

Brass Bussings

brass bushings go inside another piece, usually they only give you .003-.005 thousands of an inch of tolerance on the OD and ID.

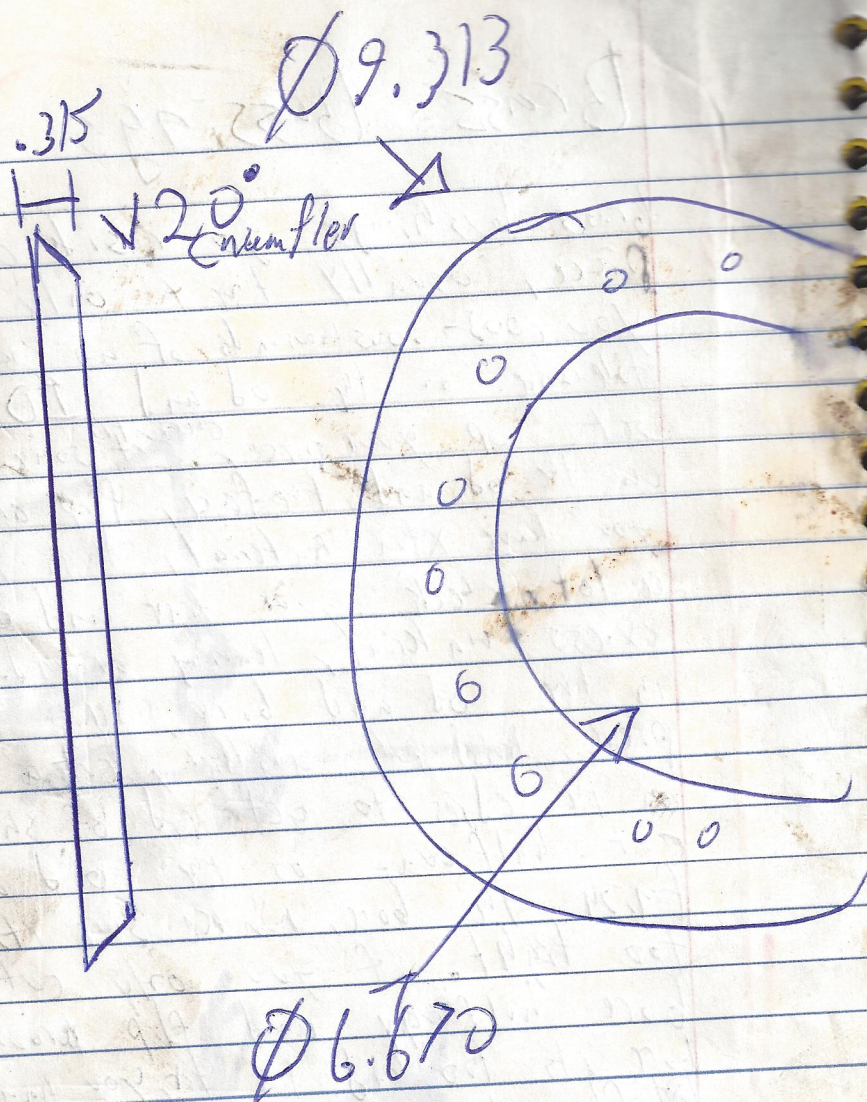
set up your piece ^{cut the} ~~cut~~ some material on the OD and the face, flip around if you have extra material, though it is a lot, check your fit and cut excess material, leaving around .100 in the OD and bore, release the piece, hold it together, cut a 45 on the edges to get rid of sharp edges.

cut halfway on the OD and finish the bore, make sure it's not too tight. if you only cut the bore halfway and flip around and bore it's too big for fit, you have to flip the piece back again and finish on that side.

if the fit on OD is bad, put the fit as close as you can to the cut as much material as you can to reach the OD on the other side.

~~Final~~ sand down the rest.

→ see guest grows.



1. Cut face 3" from the top of the piece to the center

2. ~~mark~~ Cut some on OD

3. mark 6.600 with marker and cut a groove .350 deep

4. Cut face groove slowly as deep as you can

flanges

flanges are ring shaped parts that go on the side of other parts to hold belts in place. usually are thin and require the bore to have minus of .005 tolerance and should not be edged. when they are thin they are very difficult to make.

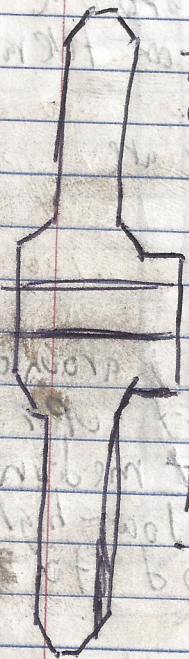
.315 wide flanges.

if the piece is thin, around .400 - .500 range, set it up, cut the face on high + medium speed on the OD on low-high speed, leave .060 on OD to cut last.

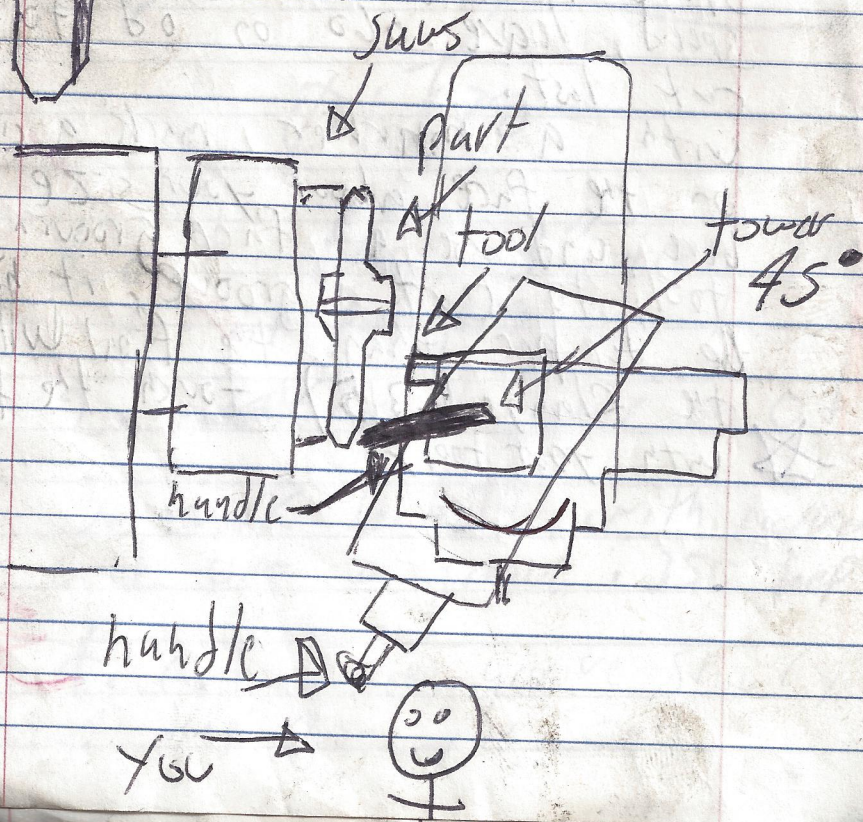
with a marker, mark a circle on the face about your size of bore and use a face grooving tool to cut a groove, it has to be deeper than the final width of the flange (.315) touch the face with your tool.

food service sprockets

these are stainless steel pieces
that must not have holes in them
if so, must be reweld.



the TIR must be perfect
on all sides, and set your
tower on a 45° angle
Facing you and to your
left, remove the handle
on the top of the tower
so it does not hit the
saw, and put a low
profile tool facing out
3 inches.



* use speed 225 for better
cuts, put a lot of water.

* if some holes show up and
you feel like can't clean them
up, stop cutting the welds and
and hub and do the rest of
the part, leave some material
so they can weld again and
you can rework it, do the
other side.

* use a good cutting tool to
make cuts of .0030 or less
each time, use a sharper and
new insert to make the last cuts
and put water to make it look
better, cut the raw weld on
lower speed but use a higher speed
to make the last cuts.

* if they send you a big stainless
steel piece for a food service food
sprocket, face it on both sides and
leave extra material in the o.d.

* the chamflier also may be ok
with the graduated tower and not
with a 15-10 degree tool.

Wheels

wheels are pre-made parts,
usually cut from flat steel
to be machine completed,
find out if you can hold the
piece by the hub, the bore (with an
arbor) or the OD, make sure that
the inside face of the wheel and the
inside of runs good on the fit,
this is important, make some cuts in
the OD and face and you can
flip the part around to have a
better grip of it, if the width of the
plate is .700, but the required
length is .500 and you must divide
the .200 that you will have to cut
and face ~~face~~ one side of the piece
100, and leave 100 for the other
side, always divide the amount
that you have to cut.

Angles  this angle must be measured backwards

this can't must
be measured
backwards

$$\sqrt{7134} - 90 = 10$$

20 this cage must
be measured
right.

73.20

ϕ must be right on top of the measure

$$71.34 - 90 = 19$$

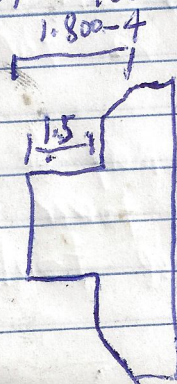
How to measure Angles.

* Some angles are measured backward because the engineers measure from the center of bore to face, but you must measure from face to OD or from OD to face. you will never know what kind of angle you need unless you try it and test it.
example, if an angle is .75 your angle may be 15 ($.75 - 90 = 15$). or if you angle is .49 your final angle may be 41 ($.49 - 90 = 41$).

* check with your supervisor to make sure that you have the right angle to cut and to measure before finishing.

* set the protractor to the right angle and measure from OD or face.

* if your angle is measured from the face or edge of the part and have a hub, set all measures to find out how much you have to cut.



$1.500 - 1.800 - .004 = .296 \pm .004$
.296 is how much you have to cut, that is the minimum, the .004 is your tolerance. These measurements must be exact or the piece will be scrapped.

When cleaning out the
saws.

sometimes trash goes inside the
saws, and accumulate in places,
making difficult to turn them,

you must take them out and
clean the saws, the third goes
out first, then the second and
number one goes out last,

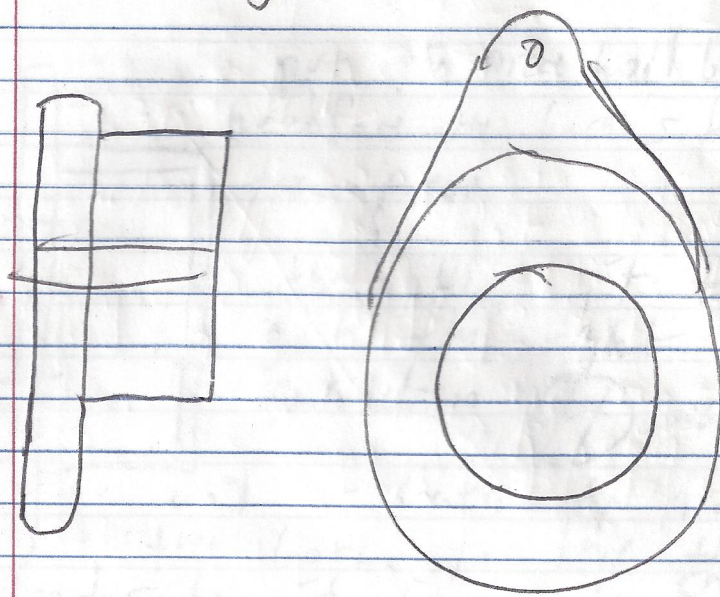
and to put them back in

★ 1 goes first, then second then
third

* put some oil or grease inside
the saws, close the lid of the
machine and start it to get rid of
the excess oil, put a bag under the
★ machine to catch the oil.

★ to put them back inside, find
the beginning of the line, align
it with the number one and
put the saw inside, turn it until
the beginning is aligned to number 2
and put the saw number 2, then
do the same to the third.

Drawing Hubs



1. put harbor. and cut
od on hub and back side

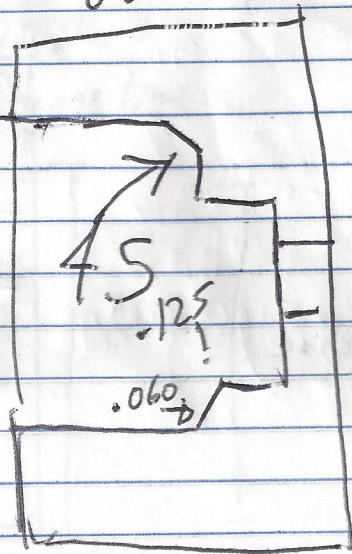
2. take out harbor

3. cut saws

4. put part and

★ finish bore

4.5 inside a
counter bore.



leave .60 if the 4.5 is

1/16 on the left and

.125 on the bore to

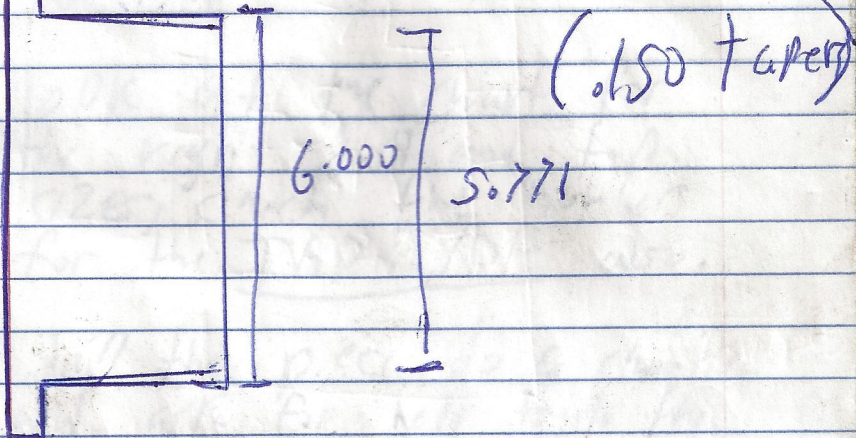
make the 4.5 on

★ ▽ the counter bore

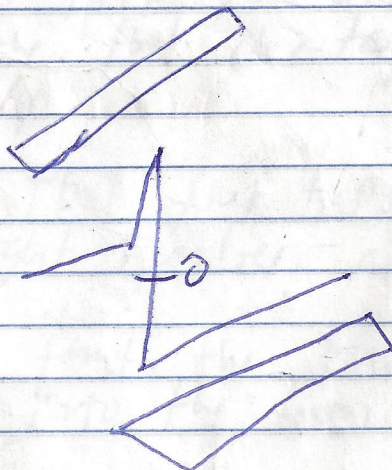
UO-STL

3/4 taper on OD
4.953 LTD

5.562 gage line
4.419



★ 1 set piece



Taps

taps are usually on/y drilled and tapped. it does not need to have the bore to size

* look at the chart for the right drill and tap sizes, check the numbers for the TAP TYPE also.

* drill the piece, do a chamfer and ask for help to do the tap.

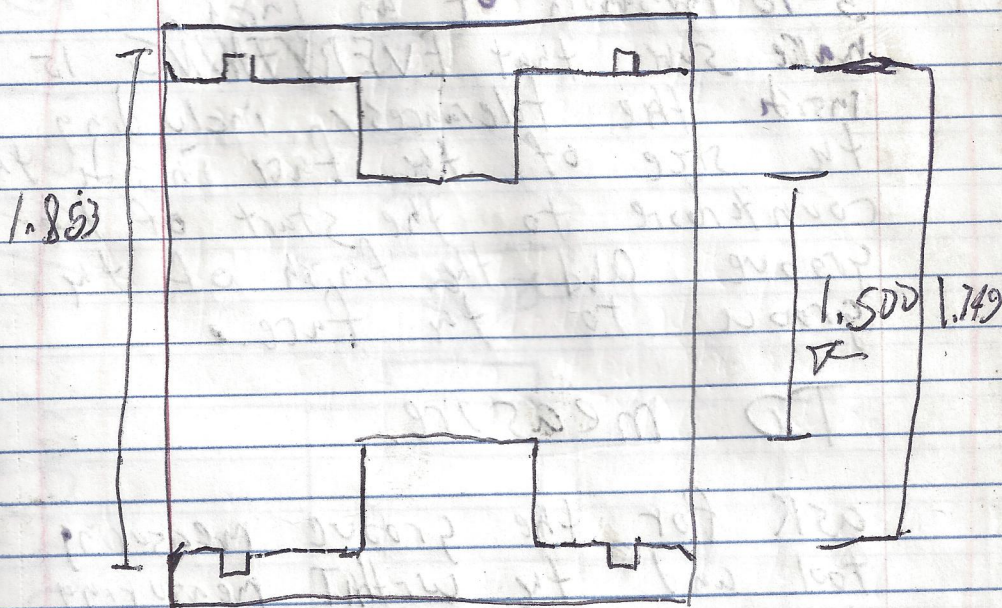
* use oil and take the tap out every once in a while to clean the trash out.

* make sure that you will not force the tool in, they break easy/y.

* make sure that your taps have the right numbers and letters

* make sure that the taps go straight into the bore

Snap Rings / grooves inside bore



Snap Rings are rings that go inside a C-bore, very tightly and hold a bearing, the counter bore for that is usually very close, 1-7/10 of an thousandth of an inch, and the snap ring bore is also very close, .017 with a and keep, measure the size of the counter bore minus the size of the snap bearing bore and that is how much you have to take in, touch the inside of C-bore, push O on X, and slowly go in, use water

example
 $1.749 - 1.853 = 103$, that is or measure

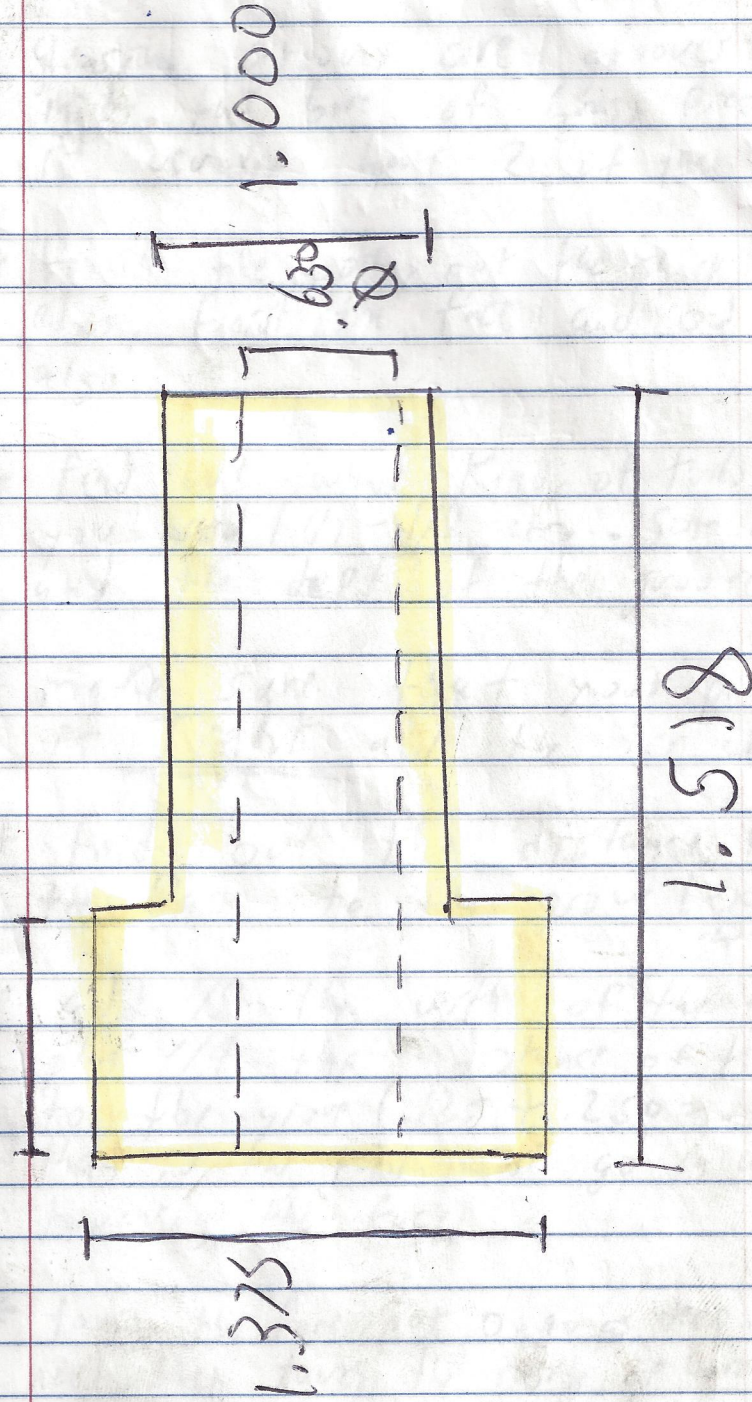
the width of the bore is usually
5-10 thousandths of an inch,
make sure that EVERYTHING is
inside the tolerances, including
the size of the face inside the
counterbore to the start of the
groove and the finish of the
groove to the face.

To measure.

ask for the groove measuring
tool and the width measuring
tool also, put them on the tip
of a 6" digital caliper and
put it straight up and down
on the counterbore, put "0"
on your digital caliper, open
it up inside the groove and
see how much do you have.
measure the width of the groove
also with the other tool,
make sure to read the right
< > or > < side of the tool.

Be extra careful that your
grooving tool does not break inside
the groove. run slow and clean out
the trash.

990 bar holder tool ©2016



Greasing grooves

greasing grooves are grooves inside the bore of brass pieces it usually have 2 of them.

* Finish the bore, cut the sharp edges, finish the face and OD also.

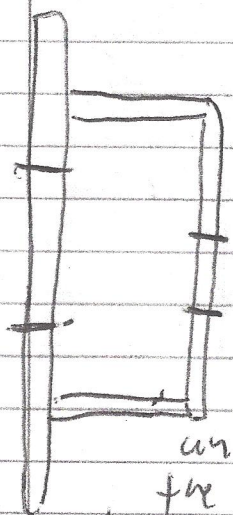
* Find out what kind of tools you need $1/2 - 1/8$ etc. $.250 - .125$ and the dept of the groove

* make sure that your piece is Tight and the fit is OK

* find out the distance from the face to the groove (example - $.250$)

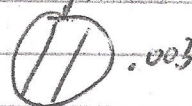
* add $1/8$ the width of the groove plus $1/4$ the distance of the face to the groove ($.125 + .250 = .375$) that is the tool must go in after touching the face

* touch the face, put O, go as deep as you need, then touch the inside of bore, slow speed go Double as needed, example $1/16$ deep $.62$ go double $1/8 = .125$



put the piece hold by the
hub face the back side of
the piece

then flip it to
get the right
parallelism



.003

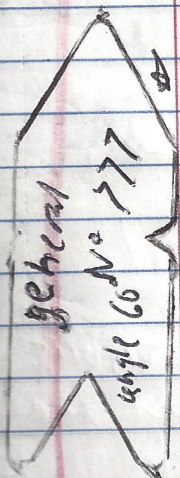
and concentricity from
the bore to the ~~face's~~
indicate it from the ~~chain rows~~
chain rows, not the hub.

1. cut jaws and put the piece,
check tir on the chain row, Do
not check tir on the hub od, if
you wont be able to clean the hub, that
is OK. Clean hub, face, and do the bore.
2. flip to the other side clean up that
face and do the bore, check parallelism
on all around the part.

Setting up for threads

- When setting up for threads,
- ★ you must find out the ^{right} gage,
- ★ find out if the ~~gears~~ threads are even or (2-4) or uneven (1-3)

- ★ make sure that you have the right cutting tip (30°) and its set up straight, use a special tool to measure that (general 777) it measure from the face of the part to the center of the tool.



- ★ find a good bar and set it leaving only one inch extra length so it does not shake, example, if the bore for the threads is 2 inch deep, leave the bar only 3" long.

- ★ set up your machine, set up the gears inside and engage the blue knob, it has 2-4-1-3 numbers and spins when engaged, make sure that you disengage it after done making threads, put the black handle underneath - it is NEUTRAL before starting.

- ★ set up your speeds, you can change them anytime if the part shakes,

- * Set up your Z on the back
any your X close to the core
- * put blue print to see the lines
of the thread.
- * the DC level must be backwards
- * CA the Chamfer on the core
before starting the threads.

How to run the threads.

- * Set your speeds on low-(65-25)
- * Make sure your tip tool is the right one and it properly attached, secured and tight up.
- * Make sure that your tip is straight, use tool to measure it.
- * If your threads are even, you must stop on 2-4, if they are uneven you must stop on 1-3.
- * Make sure that all your tools and part are tight and the Tir is OK.
- * Start your machine, take out the slack, stop your line right on the even or uneven number (1-3)-(2-4) always the same number.
- * Leave a reasonable distance from the tip of your bar to the face 4-10 inches, so you can work and stop the bar.
- * Start the machine, stop at your number using the BLACK handle, not the silver, keep your hand on that handle so you can stop it when the Z reach.
- * Take more if it does not cut do not take more than 15 if it's already cutting.
- * Make sure that you have the right threads, measure with the tool.

sure

that looks like a swiss knife
V-S-S-AMER. Nat 4-42 you can't
never trust the machine.

* Make more cuts of 5, always
start on RIGHT on your number and
stop on Z-0.

* if you need to stop for any reason
first you must move your X handle
back Before stopping the black
handle.

* Check with the gage after each
cut to make sure it fits.

* Do not stop the machine with the
emergency lever, take out your cutting
tool first.

* Do not mistake the black handle
for the silver handle.

* Do not forget to use the numbers

* Do not change settings or move the
cut after you have started the threads.

* Measure with the gage after each cut.

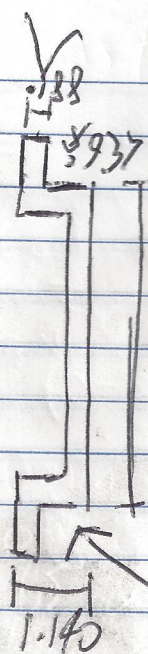
* mark where the threads start so you can
measure better.

* take out your bar backwards after
every cut (X)

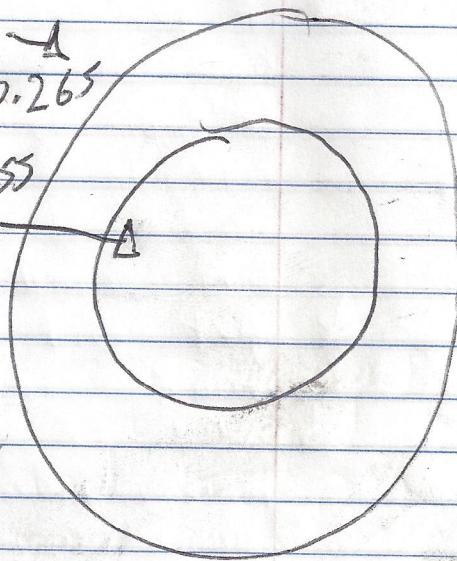
* if you miss your number, let it go
another pass, the bar does not move.

* always keep your hand on the black
handle when cutting so you can stop
it if necessary.

Ring seal



10.265
5.755



1. do od

2. do outside part and
bore

3. cut jaws and finish the
LTD and counter bore last.

cast iron

When making face counter bars
on cast iron use speed
165, use water and ~~to~~ push
the wheel with your hands slowly.
make sure that the piece will not
suck your insert in.

A cast iron flys all around,
that is a way to know it

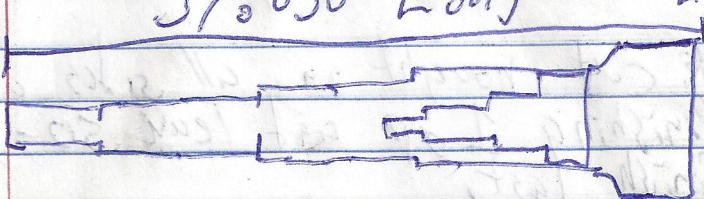
SF / 5212 22 1/2

Long axels

clean up

37.030 Long

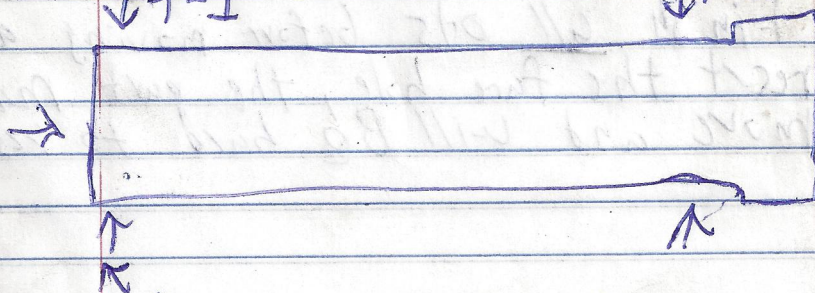
clean up



When you start making these parts you must do a long cut on the OD, all the way to the end to make sure that you have ϕ taper from the start to the end, because on the second operation it will leave a lot of taper and you will have to sand it down.

$\downarrow +1$

$\downarrow +1$ OD



OD on both side must match before cutting the face and making holes. or do a small hole on the face, make cuts in OD fix the taper and re-cut the face and hole.

6001-feswer

* put and leave it there as a safety measure

* cut rough on all sides after finishing 1st and leave 50-30 to finish last.

* try to keep your lifeline forward in case that your part moves or falls, move it forwards or backwards as needed,

* cut all ods to size and sand down the rest, sometimes they leave .0002 or .0002 tolerances.

* Finish all ods before moving and reset the face hole, the part may move and will be hard to reset.

Taper on long bars

sometimes they send you brass or metal bars and ask you to do the oil, when you have taper on the oil, example, one side of the bar is 3.500 and the other side is 3.507 you must push the life saver tower .007 against you or towards you.

do this when you still have material to cut and make some cuts to test which way you have to move your tower.

35 A100

Big thin plates

.150 thickness

1. set up and do the od on 022L (20) using slow speed and cutting only .050 thousand at the time. put plates on each side to give the piece more strength.
2. make sure that the face is not robbing more than .050 on the face, but it does not matter much.
3. cut only .050 at a time and have a lot of pressure on the part to avoid the part from moving and grinding against the other plates.
4. get it to size on od, cut sharp edges and move to a lathe machine to do the chamfer and bore.

★ Plastic Bushing and Parts

★ 1. plastic parts are hard to machine because they shrink, or ~~egg~~ while marking them and ~~expand~~ when cool down, they also shrink when inserted inside the bore of another piece (bushings).

★ 2. put the piece, clean up face and OD, and flip around to finish on the other side, that way if it falls you will have a clean side to set it up again.

★ 3. you can drill, face clean and OD all your pieces so some of them start cooling down so you can do your finish cuts, leave around 100 on bore and OD.

★ 4. always make big cuts of around .008 even when you are finishing, if you make cuts of less than .001 your insert bounces and refuse to cut, use slow speed on bore.

★ finish the bore and find an arbor if you don't have enough material on the LTD to cut the bore and OD without the piece shrinking and egging. I do the LTD last

★ if you have to sand down the bore, use a new 320 sand paper and clean the bore with alcohol if gets dirty

★