OmniTurn Sample Part

Welcome to the OmniTum. This document is a tutorial used to run a first program with the OmniTurn. It is suggested before you try to work with this tutorial that you spend some time reading the manual and gain a basic understanding of the programming and operations of the system.

The goal of this tutorial:

- first looking at the print
- tooling
- programming the part
- checking the tool offsets
- test running the program
- cutting a part
- correcting errors in the program
- adjusting the part to size with offsets

Materials needed for this tutorial -(these can be purchased from OmniTum as ”Start-up tooling kit”)
1/2” collet
1/2” x 3” aluminum, brass, or other easily machined material
LH turning tool and holder
LH threading tool and holder
1/4” drill
5/8” bushing holder
1/4” bushing
#2 center drill
5/8” bushing holder bushing for center

Starting the sample part

A note about the options for the OmniTurn

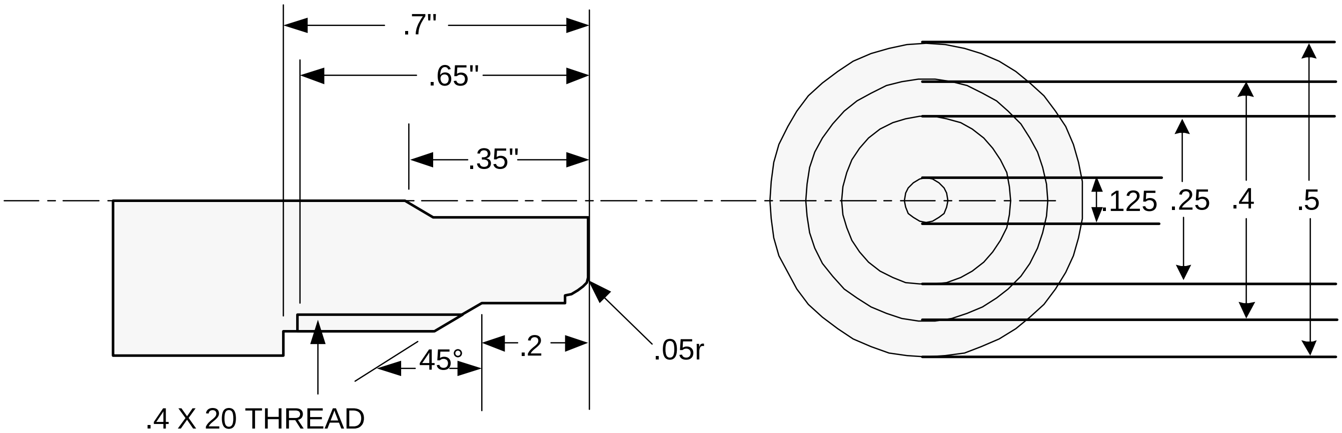
The OmniTurn has a number of options. Such as:
Spindle control - On / off
Spindle speed control - infinitely
Threading encoder
I/O additional ”M” functions

Sample part layout

Please take a look at the following part. You will notice that there are a number of operations that must be performed (otherwise what fun would this teaching part be). We will be doing the following:

Roughing the OD
Center Drill
Peck drill to depth
Finish the face and OD
Single point threading
Deburr with the turning tool
Make a single pass on the thread for cleaning and deburring

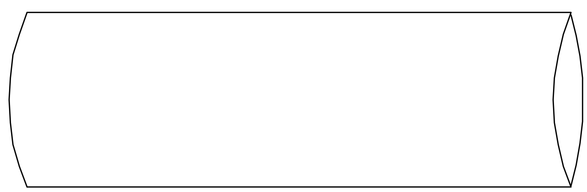
OmniTurn Startup sample part



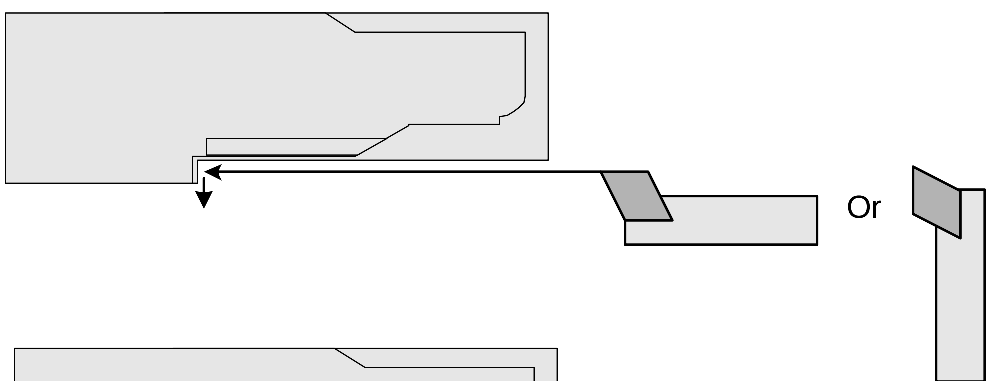
If you are familiar with machining you might want to skip to the next section, ”TOOLING”

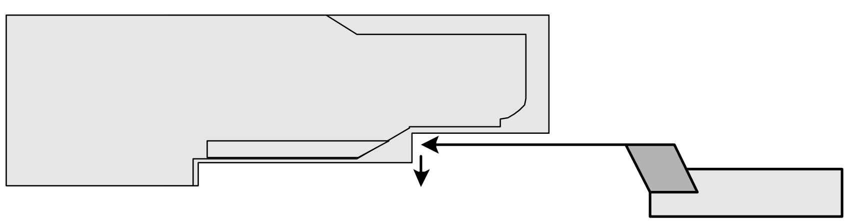
OPERATIONS

We will start with a solid piece of material 1/2” in diameter, 3” long.

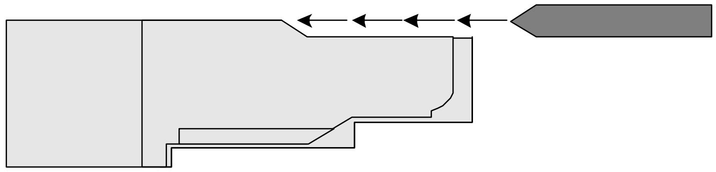


In machining this part we will use the same tool for roughing and finishing the OD. This does not have to be the case. You could setup another tool for the finishing pass. We will be doing the roughing first.

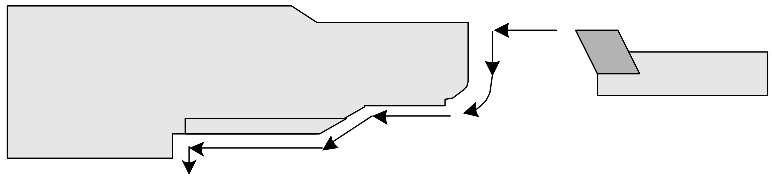




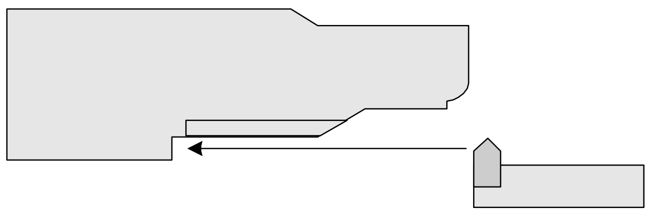
We will take two passes to rough the part. After we rough the OD and remove most of the material we change tools and center (spot) drill. Then we change tools again and peck drill the hole to finish depth.



After the drilling we change back to the turning tool to face and finish turn the OD.



With the OD finished we can thread. (if your attachment does not have an encoder for threading you will have to skip over this operation)



After the threading is complete we will change back to the turning tool to turn the major diameter of the thread to help deburr the OD. Then to make sure that all of the threads are clean and burr free we can take a single pass with the threading tool at it’s finished depth. If the appropriate threading insert is used, you would not have to do the last 2 operations.

Selecting Tooling

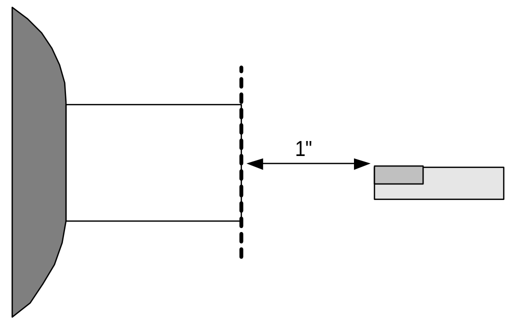
Before we start to program there are a number of decisions that must be made. Tooling is one. First you have to select the tools to be used. This has been gone over in the previous section but they have not been formally listed yet. Now we can assign tool numbers:

Tool #1	Left hand turning tool
Tool #2	Center tool
Tool #3	1/8” drill
Tool #4	Threading tool

Then we figure out where we want them to start. This does not mean where they will be on the slide. When you write the program, the physical location of the tools does not matter, this will be taken care of during the ”setup”. What we need to take care of at this point is the ”logical” starting point. This is where we will take over the movement of the tool with the program. It is important to note this information down for the person that will be doing the setup at the machine. Even if you are the one that will be doing the setup it is good practice to make a setup sheet and write down the starting locations for the tools.

For this sample part we will start tool #1 a little bit in front of the part in Z and on center in X. Please note the next diagram. We will start the tool 1” in front of the part. This is a very safe distance to rapid the tool too and it’s is not to far away so as to waste to much time. This amount will vary depending on the type of tooling used and preferences of the programmer. As you get used the to the system you can make this distance smaller to gain cycle time. Also note that we have all the tools starting 1 inch away, you can vary this. We made them all the same so that setup will be a little easier at first.

OmniTurn Startup sample part



For each of the tools we have to establish a starting location. This location is not that important to worry about much. Just pick a location for each tool that will be safe to rapid to without worrying about collisions. Now we will add some information to the tool table that we started before:

Tool #1	Left hand turning tool	X=0	Z=1
Tool #2	Center tool	X=0	Z=1
Tool #3	1/8" drill	X=0	Z=1
Tool #4	Threading tool	X=0	Z=1

Programming

Now that we have the part layout and a general machining sequence we can begin writing the program. If you want to try and write a program before you read ours, now is the time to try. Do not get upset if your program is not the same as ours. Each machinist will program the part differently, this is normal. Only concern yourself with understanding what we have done and how our program works so you can work with the general format to create your own.

In order to enter the part into the OmniTurn you will have to enter a new program:

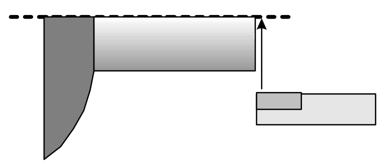
-Turn the OmniTurn on
-HOME the slide (see the section on homing in the Jog section)
-ESC back to the main menu
-Go to the Automatic menu by pressing A
-When asked "FILE TO BE PROCESSED" type the name of the new program. For this example we will use SAMPLE
-The control will say "FILE NOT FOUND, PRESS ESC TO CONTINUE".
-If you do not get the above message, that means there is already a program with this name and you should either delete it or select a new name.
-Then press F3 to enter the editor
-Press Fl to create a new file

Now you should be ready to enter the new program. Try typing in the program for the practice. If you do not want to spend the time entering in the program, load the disk supplied with the start-up tooling kit. The sample part is named SAMPLE 1

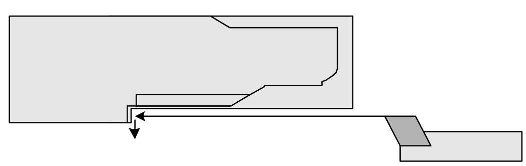
NOTE: Code in Italics are optional. Not all controls will support these lines.

G90G94F300G73	• This program is in absolute, feed rate is set at 300 ipm. Start every program like this. There are very few parts the require G91 programming. Note that this program is written in radius mode -G73
T1 (LH turning tool)	• Makes tool 1 offsets active
X0Z1	• Moves the tool to a safe distance from the face
M01	• I put in an optional stop so when I run the program the first time I can

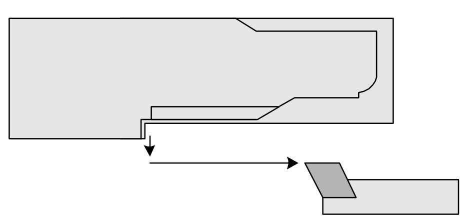
	have the tool automatically stop before it takes a cut and look to see if it came to the right location.
X.1 Z.025	• We will use the first tool to act as a material length stop. Here we are positioning the tool a little off center and in front of zero of Z. There are two reasons we stop the tool a little on the plus side of zero: when the collet is closed the part will pull back some, the other reason is we want to leave a little material to be faced off.
M00(close the collet)	• Stop the program to allow the operator time to pull the material out to the stop and close the collet. The comment will alert the operator to perform this.
M03S2000	• Turn the spindle on at 2000rpm. This command operates an optional function. If your machine does not have spindle control please do not enter this command. What you might want to do instead is to add a comment on the previous line ”and turn the spindle on”.
X.27	• This moves the tool beyond the major diameter so it can move in Z for a rough facing cut. This move is still in rapid
G96S250	• This changes the spindle speed to ”constant surface feet” mode. We will be cutting at 250 sfm. If you have a spindle drive option you can choose to cut in this mode or just leave it in RPM.
G77S3000	• Sets maximum spindle rpm to 3000 rpm so the spindle won’t go to fast while facing off the part. If you had the part in the chuck you would probably make the G77 speed much lower.
G04F1	• This is a dwell to allow the spindle to come up to speed before cutting
Z.015	• This positions the tool in Z for the rough facing move. (rapid feed rate)
G95F.003X-.015	• Here we change to IPR at .003” per rev. We move the tool past zero in X to face the part. We move past zero since there is a radius on the tip of the tool. This move assumes that the radius is .015”. Please adjust this move to coincide with the tool you are using.
X.13	• Move the tool for the second roughing pass, still in rapid



G94F300Z.025	• We move the tool off the face of the part in rapid
G97S2000	• Changes spindle speed to rpm mode, and sets speed to 2000 rpm.
X.205	• Position the tool to take the first roughing pass, this is still in rapid
G95F.003Z-.75	• Change the feed rate to IPR and move the tool to take the first rough pass
X.26	• This moves the tool out in X at the same feed rate of .003”



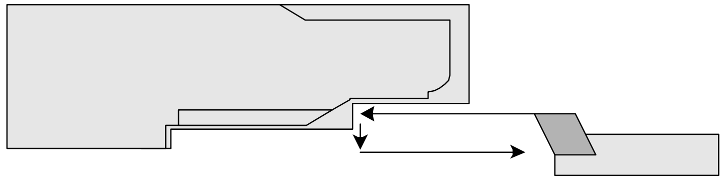
G94F300Z.025	• The tool is moved at rapid to clear the face of the part



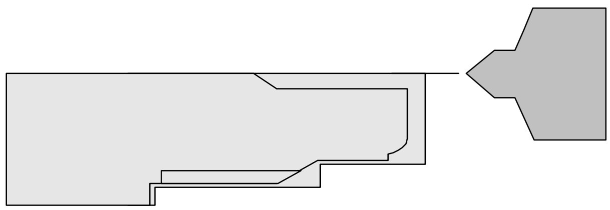
S2500	• Change the spindle rpm to a higher speed for the next smaller diameter pass.
G95F.003Z-.195	• Change to feed mode, and make the second pass

OmniTurn Startup sample part

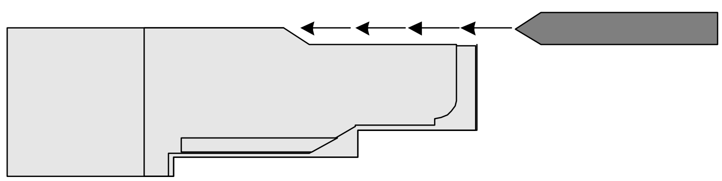
X.225	• This clears the tool, still in the feed mode
G94F300Z1	• Change to rapid and move the tool to Z 1, this is clearance for a tool change



T2S2500(Center drill)	• The center drill is called into position and the spindle speed is changed.
X0Z1	• The tool is moved to a safe location
M01	• Optional stop for tool check on the first run through
Z .025	• The tool is rapidly sent to the face of the part with about .02 clearance
G 95 F.002Z-.025	• Feed the center tool into the material at a feed rate of .002"/rev

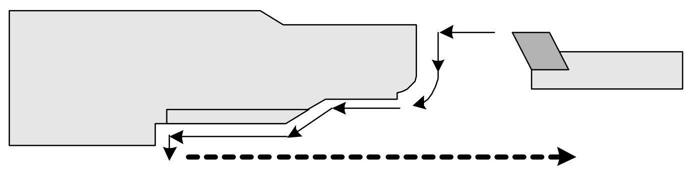


G94F300Z 1	• Rapid the tool clear of the part for a tool change
T3S2800(Drill)	• Call the drill into position and change the spindle speed
X0Z1	• Move the drill to a safe location
M01	• Optional stop for the tool check on the first run through
2.025	• Rapid the drill to the face of the part with a little clearance
G95F.003	• Change to the feed mode of IPR
G83Z-.35K.1L300	• Peck drill the hole, .l" per peck, feed rate of .003 IPR, rapid at 300 ipm

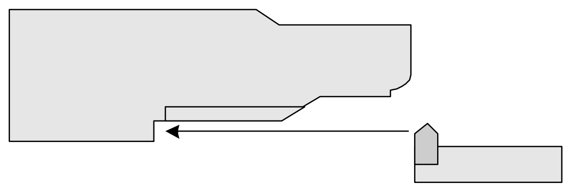


G94F300Z1 S2400	• Move the tool to clear the part for a tool change and change spindle speed.
T1 (LH tool .007 TNR)	• The turning tool is brought into position again for the finish pass
X0Z1	• Move the tool to a save location
M01	• Optional stop for the tool check on the first run through
G77S3000	• Set maximum spindle speed to 3000
G96S250	• Change spindle speed mode back to sfm @250
G04F1	• Dwell 1 second to allow the spindle to change RPM
G41	• Turn on tool nose radius compensation
Z.025D1	• The tool is rapidly sent close to the face of the part & turn on comp with D1
G95Z0F.002	• The tool is fed to the face of the part to begin the finish contour
X.075	• The next few lines turn the part
G02X.125Z-.051.075K-.05	
Z-.2	
X.2Z-.275	
Z-.75	
X .26	
G97S2000	• Spindle back to RPM mode and set speed to 2000
G94F300Z1	• This clears the tool in Z to 1 at rapid feed rate
G40	• Turn tool nose radius compensation off with move off the part

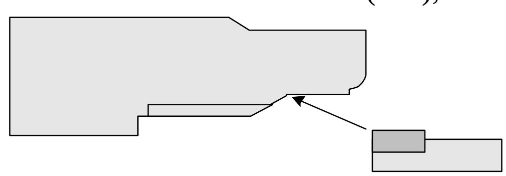
OmniTurn Startup sample part



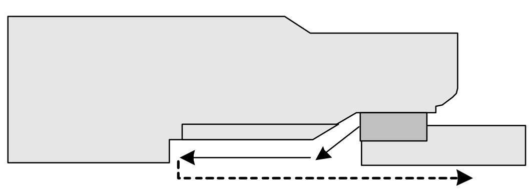
T4(Thread tool)	• Calls the threading tool offset
X0Z1	• Moves the threading tool to a safe location
M0l	• Optional stop for the tool check on the first run through
G04FI	• This is a dwell used to allow the spindle to change speed
X.2	• Moves the tool to the starting diameter of the threading cycle
Z0	• Moves the tool to .2” from the start of the thread in Z
G33X.175Z-.65K.05I .004	• This is the threading command



Z1	• This clears the tool in Z to 1 for the tool change
T 1(LH Turn tool)	• Call the turning tool for the deburr pass
X.15Z.1	• First move to location for the tool, more aggressive location than before Note: I did not put a MO1 after this tool call since this tool was already called and it was OK. So why stop it again?
G41	• Turn on TNR comp
Z.01D1	• Move tool to a spot so it can start the deburr pass
G95F01 X.125Z-.2	• Move tool to start of deburr (IPR), faster than normal cut pass



X.2Z-.275F.003	• Start deburr pass
Z-.75	



X.26	• Clear tool
G40	
G94F300Z1	
T4S2000(threading tool)	• Call the threading tool for a single pass to cleanup thread
X0Z1	• Establish tool location
G04FI	• Dwell for 1 second to allow the spindle to change speed
X.2Z0	• Moves the tool to same start location as first threading cycle
G33X.175Z-.65K.05I .0040	• This is the threading command for a single pass (note O)
M05	• This turns the spindle off. We are not waiting for the M30 at the end of the program to turn the spindle off, this saves a second or two
Z1	• this is a clearance move
T0	• This moves the slide to it’s home location **(This is not needed, you do not have to go home at the end of every part!)**
M30	• End of the program, the control will now be ready to run another part

Saving the program to the disk

- Now to save the program press Fl to get back to the main page of the word editor
- Then press F3 to save the program to the disk
- Then press F2 to exit the program

Verify the program

Before you run the program it is best to look for possible mistakes. Even the most experienced programmers make simple typing errors. The verification software will point out possible errors.

First make the program active by entering in the name when asked ”FILE TO BE PROCESSED”

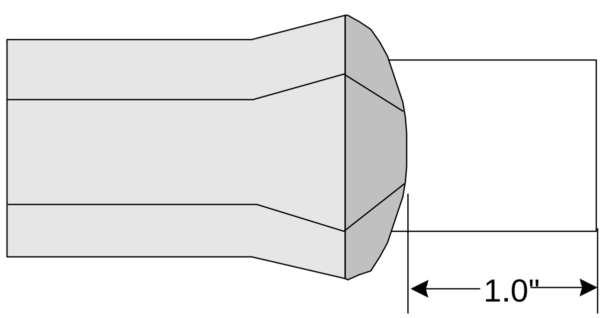
Then press F4 - Please see the notes on using the verification software in chapter 5

Setting the tools on the slide

Now that you have written and entered the program it is time to set the tools on the slide. There are a few considerations that must be made when you are putting the tools on the slide, remember:

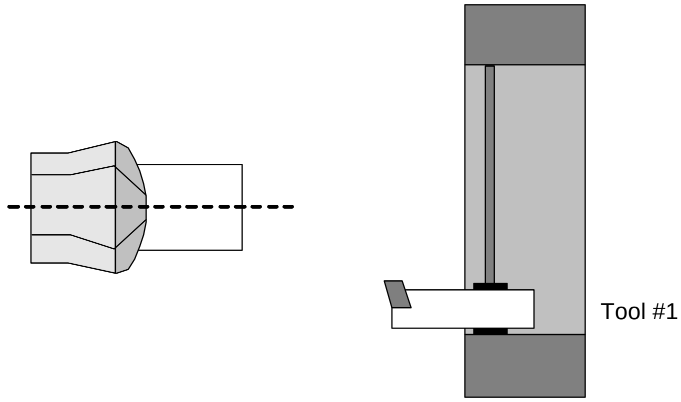
• Set the tools so that they will not interfere with one another during tool change or during a cutting cycle. Be sure to consider both the length and width clearances.
• Keep the tools as close together as reasonable so there is not to much time wasted during tool changes.

To set the tools we first have to put a blank in the collet. Set the blank so that it sticks out about 1 inch. This will be enough to machine the part and not hit the collet or spindle.



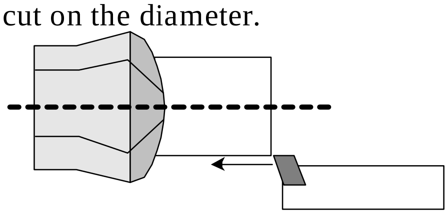
Then put the turning tool on the slide. For this example put it on the near side.

Setting X axis

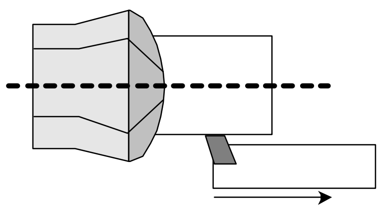


Now go to the jog mode and follow the procedure on setting up tool. Turn the spindle on, (if you have a unit with spindle control do this in the MDI mode first).

Move the slide and take a light cut on the diameter.

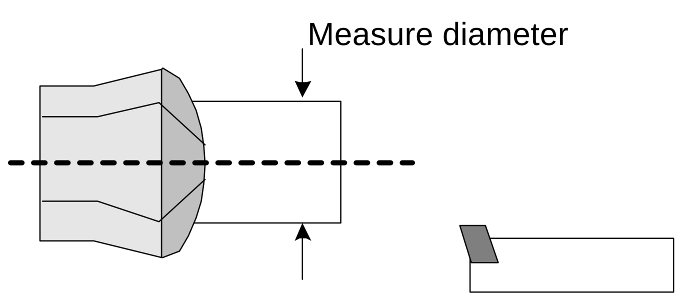


Then move the slide in Z only to clear the material, do not move in X!



Turn the spindle off. If you have the spindle control use the switch on the OmniTurn panel.

Then measure the diameter with a micrometer



Now press T -"SET TOOL" on the keyboard to start inputting the tool offset.

After you press T the control will ask you to input a tool number. Now press 1 and return.

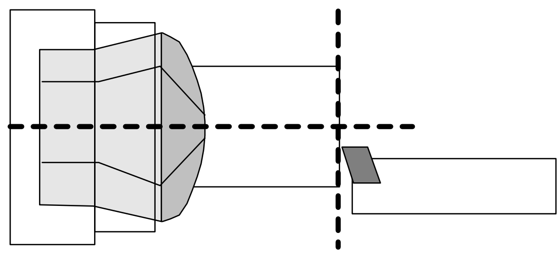
Next you have to tell the control that you are setting the X axis for the tool offsets, press X.

Now the control asks for the **diameter** of the part you just cut. Enter this measurement and press return.

To go back to the jog mode and continue with the entering of the offsets press the ESC key.

Setting Z axis

Now you have to set the Z offset for the tool. Turn the spindle on. Jog the tool until it touches the face of the part. Once you get close to the face of the part select the #5 setting for a jog speed of .001" per jog stick movement. This allows you to just "touch" the face of the part. Then press T to start the input of the Z offset for T 1.



Then press 1 when asked for the tool number, and RETURN

Then select Z

Then the control asks the location of the tool in Z. Since we are going to use the face of the bar as Z=0 you can enter 0 and press RETURN

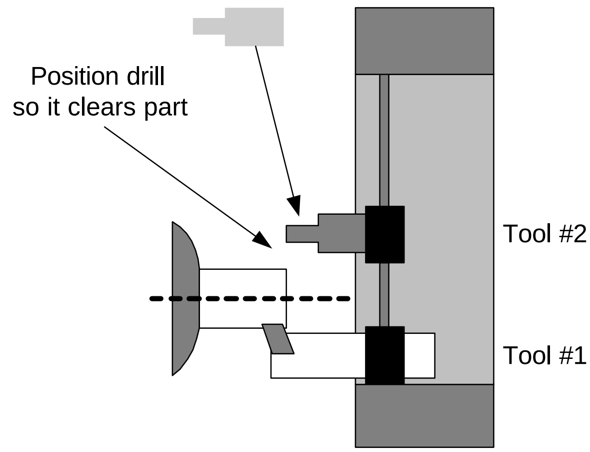
Press ESC to continue in the jog mode.

Now you can setup the center drill and drill. There are a number of ways to setup drills. We will describe one technique. You will need a 1/2" collet, bushing, bushing holder, and pin.

The first concern for this type of tooling is to establish where you want the next tool to be on the tooling plate. This is important since there is a possibility that if you position the new tool to close to the last tool that there can be a collision. Remember this is a gang tool machine. It does not have a turret. When you move a tool you are also moving all the other tools at the same time. You have to set the tools so that while you are working with a tool all of the others do not collide with the: part, colet, spindle, etc. A simple way to set the tooling is:

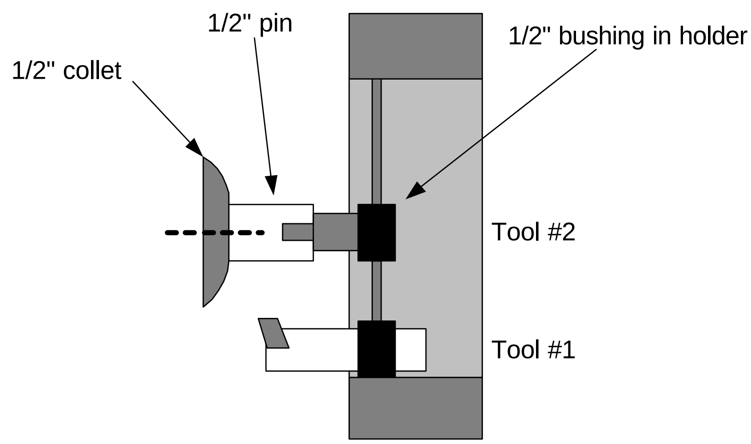
Put the control into the jog mode and move the first tool to it's worst case position and then put the next tool on the slide so there is enough clearance with all obstacles. This does not have to be very exact with most examples since it is only for clearance. When you get into high production jobs like this can be very important since if the tools are to far apart there is a lot of wasted movement and the cycle time will be to long. For this example the tools should be positioned so there are no collisions and the amount of wasted moves is limited.

Setting X for a Drill



With the slide and tooling set as shown above lightly secure the center drill and holder to the slide.

Then put a 1/2” bushing into the holder and a 1/2” collet in the spindle. Jog the slide until the tool holder with the 1/2” bushing is in line with the spindle. Now move the slide close enough to the collet so you can put a 1/2” pin into the collet and bushing holder at the same time. Loosen the bushing holder and line it up exactly with the collet. Then lock the collet holding the pin so we know that it is directly in line with the pin and bushing. Then lock the bushing holder in place.
Now we can establish the tool offset location for the center drill in X.



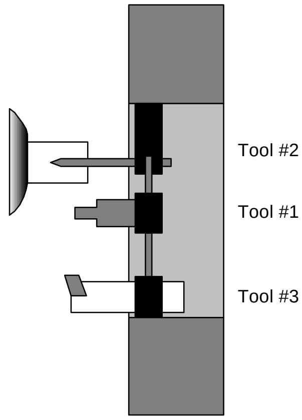
-In the jog mode select ”T”
-When asked for the tool number, enter ”2” and return
-Press X for the axis you are entering
-Enter ”0” for the diameter measured and return
-Press ”Esc” to return to the jog mode

Now move the slide back in Z and put the correct bushing and center drill into the holder.
The Z location for the tool is next.

-Jog the tool until it just clears the part, with the spindle off loosen the tool and slide it forward until it touches the face of the material. Then tighten it in place
-Press T to start the tool offset input procedure.
-When asked for the tool number, enter 2 and return
-Press Z for the axis to be set.
-Enter 0 for the current location, and return.
-The Z is set, press the ”Esc” key to return to the jog menu.

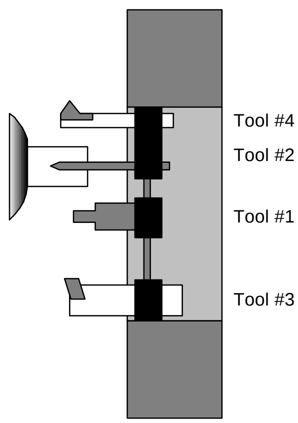
OmniTurn Startup sample part

For Tool #3 follow the same procedure with setting the drill.

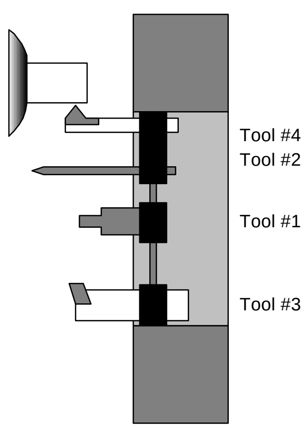


Next is the threading tool. Here we will use another technique. Instead of cutting with the tool we can use the cut surface from setting the first tool to establish the location of this tool.

First, while the slide is still on center with the drill, mount the threading tool so it does not collide with the spindle or the work piece.

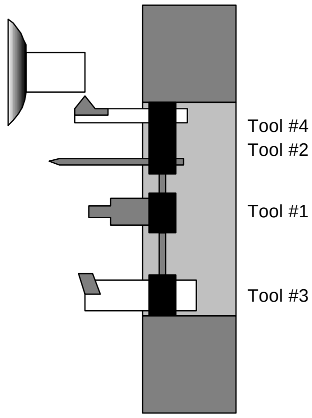


Then jog the slide until the threading tool just clears the turned diameter, it would be best to use a feeler gauge or slide the tool until it just touches the turned diameter. You can do this with the spindle off. If the spindle is on, then you would jog until the tool just touches the material and makes a chip. With the tool touching the turned diameter select "T" from the jog menu to enter the X tool offset for tool #4. Then when the control asks what the turned diameter was enter the value given for this turn from tool #1. Then press RETURN.



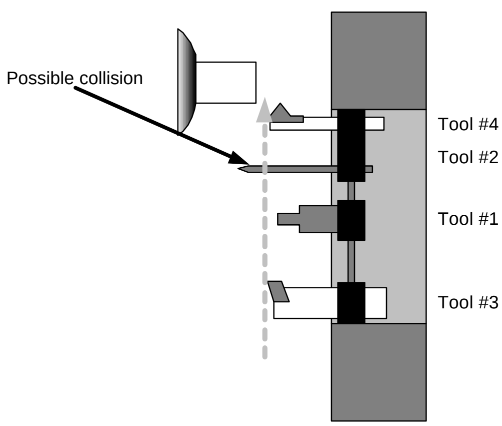
OmniTurn Startup sample part

Then press ESC and jog the tool until it is just even with the end of the material. This location is generally not that critical so don't waste to much time.

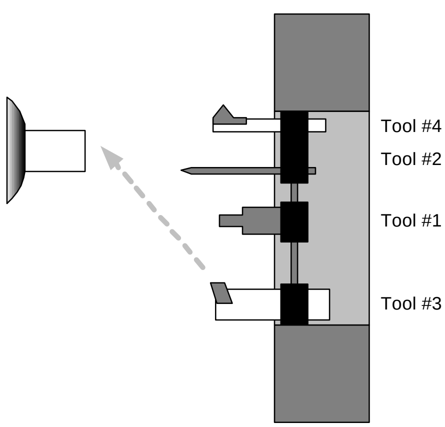


Now that you have the tool where you want it in Z you can establish the tool offset. Press T to start the entry and then select Z. Then enter 0 for the current location and press RETURN. When the control tells you that the offset is entered press Esc to get back to the jog mode.

Now you are done setting the tools. The next step will be to test run the program and make sure that it does what you expect it to do. First before we leave the Jog mode move the tools into a position that will allow the program to run without colliding on it's first move. If we left the slide as shown above we might hit the long drill on the way to the LH turning tool.



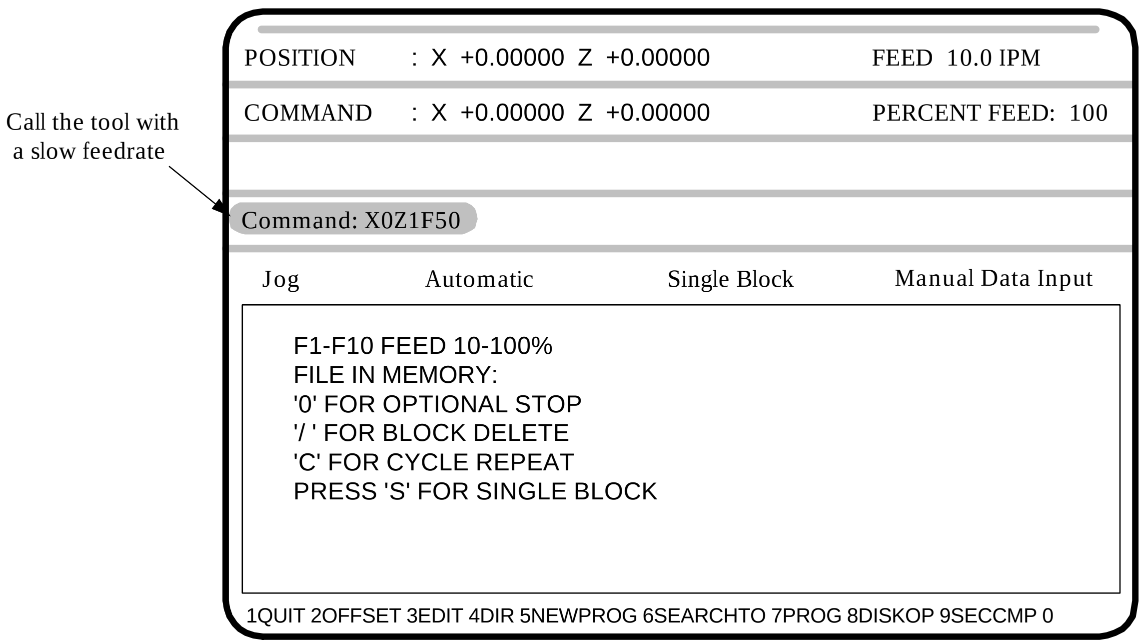
If we move the slide back and over a little there will be no problem.



Checking the tool offsets

Once you have entered the tool offsets you might want to check and make sure the tools go where you expect them to. One way to do this is in the MDI mode. Here you can call the tools and they will go to the set tool offset positions. To get to the MDI mode exit the jog mode back to the Main menu. From the main menu press M. In the MDI mode you can enter one command at a time and execute it. To call the tool into position enter the command that calls the tool:

T1, return, and cycle start. The position display will show the distance from the face and center of the work piece to the tool location. If this distance does not look correct you might want to reset the tool. Then type the command **X0Z1F50 and RETURN.** This command will move the tool to center and 1” from the face of the work piece at a feed rate of 50 ipm. After you press cycle start the slide will move. If you want to stop the motion press the MOTION STOP button on the face of the control. If you want the motion to continue press the cycle start again. To cancel the command press the ESC key while in motion stop.



After you check the first tool you can enter the command to locate the second tool: T2. If there is a problem with the location of a tool you can go back to the jog mode and reset the tool. To exit the MDI mode press F1, this will bring you back to the main menu.

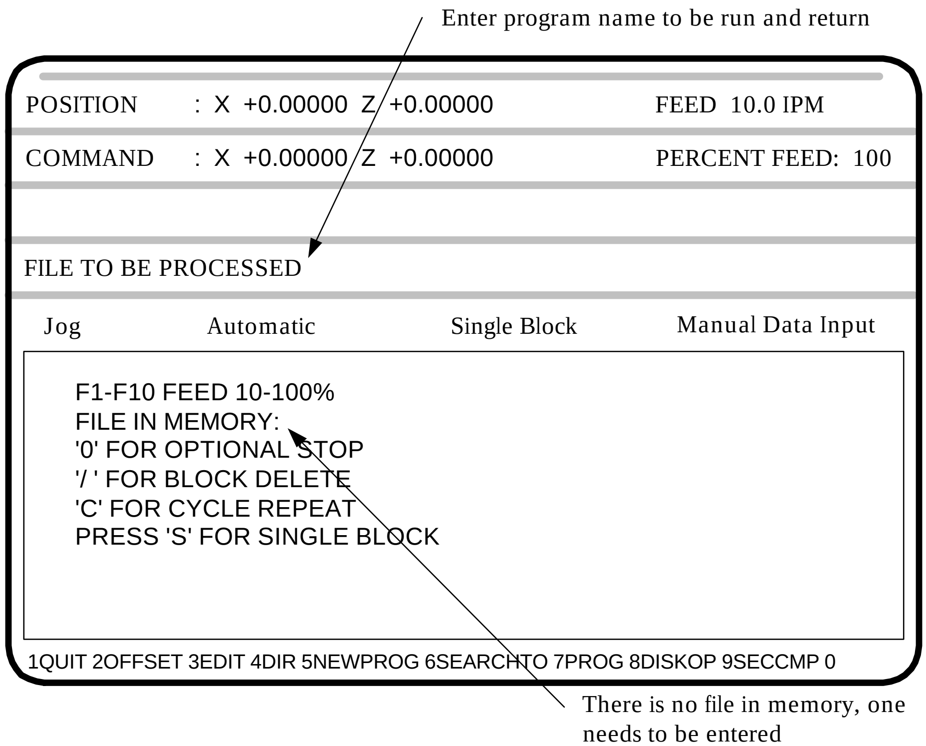
Inputting secondary offsets for TNR Compensation

Before you run the program the values to be used for the TNR compensation must be entered.
Go to the Automatic mode and then press F9. SECCMP

- When asked what secondary offset you want to adjust press 1 (for D1 in the program) and RETURN
- If there is a value in X, press C to clear it. Then enter -.014 and RETURN. This value is twice the TNR, and the direction is (-).
- If there is a value in Z, press C to clear it. Then enter -.007 and RETURN. This value is twice the TNR, and the direction is (-).
- Then enter the size of the TNR into R. .007 and press RETURN.
- Then press ESC to get out of the secondary offset entry page.,

Testing the program

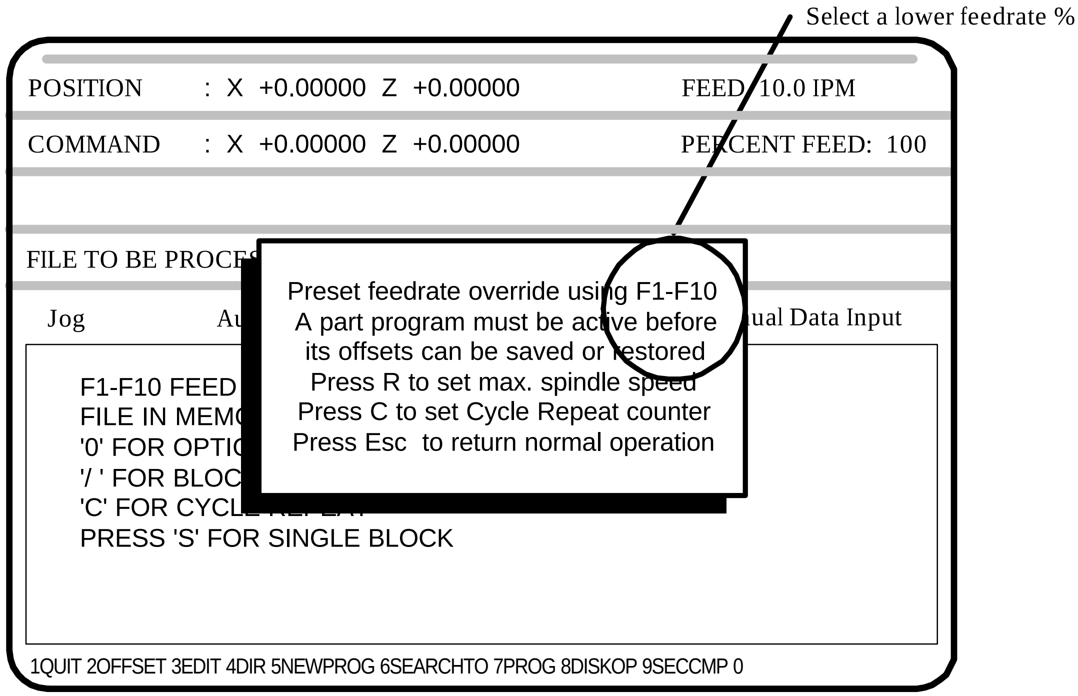
Now that you have made sure that the tools go to where you think they should it is time to test the program you have written. Go to the Automatic mode screen. If you were still in the MDI mode you could press F1 to quit MDI, and then press A from the main menu screen. Then at the Automatic mode screen you have to enter the name of the program to be run. If it is already listed as **the FILE IN MEMORY** and the control asks you to press cycle start, then you are ready to go on. If there is no file in memory then enter the name now. If there is another program active press F5 to allow you to enter the new name now.



There are a few ways to test a program. Making the program run in slow motion is one of the ways to confirm what the program will do. If the slide starts to take off in a direction you did not intend you will have time to stop the slide and correct the program without having a collision.

Feed rate over rides:

The F" keys will lower the feedrate as a percentage. If you press F1 while the slide is running a program the slide will slow down to 10% of it's normal speed. F2 will give you 20%, all the way up to F10 at 100%. This reduction is applied to both feed and rapid moves. You do not have to wait for the program to start in order to reduce the feed rates. You can preload a percentage by going into F10 from the Automatic menu. With this page you can press an F key before you start. When you press a new feed rate over ride you will notice the PERCENT FEED will update and confirm your selection. Then press Esc to go back to the Automatic mode.



FEED HOLD:

If you want to stop the motion of the slide press “FEED HOLD” on the control panel. This will stop the motion of the slide. If you want to continue press CYCLE START. If you want to exit the program and make some corrections press ESC. This will turn off M functions and put the control in the main menu.

OPTIONAL STOP - M01

In the example program we put MO I, optional stops after each tool change. This enables you to run the program with slower feedrates as describe above, and have the program automatically stop after each tool change so that you can check to make sure the tool is in the correct position. If the tool is not where you expected it to be, you can press ESC and then go to set the tool again. If the tool is in the expected location, then press cycle start and the program will continue.

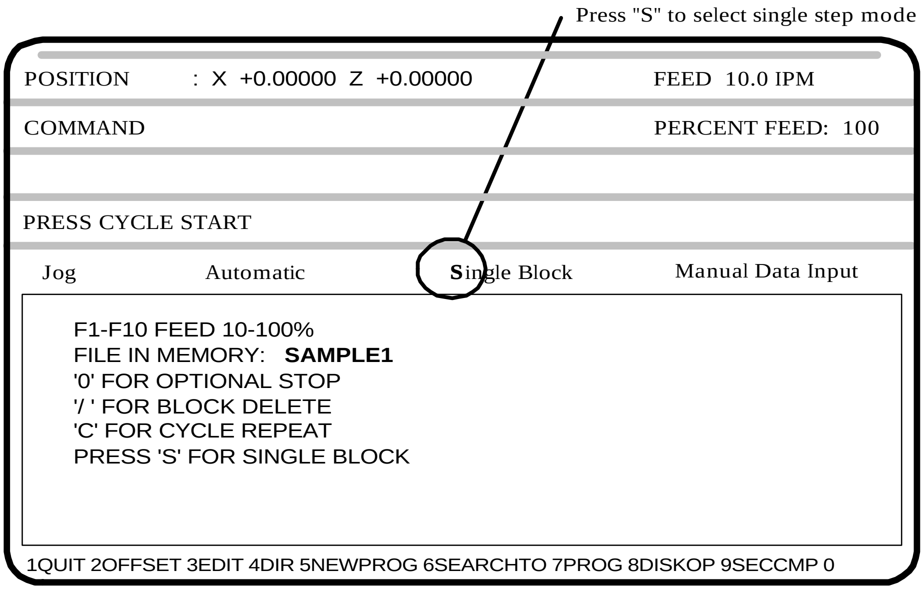
To activate the Optional stop mode press the letter O when in the automatic mode before you start the program.

Single Block mode

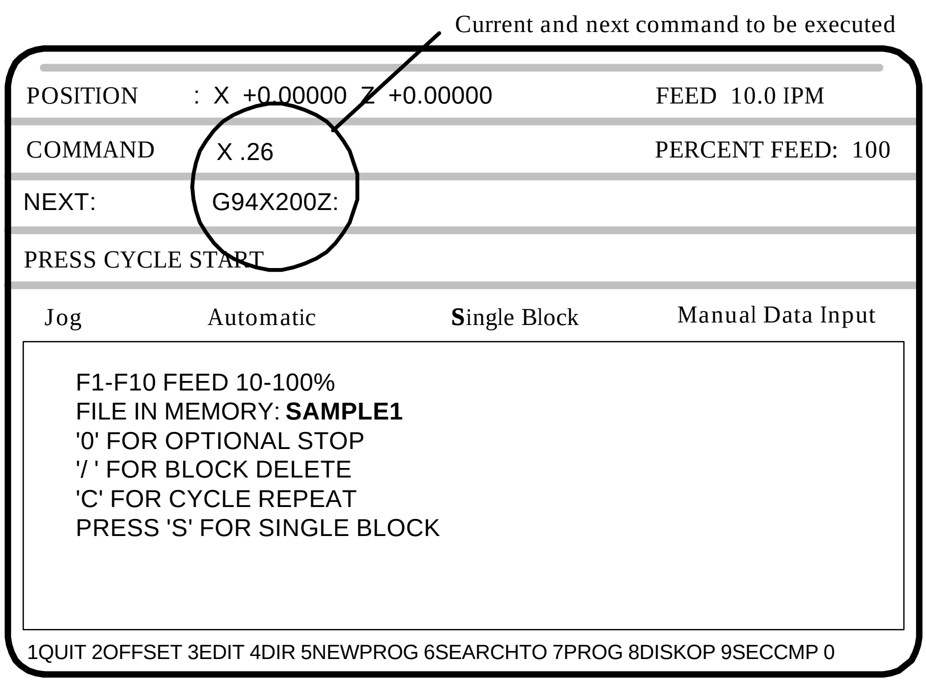
This mode will execute one line of the program and wait for you to press cycle start again to execute the next line. This way you can see what will happen with your program before it happens. This is really good to do if you are not sure what the program is going to do. After you gain some experience with the control and are comfortable with stopping the control you might skip using this option.

To activate this mode press ”S” from the Automatic mode page.

OmniTurn Startup sample part



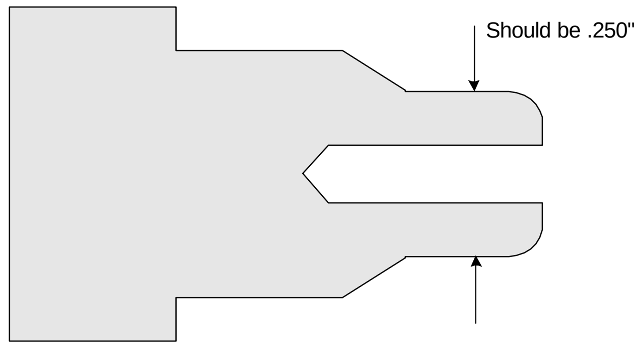
This mode will show you the line of code that you are currently executing and the next line that will be run. This way you can make sure that the code you are running will go where you think it should. If you find and error and want to edit the line before you run it you can press ESC and then go back and edit the program before you run the line.



Making corrections

Once the program has been tested and it is making the correct motions it is time to make sure that the part is the correct size. In our example we have a close diameter to maintain. It is the .250” turn.

OmniTurn Startup sample part



If we now make a part and measure it there will probably be variations from what you program and the size of the finished part. These differences can come from a number of sources, minor errors in establishing tool offsets, tool deflections, material deflection. If the error is small it is possible to change the tool offset to try and correct for the error. The correction is made by pressing F2 from the automatic mode page. This will bring up the tool offset correction screen:

1	**X: +0.86480 Z: -1.25340**	**17**	**X: +0.00000 Z: +0.00000**
2	**X: +1.65025 Z: -1.99200**	**18**	**X: +0.00000 Z: +0.00000**
3	**X: +2.91130 Z: -0.93885**	**19**	**X: +0.00000 Z: +0.00000**
4	**X: +0.00000 Z: +0.00000**	**20**	**X: +0.00000 Z: +0.00000**
5	**X: +0.00000 Z: +0.00000**	**21**	**X: +0.00000 Z: +0.00000**
6	**X: +0.00000 Z: +0.00000**	**22**	**X: +0.00000 Z: +0.00000**
7	**X: +0.00000 Z: +0.00000**	**23**	**X: +0.00000 Z: +0.00000**
8	**X: +0.00000 Z: +0.00000**	**24**	**X: +0.00000 Z: +0.00000**
9	**X: +0.00000 Z: +0.00000**	**25**	**X: +0.00000 Z: +0.00000**
10	**X: +0.00000 Z: +0.00000**	**26**	**X: +0.00000 Z: +0.00000**
11	**X: +0.00000 Z: +0.00000**	**27**	**X: +0.00000 Z: +0.00000**
12	**X: +0.00000 Z: +0.00000**	**28**	**X: +0.00000 Z: +0.00000**
13	**X: +0.00000 Z: +0.00000**	**29**	**X: +0.00000 Z: +0.00000**
14	**X: +0.00000 Z: +0.00000**	**30**	**X: +0.00000 Z: +0.00000**
15	**X: +0.00000 Z: +0.00000**	**31**	**X: +0.00000 Z: +0.00000**
16	**X: +0.00000 Z: +0.00000**	**32**	**X: +0.00000 Z: +0.00000**

OFFSET NUMBER:
Press Esc to exit offset adjustment screen

If we measured the diameter in question and found it to be .252” we would have to correct the diameter by making it .002” smaller. To do this we would have to remember what tool was used to cut it. In this case it was tool #1, the LH turning tool. So we would enter ”1” when asked what offset number. Then for the correction in X we would enter ”-.002” to make the diameter smaller. You will notice that the value of the tool offset location will be modified.

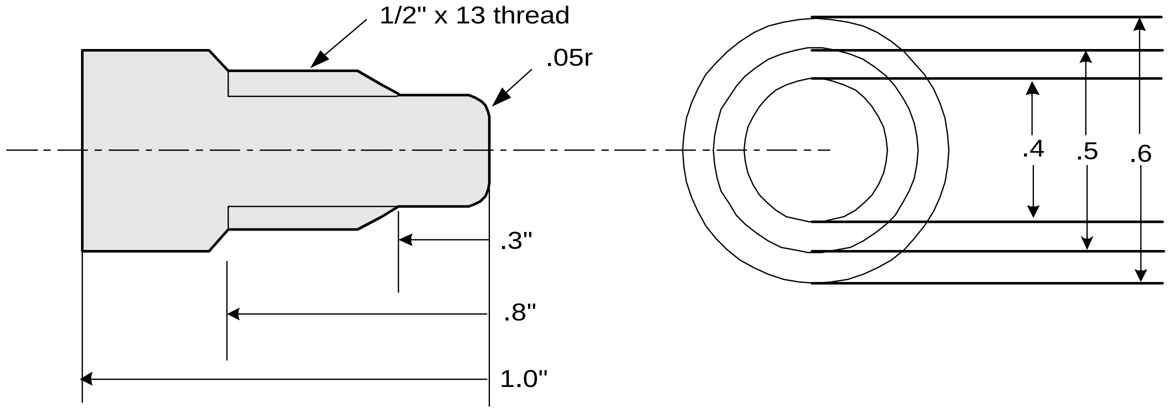
1	X: +0.86480 Z: -1.25340	17	X: +0.00000 Z: +0.00000
2	X: +1.65025 Z: -1.99200	18	X: +0.00000 Z: +0.00000
3	X: +2.91130 Z: -0.93885	19	X: +0.00000 Z: +0.00000
4	X: +0.00000 Z: +0.00000	20	X: +0.00000 Z: +0.00000
5	X: +0.00000 Z: +0.00000	21	X: +0.00000 Z: +0.00000
6	X: +0.00000 Z: +0.00000	22	X: +0.00000 Z: +0.00000
7	X: +0.00000 Z: +0.00000	23	X: +0.00000 Z: +0.00000
8	X: +0.00000 Z: +0.00000	24	X: +0.00000 Z: +0.00000
9	X: +0.00000 Z: +0.00000	25	X: +0.00000 Z: +0.00000
10	X: +0.00000 Z: +0.00000	26	X: +0.00000 Z: +0.00000
11	X: +0.00000 Z: +0.00000	27	X: +0.00000 Z: +0.00000
12	X: +0.00000 Z: +0.00000	28	X: +0.00000 Z: +0.00000
13	X: +0.00000 Z: +0.00000	29	X: +0.00000 Z: +0.00000
14	X: +0.00000 Z: +0.00000	30	X: +0.00000 Z: +0.00000
15	X: +0.00000 Z: +0.00000	31	X: +0.00000 Z: +0.00000
16	X: +0.00000 Z: +0.00000	32	X: +0.00000 Z: +0.00000

X DIAMETER ADJUSTMENT:
Press Esc to exit offset adjustment screen

After you correct the X adjustment then just press return to get past the Z input. Then press Esc to go back to the Automatic mode to run the next part.

Now you are ready to run parts.

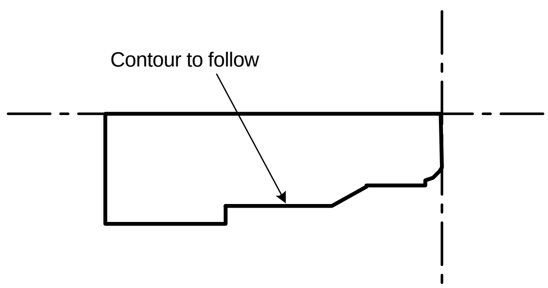
Worked Examples



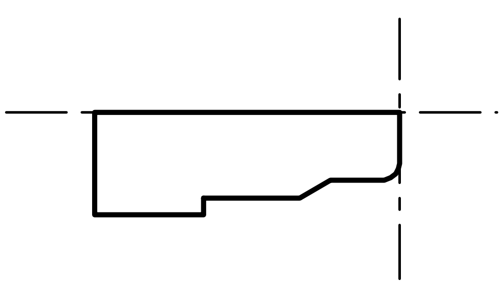
In the above example, the finished part will be made part from a cutoff blank .61” diameter by 1.1” long. The first item to take care of is the layout of the job. This will entail what our sequence of operations will be. Then select the tools to accomplish this sequence. Once the tooling is set it is possible to write the program and then cut the part.

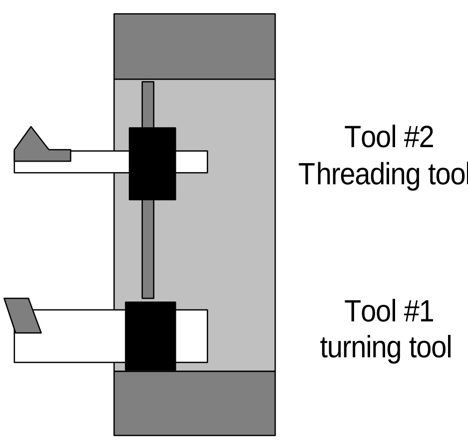
The machining operations that we will be performing are:

Rough turn the OD twice, Tool #1
Face the part to length, Tool #1
Profile the contour to finished size, Tool #1



Now, the tools can be selected for the operations listed above. See below the tool selections that will be used:





Before the program is written we will setup the tools and establish the starting point of the tools from the T command. The tools will start at:

Tool #1 The turning tool will start at the center of the part in X and .2” away from the face in Z. This .2” in Z will give enough room in Z so that the tool will not crash into the face if the part is a little long.

Tool #2 This tool will start at a radius in X equal to the major diameter of the thread less the amount for the first cut.

Ø.5”+2 = .25” radius

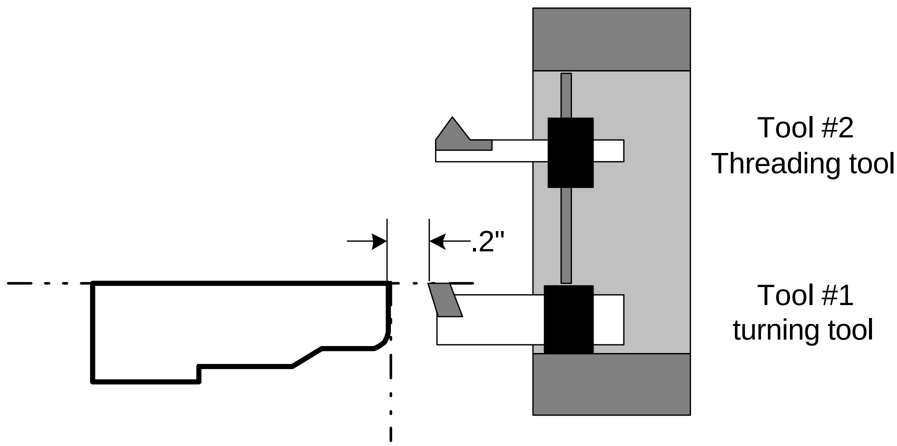
This radius less the first cut of .005” is .25” -.005” = **.245”** starting location in X

In Z we will start at least .2” away from the start of the thread to allow the servos time to ramp up to speed.

We need these starting dimensions now because these locations are used in:

1. The setting of the tool offsets in the Manual mode.
2. The writing of the G92 statements after the Tool change in the program

Note: Care should be taken that the offset location will not cause collisions during tool changes.



Above is an illustration of where the T1 command will locate the tool. We have set the location so that the tool will come to the center (X = 0) and .2” from the face of the finished part, or. 1” from the face of the rough blank. The setup can be done with either the finished or blank part and compensated for during the establishing of the Tool offsets.

Before we start to program there are a few more comments to be made:

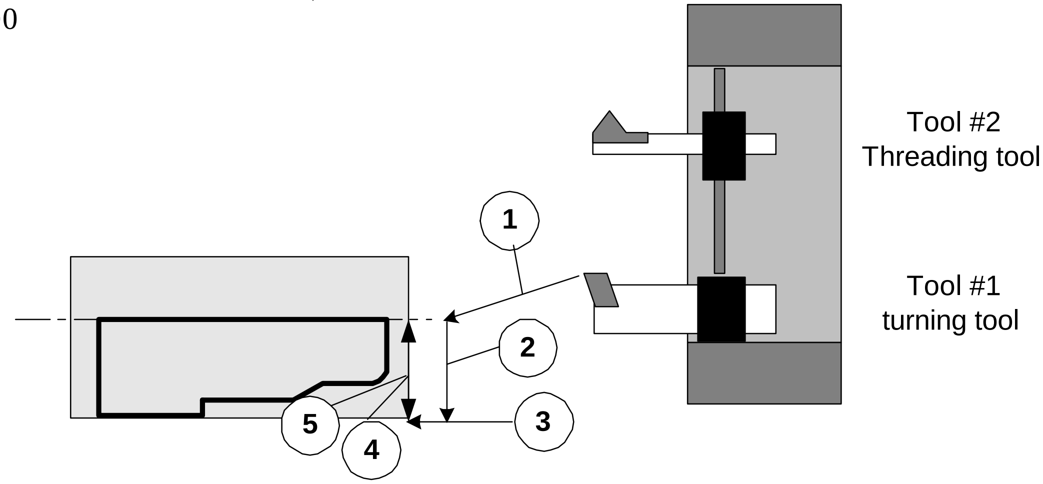
G90 or G91: we will be programming in G90 (absolute). This means that all dimensions given will be from the Reference zero for the part. The location of this Zero is established by the G92statement after the Tool change.

G94 or G95: This part will be programmed in G94, Inches per Minute.

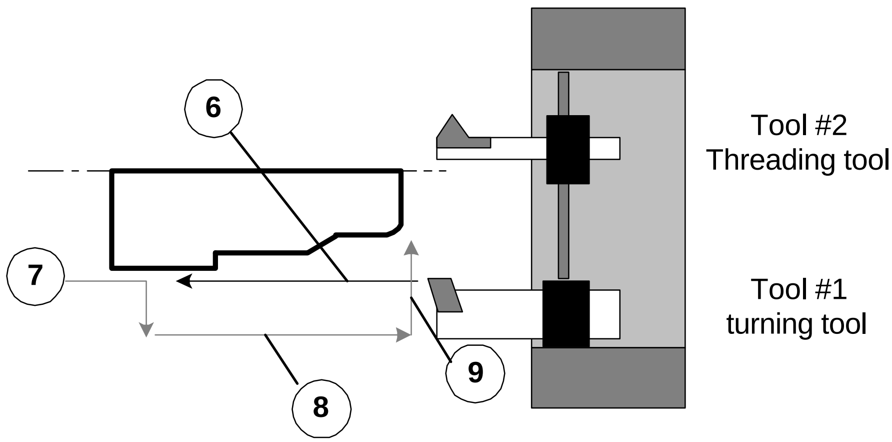
Worked Examples

(THIS IS A SAMPLE PART FOR OMNITURN)

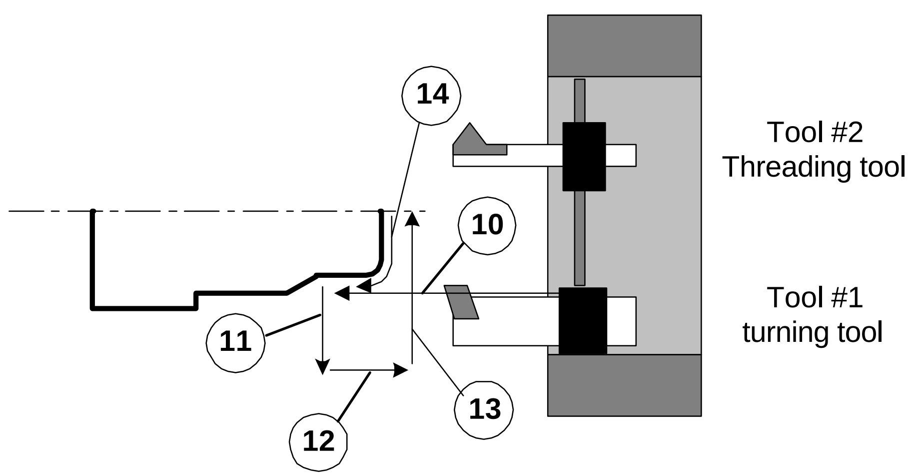
	G90 G94 F200
(1)	T1
	G92 X0 Z.2
	M01
(2)	X.35
(3)	Z.01
(4)	X0F3
(5)	X.26F200



(6)	Z-.79F3
(7)	X.27F200
(8)	Z.1
(9)	X.21

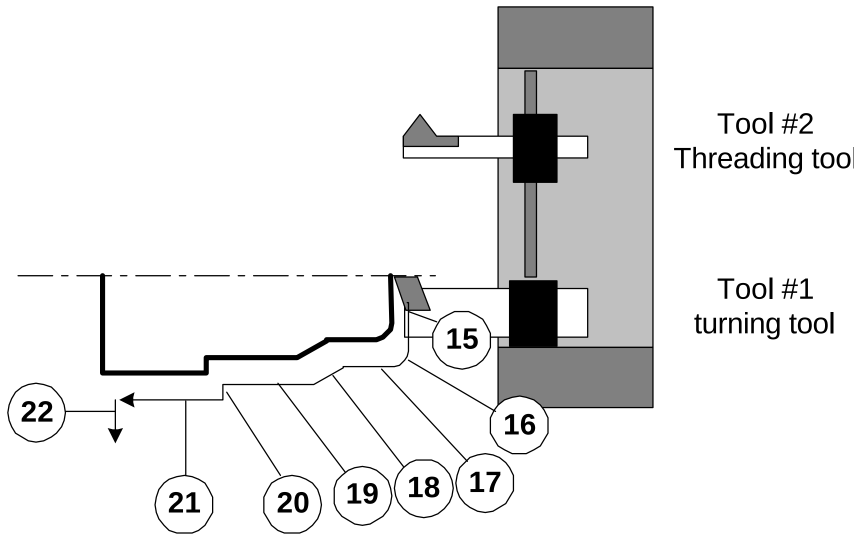


(10)	Z-.29F3
(11)	X.22
(12)	Z.1 F200
(13)	X0
(14)	Z0F3

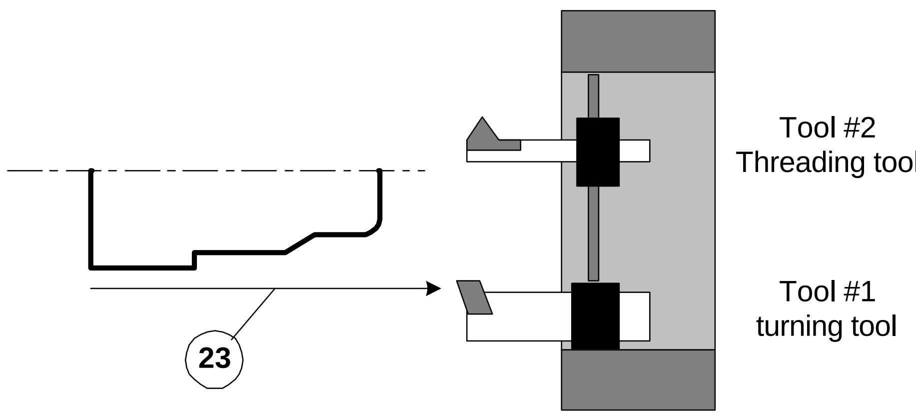


Worked Examples

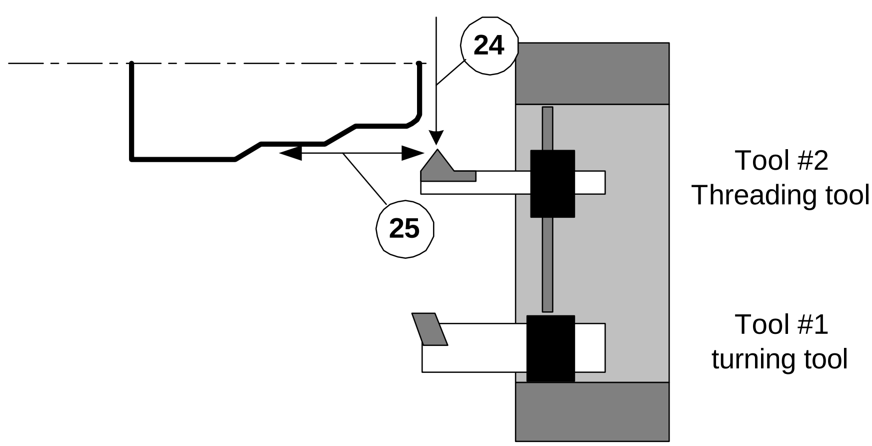
(15)   X.15
(16)   G02X.2Z-.051.15K-.05
(17)   Z-.3
(18)   X.25Z-.35
(19)   Z-.8
(20)   X.3Z-.85
(21)   Z-1
(22)   X.32



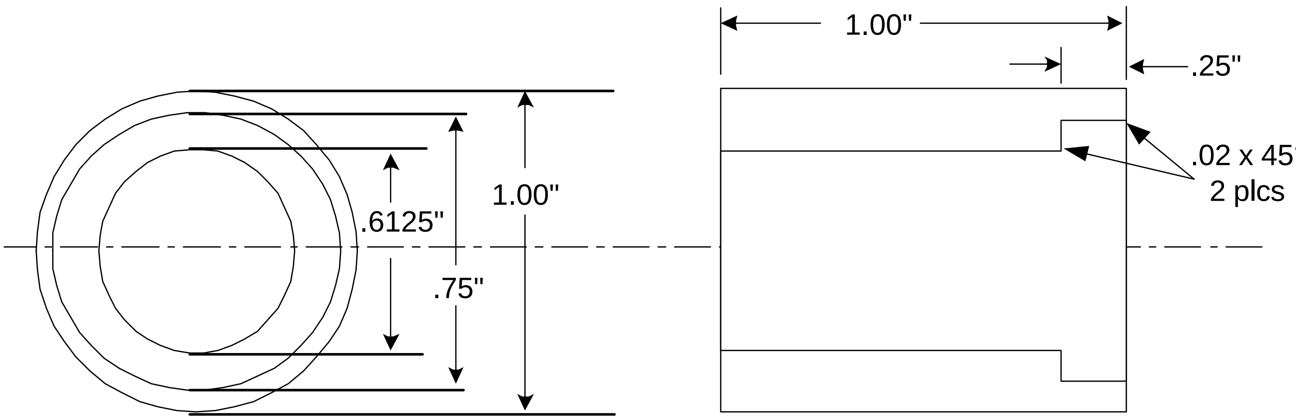
(23) Z.5F200



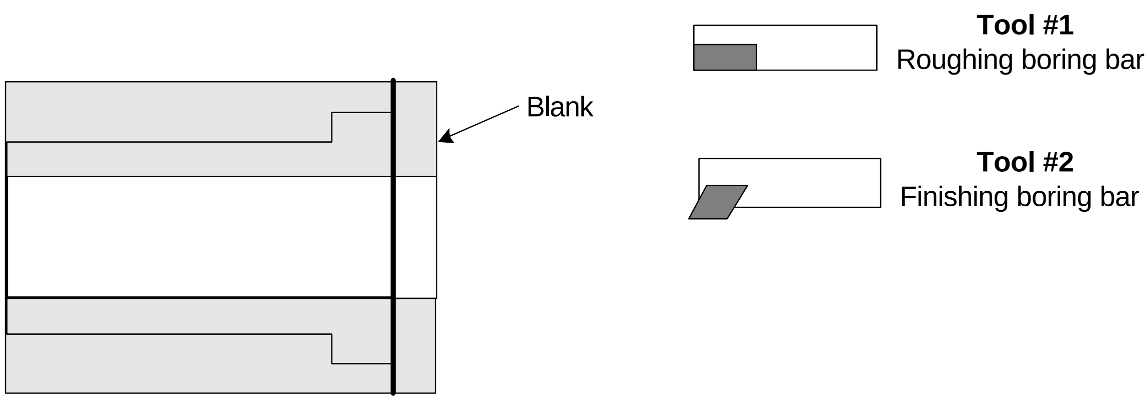
(24)   T2
       G92X.245 Z.2
       M01
(25)   G33X.22Z-.81.005K.0769P
       T0
       M30



Worked Examples



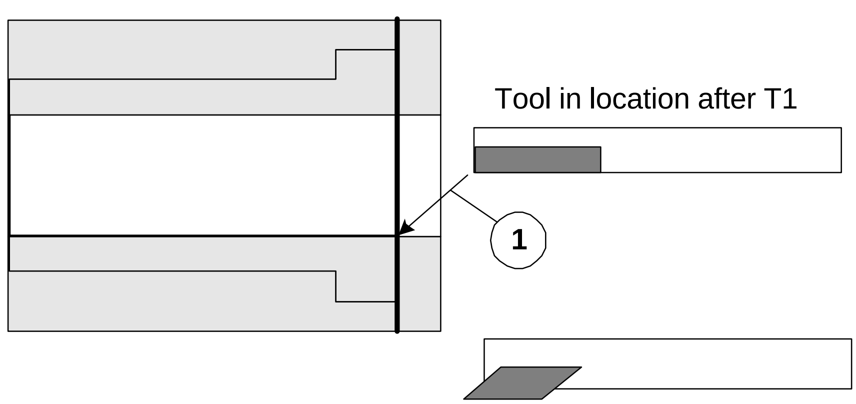
In the above example we will be using two boring tools to finish the face and ID.



The blank has been predrilled and is a little longer than the finished part.

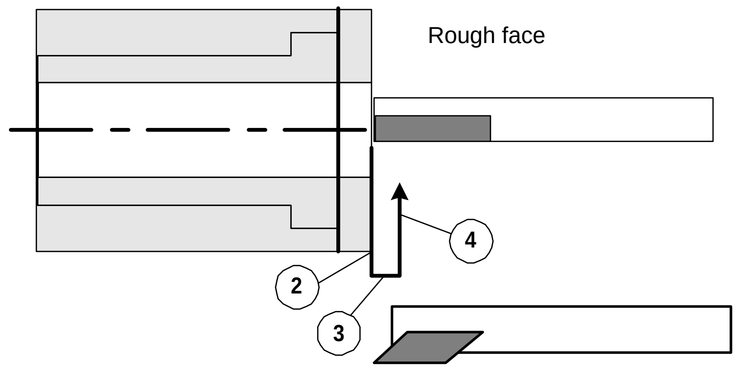
```
G90 G94 F200
T1
G92 X0 Z.2
M03S2000
```

(1) X.28 Z.01

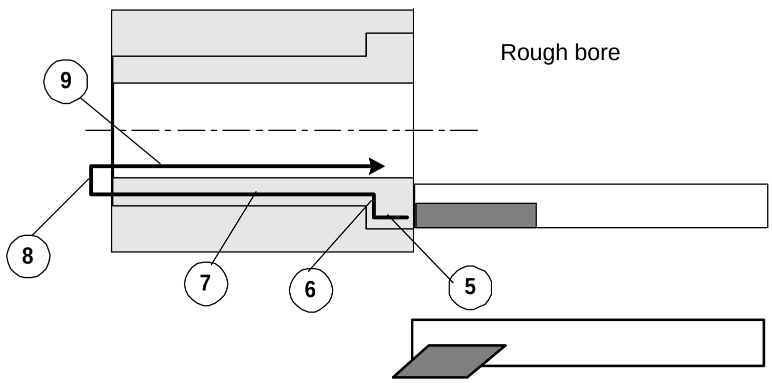


Worked Examples

(2) X.55 G95 F.003
(3) G94 F200 Z.015
(4) X.34



(5) G95 F.002 Z-.235
(6) X.3
(7) Z-1.05
(8) X.29
(9) G94 F200 Z.5
T2
G92 X.55 Z.1
S3000
Z0



(10)	G95 F:002 X.395	
(11)	X.375 Z-.02 D1	Generate chamfer with correction to effect first bore
(12)	Z-.25 D2	Bore first diameter with correction for shoulder length
(13)	X.32625	
(14)	X.30625 Z-.27 D3	Generate chamfer with correction to effect second bore
(15)	Z-1.05 D4	Bore second diameter with correction for back of bore
(16)	X.29	
(17)	Z.5 G94 F200	
	M05	
	T3	Position the tools away from the part for loading, it also Turns off the secondary offsets
	M30