Boring Bar Thoughts, version 1.2

By R. G. Sparber

Copyleft protects this document.[1]

When turning an outside diameter (OD), using a cutter is rather predictable on my little hobby lathe. But when I want to use a boring bar to enlarge an inside diameter (ID), things can get a bit unpredictable. Recently I have been developing a program to help me turn ODs and IDs and it caused me to think long and hard about how a boring bar works. This article makes my thoughts visible to the community. Only by such peer reviews will the fact be separated from my fiction.

A boring bar is like a diving board. The more force put on the end, the more it bends. But unlike a diving board, the amount of force is a function of how much it bends. This may sound like "circular reasoning" but it is actually a form of mechanical feedback. In all cases the force and the amount of bending balance out. This article presents the logic and equations that predict this balance. The result is a more accurate finish pass with no need for "spring cuts".

A companion article exists which details a program which contains the logic and math described here. A copy of the program is available from me. It can be uploaded into a TI-83 Plus.

Conclusion

In a sample of 1 shop test, I specified an ID of 1.000" and the program guided me to cut an ID of between .9995" and 1.000". This is not a proof although it is typical of my experience with this program.

[1] You are free to copy and distribute this document but not change it.

Theory

As I was trying to understand how a boring bar bends during cutting, I came across an excellent reference

http://ctemag.com/pdf/2006/0606-boring.pdf

The gist of this article is that there are two forces acting on the cutter while boring a hole. There is a downward force which pulls the cutter off center. This effectively moves the cutter away from the surface. There is also an outward force which also moves the cutter away from the surface. The take home message here is rather simple: the action of cutting deflects the cutter. We all should know this. What is not as obvious is that the amount of deflection is a function of the boring bar geometry, feed, speed, material, and depth of cut (DOC). So not only does the cutter deflect, it deflects by an unknown amount.

All is not lost. While boring a hole by making multiple passes, we can stay with the same bar so the geometry does not change. Same goes for feed, speed, and obviously material. If we can keep the DOC consistent, then we can accurately predict the next change in radius of the bore. I don't need to know these parameters. I just need to keep them constant.
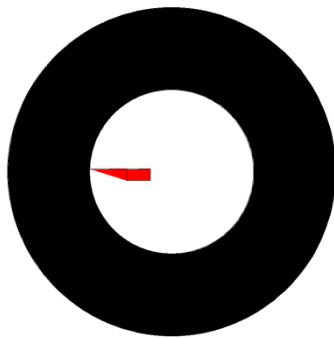
It is standard practice to make as many "hogging" passes as possible before starting any finish cuts. How much you can hog depends on the power and rigidity of your lathe. I can hog 50 thou in aluminum but not much more.
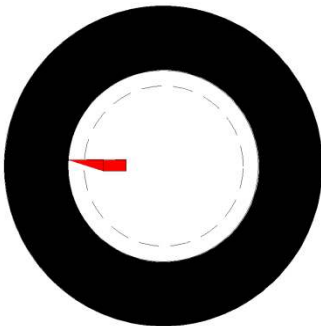
When as much material as possible can be removed by hogging, ideally one more pass is made with a smaller DOC. This is the finish pass. With a hobby lathe, there are advantages in making more than one finish pass. More on this later.
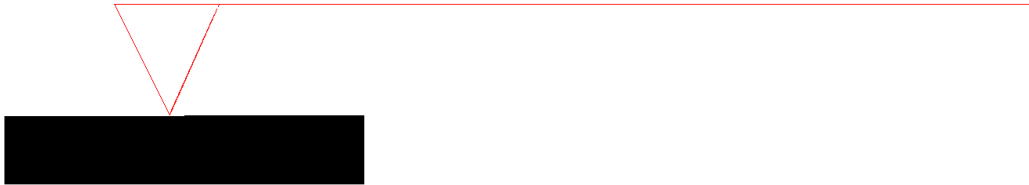
Let's focus on what is going on with the boring bar rather than the bore. I will simplify the story by only showing the boring bar in two dimensions.

top cut away view of the bore

The red line is my boring bar. It ends on the right where it enters the support block. The red triangle is my cutter. I show the cutter just resting on the black box which is the wall of my workpiece. The bar is straight because the cutter is just touching the wall.

When I advance the cutter into the bore's wall, I start to bend the bar. My cross feed dial changed by "A", but the resulting change in Inside Radius (IR) is "B". B is always less than A. The difference, A-B, is the amount the boring bar has bent.
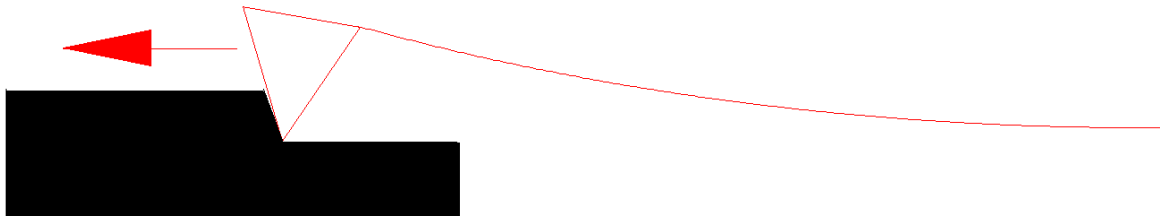
top cut away view of the bore
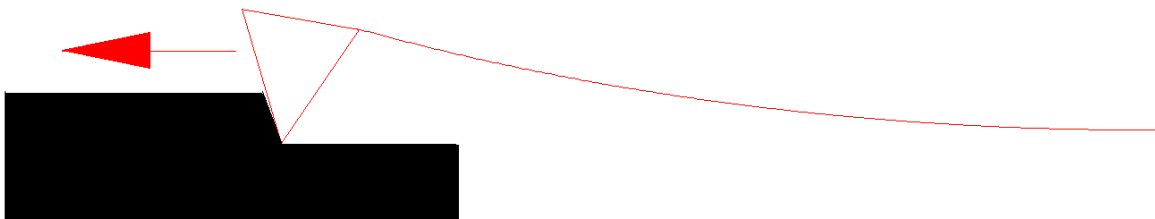
Say I take a "large" number of equal DOC passes. I then measure the IR, take one more equal DOC pass, and measure the IR again. Ideally, the DOC will equal the change in IR because whatever bending is in the boring bar, it does not change. This is just a restatement of my original speculation.

top cut away view of the bore

Do see what we have here. We can get an accurate change in internal radius as long as we always take the same depth of cut per pass. That will cause the same deflection of the boring bar. The trick is to plan ahead so all passes are as close to the same depth as possible. More on this later.

Consider what is going on when we first start to make roughing passes. To keep things manageable, assume all DOC are equal.

Before the first pass, the cutter is resting on the ID and I zero the dial. There is no bending force on the boring bar.

I then feed out, say, 50 thou. Some of this out-feed will cause a change in radius and the rest will deflect the boring bar. All I really know is that the boring bar *will* bend so my DOC will always be larger than my actual change in radius.

I will stick my neck out here and say that the amount of boring bar deflection is a function of the actual change in radius. Maybe the boring bar bends 10 thou resulting in a change in radius of 40 thou due to my out-feed of 50 thou.

I make my second roughing pass so out-feed by 50 thou again. The boring bar has already been deflected by 10 thou. Say it bends an *additional* 3 thou. This means that my actual change in radius will be 50 -3 = 47 thou. In other words, I have made a deeper cut even though my change in the cross feed dial is the same.

Given that my change in radius went from 40 to 47 thou, I expect that the bending force on the boring bar is larger. So it makes sense that the bar was deflected 10 thou during the last pass with a change in radius of 40 and is now bent (10 + 3 =)

13 thou when my change in radius is 47. The numbers may be wrong, but the trend is (hopefully) correct.

top cut away view of the bore
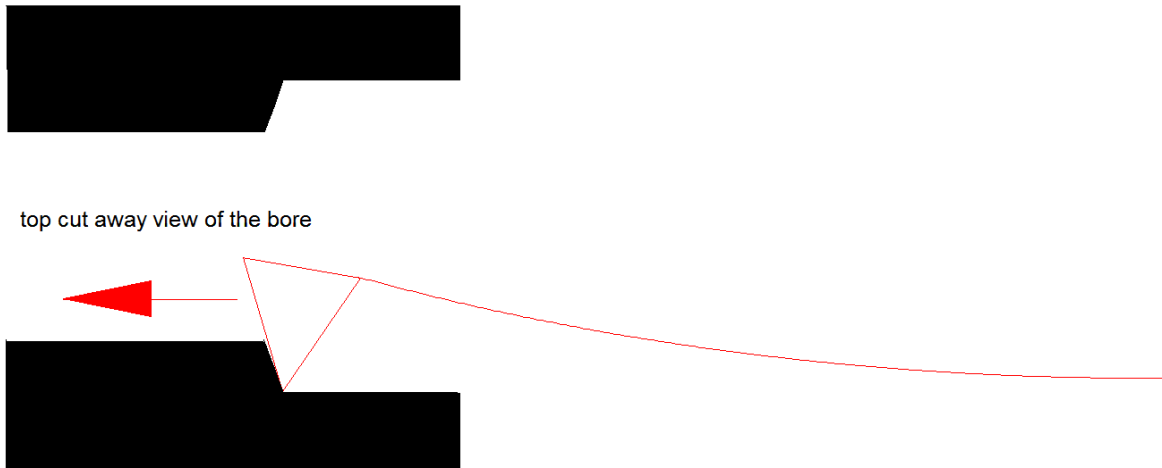
With each equal roughing DOC, the change in radius compared to the previous pass becomes smaller. This is the same as saying that the **change** in deflection of the boring bar becomes smaller. Eventually, the change in radius will be the same as the roughing DOC assuming that there is no error in the lathe. I dial out by 50 and my change in radius is 50.

After making a "large" number of roughing passes, I start my finish pass phase.

The big "ah ha!" for me was realizing that my boring bar is sitting there bent some number of thou due to the last change in radius. Say, after 10 roughing passes of 50 thou each, my boring bar deflected by 15 thou.

Typically, my finish pass DOC is less than my rough pass DOC. Well, given that increasing the DOC from zero to the roughing pass DOC eventually built up a deflection in the boring bar, if we now start to make shallower DOC, the boring bar will give back some of that deflection. So I may change my cross feed dial by 10 thou in preparation for my finish cut and get **more** due to the un-springing of the boring bar. After all, it was deflected, say, 15 thou due to a 50 thou DOC. Going to a DOC of only 10 is a big change in the change in radius which means a big change in the force deflecting the boring bar.

So maybe when I change the cross feed dial by 10 thou I get a change in radius of 15 thou. That is 10 from the cross feed dial and 5 from the un-springing of the boring bar. This is why I don't like to have only one finish cut when boring a hole.

One solution to this problem is to just take finish cuts of equal depth and not take any roughing cuts. Not something I would want to do. Too much like work and it would put me to sleep.

Instead, I suggest we still take as many roughing passes as possible but leave enough metal to permit at least 3 finish passes of at least 5 thou each. Measure the IR before each pass.

Let me define a useful number that I call the Depth-of-cut Correction Factor (DCF):

$$DCF = \frac{change\ in\ radius}{DOC} \qquad\qquad (1)$$

In the above case, my DCF equals $\frac{40}{50} = 0.8$

If I multiply my DOC, the amount I turn my cross feed dial, by the DCF, I will get a prediction of the change in radius for the next pass. The closer the next DOC is to the DOC made when I calculated my DCF, the better the result. Just to check, my out-feed was 50 thou and with a DCF of 0.8, I should get a change in IR of (50 X 0.8 =) 40. OK.

I am going to calculate my DCF from data related to my first finish pass. It will be applied to the second pass. Then the data related to the second finish pass will be used on the third pass.

Here is a theoretical example:

My goal IR is 1.5" and my current IR is 1.0". I need to change the radius a total of 0.5". Say my roughing DOC is 0.05". I will take 9 roughing passes which would get me to an IR of 1.45" if the boring bar did not bend. But say it was deflected 15 thou so my actual IR is (1.450" - 0.015"=) 1.435". The remaining (1.5" - 1.435" =) 0.065" will be removed with 3 finish passes. If the deflection of the boring bar did not change due to a smaller DOC, I would take (0.065"/3 =) 22 thou for each pass.

My boring bar had a deflection of 15 thou because my roughing DOC was 50 thou for 9 passes. Say my deflection for a 22 thou DOC is 7 thou. I dial in a DOC of 22 and my IR changes by (22 thou + 7 thou=) 29 thou. Ouch! This means that my IR is now (1.435" + 0.029" =) 1.464".

Since this is not my only finish pass, I can recover. I calculate my DCF and get $\frac{change\ in\ radius}{DOC} = \frac{29}{22} = 1.3$.

In preparation for my second finish pass, I calculate my next DOC by taking the remaining (1.5" - 1.464=) 0.036" and dividing it by the remaining 2 finish passes. So if all stayed constant, each pass would be $\frac{0.036"}{2} = .018"$. My last change in radius was 29 and I want this next one to be 18. This means that my boring bar will spring back a bit more.

In order to get closer to the right answer, I will use my DCF to generate a closer guess at the best DOC. Recall that

$$DCF = \frac{change\ in\ radius}{DOC}$$

If I have a desired change in radius, I can find the corresponding DOC by rearranging terms

$$DOC = \frac{change\ in\ radius}{DCF} \qquad (2)$$

In our example, my last DCF was 1.3 and I want a change in radius of 18. So

$$DOC = \frac{change\ in\ radius}{DCF} = \frac{18}{1.3} = 14\ thou$$

This estimate is not perfect but should be better than ignoring the fact that the boring bar is springing back due to smaller DOCs.

I take my finish pass with a DOC of 14 and again measure the IR. I was shooting for an IR of (1.464 + 0.014=) 1.478" using my Depth-of-cut Correction Factor. Say there is still some spring left in the boring bar so I get an IR of 1.480". That is a change in radius of (1.480" - 1.464=) 0.016"

It is time for the final finish pass. I have (1.5" - 1.480"=) 0.020" to go. My DCF is $\frac{16}{14} = 1.1$. I have to remove 20 thou so employ equation (2) again to get

$$DOC = \frac{change\ in\ radius}{DCF} = \frac{16}{1.1} = 15\ thou$$

I out-feed by 15 thou and take my final finish pass. If all goes as planned, I will be at an IR of 1.500".

I have described what happens when turning an ID. The logic works for OD too. The DCF was greater than 1 for ID but will be less than 1 for OD. If the lathe is ridged enough, then a single finish pass can work and you don't need to use a DCF at all.

If you have made it this far, I suspect you are thinking how impractical it would be to do all of this math just to turn a diameter. I completely agree. That is why I have developed my "Lathe Assistant" program that runs on a TI-83 Plus pocket calculator. You feed it measurements and it tells you where to put the cross feed dial. Details can be on my web site soon.

This approach assumes you have an ideal lathe. In fact, it really assumes you have a lathe with a fixed error. Any fixed error between depth of cut and change in radius will be canceled by the Depth-of-cut Correction Factor. What can't be done is compensating for random error. You would need a time machine to tackle that one.

Shop Testing

Although a sample of one shop test is far from a proof, it is sufficient to disprove my speculation. I will use my program which deals in inside diameters rather than inside radius. I am machining aluminum and selected a surface feet per minute of 100. My roughing depth of cut is 50 and I will take 3 finish cuts.

My goal diameter is 1.000".

I started by drilling a 5/8" hole so my boring bar would fit. It measured .637". The program told me to use an RPM of 470. The closest I can get with my lathe is 500. The cross feed dial was set to 0 with the cutter just touching the inside bore.

The program told me to take 3 rough passes. I then sprayed on some alcohol to cool the aluminum and then measured an ID of .908". This is a change of ([.908" - .637"]/2=) .136" in radius. If the boring bar did not deflect, the change would be (3 X .050" =) .150". So my boring bar has deflected (0.150" - 0.136"=) .015".

I have ([1" - .908"]/2 =) .046" left to go on radius. Given 3 finish passes, that is a DOC of (.046"/3=) .015" for my first pass. There is no DOC Correction Factor on the first finish pass.

The cross feed dial is now at 50. The program tells me to go to 35, a change of 15, and take a finish pass. After the metal cooled, I measured .945". This is a change in radius of [(.945"- .908")/2 =] .019". I dialed in 15 thou but since the boring bar will un-spring a little, the change in radius was 19 thou. This means a DCF of (19/15=) 1.267.

I have ([1" - .945"]/2=) .028" left to go on radius. I have two passes left so will divide by 2 to get .014" for the next pass. The boring bar still has some spring in it so I want to dial in a smaller value. Using the DCF, I estimate that I should change my dial by (14/1.267 =) 11.05. My dial was at 35 so by turning it out by 11.05, I get to 23.95.

The program told me to set the dial to 23.9 so I went with 24. After taking the finish pass and letting the aluminum cool, I measured an ID of .9665.

I have ([1" - .9665]/2=) .017" to go on radius. This is the last finish pass so it will all be taken this time.

My measured change in radius was ([.9665" - .945"]/2=) .011" for a DOC of .011" so the boring bar is finally at equilibrium and is not adding any more to the cut as it un-springs. My DCF should therefore be (11/11 =) 1. My dial was at 24 and I want to feed out 17 which puts me at 7 on the dial.

The program tells me to set the dial to 6.6. It carries more digits so has less round off error than me.

6.6 is a change of (24 - 6.6=) 17.4. I set the dial to 6.5 and took my last finish pass. After waiting for the metal to cool, I measured 1.000". A second measurement showed 0.9995".

Note that if I had set the dial to 7 rather than 6.5, I would have ended up .001" over. So the program did a better job predicting the depth of cut needed than I could with pencil and paper.

This single application of the program is no proof. But from an engineering standpoint, if I use the program many times and it always gives accurate results, it is "good enough". No rigorous proof is necessary. Time will tell. I've got a lot more turning to do in my shop.

I welcome your comments and questions.

Rick Sparber
Rgsparber@aol.com
Rick.Sparber.org