-------------Extracted from: http://www.chaski.org/homemachinist/viewtopic.php?t=75898&highlight=grind

If you want to study some a good book that maybe your library might have or might be able to get on interlibrary loan is
“ Design and Use of Cutting Tools |” by Leo J. St. Clair
or like me maybe you could find one on ebay.

Any chance you can find an old hand that came up using HSS for machining? There's a lot more to grinding truly functional tools than can be covered quickly----and seeing it done is far more instructive than talking about it. Be certain that your mentor, assuming you're fortunate enough to find one, has proper experience. Try to avoid learning from someone that's self taught and has no clue---all that does is reinforce bad habits.

Unless you have a lathe with a high speed spindle, and aren't afraid to run it that way, you'd be far better off to lose the idea of running carbide. It has a place, but you may not get close to proper justification in making the choice. That would be particularly true of finish machining carbon steel.

Mild steel, be it hot or cold rolled, will cut with a great surface finish so long as you're taking a good depth of cut and using a reasonable feed rate, but when you start with finish cuts, it generally tears. Unlike some alloys, mild steel does not respond well to light cuts. A properly sharpened HSS tool bit will generally yield better, although not perfect, results.

One word of advice. Do not short-change yourself the opportunity to learn to sharpen proper HSS cutting tools. You will be held captive as long as you rely on inserts------and will generally be poorly served unless you have large equipment that can benefit from carbide.

That would be particularly true for those with fractional hp machines, that simply do not have the necessary features to benefit form the application of carbide except in rare instances (C2 for cast iron, to get around abrasion, for example). Shortcutting the learning curve of proper tool sharpening is no different than never learning to land an airplane. You must do it, and do it you will-------so it's best to get it down from the start. Substituting insert carbide for the learning curve is the single greatest mistake the home shop type can make.

Your grinder should be equipped with an aluminum oxide wheel of the proper hardness and grain structure. The typical bench grinder comes equipped with wheels that are far and away too hard for grinding HSS. The end result is little metal removal and burned tools. You must also learn to dress the wheel properly.

It might be wise for you to discuss this before investing in yet another wheel that won't serve you well. The best choice of grinding wheels is available in the 7" x ½" x 1¼" configuration. That's due to wheels that size being used on cutter and surface grinders. You might consider building your own grinder to accommodate such a wheel. It's a nice project, and you end up with a functional grinder suited to your needs.

Machining, particularly mild steel, often does not yield pretty cuts. It's the nature of the material, especially when turning. While you can improve the surface finish by various methods, it's nearly impossible to machine with precision and get good results. Carbon steel does not cut cleanly, particularly when shallow cuts are taken. Cuts deep enough to improve finish are generally too deep to allow for control of size---so you're stuck between a rock and a hard place. You must learn to take reasonable cuts, repetitively, consistent in depth from cut to cut, and to shoot slightly oversized (.0005", assuming you have a decent finish), allowing for some minor polishing with abrasive strip. I caution against the use of files for sizing on a lathe. Regardless of how skilled a person may consider themselves to be, you're likely to do more harm than good with files. Your only reasonable alternative to that is to leave material for finish grinding, which would be totally unreasonable if you don't have a cylindrical grinder at your disposal. A toolpost grinder would prove to be a long, hard battle, worsened by grinding soft material, and without coolant. Not a good combination.

Tool geometry can get you in trouble if it's not well understood. I offer you, as an example, turning leaded brass with excessive positive rake, particularly on a light duty machine. Rake angle (or, as in this case, the lack of it) is ultra critical to avoid hogging. Broken tools and scrapped parts are often the result of not knowing what works, and what doesn't.

You can learn the theory from books, which should talk about rake (and clearance) angles as they relate to materials. Do not assume that everything is carved in stone. There are exceptions to some of the rules, which you will come to understand with experience.

Grinding these angles, particularly in combination with chip breakers, is a whole different matter. That takes some time and dedication, but it's not beyond your ability----you just have to be determined. I can help you with the learning process, but I use a very unconventional system, which, at first glance, leads you to think I don't know what I'm talking about. Before you jump to conclusions, remember that I mentioned that we stand on the shoulders of those that went before us. My tool grinding method was assimilated a full eight years after I entered the shop. It is, hands down, the best grinding system going, but it requires skill on the part of the operator. You can't fake it. I'm willing to guide you through the process, but you have to think differently about your grinder. NO work rest, for example. I'll leave that up to you.

Even well seasoned hands on lathes tend to be weak in grinding good tools. It's an art that requires a little dedication, thus is generally not pursued. We're used to instant gratification in this country, so far too many won't make the effort. One's expectations may be being met, but that doesn't mean there's not room for improvement. It's not enough to scratch material off with a reasonable finish----when it's time to move material, it's nice to know how to do it-----and to grind a tool of such a nature that it is capable. The application of rake/chipbreaker combination is the solution to that problem. Yeah, it takes a little effort, and maybe even the ability to put one's ego in neutral and learn something from others with practical experience.

While it's true that honing is a critical part of proper tool sharpening, if the tool comes off the wheel requiring honing for anything beyond improving the surface finish of the tool, the job on the wheel was not executed well. There's more than enough room for improving one's skills in that example.

Hand honing of cutting tools is an open invitation to creating tools that perform poorly. It's all too easy to round off the cutting edge ever so slightly, so it drags below the point. That's a common ailment with ham handed "machinists" that work with a closed mind and can't see room for improvement. I've worked with them, and I'm sure Kap has worked with them as well. A casual glance at their tools is usually adequate to see the problem.

It's important for the operator, be he/she a hobby machinist or one that works in the trade, to understand tool geometry. That point can't be driven home enough. How can one grind a functional tool when it can't be identified? You should be able to pick up a cutting tool and know that it will, or won't, cut, and why.

You know, I wanted to wade in on this one and provide some decent guidelines, but it's not an easy thing to do when you're dealing with a vast array of people with varying depths of experience. For me to talk to another machinist that is adept at grinding tools by hand----hey--no problem. We more or less speak a common language. Fact is, I recall reading some comments on chip breakers on one of the other boards recently and enjoyed a nice little smile. I thought to myself, "yeah, this guy understands!"

Basic tool grinding is a must before one moves forward with trick grinds, and learning the basics without a tutor is no small feat. You can read the book and make the grind, but unless you're familiar with how each material machines, you often have no clue if your tool is performing well, or not. Problem is, it's not hard to get a tool to cut-----it's just hard to get one to cut well.

I consider myself reasonably fluent in language, but trying to describe grinding operations to one that may not have a clue (that's not meant unkindly) is a tough chore. Terms used are often reversed, often misidentified, and frequently not understood. It's important for the trainee to put forth at least minimal effort and try to learn proper terminology, which would make the chore much easier.

Considering at least one person feels little of value has been stated, would anyone be interested in me talking about grinding toolbits? Before making a decision. please consider that I am very narrow of mind where machining is concerned, and have certain things I do and don't do----and I'm not interested in compromising on any of them. For example, If you insist on using a rest for grinding, you may find you can't duplicate the things I discuss. I worked the way I did because it was successful----I did not blunder along. It can work for others just as well, assuming they have an agenda in common with mine.
------------------ End of first thread "Grinding HSS Tool Bits

Grinding Wheel Choice

--------------- Extracted from: http://www.chaski.org/homemachinist/viewtopic.php?t=75985&highlight=grind

Successful grinding of HSS toolbits is best achieved by the use of a wheel that is readily friable and open in structure to insure free cutting and proper removal of swarf. Grain selection can be a matter of choice, but is often dictated by the availability of wheel selections. If more than one wheel is at one's disposal, a 46 grit wheel is a good choice for roughing a tool, but the finish is not as desirable as one produced by an 80 grit wheel. 80 grit wheels tend to be a poor choice for roughing, cutting slower and hotter than a 46 grit wheel, so it's not a good choice if only one wheel is available. It has long been my custom to use a 60 grit wheel, which serves both purposes adequately, and is an excellent compromise.

Hardness rating of grinding wheels is based on surface speed. The typical vitrified wheel is generally rated @ 6,000 sfpm, and behaves softer as the diameter decreases. Armed with this idea, it's wise to select wheels with the largest possible diameter, without exceeding their recommended maximum speed rating. As wheels get smaller, useable life diminishes rapidly, but it can be recovered to satisfaction if wheel speed is increased. If you're fortunate to have control of the speed of your grinder, insure that you don't exceed the 6,000 sfpm threshold, to insure that the wheel doesn't come apart. Most bench and pedestal grinders don't have the ability to change speed, but many cutter grinders do.

The typical bench or pedestal grinder is normally equipped with wheels that are too hard for such an application, and wheels are not readily available in the more desirable grades for such use. To resolve this problem, a 3600 rpm grinder can be adapted to run wheels that have a 1¼" diameter hub. Such wheels are commonly used on cutter and small surface grinders and are readily available in a broad variety of styles and selection of grits, hardness's and structure.

Type of grinder is beyond the scope of this post, but it's important that the reader understand that the vast majority of problems one encounters when grinding HSS are related to wheel choice, and while success of sorts can be achieved with almost any wheel, there are solutions to the problems that can turn a rather unpleasant task into one that is quite tolerable.

Rule of thumb for grinding is to use a hard wheel for soft materials, and a soft wheel for hard materials, and further dictated by silicon carbide's ability to dissolve in steels at high temperatures such as found when grinding. While silicon carbide far exceeds the hardness of aluminum oxide, it is not suited to grinding HSS for that reason. In keeping with these guidelines, aluminum oxide wheels become the acceptable choice for grinding HSS---with, perhaps, the odd exception, which is unlikely to be an issue. Silicon carbide wheels are rarely used in the shop unless non-ferrous materials are to be ground. They are not intended to be applied to steels or steel alloys.

Wheel hardness is not related to the hardness of the grinding media. Hard or soft wheels share the same material, but are assembled with more or less bonding material to dictates the hardness of the wheel. A wheel that requires greater effort to dislodge dull bits would be considered a hard wheel, while one that permits the bits to be dislodged easily would be considered a soft wheel. That is the friability index of a grinding wheel, and is noted as a letter of the alphabet, A being soft, Z being hard.

Another feature to be considered when making a choice is the structure index. Abrasive grains are spaced close together in a tight wheel, and spaced at greater distance in an open wheel. This index is referenced by numbers, with 0 being a closed wheel, and 12 being an open wheel. Open wheels provide a space for swarf and help in keeping the temperature of the grind at a lower level. A wheel that is not open enough will "load" and not unload, yielding a wheel that displays all the characteristics of a wheel in need of dressing because it's dull. It's clear, a balance of hardness and structure are important to successful grinding, so they should be considered when selecting a wheel.

Wheels are bonded by various methods, but the one that is of most concern to the home shop type would be a vitrified bonded wheel. Vitrified wheels are bonded with selected clays, which are high fired, and are identified by a V in the wheel designation, following the hardness and structure designation. The process builds minute lengths of glass that bond the bits to one another upon vitrification. Hard wheels have a greater amount of bonding agent than soft wheels.

Others bonding systems serve specific purposes, but are unlikely to be needed in the home shop for the most part. Resinoid parting wheels can be an exception. Other types of bond include silicate, rubber and shellac.

Abrasive designation type within a media is an indication of the process used to produce the media. One might expect to see markings such as 19A, 32A 0r 38A on a Norton wheel. All are aluminum oxide wheels, although each is better suited to different types of grinding. The 38A abrasive has excellent friability and a porous nature, making it the wheel of choice for heavy stock removal and a good choice for HSS grinding.

Assuming one was to order a Norton wheel for grinding HSS, a designation such as 38A60H8VBE would be considered an acceptable wheel. Grinding techniques vary, so some may prefer a slightly harder wheel, so experimentation would be encouraged.

Grinding wheels should always be given a "ring" test before mounting, immediately after a physical examination in which one looks for signs of cracking or pieces having been broken from the wheel. By placing the wheel on a solid object (screw driver, for example), and tapping the wheel with the likes of the handle of another screwdriver, a distinctive ring should be heard. If not, the wheel should be immediately destroyed to avoid an innocent party from mounting what is very likely a wheel that will explode when spooled up. It should be the policy of anyone running a grinder to destroy damaged wheels immediately.

Once it has been ascertained that the wheel is in good health, it should be mounted without a struggle on the arbor. If the wheel is a force fit, a risk of cracking is ever present, particularly when the flanges are tightened, so make sure the wheel fits properly, and can shoulder against the fixed flange (if so equipped) readily.

A wheel that has a bore out of perpendicular with the sides can be cracked when tightened. Should one encounter such a wheel, the bore can usually be scraped adequately to permit proper seating. It should never be horsed.

Always stand aside when spooling up a wheel, even one that is known to be good. The first few moments should serve to shake out a wheel with problems, so allow about a minute of full speed operation before standing in line with it's rotation. That's particularly important in a shop where children have access, or objects are moved about, where reasonable risk of cracking a wheel exists.

Proper dressing of grinding wheels is key to success. Wheels that don't run true and are not relatively flat are useless for tool grinding. You should be able to rest the tool being ground on a wheel and have it stay where placed, without bouncing away. The wheel should be trued immediately upon installation, then dressed on a regular basis as it's being used. One might dress the wheel several times in the process of grinding one tool, but the benefit of keeping the wheel properly dressed pays serious benefits in cooler grinding and faster stock removal. It becomes evident that a wheel needs dressing, particularly if it's too hard.

Truing a wheel can best be accomplished with a mounted diamond, or diamond cluster, but a wheel so prepared is not well suited to offhand grinding. Diamonds tend to leave a wheel quite smooth, increasing area in contact with the grind, and behaving somewhat like a bearing. Grinding with a wheel so dressed tends to be slow and hot. In order to disrupt this pattern, roughing the surface by different means is normal procedure.

The best possible surface for hand grinding is achieved by the use of a star, or impact, type dresser. They function by hammering the wheel and dislodging the old bits of abrasive, exposing new bits that are sharp and ready to grind. Problem is, this type of dresser requires a bit of skill to apply successfully, and often wastes a good amount of useful wheel as the user attempts to get the wheel running true and flat. As a result, I do not use, nor endorse such a dressing tool. Further, if you use the same method of tool grinding that I do (no tool rest), there is no place for the dressing tool to rest as it's applied.

There are available on the market items known as dressing sticks. Two varieties should be of interest for those that intend to grind HSS tools. One is a solid boron carbide stick, typically 3/16" thick, ½" wide, and 3" long. They are excellent for dressing wheels when new, but as the corners slowly round off, they begin dulling the wheel more than dressing---so I don't recommend one unless you have the ability to keep it sharp.

The next choice, and the one I recommend highly, is the use of a vitrified silicon carbide stick. They are typically 1" square and 6" long, and inexpensive (roughly $10). They are readily available from machine shop supply sources. One should last the average home shop type for a life time. Choose one that is coarse grained, anything from 16 grit up to 24, depending on availability. Avoid finer sticks. A wheel, once diamond dressed, can be fine tuned with such a stick in seconds, and done freehand without the use of a rest. It is also applied when redressing is required, keeping the wheel flat and sharp.

++
Will these silicon carbide sticks be useful for renewing the wheel surface?
A deal at MSC ~ $2.90 ea.;

http://www1.mscdirect.com/CGI/NNSRIT?PMPXNO=1747275&PMT4NO=16320797

From all indications, an excellent choice!

I normally use nothing more than a dressing stick.

A friend also suggested using the stick dresser to rough radical changes to the wheel shape before finish dressing a radius, angle, or similar with the diamond.

That has always been common practice. Fact is, unless one has a profile dresser, wheels are commonly hand formed without a diamond. It's just not as hard to do as it may sound, particularly when the vitrified dressing sticks are used. Depending on the intended purpose of the wheel, it is often desirable to not diamond dress at all.

Quote:
He was also kind enough to provide a couple of suggested sticks, one for diamond and one for AL wheels.

Are these familiar to you? He says the first one lasts nearly forever and the second (for diamond) goes REAL fast, as it is extremely soft.

Norton 37C24-52VK (AL)

That is the exact stick I recommend----if not one of the more economically priced imports.
++

Grinding toolbits, unless very large, say, above 1/2", generally does not require a lot of hp. I've used a ½ horse Craftsman motor that belonged to my father as long as I've had my own shop, and have yet to exceed it's ability. What you'll come to understand when you select the proper wheel is that the wheel cuts readily, and requires very little pressure. If pressure is required, you have the wrong combination.

Please keep in mind that I do not use my grinder for heavy grinding, such as grinding welded components, nor should anyone. Wheels for grinding HSS should be more or less reserved for that purpose alone. They aren't really suited for much else. A second grinder with harder wheels is very important for those with needs beyond grinding HSS.

The problem you're likely to encounter with your 6" grinder is the inability to buy a wheel that's suited to grinding HSS--- which I have already addressed. If, by chance, the grinder runs @ 3,600 rpm, and you're willing to make adapting arbor(s), it could prove to be a very nice machine. In your case, it may require building new wheel guards as well, but they can be easily made from short lengths of steel pipe.

My comments are sure to anger a few, but please hear what I have to say and try to understand that there is a serious amount of logic in my position.

Depending on the nature of the task at hand, I normally speak against the use of belts for sharpening cutting tools. There are exceptions.

The problem with belt use is that belts do not run flat. If you'll pay attention to what a belt does just above the point of contact, you'll notice that it buckles outward. That is the worst possible scenario I can imagine. Unlike grinding with the radius of the typical type 1 wheel, which provides for a slight hollow grind, a belt tends to round off the cutting edge ever so slightly. It is akin to starting with a dull cutting tool, and often is the source of horrible finishes and difficulty in holding size.

Do I use a belt for some grinding? Yes----I do-----but I usually regret the decision, often because of the diminished performance of the tools so ground.

My suggestion for those that prefer a belt (they tend to move material fairly rapidly), is to use one for roughing, but switch to the proper wheel for finish grinding. Properly applied, a wheel will beat a belt every time.

One other issue with belts as opposed to wheels: You are drastically limited by the use of a belt, very unlike a wheel. That is likely not evident to the novice, who has great difficulty in grinding cutting tools. However, as one's skill level improves, it quickly becomes evident that not only belts, but tables become a limiting factor.

I encourage every person to learn to grind free hand----without the use of a rest. It's harder to do at first, but easier when mastered. Again, I'll address that in a future post.

One does not "dress" a diamond wheel in the common sense of the word. Diamond wheels tend to load over time, particularly if they're run with improper or insufficient coolant. Also, a diamond wheel that has been subjected to steel benefits greatly from the application of such a stone. They are soft and friable, to break down quickly under the diamond. The purpose is more to clean the matrix of the diamond wheel, never to reshape it. A touch, long enough to see a change in the appearance of the surface of the diamond wheel (cleaning the crud) often restores the wheel to excellent performance.

Quote:
Also, I just bought some wheels off Eaby for the cutter grinder. These are 6x3/16" wheels I intend to use for gashing endmills. 38A60-K8VG. I wanted something hard enough to hold a corner when splitting points on two and four flute center cutting endmills. Does this sound about right?

While they would serve the purpose, you may find you'd have better success using a parting wheel for gashing. Parting wheels tend to cut much faster and cooler, although there may be exceptions.

As is referenced in grinding handbooks, heat checking of grinding heat treated objects is an ongoing problem. It has customarily been addressed by hand dressing wheels, so the moment of contact with the wheel is brief, and cutting is fast and cool. Rapid traversing of the object past the wheel is also instrumental in reducing the problem.

Your choice of wheel has reasonable designations (were it 7" diameter, I'd suggest it as a wheel for grinding HSS toolbits), but one that is diamond dressed would tend to cut quite hot. The wheels you chose would still need to be reduced considerably in width for your intended purpose, so the benefit of using such a wheel can be lost. Grinding wheels that are fine enough respond fairly well to having their cross-section reduced, but they tend to be quite fragile. By sharp contrast, parting wheels, which are not vitrified, but resinoid bonded, withstand much greater speeds, and are, in fact, very resilient compared to vitrified bonded wheels.

If you're familiar with grinding, you know that wheels can fail to cut for various reasons, most of which are very apparent. However, it's not always clear when a wheel is dull, or loaded.

A silicon carbide wheel that has been applied to steel loses its ability to grind almost instantly, and displays the same qualities of each of those conditions. What has happened is the silicon carbide has been absorbed into the steel, rounding the abrasive such that it no longer has cutting edges. Grinding wheels are, in a sense, milling cutters with a huge number of teeth. If these teeth are dulled, they cease to cut, or cut under considerable protest, with lots of heat. Such a wheel requires considerably more pressure to force the cut, if it cuts at all. All in all, the wheel experiences a total collapse in performance. You end up with a glazed surface, often heat checked, and can have a series of waves produced.

Silicon carbide is used extensively for honing, and also for lapping. In both cases, surface speed is so low that the problem doesn't exist. If you can duplicate that condition, there's absolutely no reason why you can't, or shouldn't, use silicon carbide for steel. Fact is, it will outlast aluminum oxide by a serious margin.

Quote:
On the parting wheel.. what exactly am I looking for? Surely you don't mean a fiber reinforced zizz wheel like we use in a die grinder?

No-----the wheels of which I speak do not have any reinforcement, and are available in very narrow widths, for sure 1/16", and could be even narrower. They have a 1¼" bore. They are available in fairly large diameter, at least 6", perhaps larger. It may require you to check with a supplier of wheels that has a large inventory----but I know they're out there. I haven't purchased one in well over 25 years, so I have little to offer in the way of information. You should be able to find them included with other parting wheels-----just look for thin wheels that are resinoid bonded and have no reinforcement.

For your information, you can usually part a shank, 1" diameter, heat treated, in something like 15 seconds with such a wheel. The shank should be spun while being parted, which can be accomplished by hand with a small crank. These wheels are nothing short of magic.

Dave wrote:
I use the rest to steady the edge of my hand, and the fingers of that hand to hold the tool bit at the proper attitude towards the wheel, up above the rest. The fingers on the other hand serve to help hold the tool bit and apply pressure towards the wheel.
Dave

Dave,

That's the exact process I used (ten years in the trade at that point) when I was forced to using the system I now use, one without a rest. I was hired into a shop that had a solitary grinder mounted on a steel pole, one of the roof supports in the shop. The grinder was mounted about chest height. I hated it at first, but quickly came to see its benefits, which were grinding most any form without the rest getting in the way. you can grind farther down on the wheel when there's no rest in the way, so you can see what you're grinding. You quickly learn to use the wheel as a guide, which works very well if it isn't bouncing.

As to the guard, I made my grinder, specifically for this kind of grinding. While you could get caught between the guard and wheel, it's far enough down around the wheel that it's never been a problem, and I've used the grinder since '67, so it's had a good shakedown. I'm also steady as a rock when I'm grinding tools. Don't know how that's going to work as I continue to age.

This system is particularly useful in grinding chip breakers, which I've done as long as I can remember. I'll probably repeat much of this when I post on the grinder in the near future. It's valuable enough that everyone should be exposed to it. Those with talent and a yearning to grind better tools should give it a go before dismissing it. It's really that good!

MikeC wrote:
I roughed it in on the big grinder and then looked over my shoulder at the T&C grinder sitting back there with a narrow sharp wheel on it. It is indeed pretty handy! I think I'll probably still rough out

on that big 1750rpm pedestal grinder, but I'll definitely save a couple of special rocks for the cutter grinder to final finish and shape.

You've grasped the point I've tried to make. A grinder with a rest is too restrictive for many of the operations that make for a better cutting tool. How you arrive at the basic tool should be done with the most comfort and confidence of the workman, and need not be done as I do mine.

Adapting to the system I use didn't come easily at first-----but as you're observation of those special rocks suggest, you get to the point where you benefit greatly by getting away from convention. It's not for everyone----some folks will never master the art of grinding tools by hand-----let alone learning to do so without a rest. However., for those that accept the challenge, mastering the process is a major step forward.

What's important is for each individual to understand the process, whether it's adopted or not. That way, when a complex grind is a necessity, you have a sense of direction in how to accomplish the task.

Marty_Escarcega wrote:
Question though for Harold. Your arbor calls for 2-3/4" flange built onto the arbor. In order to save time and material, could one start with a 2" piece of stock turn a shoulder and use 2.25" dia. x 5/16 thick washers?
Or weld on a piece of metal then turn/face the flange?

Harold, I know you were specific about the flange diameter. Just asking if there may be an alternative to buying or scrounging 3" stock.

Certainly, pretty much any combination of things could work--but remember that you'll be working directly in front of a wheel that is all too capable of killing you. The large flange size is in keeping with proper flange support of the wheel----I would not recommend a reduction, although you'd be pleasantly surprised to find a flat washer under a nut would work, too. I just wouldn't want to stand in front of it. The whole idea is to promote as much safety as is possible. The large flange diameter supports the wheel properly for those times when you have to use the side of the wheel, which is often.

The advantage of making the adapter in one piece is that you have maximum rigidity and alignment. There's no reason why you couldn't turn the adapter, sans the fixed flange, from a small diameter piece of stock, providing a shoulder for what would have been the fixed flange to locate properly. The flanges would then be made from flat stock, even torch cut before machining. If you did your work well, it would serve equally as well as the one piece method I suggested. Making the flanges without soft jaws could prove challenging---but not impossible.

A grinder for HSS Tool Bits

---------------- Extracted from http://www.chaski.org/homemachinist/viewtopic.php?t=76065&highlight=grind

've commented often on grinding toolbits without the use of a rest. The process requires a little skill, but that comes quickly with practice. The benefit is full access to the wheel, which, if you're grinding chip breakers, can often be a necessity. The work rest often limits your ability to address the wheel with the tool such that you can view the grind as it progresses, and limits the degree of mobility available. In both cases the end result is often a poorly configured tool or chip breaker, with less than good machining characteristics, and lots of frustration. Should you choose to explore this method of grinding tools, you'll come to realize that tools are easier to grind, with or without a chip breaker.

Being able to see clearly what you're grinding is key to success. Leaning over a low mounted grinder, trying to view the grind when it's wrapped towards the back of the wheel to avoid the rest, tends to be troublesome, but there's a simple solution. A small grinder, with no rest, can be mounted easily to a wall or on a rigid post, with the center of the wheel at approximately chest height. The height of the grinder is particularly enticing when grinding a larger tool (½", for example), which takes considerable time. The height allows for the operator to stand erect, eliminating fatigue from leaning over.

Considering you have no point aside from the wheel to steady the grind, the wheel should run dead true, with no bounce. Once it starts breaking down, it's desirable to dress it with a dressing stick to keep the surface smooth and free cutting. It's been my experience that once a wheel starts shedding media, it tends to be somewhat selective, shedding more in one area than from the balance of the wheel, quickly forming a wheel that is not round. Grinding with such a wheel becomes almost impossible by the no-rest method. Remember----the wheel becomes your rest. It must run true.

The typical bench or pedestal grinder rarely has a wheel that is acceptable for successful grinding of HSS, and one that is acceptable can be difficult, if not impossible, to obtain. In order to get around that problem, wheels that are intended for cutter grinder and/or surface grinder use are an excellent choice, although they won't mount directly to the typical bench grinder. These wheels are sized 7"x ½" x 1¼", are readily available in a broad selection, and are stocked by most machine shop supply houses. They're also commonly found on ebay. A little work in building a dedicated grinder that provides for mounting them can be accomplished at reasonable cost.

To build such a grinder, a ½ horse, 3,600 RPM motor with ½" shafts protruding from each end is more than adequate. Larger is better, but not necessary. By choosing a motor with double shafts, adapters with an integral fixed flange and locating boss can be made that accept the 1¼" bore wheels, providing a grinder that can be equipped with a roughing and finishing wheel, or even one side fitted with a diamond wheel and table for grinding brazed carbide tools. If the grinder is used in that fashion, it's desirable to be able to reverse the motor, and to have a double pole double throw switch for turning the motor on, with or without the coolant pump. Diamond wheels for such use should not be operated dry.

Wheel guards can be fabricated from large pipe, and mounted with appropriate brackets. If there is no place to attach the brackets to the motor, the motor can be mounted to a base plate, which in turn would be mounted to the wall or pole. Some welding would be required to enclose the inside of the guards. ¼"' plate can be used for the purpose. Note: The typical 7" grinding wheel has a maximum recommended speed in keeping with a 3,600 RPM motor. To assure your safety, limit the guard size such that a larger wheel can not be mounted.

Considering you grind lower on the wheel, the bottom edge of the guard is best held back such that it does not interfere with your grinding. The top end can closely approximate the typical grinder, considering you now use the centerline of the wheel instead of the upper portion, so visibility is not an issue.

The importance of running vibration free can't be emphasized too much. Thus, adapters that are adjustable to a small degree are advised. Boring the hole for the shaft should be accomplished concentric with the balance of the adapter, with a slight amount of oversize provided. The adapter then has a series of two sets of three holes spaced @ 120°, drilled and tapped either 10-32 or ¼"-28 where they will contact the shaft of the motor near the end of the motor shaft, and again near the bearing of the motor. Wall thickness of the adapter where it mounts to the shaft should be no less than ¼", and set screws should finish below the surface to avoid harming one's hand should they use the adapter to stop the grinder. The adapter can then be dialed in perfectly by adjusting the six socket head set screws while monitoring the adapter with a DTI. Excellent shop practice should be exercised when making the adapters. They should be dead concentric and perpendicular in order to minimize vibration and runout. The opposite end of the adapter should have a flange diameter of 2¾", no less than 3/16" thick. The flanges should be relieved so only the outer ½" bears on the blotter. The movable flange should also be no less than 3/16" thick, and have a small shoulder where the nut bears. It should be bored with minimal clearance on the threaded shaft so it is self aligning, minimizing vibrations.

The locating boss portion of the adapter (the part that accepts the wheel) should be held to 1.250" +.000/-.002" in diameter. That allows for precise centering of the wheel without risk of cracking should the bore not be perpendicular to the wheel. Length of this diameter should be held short of the widest wheel you intend to run. Considering these wheels are available as narrow as ¼", you may wish to keep the length as short as 7/32". Likewise, you should allow for enough thread so that a wheel that is ¾" or 1" wide can be mounted. That, along with the thickness of the nut of choice, will dictate the length of the thread you'll generate. I've found that a ¾"-16 thread serves very well, but any thread larger than ½" in diameter would be adequate.

Provide a set of shallow flats on the body diameter of the adapter to allow for a wrench to remove stubborn wheels. Hand holding the wheel while tightening the nut with a wrench will provide more than adequate holding power, even when reversing the motor. The motor should not be capable of instant reverse, however.

Commercial nuts may or may not have threads that are perpendicular to the faces of the nuts, so it's a good idea to single point a thread that is short of the length of the nut you choose, then mount the nut on the thread and face both sides so it will run without attempting to tip the wheel. Break the corners of the nut, or if you desire it to mount the nut in only one direction, face the corners until you have a shoulder that's about 1/64" long, then chamfer the corners of the opposite side. Deburr well.

Always run proper blotters on your wheels, and never mount a wheel that doesn't ring when tapped. Any wheel found to yield a dull thump instead of a clear ring should be destroyed to avoid an unsuspecting person from mounting a wheel that could very well kill them.

Always stand clear of grinding wheels when spooling them up. ALWAYS! Even when you know that the wheel was good when mounted. You never know when a wheel might be bumped and cracked by the activity of others. Allow the wheel to run a half minute or so before standing in front of it. Wear eye protection.

If you've been struggling with tool grinding, give this process a try. You may be pleasantly surprised. I know I was when I was introduced to it by my peers.

Truth is, there's five kinds of bond, which I touched on briefly in my post on wheel selection. The five varieties are vitrified, the wheel that is of most interest to the home shop type, and used for grinding of HSS, shellac, rubber, synthetic resin (resinoid) and silicate (of soda). Each have particular properties, and lend themselves to specific operations. Vitrified wheels are most common, and will serve the needs of the home shop type with rare exception.

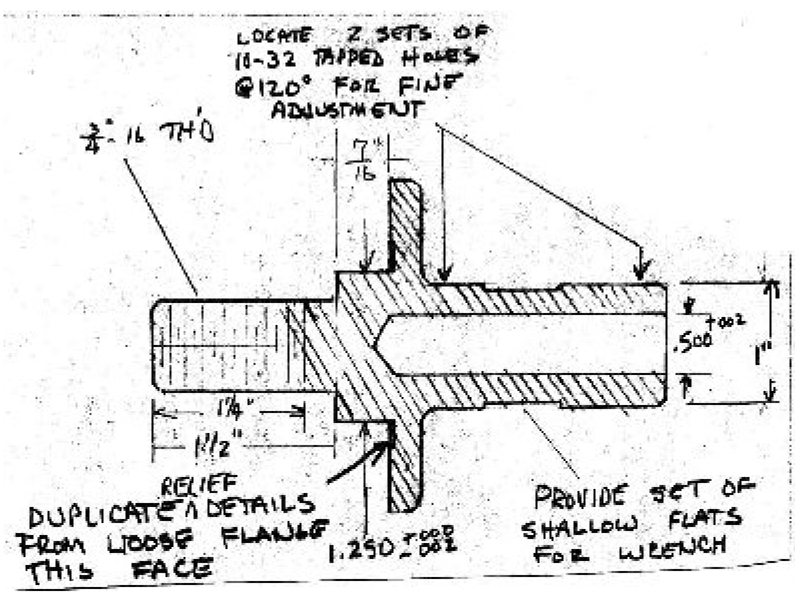
Resinoid bonded wheels have incredible strength, and are designed to operate in a range that exceeds 9,000 SFPM, yielding exceptionally fast metal removal rates. Parting wheels are generally this type of construction, and are designed at operating speeds as high as 16,000 SFPM. As you allude, a ring test has no value for this
type of wheel.

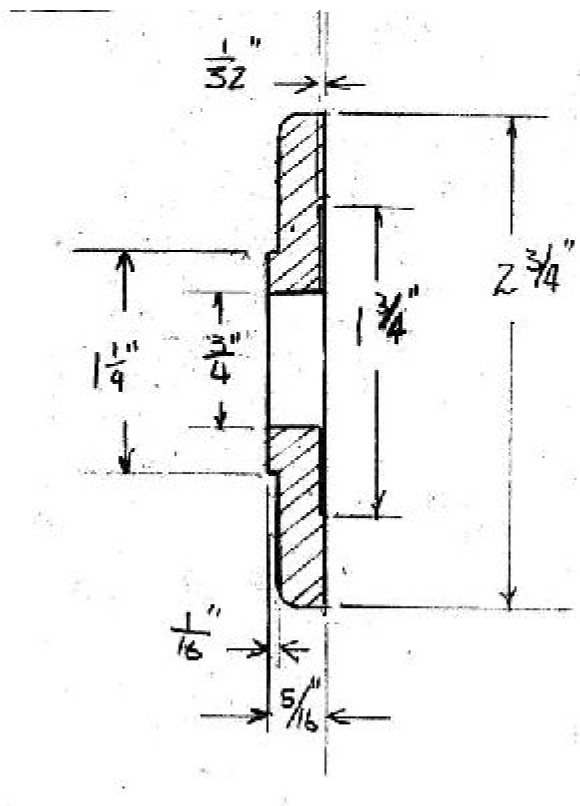
I don't have a pic of the adapter available, but I pencil sketched cross sections of the pieces a couple weeks ago for our very own SteveM, so I've copied the sketches, below. They're poorly done, chopped out in haste, but convey the message. Please note that the thread, in this case, is ¾"-16. Thread size is, of course, optional.

The dimensions presented are adequate for mounting a wheel no thinner than ½" thick, but allow for a wider wheel, up to 1". You would narrow the 7/16" dimension if you wanted to mount wheels of less than ½" thickness. Should you make any alterations, make sure that the nut can tighten the flange against the narrowest wheel you choose to run. The grip length in any case should be slightly shorter than the thickness of the moveable flange.

You may have to do a compete redesign for your needs, but adhere to the flange diameters, relief and locating boss diameter, to keep you out of trouble. The balance of the features aren't critical. Concentricity and perpendicularity is always important.

There are almost no dimensions on the side that mounts to the motor, although I did dimension it for a ½" shaft diameter. The OD of the adapter should be adjusted so regardless of motor shaft diameter, you can install set screws that are at least ¼" long. Regardless of diameter, insure that the set screws are flush, or below the surface, when tightened. Lengths should be determined by the length of the shafts on your motor, thus there are none included.





I use a ½" wide wheel, which allows for grinding (relatively) short boring tools. As the wheel gets wider, it forces you to grind a wider and wider relief area behind the cutting edge, or, said another way, the shortest tool you'll be able to grind will be a function of the wheel width. It's nice to be able to grind them short, especially if you use small tools, say 3/8" or less. That keeps the tool rigid and chatter at a minimum. You can get around that problem by using the corner of the wheel to grind the relief, but that leaves a tool that looks like it's been chewed out by a gopher. I don't like my tools to look like that. Sort of detracts from the level of accomplishment I've achieved in my trade. If I was to suggest a "perfect" wheel, it would be almost identical to the one on ebay: 7" x½" x1¼" 38A60H8VBE, Substituting an I for the H would be very acceptable. Fact is, until I mounted the H I'm running now, I'd always used an I. Found I liked the H a little better, although it's quick to break down. Keeps the wheel cutting really fast, though, so roughing a tool goes very well.

Grinding chip breakers on HSS toolbits

-------------- Extracted from: http://www.chaski.org/homemachinist/viewtopic.php?t=75969&start=0

I had offered to talk about HSS tool grinding, and did so with hesitation. My reservations were well founded. It's interesting how a person can do things on a routine basis, with success, and be at a loss to explain how or why, but that's where I find myself. It's nothing short of shocking to be without words, particularly when I rarely have a clue that it's time to shut up.

Grinding toolbits, for me, has been routine for so long that I have no memory of not being able to grind them. The skills involved were accumulated in practice, over a period of time, with the tools put to use in a production environment. By applying the tools under demanding circumstances, tool design was determined and performance evaluated on the machine. From this, I'd like you to understand that there is no substitute for experience. You can not learn the things that make one a good tool grinder from reading, you must experience them, and see the end result in order to understand the effects of subtle changes in tool design that spell success, or failure.

Functional turning tools are ground to a mental picture, with relief angles ground to fit the requirements of the task at hand. While these angles are important, they are not carved in stone, and vary with circumstances. It becomes a matter of course to grind in such a fashion, so no guides, aside from profile gages, are necessary. That's not to diminish the importance of relief angles, but they are flexible enough that they are not a serious concern unless one finds himself in a production environment, where a maximum performance over a long period of time is a requirement.

Often times, an alteration of an angle, particularly a rake angle, will allow for better performance for the short haul, often long enough to complete a rather unpleasant task. Turning the corners off a stainless plate cut from a square is a good example. Excessive positive rake, coupled with a very light feed, using a plunge turning technique changes the job form one of misery to one with few, if any, problems. It's easy enough to achieve functional angles without gauging, so the eye should be trained accordingly.

Having said that, I still need to say enough to help those that struggle. Which ones, and for what reasons, remains a mystery, so I'll start the ball rolling and try to answer questions if there are any. Best thing for me to do at this point is to talk about basic machining concepts, and what influences performance. The principles remain pretty much the same as you move from one type tool to another, taking into account that working in bores requires a serious rethink of front relief angles, which are influenced by bore size.

Tools that are ground with simple angles perform reasonably well, but often with higher cutting pressure than might be desired, and they rarely have any features that manipulate the chip. Surface finish and production, under those circumstances, can suffer. Chip control is often just as important as the cut itself----and must be dealt with accordingly. Faster operations are one of the benefits, as is safety through the elimination of stringy chips.

The mechanism by which this occurs is a ground chip breaker. Such an addition to your tool opens the door to greater rake angles, lower cutting pressure, and chip control Chip breakers can be ground with sufficient positive rake to seriously improve cutting characteristics and still have excellent edge strength, due in part to reduced heat at the point of cut, a result of positive rake. An offset to the positive rake is accomplished by grinding a cylindrical breaker, in which the chip reverses direction quickly after leaving the cut, more or less counteracting the effects of positive rake (hogging). The radius of the chip breaker, in conjuntion with the proper amount of front relief, tends to control hogging and chatter. Properly ground, the breaker will roll the chip, yielding either C's, 9's, or an endless coil. The coil isn't as desirable, but from the standpoint of safety, is much better than strings, and is often much easier to dodge than hot C's or 9's. There are times when a coil becomes the chip of choice, such as when boring a hole that has minimal bar clearance and it's desirable for the bore to not accumulate chips.

Metals, for the most part, respond to being "peeled" (shorn via positive rake) much better than they do to being displaced by brute force (negative rake). For those that use HSS, not carbide, the advantages to using reduced rake are few. It is not an option when machining steel, and offers no advantages when machining aluminum, so it's pretty much restricted to use in soft cast iron and free machining copper alloys, in both instances to control hogging, with no real advantages in machining aside from a prolonged edge when machining cast iron. In those examples, it is generally accepted that rake be reduced to 0, with no real need to go negative. Chip breakers, by sharp contrast, can be applied under almost all circumstances with huge benefits, except for machining free machining copper alloys, and, once again, soft cast iron.

Chip breakers are sensitive to speed, depth of cut and feed rate. Therefore it's nearly impossible to describe one that suits all purposes. Again, the basic principle remains a constant, so all one need do is make subtle changes to the breaker to improve performance when it does not perform to expectations. Likewise, a change of feed rate, spindle speed or depth of cut often will yield good results with an existing grind.

A narrow, deep breaker will usually yield poor performance, although it might serve for finishing cuts. Armed with that idea, one would widen a breaker that did not permit free flowing of the chip. It is also desirable for the chip to have the ability to flow freely, with a subtle change of direction such that there are no corners to trap the chip and prevent flow. A gentle radius is the most preferred design, although other configurations can work. By choosing a radius, positive rake can be introduced to the cutter without building any traps, with minimal effect on heat transfer. .

Please take note: The information I'm providing is pretty much useless for rocker tool post use. Between the included positive rake of the post, and the angle necessary to keep the post away from the cut, the geometry is totally different. Tools I recommend are intended to be mounted in a holder without rake, and at right angles, or very near right angles, to the turn. Both of those features become impossible with a rocker post. The details on chip breakers can, indeed, be applied to tools ground for use in a rocker tool post, but the extreme angle differences must be taken into account. In general, poorer performance can be expected as compared to holding cutting tools in a different post.

In order to facilitate the presence of a chip breaker, tool design is usually slightly different from conventional grinds. Where the included angle of the top of a right hand turning tool might have been in the vicinity of 60° for conventional grinds, an angle of 80° is more desirable, assuming it isn't in the way of the use of a center. In such a case, the tool is generally relieved to clear the center, leaving the bulk of the material that isn't in the way. By providing the greater angle, there is room for the full chip breaker to be ground without sacrificing much, if any, of the rear portion. That keeps the chip breaking properly, but plays no role in how the cut behaves otherwise. The bulkier tool is somewhat better at handling heat, so that is a side benefit.

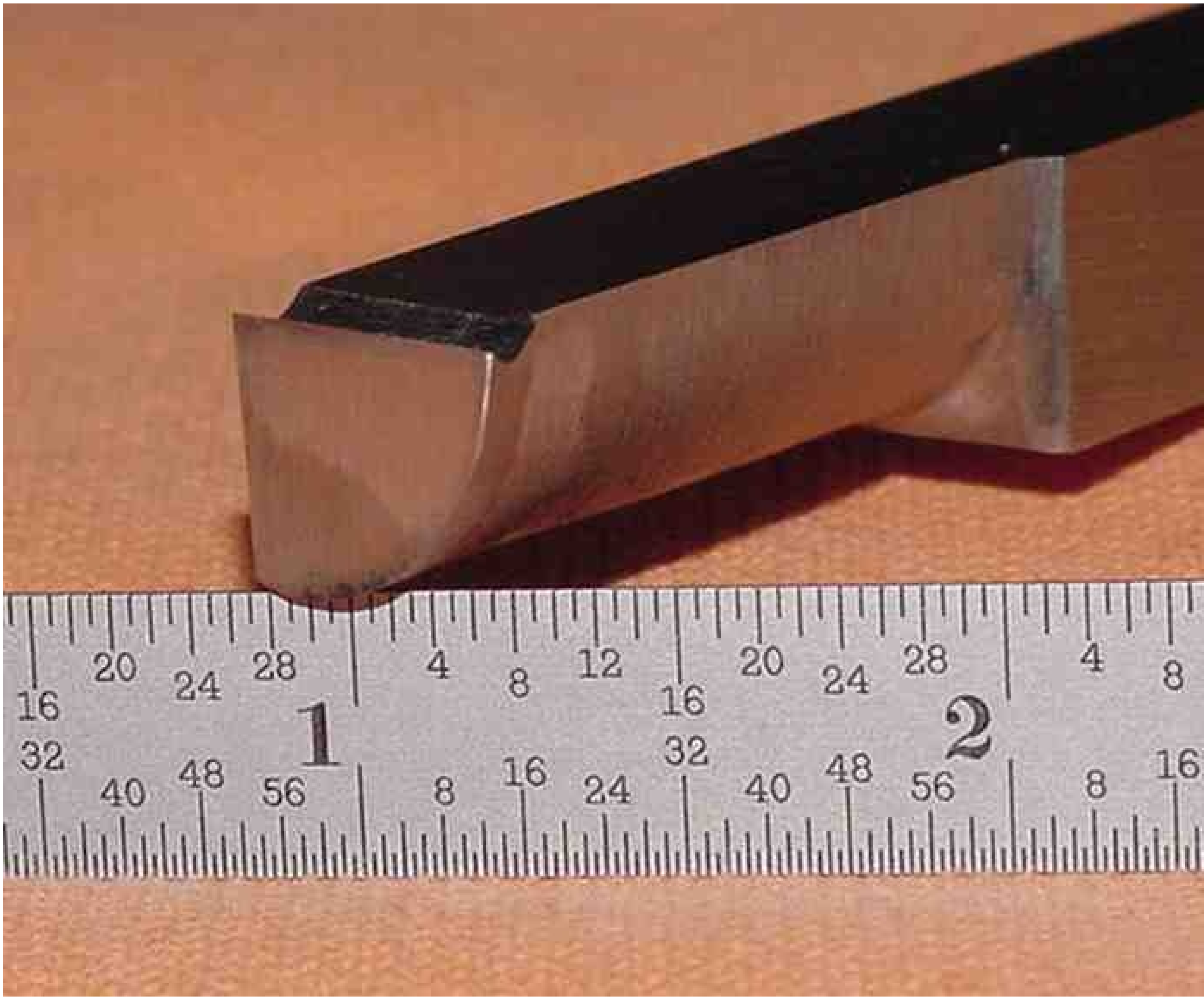
For starters, lets consider the tool, below. It's a boring tool, one that is very useful in removing stock rapidly. The cutting edge is ground at a slightly less than 90° to the shank, so that the tool is able to bore to depth and face. For through bores, the tool can be mounted at a slight angle, so the cutting edge has a minor amount of lead. This design works exceedingly well in both circumstances, so there is no need for two different tools for boring. The cutting edge in this case is the end of the bar, which I will reference as the front. The relief angle will work as low as 5°, and up to 8°, and is affected by feed rate. You'll notice as you alter the angle, the tool will have more or less tendency to hog, and often chatter. You can use that feature to help control the performance of the bar.

Side clearance is in keeping the requirement for the tool of choice to enter the hole for which it's intended. There is no point to giving specific guidelines, for each hole has it's own demands. Just make sure that the heel of the tool, with the cutting ege in contact with the bore, has clearance. 5° or more beyond the radius of the part is very acceptable.

Pay particular attention if the tool is very close to bore size, so the back bottom edge of the bar doesn't drag in the bore. Remember, none of the angles are critical, so long as the tool is kept within reasonable boundaries, it will function very well. Avoid extreme clearance angles to help control hogging and to provide the maximum edge strength possible. If, by chance, you find the tool does not cut freely, inspect the tool for signs of dragging in the bore------and check the chip breaker to insure that the chip being generated is flowing freely. Chip stack up will quickly snap a tool if left unchecked. Chip welding alone can be the source of much grief. Use a tool of this design with lubrication. Brush application is adequate, just don't run a chip breaker dry. If, when machining aluminum, you find it has a tendency to streak off on the breaker, it's always a good idea to spin a small cylindrical stone in a chuck and apply the breaker to the stone to polish the surface. The smoother the better. Be careful to not round the cutting edge should that be necessary.

The chip breaker is ground freehand, using a standard grinding wheel with a radius dressed on the corner. Alternately, an old wheel can have a radius hand dressed and used exclusively for such grinding. That eliminates rounding the corner of your wheel, prolonging the useful life. The radius on the wheel can be smaller than the one you desire, but it is not as easy to get a uniform grind.

I also suggest that tools be ground without the use of a rest, which I will address in a different post. Lets see if this has helped anyone, or if it has raised any questions that may need to be addressed. The next tool will likely be a right hand turning tool. Once you have seen a couple designs, you should be able to envision a tool of almost any design, and grind accordingly.




4 8 12 16 20 24 28
32NDS
64THS
8 16 24 32 40 48 56
1
4 8 12 16 20 24 28
8 16 24 32 40 48 56
2

Material? Thickness of wall? Diameter of bore? Tolerance? Fighting chatter? Through bore, or to a shoulder?

Quote:
I normally use brazed carbide boring bars and when sharpened right they aren't bad but they require a lot of pressure and high rpm.

That's because they generally lack proper geometry for free machining. If readers come away with nothing more than the understanding that a proper chip breaker changes the way a tool performs, almost always for the better, each will have learned a valuable lesson. Relief angles on tools play a far less important role than some may have you believe. What really matters is how the tool sees the work-----rake. It also often dictates tool life----so there are times when one must compromise.

Quote:
I've got one of the bars for HSS that uses a short piece of 1/4" toolbit sticking out at 90*. I've tried a lot of different profiles but haven't had the success I should. I think I know what you will say...I should grind a narrow chipbreaker groove along the length of the short bit...correct?

I'm inclined to say yes, but without knowing more, I hesitate. Thin wall work can be exceedingly tricky---whether you're turning or boring. I'd want to know a lot more about your setup, including the nature of the bar, diameter (of the bar) , if you're running on center, speeds and feeds, and, of course, what material, what tolerance, and what kind of surface finish you're trying to achieve. Sort of a repeat of the above things I mentioned. It's all important if you're trying to solve the mystery. Some things can be unreasonable and make the task border on the impossible without addressing the job from a different perspective.

Quote:
That's what you told me for a facing tool for alu and it worked well. I never thought of doing this with a bit for steel.

That's the beauty of chip breakers. They work on almost everything----although hard materials shorten their lives considerably. Tough materials can yield surprising results, however. It's important that you understand the difference between hard and tough. They're not the same thing.

The nature of chip breakers for steel as opposed to aluminum would be a slight reduction of rake angle----and if the machine is light duty, width and depth may have to be adjusted accordingly. A shallow cut with fine feed would require a narrow breaker, and shallow, in order for the chip to curl. By contrast, a deep cut with heavy feed would require a wider and deeper breaker. Rake angle may have to be altered to prevent edge failure----which can be accomplished by using a larger radius for the breaker. As the radius increases, the angle diminishes, and it isn't nearly as deep. These features are best determined by trial and error, so the tool suits the exact conditions in which it's being employed.

Hyper wrote:

Which do you grind first, the chip breaker or the cutting edge.

My approach is to grind the entire tool first, leaving room for touchup.

Reason? Even when using a grinder that I recommend, you can't always see what you're grinding when you're grinding the chip breaker. You run the risk of rounding the cutting edge, or generating a breaker that doesn't parallel the edge as you might like. By leaving a small amount (locating the breaker slightly back from the desired cutting edge), you can then grind the cutting edge to the chip breaker. It's clearly visible, and easy to follow.

By the way, the tool that Kap mentioned for roughing steel, would be ground exactly that way, and it was an excellent tip from him. I rarely grind my (turning) tools that way, but only because I usually go to negative rake carbide, using molded breaker inserts. Interestingly, they are made the same way---with a shallow breaker behind the edge. It need not be at the edge to function----I do that only to increase positive rake and reduce cutting pressure. It can work against you in steel, as Kap so kindly mentioned. Should I use my design in steel and have a problem with edge life, I usually reduce the rake angle instead of move the breaker. Not better or worse, just my way of working.

Quote:
Also, do you have any suggestions for chip breakers when working with something like Delrin? (for turning & boring).

Absolutely! Duplicate what you see in the posted pics, and, if anything, increase the rake angle. Make sure the chip can flow freely, and run the machine flat out, assuming that doesn't present any problems. Delrin is one of the finest of all plastics to machine and responds to positive rake in ways that are hard to understand. Keep the tool exceedingly sharp to avoid chipping the material. It's not uncommon to have a string chip flying over your shoulder, landing behind you. It may or may not break, but that is of little consequence with Delrin. If you insist on breaking it, reduce the width of the breaker (or deepen it), but stay with all the positive rake you can achieve. There are no heat issues with turning or boring Delrin, so the fragile edge isn't a problem.

Quote:
I had thought about a chip breaker that was more angled instead of radiused, but haven't tried that yet.

Regardless of the nature of the breaker, you need a slight radius at the point of contact that determines the diameter of the cut. The radius should be a function of the feed rate, to help blend the feed lines. The angle you spoke of can work for or against you, depends on circumstances, but a sharp corner will still yield a relatively poor finish. Only with a narrow trailing straight portion or a radius would it not.

Because of the serious amount of positive rake, the compound angles you generate when applying a radius to your tool can create problems. The cutting edge tends to fall well away from center. If you keep the tool sharp, and the radius to a minimum, you should have excellent results. Unless you're machining a material that is notoriously bad for finish (C10108, for example), this machining procedure should yield good to excellent finishes. It stands to reason that you should select proper feed rates for finish cuts.

Should you fail to get a decent finish, a quick look at the tool will usually tell you what's wrong------ and if it doesn't -------you need to keep studying until it does. Remember---you will not learn to grind functional tools by reading---that serves only to guide you while you learn to apply the geometry. You can follow the recipe and still get a non-functioning tool. Experience helps identify why.

Quote:
Can you show (when you have time of course) examples of left/right bits

I will likely grind and discuss a second tool, a right hand turning tool. By then, if the reader is still having trouble, all the discussion in the world won't help. Grinding toolbits is no different from playing the piano. You can read and read, but until you tickle the ivories, you won't know how to play the instrument.

The guidelines I've set forth, and will mention again when I get to the next tool installment, rarely change. The basics of cutting tool geometry repeat, regardless of how the grind is applied to the tool. Armed with that idea, you should understand that if you desire to grind a tool that has not been discussed, the only decision you need make is how you want the breaker ground on the tool. The rest is up to the circumstances at hand----material type----depth of cut, feed rate, spindle speed----all of which drive the design. Again, this is something that comes from experience, and the only way you can get that is to get your hands dirty. Once you understand the concept, grind away and look for the results you desire. If you don't get them, alter features until you do. It's something that you learn quickly if you have the opportunity to use it long enough -------which was the real advantage in learning on the job. Some jobs we ran would last for a few days-----an ample learning curve, which provided knowledge for the next job. You'd be surprised how quickly this thing comes together if you give it a go.

Quote:
and threading bits as the way you would sharpen them? Thoughts on the Aloris threading bit that comes with their QC sets?

I am not familiar with the Aloris threading bit, so I hesitate to provide the slightest input. I don't use an Aloris type holder, and haven't for more than 39 years. My preference is a specific (OK Rubber Welders) square indexing block. Perhaps someone like Kap could provide some useful information. Kap has some serious experience to his credit and is very knowledgeable about the ways of good and proper machining.

I've always hand ground my threading tools, no work rest, using a fishtail as a guide. In order to see the true profile of the tool, it should be viewed with a light behind the tool and gauge, looking for light that spills through any gaps that are found. Trusting a view without this process tends to yield forms that are not true.

The sole exception to my hand grinding threading tools would be those rare times when I had a surface grinder at my disposal, with a threading tool block that established the proper angles and clearances. That makes the task real easy, although it does nothing toward promoting your skills. When my shop is setup and running, I fully intend to make such a block. I don't need any more practice. It's time for my tired eyes to do some things the easy way.

Know one thing about threading, or any form tool that is not a right angle. The moment you introduce rake, you move away from the proper configuration, and risk losing acceptable form. With threading, that is a particular problem, due in part to the rather narrow band of tolerance, plus there are some trick things you can do to end up with a very nice thread, but plunge cutting is very much a part of it-----so any rake ground on a threading tool not only affects form, but limits the way you can use the tool. Unless you're using insert threading tools that are ground with compensation for any deviation from 0 rake, never use a threading tool that has rake applied if you expect proper results. The tool may function, but the results will be questionable.

Harold, Could you show a sketch of how the bit is orientated wrt the work? I guess the bit is parallel to the axis of rotation of the part , due to the curve on the side but is it also "tilted" sidewise? For a deep bore how
does this work?
Thanks.
...lew...

Lew,

As you alluded, the tool that is posted would be mounted very close to parallel to the bore, with no lead if boring to a shoulder. The tool shown addresses this issue, considering the included angle of the tool is less than 90°, which allows for facing. My usual practice is to mount the tool so it is not quite parallel to the intended face, with the tip the highest point. That permits machining a square corner and a facing cut towards center. By proper configuring the tool, that allows for a slight angle when mounting the tool in the post---with the heel farther away from the operator than the point. That helps prevent chip accumulation by providing additional clearance between the bar and the bore towards the shank.

Through hole boring is a different matter, but can be accomplished with the same tool. Depending on bore size, you may or may not be limited by the relief ground on the tool. Often the angle can be set such that the tool can enter the bore past the relief, but that is determined by bore size. A large bore that provides ample clearance makes that possible, whereas a bore that is small as compared to bar size would not allow for the same setup. Setting the tool in this manner provides a lead that is quite desirable when moving metal. It also often helps direct the chip forward of the cut, a very desirable thing if you don't want chips in the bore. Sometimes you just have to tough it out-----chips are known to do as they please.

You should have total flexibility in the way you use a tool of this nature. As long as it will accomplish your goals, there are few restrictions. One important thing to keep in mind is to keep the tool on center at all times, but if you must deviate, go above slightly, never below. Boring above center often yields a few benefits, especially when the bar is a tight fit in the bore. Use it with restraint-----you're always better off on center. And don't hesitate to experiment-----that's how we learn.

Boring tools such as I have posted have limited depth capabilities, and are in no way intended to replace a long boring bar-----but they replace them with far better success when the work allows for their use. They also often open up a position on an indexing block that otherwise would be covered by a too-long boring bar. The design of tools of this nature are famous for moving metal. That's particularly true for the guy that has a fractional hp machine that isn't otherwise capable of serious cuts. Small machines benefit from this tool design to a huge degree. The only caveat is if the machine is too light and the positive rake creates hogging. Guys with small machines should approach the use of this design much the same way one would when machining brass. If hogging is a problem, it can often be corrected by altering front relief, and a lessening of rake angle and width of the breaker*. As I said, experiment. You won't learn much from reading.

spro wrote:
The chip relief info is then going to how to generate that without a straight wheel to start that cut and what happens after as radius increases on the wheel edge. This does not mean to distract but a thought about your grinder.

Spro,

If I follow your point, which is a good one, you're suggesting that a grinding wheel would gradually evolve with a radius that is greater than may be desired for a chip breaker to be ground to desired specifications. If so, that isn't usually a problem. When you use a wheel that is suited to grinding HSS, it gets dressed on a fairly regular basis. While I didn't count, I dressed the wheel no less than five times when I ground the tool pictured. As a result of the fairly constant dressing, the wheel tends to keep a relatively sharp corner. Fact is, if you can grind such a tool without dressing the wheel, the wheel is magnitudes too hard, and grinding will a slow, laborious process. You'll come to understand that HSS grind quite well if you use the proper wheel, which I'll address in a future post.

Quote:
Many as you know have on the right side a thread which fits a Jacobs chuck. This secured chuck can hold what you need for individual reliefs by means of expendable grinding or cutoff wheels. This would not be promoted but is done.

How one achieves a chip breaker isn't important. I grind almost all of them with the corner of a wheel, or by installing an old, worn out wheel that has been dressed with a desirable radius, which was the case with the tool in the original post. The negative in these instances is that wheels are formulated to run @ given surface speeds, and start losing hardness (not really---but they behave as if they do) as surface speed diminishes. That's not all bad in that you can often grind a chip breaker without overheating the edge. You experience rapid degradation of the wheel, but that's a small price to pay for the successful grinding of what is usually an excellent cutting tool.

There are times when the side of a wheel is not capable of grinding a desired breaker. I've included a picture of such a case, below. Sorry for such a lousy looking tool, but it has sat for years in less than ideal conditions, and it shows. The change in angle of the tool eliminated the possibility of grinding it as I normally would. It was ground many years ago, so I can't say with certainty how I achieved the breaker, but I am inclined to think I used an air grinder with a mounted point. Any port in a storm, so to speak. How you get there isn't important, not so long as you achieve success, without rounding the cutting edge.

The Jacobs chuck mounting you speak of would be particularly bad, although functional, if one uses mounted wheels for grinding chip breakers. Small wheels of that nature require speeds well in excess of 10,000 rpm in order to perform properly, but, again, that's of little significance if they serve the purpose.

It is 2" sch40 pipe(2.20 od if I remember right). The bore is 4 1/2" deep.The final wall thickness is .090.
I have about 3" sticking out of the chuck. I turned the od first....maybe not right but I wanted to see if it would distort before going any further.

Pipe is inclined to machine quite poorly, so you may be experiencing some of what goes with machining material that isn't really intended to be machined.

Finish turning the outside before roughing the inside is always a mistake. Parts, regardless of their nature, are best fully roughed before any finish cuts are taken. That allows any stresses that cause movement in material to be relieved as much as possible, then the error created by the movement is removed in final machining. That's pretty much industry standard, at least prior to CNC operations. I worked light production work for years, and that was pretty much the accepted practice. The added benefit is that you remove scale prior to applying your finishing tools. It's hard on edge life.

A tool in keeping with the design I showed will improve your performance, particularly for finish cuts. Again, it's important that you get all the scale out of the pipe before attempting finish----it's terribly hard on tools, especially HSS, which is far softer than carbide. The positive rake helps shear the material------instead of tear it.

You might consider following Kap's advice in this instance, at least for roughing. His modified flat chip breaker increases edge life considerably under harsh conditions, although you lose the benefit of shearing action from positive rake. The chip breaker would allow for chip control, and the narrow flat area will provide a much more resilient cutting edge, albeit at the cost of higher machining pressures. Go to the high rake for finish, after the scale is removed.

By the way, this is a great place for using soft jaws. You're going to have fits with a wall that thin, holding the part adequately without crushing it. Soft jaws (wide ones) prevent crushing, with the added benefit of minimizing chatter. Give it some thought.

'd like to point you to a post I provided long ago, in which I discuss the use of soft jaws. They are, in my opinion, the magic bullet of lathe work, and would be the perfect solution to holding your project. If you're inclined, please check this link:

http://www.chaski.org/homemachinist/viewtopic.php?t=4266

I had posted some pics originally, but I don't think they'll display now due to changes in the software the board has used. If there's any interest, could be I can re-post them. I still have them available, but I don't know if I can add them to such an old post. If not, I'd be pleased to send them to you on the side if you feel they'd be useful.

In the case of pipe, I'd make wide and long jaws that support it almost 100%--and use the "spider" I spoke of for fine adjustment. With the soft jaws bored to accept the rough OD of the pipe, I'd bore the pieces within .03" of finish size, then make a setup to rough turn the OD, again, within .03" of finish size. At this point you should have relaxed the majority of stress pent up in the material, and can go for finish sizes, starting with sharp tools. I'd turn the OD to size first, then re-bore the soft jaws to fit. Finish cuts would be virtually perfectly concentric (should be within .0005" if you've done your work well), and chatter would be a non-issue, due to the full support of the tube. Because you would desire full contact of the jaws to material, it's important to hold all the pieces identical in size, say within ± .001". That way the radius of the jaw will match the radius of the pipe and not distort it. I talk about jaw to part fits in the referenced post, so that will give you proper guidance on fitting the parts to the jaws.

HSS Right Hand turning tool with chip breaker

---------- Extracted from: http://www.chaski.org/homemachinist/viewtopic.php?t=76447&highlight=grind

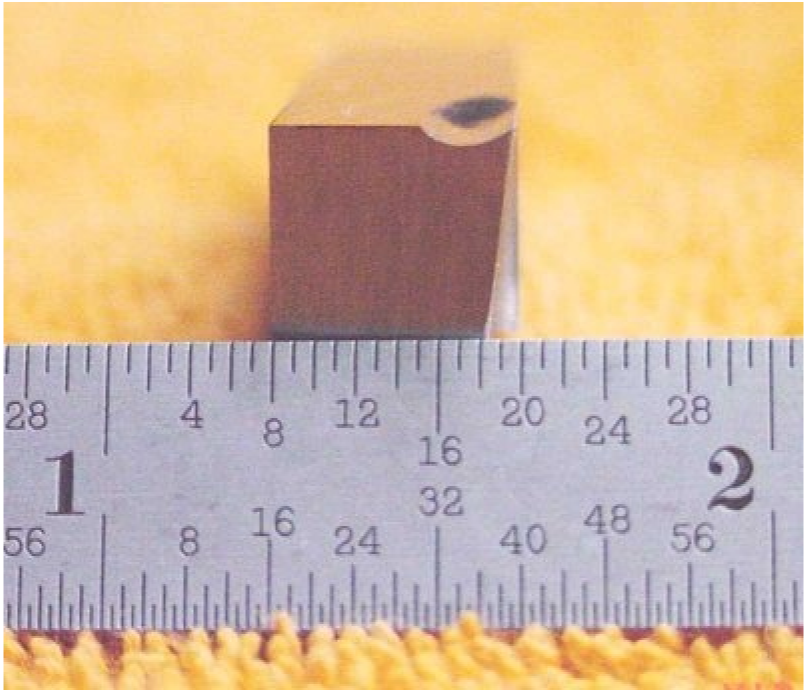
A while ago I posted on grinding chip breakers, and threatened to post a picture of a right handed turning tool. In the previous article http://www.chaski.org/homemachinist/viewtopic.php?t=75969&start=0 I mentioned that regardless of the nature of the tool, features are almost identical. The sole exception being boring tools, where front relief must be greater to compensate for the radius of the bore, so I felt there was little to no urgency to post further pictures in that anyone that understands the principles involved can easily grind any type tool needed, using the guidelines I had set forth.

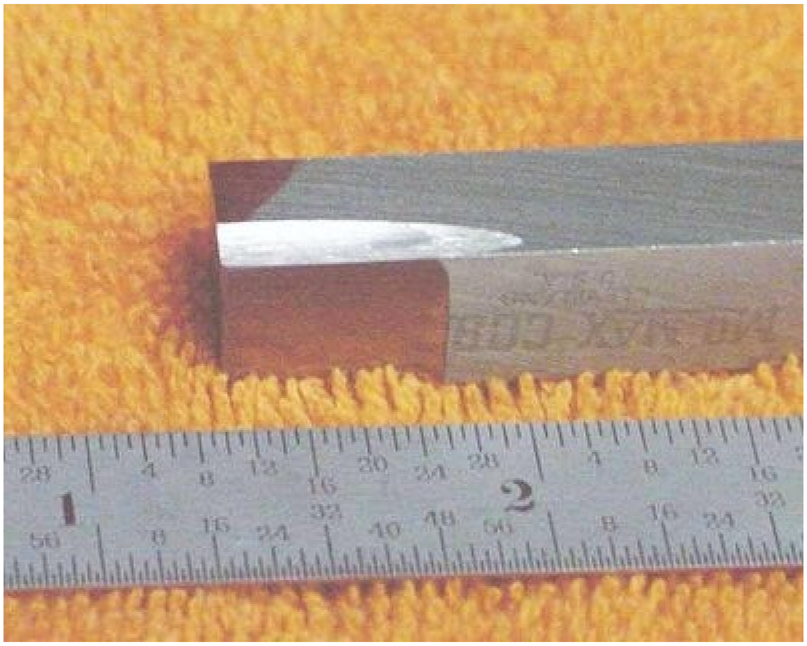
Due to a recent gentle inquiry, below you'll see three pictures of a right hand turning tool, ground on a piece of 3/8" square HSS tool stock.

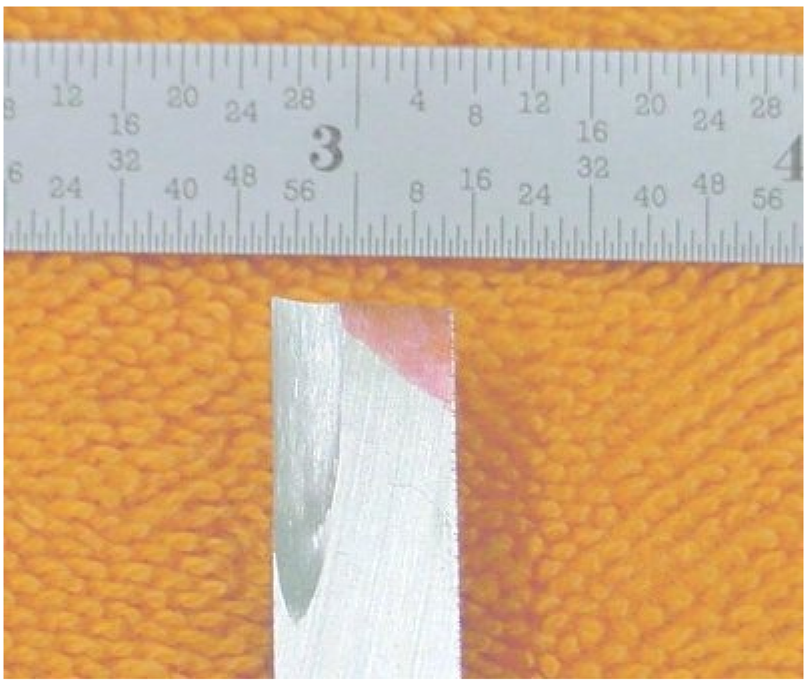
The reader should understand that the tool shown isn't ground for any particular purpose, just an example of what a tool might look like for a given project. Width and depth of the chip breaker will vary according to needs-----wider and deeper for some materials, feeds, and depths of cut, shallower and narrower for others. It's virtually impossible to set rigid guidelines due to the wide array of machine tools in use, plus the variety of metals on which such a tool might be applied. Please follow the above link for more information in that regard.

Please note that in the top view of the tool, there appears a red area. Should you use such a tool with a center, and find it doesn't provide the necessary clearance, the red portion can be removed to accommodate the center. It's a good idea to leave the entire chip breaker portion to assist in rolling and breaking the chip, but the relief can be ground almost to the point with some success. Leave as much as possible if you must grind the relief. If nothing else, it acts as a good heat sink when machining steels.

You will get good at grinding such a tool only by practice. There are no magic bullets, and grinding fixtures will lend little help-----it's a matter of training your hands and eyes to do the work. The chip breaker is best ground on a grinder without a work rest----as is the entire tool.







Resharpening is generally accomplished by grinding all faces a little, or only a face in need. The condition of the tool when resharpening would be required would dictate how I'd sharpen it. Pushing back the end would work, assuming you haven't dinged the sharp edge behind the area in use, but that's usually a tall order if you're not machining aluminum. By regrinding all faces slightly, you can keep the tool sharp and extend its useful life through several cycles. The one negative is that you have to reposition center each time the tool is ground.

Assuming you are machining aluminum, the tool will usually undergo very little wear----so a honing is all that is required to keep the tool sharp through several cycles.

You may have noticed (perhaps not) that I grind my tools with only one face-----they are NOT chopped up with a multitude of angles. The advantage, aside from looking less bush-league, is that you can use the ground surface to great advantage in honing. The grind is almost always accomplished with the periphery of the wheel----so the bottom and top edges are parallel, with a hollow grind between them. When you hone, the stone will ride on the top and bottom edges only, preventing rounding, but most importantly, presenting a very small area of material that removes reasonably quickly. That allows for keeping a keen edge on the tool without the removal of excessive metal.

Tools of this nature, when properly ground, work very well on steel---although you must play with the dimensions of the chip breaker to establish what works best. As I've stated on other occasions, speed, feed, depth of cut and rigidity of the machine in question all play a serious role in the requirements. Only experimentation on your part will solve the riddle.

awake wrote:
1) Does this design permit a nose radius? I don't see a radius in the pictures, and it would seem that putting a radius on the tip would mess up the geometry of the cutting edge, given the large degree of rake induced by the chip breaker.

Good observation. You have to be a bit easy on a corner radius with this type grind, for the exact reason you mentioned. The tool pictured has no more than a .005" radius honed on the tip. You can get away with more, but not much. It comes with the loss of rake angle, but only as deeply as the radius is large. Pressure wouldn't increase substantially, so with a deep cut, not much is lost. The tool would tend towards falling flat for shallow cuts.

For the record, such a tool isn't really intended as a finishing tool, although very similar grinding will yield a fine tool for finish work. The broad chip breaker is intended to curl a reasonably heavy chip that comes from a deep cut------sort of a lesson in moving metal, if you get my drift. Finish, in this instance, isn't critical, and won't be great. That's not the tool's purpose.

Quote:
2) Would you feed this tool in angled slightly to the right? Otherwise it would seem that one would need to grind more of an angle from the nose back into the red area (side relief?? I can't remember what the right term is).

The tool, as ground, could be used for turning and facing, so that should tell you about the angle it would be mounted for use. A slightly greater angle on the trailing edge could prove to advantage if you found you experienced problems with chips getting caught behind the tool. That's not uncommon if the breaker fails to produce nice C's or 9's. Again, such a tool is intended for roughing----a finishing tool would require more clearance behind the cut to minimize the chance of chips ruining the finish.

Surprisingly, the tool will face in both directions, but you get a huge area in contact when facing out.

Quote:
3) As I have done a little experimenting with this chipbreaker design (in a boring tool), I have found that it seems to cut well but the edge does not hold up as well. Should I expect that, or is that evidence that I'm doing something wrong?

You're likely doing nothing wrong. I've already commented on the fragile edge created by the positive rake chipbreaker.

If you're inclined, go back and read the original thread, looking for a post by Kap. He mentioned leaving a narrow flat between the breaker and the cutting edge. That's common practice to increase tool life, but it comes at a price. For one, the tool will cut with greater pressure, so more heat will be generated. The advantage of the design I've demonstrated is to reduce cutting pressure, and to control chips in the process. It's sort of a balancing act between stock removal and tool life, plus safety.

It has been my policy when using such a tool to take advantage of the positive rake, so I don't grind my tools as Kap suggests. It's not that it's not a good idea to do so, it definitely is-----but I also have a lathe that is suited to running negative rake carbide, which I use to great advantage when roughing. I'm more inclined to use the HSS bits for aluminum work, where I can take huge cuts, often greater than the insert tooling I have is capable of taking with the same degree of success. I think that each person will experiment and come to terms with what works best for them under the circumstances at hand.

One thing is sure------the tools I've demonstrated, when properly ground, will be very revealing for those of you that have always struggled with stock removal. I can't think of anything I hate more than facing the removal of a couple inches of chrome moly, knowing I can't take any more than a few thou per pass. I generally handle such a need by using a feed in the neighborhood of .012", and a depth of cut of .200" (per side) @ 500 RPM with material in the 2½" diameter range. You can get metal off in a hurry that way. That, of course, is using negative rake carbide!

I worked in precision grinding for several years, and am very comfortable with hand dressing wheels. I don't buy thin wheels, but use my old worn wheels instead. When they're too small in diameter to grind efficiently, I'll put them aside and use them for grinding forms---including chip breakers. The small wheel is often the key to getting in, plus the slower surface speed grinds cooler, so you don't overheat the thin section of the tool as easily. Remember-----grinding wheels behave softer as surface speed decreases.

I also use the corner of the wheel on many occasions. The tool configuration often dictates the method I choose when I grind the breaker------for a boring tool, it's always with the corner of the wheel. The wheel need not have a proper radius, you can grind the desired form with a reasonably sharp corner on the wheel if you practice offhand grinding regularly.

On Parting Tools:

I use the longitudinal chipbreaker on the top of the tool for the smal lathes at work. It seems to help pull the chip out for a freer cut with limited power. The L&S doesn't care.

Harold:

"I've hand ground parting tools as long as I've been in the shop. The longitudinal grind is exactly what I use, and have done so with outstanding success. It not only provides for modest positive rake, it also narrows the tool slightly so it doesn't bind in the cut. Given the proper speed and feed, its not uncommon to see the chip come off the machine in a full coil, turn on top of turn, building in the groove that's being machined.

I rely on all faces of the tool to establish proper clearances so as to maintain the greatest amount of rigidity in the tool. Parting, especially in sticky materials (cold rolled, for example), can be very trying. You learn to use every trick in the book to get tools to cut properly. "

These tools won't cut well sometimes, in their original shape. I changed the front relief to nearly zero degrees and added a grind to increase the back relief angle, acting as a chip breaker or chip

curler.
The trouble with grinding into the top surface of a tapered blade is, that the width is reduced slightly due to the taper. The tip of the cutter needs to stay very near the top edge.
While I have few problems with parting now, the finish on the faces of the stock looks rough. I'll need to try some other changes to see if that can be improved.

Reducing the front angle essentially prevents the blade from being able to dig into the workpiece, which can happen with some steels and softer materials. If the blade gets to the point where it can dig in, the nearly flat or perpendicular front angle can start to rub, preventing a stalling jam. The rubbing could be a big problem with materials that can workharden, but feeding just enough to keep the curl continuous prevents the rubbing.

Slow speeds and an adequate amount of cutting lube give the best results for me. No surprising jams, which would happen sometimes with manual cross feed. The rate of cross feed on a power feed lathe would very likely be a factor for successful parting, for those that have power feed. Having variable speed for the spindle is especially useful for parting.

Another practice that makes parting easier, is for parts that will have a center hole, make the center hole before parting, and this also eliminates the protuberance or nub from parting solid stock.