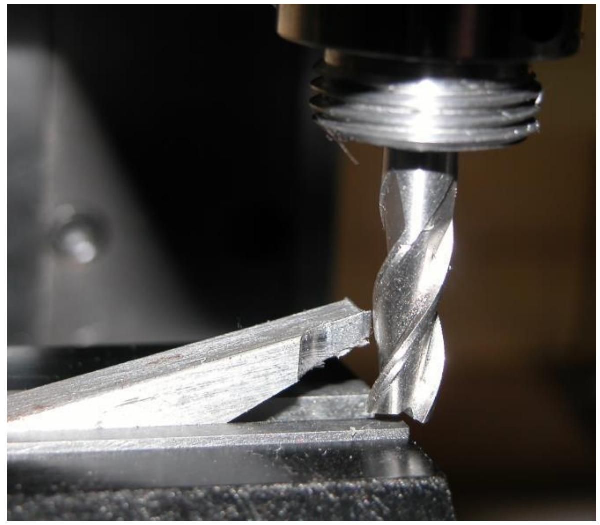
End mill is positioned to machine front relief square. Once front relief is machined square, establish zero position for the front relief as described above. Milling machine is then turned off.



Next, traverse end mill along front facing side Distance A.1.

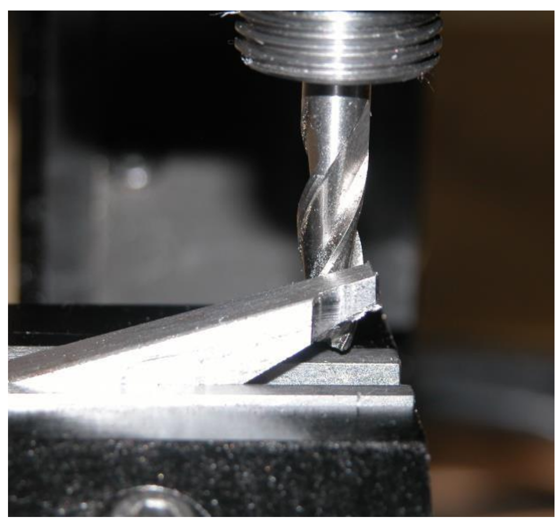


Turn milling machine on and traverse end mill into side of cutter blank until contact is made with front facing side i.e. JK's bumping technique to locate zero. Handwheel is zeroed. End mill is then traversed Distance B into the side of the cutter blank.

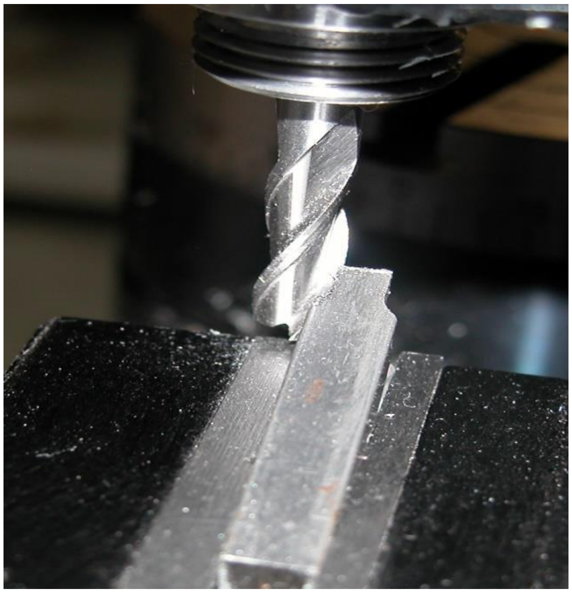


End mill is then traversed Distance A.2 back along front facing side to remove material. Again, turn off milling machine in between cuts.

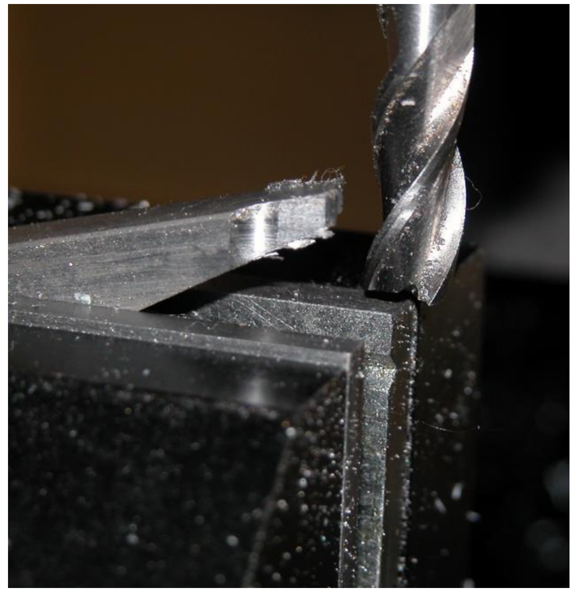
Traverse end mill across front relief. Traverse end mill along the back facing side (front to back) Distance A.1.



Turn on mill and traverse end mill into the back side of the cutter blank until contact is made with back facing side i.e. JK's bumping technique to locate zero. Handwheel is zeroed



End mill is then traversed into the side of the cutter Distance C.



End mill is then traversed distance A.2 along the side back to front. The third cut is now finished. Next, take a new reading of the cutter blank's just machined end with Vernier Calipers. Record that new reading onto the worksheet and new work point offsets will be calculated for the fourth cut. This step will account for any errors that have accumulated up to this point.

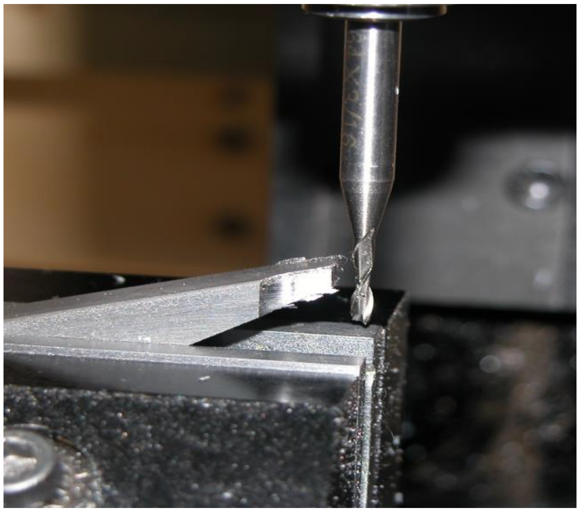
- **Machine fourth cuts: Front relief, front facing side and back facing side**
  - Critical cuts using the end mills size that will replicate the tooth radius. The use of the smaller diameter end mills and climbing cuts will require more careful positioning and a slower feed rate. Repeat, this is a critical cut. The resulting material that is left is in fact the cutter tip and tooth curve i.e. radius. Any shape other than the calculated shape will not match the original tooth.

  - Fourth cuts will require changing out the ¼" end mill for the smaller diameter end mill and therefore one will need to re-establish its "limiter" position. Repeat above positioning tip to establish the top of the jaw inserts as a limiter. Position end mill in the collet slightly above its flare. This is the strongest section of the end mill's cutting surface. Sideways cutting forces are significant.

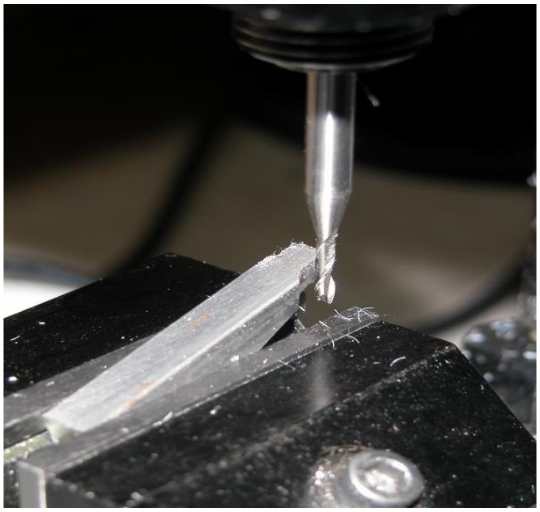
  - Front relief
    - Material removal: .002" - .004" per pass
    - Machine RPM: 1,000 RPM (approx.)
    - Feed rate: slow
    - Establish zero position at front relief position…Zero the handwheel, then make the final pass.Once the front relief is machined square, loosen the resettable hand wheel and mark it .003" and then make the final pass to arrive at "0".

  - Front facing side and back facing side
    - Material removal: Remove full amount of material in one pass
    - Machine RPM: 1,000 RPM (approx.)
    - Feed rate: slow
    - Bump each side to establish zero point

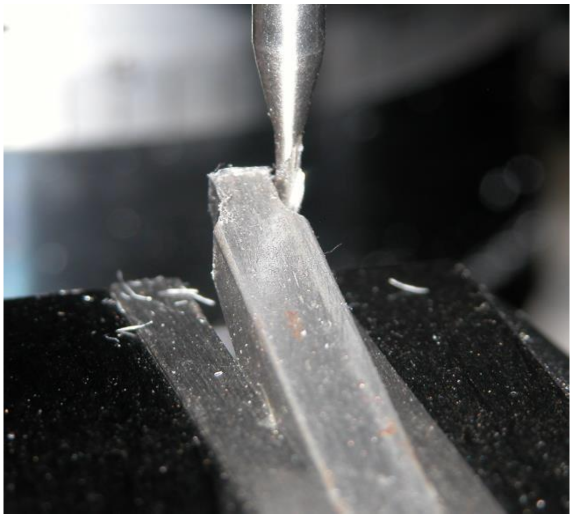
- Below pictures depict the sequence for machining the fourth cuts. The narrative that accompanies each picture follows along with the instructions as shown on the "Wheel Cutting Worksheet/Calculating Work Point Offsets".



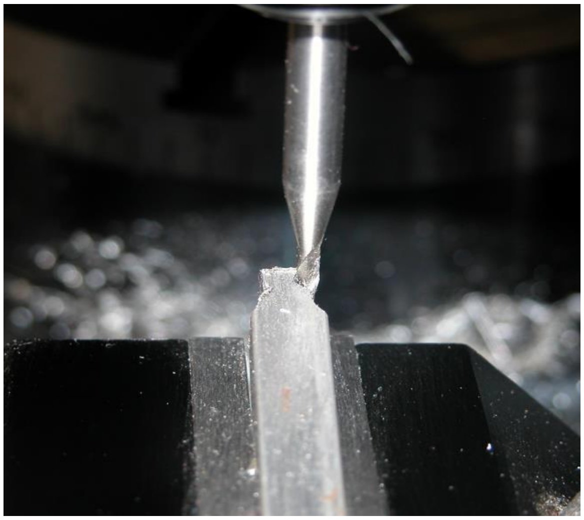
The ¼" end mill has been replaced with the end mill that will machine the tooth radius. This is a critical cut and starts with re-establishing the bottom of the end mill for clearance and then progresses to machining the front relief square and then to zero the end mill.



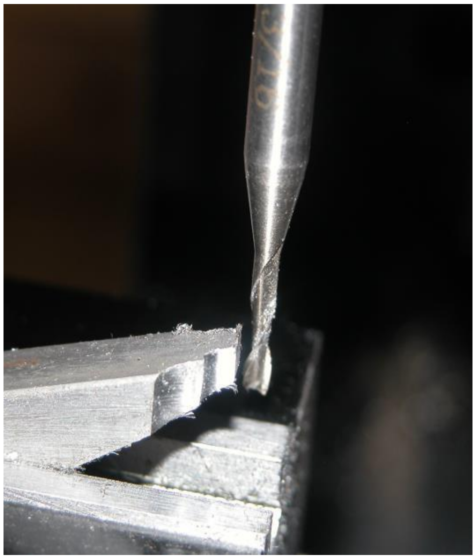
End mill is then traversed along front facing side (front to back) Distance D.1.



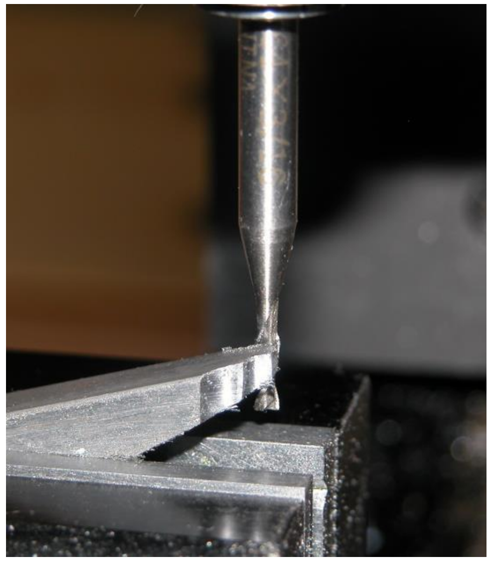
Turn milling machine on and traverse end mill into the side of cutter blank until contact is made with front facing side.
As before, this is JK's bumping technique to locate zero.
Handwheel is then zeroed.



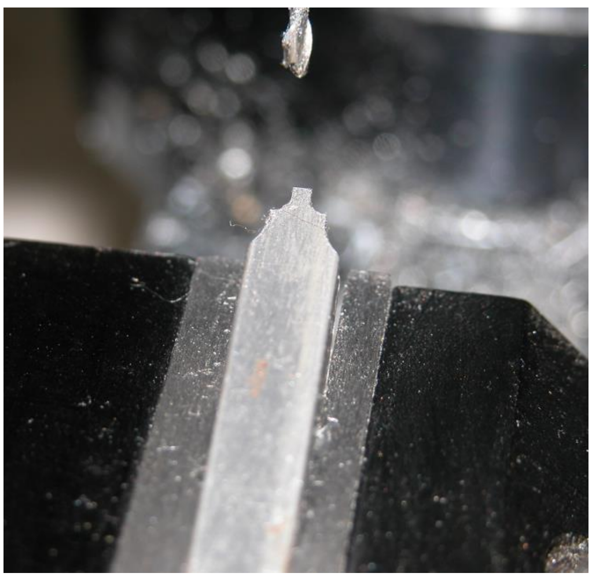
End mill is then traversed Distance E into the side of the cutter



Traverse end mill Distance D.2 along front facing side (back to front) to remove material. Again, zero hand wheel and turn off milling machine between cuts.



Traverse end mill across the front relief and then to the back facing side Distance D.1.(front to back). Traverse end mill to contact the back facing side and zero the handwheel. Traverse the end mill into the side of the cutter Distance F. Finally, traverse end mill along the back facing side (back to front) Distance D.2 to remove material.



The finished product! Well almost…

Notice that the tooth spaces have round bottoms. The two sharp corners on the cutter's front relief's needed to be stoned and rounded to complete the fit. Now we are done ☺.

JK Tip: Round Bottom Tooth Spaces:

Above instructions will leave one with a square edge cutter tip which is perfect for many of the square bottom tooth spaces one will encounter. Periodically, however, one will encounter a round bottom tooth space. Another conversation with Jerry reveals his technique for handling this is to stone by hand both corners of the cutter tip's front relief so that the corners are rounded instead of sharp. While it is not possible to cover every scenario you may encounter, here are several approaches that should cover most all your situations. For most clock wheel teeth, one would stone the corners before the cutter is hardened. A good stone for this is an 8" oiled India stone as it provides a longer surface to stroke with. It is more forgiving if you will. For the best alignment results, set up to stone the entire length of each of the front relief corners even though only about 1/3 of the front relief is a cutting edge. To maintain control while stoning, place the stone on your bench and rest your hand alongside it on the bench. Then grasp the cutter firmly between your fingers, place the full length of one of the cutter's relief corners onto the stone and stroke the full length of the corner back and forth without lifting it off the stone. Repeat with the other relief corner. It doesn't take very long and, of course, periodically check the fit between the cutter tip and the tooth space. Repeat each of the strokes as necessary. An alternate method, for those good with their hand-eye coordination, is to hold the cutter in one hand and the stone in the other hand and just stroke. If one is machining a larger round bottom tooth form and a highly accurate fitting is required, you may want to stone the tip free hand before it is removed from the mill. Again, these are all done before the cutter is hardened. Then there are those occasions when you have a very small single point cutter tip. JK provides more insight for those instances... "when you have a very small single point cutter tip, it is well to remember that it is very easy to remove too much metal without even trying and that metal removal is more controllable when stoning after hardening. Consequently, in those instances stoning after hardening will greatly help control metal removal".

On the left is a cutter with a rounded tip and on right is a square cutter tip. A top down look.

Another look from the front. Note the front relief corners. the Rounded on the left cutter and square on the right cutter. For some of us, our hands are not perfect machines. So, when stoning their full length, it might be challenging to maintain constant, square surface contact. Thus, at a minimum, keep your focus on the first 1/3 of the corner to ensure its surface maintains contact with the stone as this is the key area of the form cutter.

Task 8: Harden single point cutter:

- No need to temper cutter for brass. If machining steel, one will need to temper the cutter.
- Reminder, that tempering removes the brittleness of material while toughening the material.
- Before hardening, scrape off any burrs or fuzz thrown up from the machining with a good knife blade or similar.
- Choice of heat: propane will <u>not</u> work; MAPP gas works good; Oxy-Acetylene works best but a good job can be done with MAPP gas
- Have a large pot of water at the ready
- Hold the single point cutter at the end opposite the newly made cutter tip with a pair of Vice Grips
- Focus the MAPP gas flame at the middle of the cutter blank. Do not allow flame to touch the cutter tip as there is a chance it will burn it. After a short while the cutter tip will reach an orange color and hold it for 5 seconds and quickly quench the cutter into the pot of water.
    - Note that JK used Oxy-Acetylene at the NAWCC class and he removed the flame the second the tip became orange and quenched the cutter immediately. I believe that is because the Oxy-Acetylene burns much, much hotter and one needs to be very careful not to burn the cutter tip, particularly if one is working with a very small cutter tip such as for a watch wheel or very small clock wheel
- Again, the beauty of this method is the ease with which one can remove the carbon after hardening. Comes right off during the sharpening process.
- Special JK instructions for very small items to harden
    - Obtain stainless steel tubing of a diameter close to the size of the item needing to be hardened.
    - Obtain steel balls that will provide an interference fit for both ends of the stainless steel tubing.
    - Place the item to be hardened inside the tubing with a piece of paper about the size of the small item. When heated, this paper will burn up and consume the oxygen inside the space and ease the carburization.
    - Jam the steel balls into each end of the tubing.
    - Heat the tubing to cherry red and quench. The stainless steel tubing is a good indicator of what is also happening to the item inside. If the tubing is cherry red so is the item being hardened.
    - Cut off an end of the tubing to remove the item.
    - Material makeup of the stainless steel tubing eliminates the scale (carbon) on the inside.
- Special JK instructions for annealing small items or items
    - Place item on a piece of copper. Apply heat to the item and copper at the same time. Allow it to cool. It will cool slowly on the copper and better anneal.

Task 9: Sharpen single point cutter:

JK's modern means and methods makes for very easy sharpening and at the same time removal of the surface carbon or scaling as it sometimes is referred to. The profile of the cutter has a flat surface and thus the carbon is removed as part of the sharpening process. Oil up the 8" India stone, place the cutter flat side down onto stone, place ones fingers onto the cutter tip and it rub back and forth until the cutter's flat surface presents a clean, shiny surface. It is then sharp and ready to use. Intermix this rubbing with inspections and note any dull sections. Apply more pressure to those areas and continue to rub until all dull surfaces are gone.

Task 10: Milling machine, CNC Rotary Table (RT) and Programmer Setup

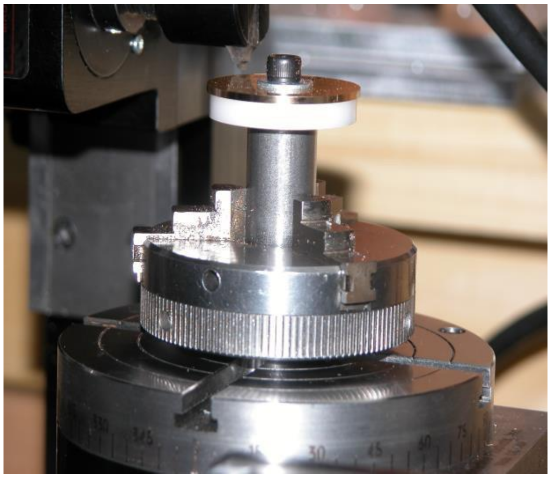
- All this time the Sherline milling machine has been in its vertical mill setup. To ready it for wheel cutting it needs to be converted to its horizontal milling machine setup. This is a very useful feature of this micro machine. As JK says, it is another reason why he loves these machine tools. Add to that he has never had to tram the milling machine. You can't say that for many milling machines. The horizontal position allows one to see exactly how the setups and wheel cutting are progressing, allows for precise adjustments, etc.

- The milling machine and Sherline CNC Rotary Table/programmable controller are modern day dreams come true for indexing wheels. It is so easy to be precise and accurate. There has been much written by the masters and amateurs alike about the use of direct indexing and universal indexing tools. This is all made moot by this modern day machine tool co-designed by Bryon Mumford of Mumford Micro Systems & Joe Martin of Sherline Products Inc.[22] As J. Malcomb Wild says, "it is the ultimate in dividing".[23]

- Milling Machine Setup
  - When wheel cutting, both the Sherline machinist vise and RT are mounted on the X table. Vise on the left and RT on the right. Both need to be indicated into their zero position and securely tightened down. For reference, I use the Starrett "Last Word" indicator.

  - Fit up the cutting arbor into the milling machine and tighten. Then fit up the single point cutter into the arbor cutter holder with its cutting face (flat face side) facing the direction of rotation. Tighten the set screws onto the cutter body very securely.

- Sherline CNC Rotary Table Setup
    - The arbor with its backer plate, wheel blank having been left tightened in the 3 jaw chuck on the lathe is now removed from the lathe and the 3 jaw chuck with arbor and wheel blank is screwed tightly onto the RT adaptor plug.



Arbor secured in 3 jaw chuck with its wheel blank and backer plate

- To tighten the chuck onto the RT, place Tommy Bar in one of the holes of the <u>upper chuck body</u> to tighten (see picture). Placing a Tommy Bar in the lower chuck body will loosen the chuck's jaws which will loosen the arbor and you will lose the concentricity.

Machinist Vise, RT & CNC programmer setup. Note this setup is for a mainspring barrel. It is the same setup for a wheel blank.

- Connect the RT plug into the Sherline CNC Controller.
- Check to be sure the CNC Controller is in the "Off" position
- Set RT table and hand wheel to "Zero"
    - Using the RT hand wheel, turn it manually until the RT table is at the "0" position.
    - Zero the RT hand wheel
- Programming the CNC Controller
    - Check to ensure the RT and hand wheel at their "zero" positions
    - Press # 9 and hold it down while turning on the controller switch. This erases the settings and prepares it for new settings.
    - Press "Mode"
    - Division Mode: "Press Enter"
    - "Press Enter"
    - Enter # teeth (Divisions) (Fill 3 numerical positions) Class example "066"
    - "Press Enter"
    - Press "Nxt" to progress the RT or "Prev" to reverse the RT.

- Understanding the Design of the Sherline Rotary Table ratios for indexing
  - Worm gear ratio is 72:1 requiring 72 turns of the hand wheel for one complete revolution of the table.
  - Hand wheel has 50 graduations: 50 graduations multiplied by 72 turns of the hand wheel = 3600 parts of a circle or said another way…3600 graduations of the hand wheel or said another way…if this was an index plate, this would be an index circle with 3600 holes
  - Some examples based on the above understandings:
    - Cutting a wheel with 60 teeth would require turning the hand wheel 60 graduations for each tooth to be cut: 3600 graduations divided by 60 teeth = 60 graduations or one full turn of the hand wheel (50 graduations)+ 10 graduations.
    - Cutting a wheel with 72 teeth would require turning the hand wheel 50 graduations for each tooth to be cut: 3600 graduations divided by 72 teeth = 50 graduations or one full turn of the hand wheel
    - Cutting a wheel with 84 teeth would require turning the hand wheel 43 graduations for each tooth to be cut: 3600 graduations divided by 84 teeth = 42.857 or 43 graduations (rounded) of the hand wheel.
    - Cutting a wheel with 56 teeth would require turning the hand wheel 64 graduations for each tooth to be cut: 3600 graduations divided by 56 teeth = 64.285 or 64 graduations (rounded) of the hand wheel or one full turn of the hand wheel (50 graduations) + 14 graduations
  - In CNC mode this would be as straightforward as entering the number of needed teeth in the "Division Mode" i.e. (60,72,84,56) for each of the above examples. Reminder that when the programmer is turned off it does not remember its settings. Always, reset to zero when turning back on and enter settings.
  - As is evident above, the Sherline rotary table can also be used in manual mode for an accurate index even though it is not near as convenient as the CNC mode.

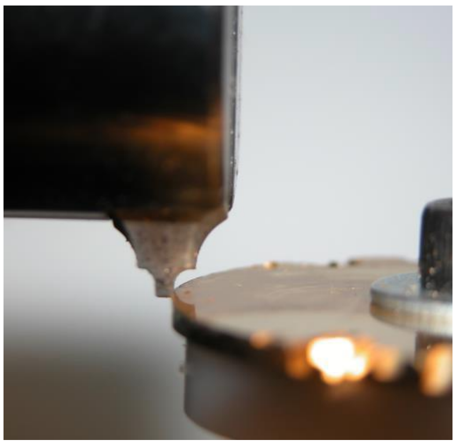
- To summarize how to operate the CNC RT and programmer:
  - Ensure cutter is adjusted for center and depth of cut; turn on CNC RT and programmer; erase previous settings; enter # of teeth; using the Z axis handwheel bring cutter thru its first full cut and then bring the cutter fully back out of the cut clearing the wheel blank; press "Next" on the programmer to progress the wheel blank to the next space and repeat

bringing the cutter thru the second space via the Z axis handwheel and then back up clearing the wheel blank. Repeat this all the way around the wheel. The programmer will maintain a constant point of reference.

Task 11: Wheel Cutting

Step 1: Position the cutter tip at the dead center of the wheel. With the Sherline milling machine in the horizontal milling, center the cutter as described below. Note that this is a critical step.

Align the cutter tip at left edge of the wheel blank as close to the halfway point of the wheel as your eyeball can judge. Using X, Y & Z hand wheels bring the cutter to the left edge of the wheel blank. Eyeball the cutter tip at the cutter tip's halfway point and just touch the wheels left edge. At this point, in effect one is at zero. Using Y hand wheel, bring the mill saddle forward to clear the wheel from the cutter's path. Zero the X hand wheel and then using the X hand wheel move the X table the Calculated Distance (see below) and then lock the table.



First step in centering the cutter…position
the cutter tip precisely at left edge of wheel blank.
Touch the blank with the cutter tip.

Calculate dead center: wheel blank OD + cutter tip diameter and then ÷ 2 = Dead Center

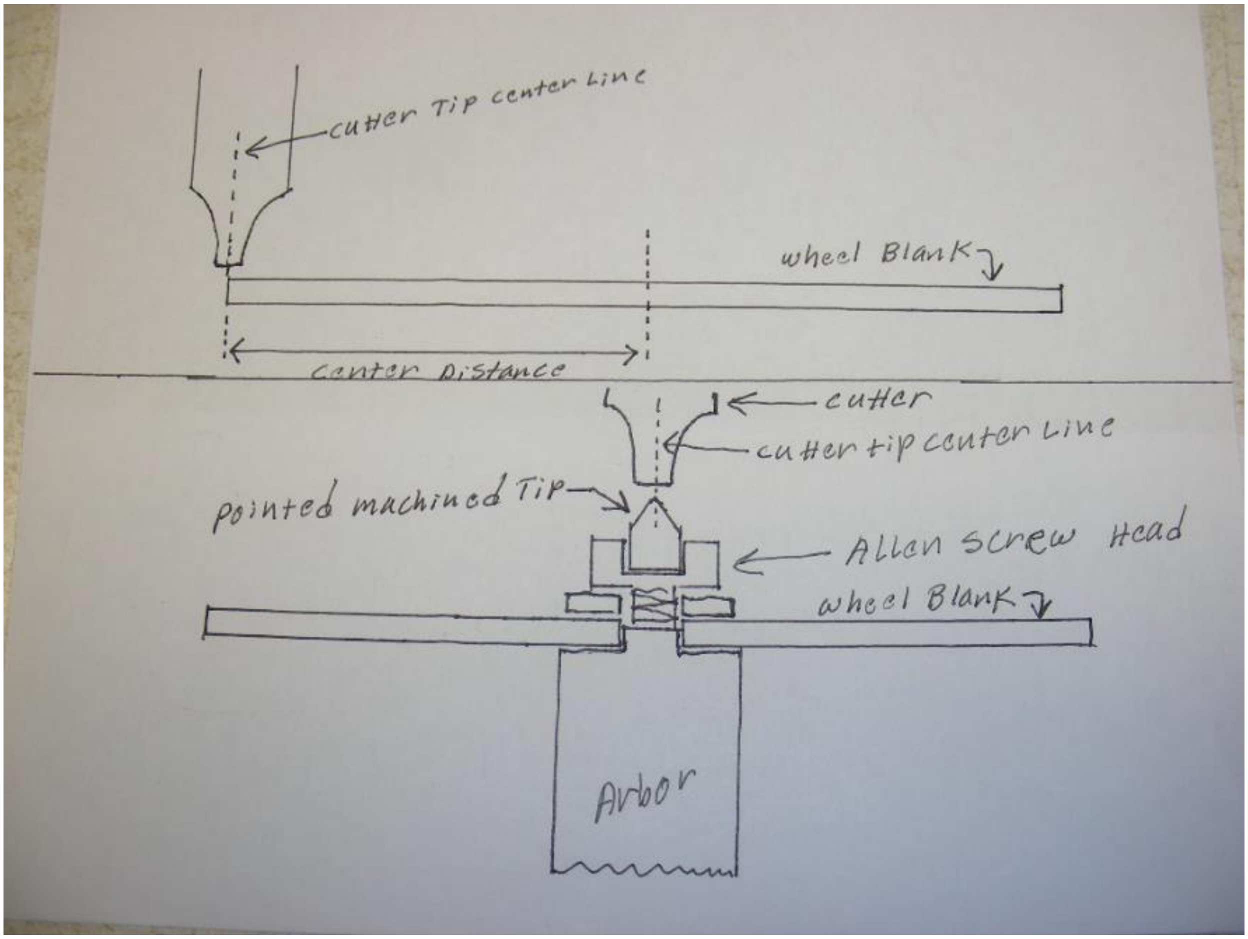
Ex. 1.375" + .042" = 1.417" ÷ 2 = .0785"

Convert calculated dead center to turns of the hand wheel. The result usually requires two steps as in this example:
Step 1: .7085" ÷.050" (one full turn of the handwheel is .050") = 14.17 turns

Step 2: Convert the fraction of the turn to gradations. In this example:

.17 x .050" = .0085" or 9 gradations (rounded up)

Calculated distance for dead center: 14 complete turns of the handwheel + 9 gradations = places the cutter tip at the dead center of wheel. Upon positioning the cutter at dead center lock the X table.

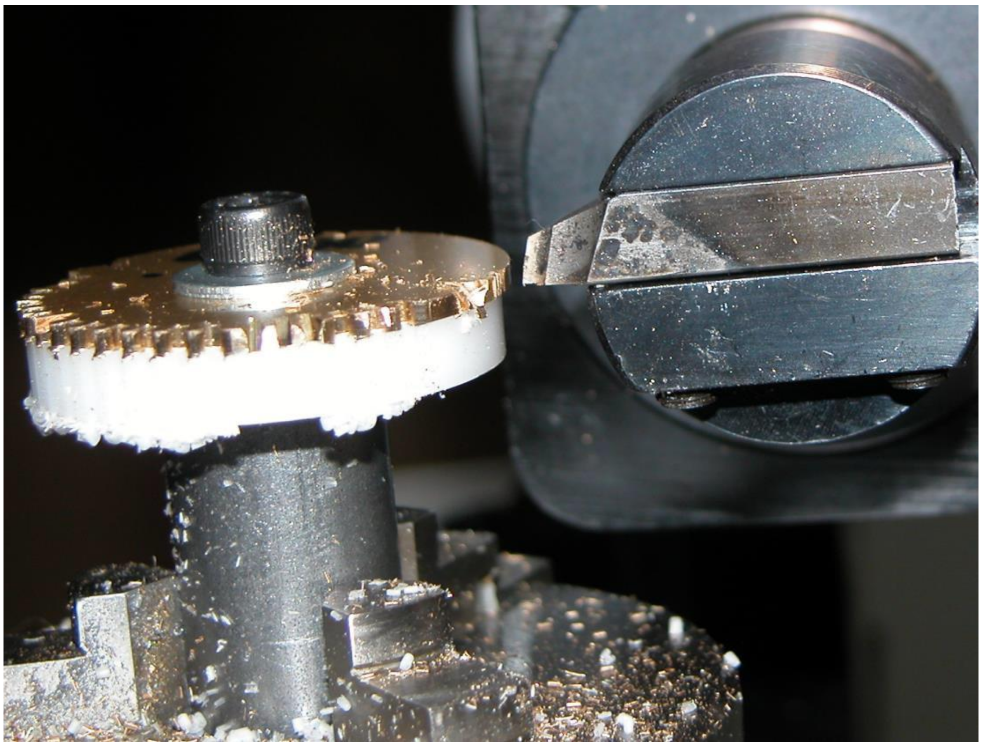


Hand drawings by Jerry K. In the top sketch he provides a visual of the dead center calculation. In the bottom sketch, he shows a way to judge "by the eye". "However, if you use an Allen screw to secure your wheel blank, you can machine a stubby little pointed center and set it in the socket cavity per the lower sketch. This will allow you to center the cutter off of the point without the math". Comments and drawings courtesy of Jerry Kieffer

You will find it helpful to understand the math as you will encounter numerous occasions to use this method of centering cutters, drills, etc. when using a milling machine.

Step 2: Cut the first tooth…Determining the final depth of cut:

A. Gain control of the cut. With the table locked, move the saddle forward using the Y handwheel enough to reposition the cutter as shown in the picture. The cutter tip needs to be at full depth pointed horizontally to the bottom of the wheel's edge i.e. bottom of the thickness of the wheel. In this position the cutter tip is on the same plane as the wheel blank. Leave cutter tip in this position.



Cutter positioned horizontally on same plane as wheel blank. This is a practice wheel setup, but it does correctly depict how the cutter tip is to be positioned to the wheel blank a/k/a on the same plane.

B. Turn mill on and with very slow feed speed bring the saddle i.e. wheel towards the cutter tip until it just starts to make contact. You will hear a faint cutting sound.

C. Using "Z" axis handwheel bring cutter all the way thru the first cut, then bring "z" axis back up and fully out of the wheel's path. Otherwise when you progress the wheel the cutter will be milling the wheel blank and ruin the wheel blank.

D. Progress the table inwards .005"/.010", bring "z" axis down thru the space. Repeat this sequence until the space begins to be formed. Being sure to bring the "z" axis back up, clearing the cutter from the wheel's path.

E. Remembering that one forms the first tooth by cutting two spaces, after being sure the cutter has cleared the wheel's path, progress the RT one space and repeat the above sequence. You'll know you are nearing the final form when the tooth top begins to take on a curved shape.

F. Knowing when the proper, final depth has been reached…"sneaking up on the depth". The first indication is when the cutter depth reaches the point that the tops of the teeth start to take on the shape of the curve. Once the curve shape appears, go in increments of .0025"/.005" and check progress both visually and with the dial calipers. The desired finish depth is when the flat on the tooth is about .005". It is a bit hard to see the flat on the tooth, thus a 10x loupe magnifier is recommended. Having the original wheel nearby so that one can compare tooth tips will certainly help. Another visual aid is to apply layout dye or permanent marker to the edge of the wheel. If need be, progress the RT around so that one can get a good look at it from the side. The CNC RT maintains a constant point of reference and makes it easy to revolve the wheel forward or backward to allow for observations, recuts, etc. with just a push of either the "Next" or "Prev" button.

All above said, JK reminds us that we are seeking to duplicate, replicate the original teeth. To that point, the original teeth may not have a small flat on top as many of the newer, later wheels oftentimes have. Consequently, this will need to be considered in arriving at the proper, final depth.

Step 3: Cutting the remaining teeth:
Once you are satisfied that the depth of cut is correct, lock the X and Y tables. Locking all the axis is a good habit to get into when machining. It minimizes backlash and potential for movement and proceed to cut the remaining teeth full depth in one cut; speed 500 RPM using the CNC RT to index the rotation.

Wheel cutting comments from W.J. Gazeley: *"Lead the cutter in until the curve of the addendum just shows on the blank, then cut the next two teeth. Index the wheel around until the three cut teeth are easily seen. If not quite up, fetch the plate back and then bring the cutter a little nearer (interpret that as .002"/.003") and cut the three teeth again. As soon as these three teeth are just up, proceed to cut all the teeth".*[23]

"If, on examination, the front of the wheel teeth are up and the back are not, the slide rest (something) is out of square. If only a little, leave it alone, but if much out of square, (re) line up on the blank; it will mean spoiling the wheel but it is worthwhile, as the next one will be satisfactory. It is always a lot of trouble to set up for an individual wheel, but once it has been carried out wheels can be cut to that particular size indefinitely".

Pinion cutting comments from W.J. Gazeley: *"With clock pinions, and some of the larger watch pinions, a slotting cutter is used first to save the heavy wear on the shaping cutter".*[24]

Wheel cutting comments from George Daniels: *When the origins of the cutter are known the wheel blank can be turned to the correct full diameter and the cutter passed through at a depth that will allow the curves to meet without reducing the diameter of the blank. When the origins of the cutter are not known the full diameter is best determined by trial on a test blank*[25]. Mr. Daniels was making the point that there are different systems a/k/a standards by which cutters are made and one may not be familiar with the source of the specific cutter one is using. Thus, while his passage is referring more specifically to cutters, I included this passage as it helped me to better visualize how to know when one has reached the correct finish depth with the cutter.

…and finally from Jerry Kieffer, "This is why in class we stressed that depth is determined by the shape of the tip of the tooth."

…after machining many cutters and cutting many wheels, you may arrive at my same conclusions 1) two or three end mills will satisfy most tooth curves 2) getting the tooth space and the tooth height right is much more important than getting the tooth curve right 3) tooth curves can be approximate and have no impact on clearance. On the other hand, the tooth space or tooth height, if not just right, will have an impact on clearance.

In this journey on wheelcutting, I have observed three general categories of tooth top curves. There are curves which tend to be pointy, curves that are flatter and an in between curve shape. The pointier curves tend to be emulated with the larger end mills of 3/32", 7/64" or 1/8". The flatter types of curves tend to emulated with the smaller end mills in the 1/32", 3/64" or 1/16" range and the in between curve satisfied with a 5/64" end mill. I share all this so you will concentrate more on the most important parts…the tooth height and the tooth space! Happy wheelcutting!

- **Resizing wheel teeth**
  After having cut a wheel, one must do a quality check of the wheel and its mating pinion by checking the wheel and pinion via the "meshing test". If it is determined that there is stickiness, teeth "catching", teeth "jumping out" and that teeth are still a bit too wide the tooth width can be resized i.e. machined smaller. One can still "save the job".

  The "meshing test" can be done right on the same setup by placing the RT in continuous mode. Full rotation mode is achieved in "Division" mode by entering "000". In this mode, the RT will turn continuously, one can then place its mating pinion's leaves against the new teeth and feel and see how smoothly it meshes. Note that by entering "000" one has lost the original indexing point of reference and one will need to re-establish this original indexing position prior to making the further adjustments to resize the tooth.

  So, if the teeth need to be resized, take these steps…
    - The X table and Y saddle are already in the locked, correct position, keep them in that position.
    - Turn off the CNC Controller.
        - Note that leaving the controller on while manually turning the RT hand wheel could damage the controller.
        - Note that when turning off the CNC Controller, it <u>does not</u> remember the last setting. Zero when restarting and reenter desired divisions.
    - Judge how much more material should be removed from the tooth shape and reposition the RT that much using the RT hand wheel. Typically it is a fraction of a .001", probably at most a .001". Turn on the mill and the CNC Controller, reenter the divisions, then cut one wheel tooth and observe the result. Remembering that to cut one tooth requires two spaces to be cut, progress the RT to the next tooth and cut that tooth. Note that if you have turned the handwheel .001", you have just removed .002" from one tooth i.e. .001" from each side of tooth. Observe the result and/or retry the meshing test. Repeat as necessary the sequence of turning the handwheel a fraction and cutting two spaces until the tooth shape is judged adequate and/or meshes correctly.