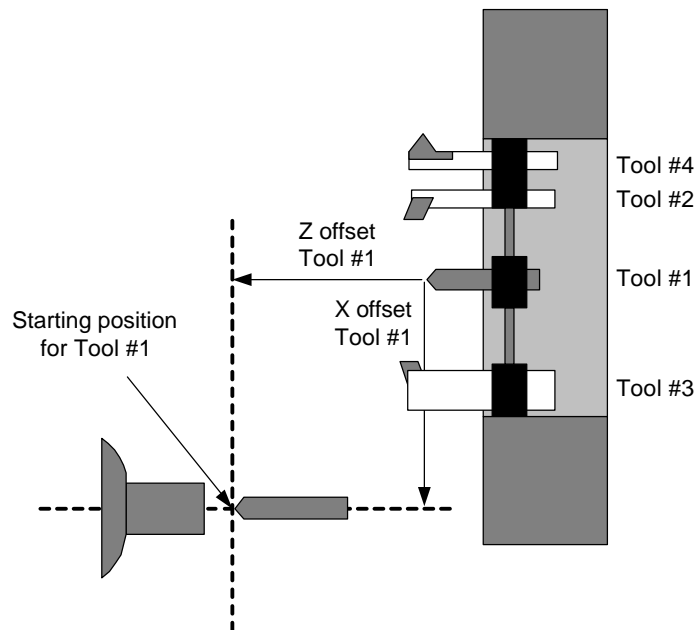# Tool Offsets

After a program is written, the control has to be told where the tools are on the slide. Tool offsets are used by the control to position the tool and know where it is. There are three steps when using the offsets.

1. **Establishing the offsets:** This is done in the Jog mode before the program is run

2. **Call the tool offset and establish what the location is:** This is done in the program you have written and used when the program is run

3. **Adjust the offset:** This is done while you are running a program and find the tool is not located where you want it. Fine adjustments are available down to .0001 "/diameter in X and .00005" in Z.
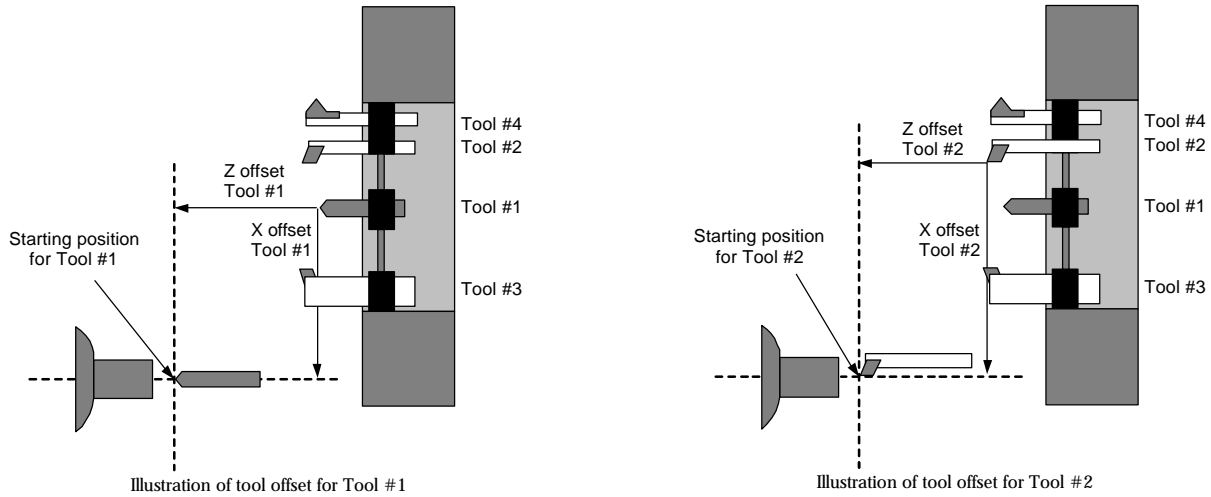
What are tool offsets?

When you turn on the OmniTurn it does not know where it is. However the position commands used in the program assumes that it does. In order to run a program the control needs to know where the slide is and where the tools are. So the tool offset is part of a two step procedure. First the slide has to establish a HOME, Reference Zero (See Establish HOME procedure). Then from HOME it is possible to find the distance of the tool from the work piece. This location will be the starting point for the tool. These procedures are required for all CNC machines, not just the OmniTurn.

In the example below, we show a slide that has four tools. The slide has been sent "Home" and is now in that location. The tool offsets that are shown are the distance from HOME to the starting point for tool #3. When the program calls tool #3 it will move the slide to the distance from the reference zero given in the offset table. It does not matter where the slide is when the offset is called.

When the control is turned off the offsets are not forgotten, they are remembered on the disk that holds the OmniTurn software. When you turn on the OmniTurn, it automatically loads the tool offsets from the last time the machine was setup. This means that once a job is setup, and the tools have not been moved, you can run it without having to setup again. The only thing to be done is Reference Home the slide so that it has the correct starting point with which to apply the offsets. Each tool will have its own offsets. See the fig below as examples of additional offsets.
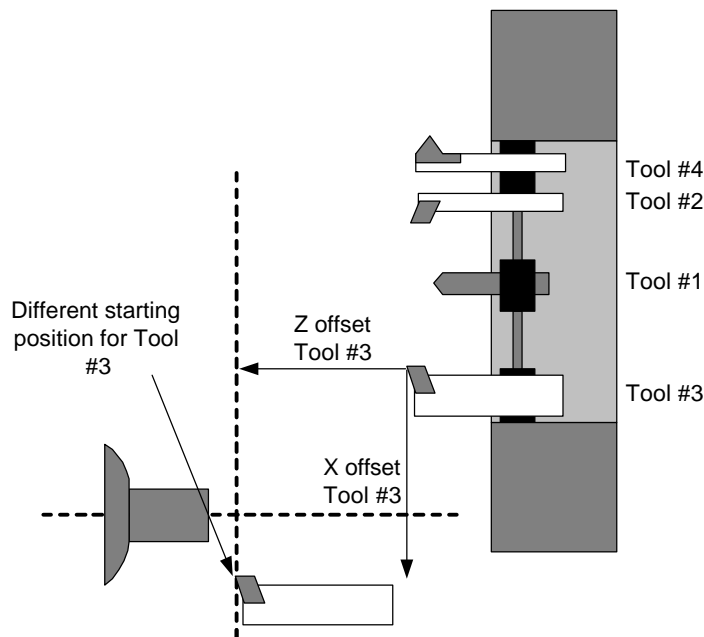
Tool Offsets

Tool #4
Tool #2

Z offset
Tool #1

Tool #1

X offset
Tool #1

Starting position
for Tool #1

Tool #3

Illustration of tool offset for Tool #1

Z offset
Tool #2

Tool #4
Tool #2

Tool #1

X offset
Tool #2

Starting position
for Tool #2

Tool #3

Illustration of tool offset for Tool #2

For all of the above examples, the tool is started at the center of the part and some distance from the face. This is being done in the examples only by choice. The tool can be started anywhere. To save moves and time you might have wanted to start tool #3 close to the diameter of the blank and not at the center. While learning to program, it might be helpful to always start the tool from the center of the part and then move it where ever it is needed. This will make the setup and establishing the tool offset a little simpler.

After you have told the control where the tool is going to start (Tool Offset) then the program tells the control what the value of the starting location is.

Look at the examples below. In this example we are assuming that Z zero is at the face of the part and X zero is at the center. We have picked a different starting location for tool #3. Both are valid starting points. The only requirement is to tell the control what the value of the starting point is. This is done in the program on the line after the tool call.

Tool #4
Tool #2

Tool #1

Different starting
position for Tool
#3

Z offset
Tool #3

Tool #3

X offset
Tool #3

The starting point for the tool is the X and Z values immediately after the Tn command. This is where the tool will move to.

Tool Offsets

How tool offsets are used

There are now two different ways to establish the location of the tool after a tool call (tool offset):

      G92 statement -original system software available on all OmniTurn's

      Non G92 statement -This code is only available on system disks dated after 11/96.

This manual covers only the new method. It is suggested that if you are programming for the first time, or have experience with Fanuc type controls that you use the Non G92 type tool calls. The documentation for the older system is available on request from the factory. If you need these pages please just ask for them, they are available for no charge.

The tool offset is automatically used when a tool call is made, i.e. T1. The slide does not move. Instead the absolute position display changes to show distance of the tool to the absolute zero of the part. Then the next line has to be a move to location with both a X and Z value.

T1
X.5Z 1

This will move the tool to 1/2" or 1" diameter depending on whether the program is in the diameter or radius mode.

Setting Tool Offset Examples

Tool offsets can be established only after the slide has been HOMED Once that has been done, the offsets can be setup.

There are a few types of tools that are common. Below, you will find a few examples covering these common types.

Tool #1   .3" drill
Tool #2   Right hand boring tool
Tool #3   Left hand turning and facing tool
Tool #4   Threading tool

Tool #4, Threading tool
Tool #2, Right hand boring too

Tool #1, .3"Ø drill

Tool #3, Left hand turning tool

.5"
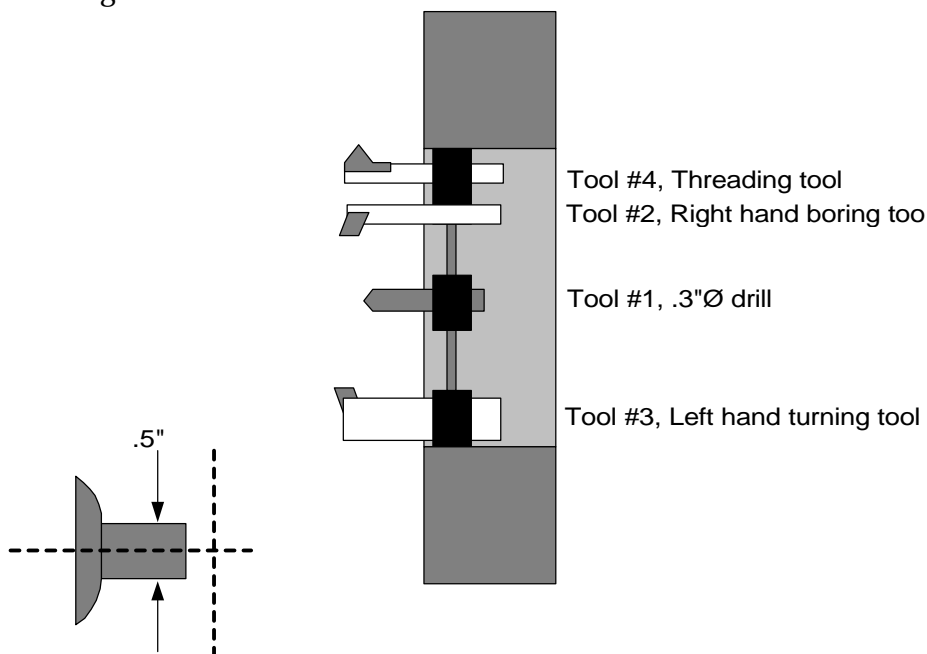
There is no order that the tools must be set up on the slide. It is not important in what order you do the offsets. It is possible to only do one offset and call it #3. The control would not care that there were no values in unused offsets.
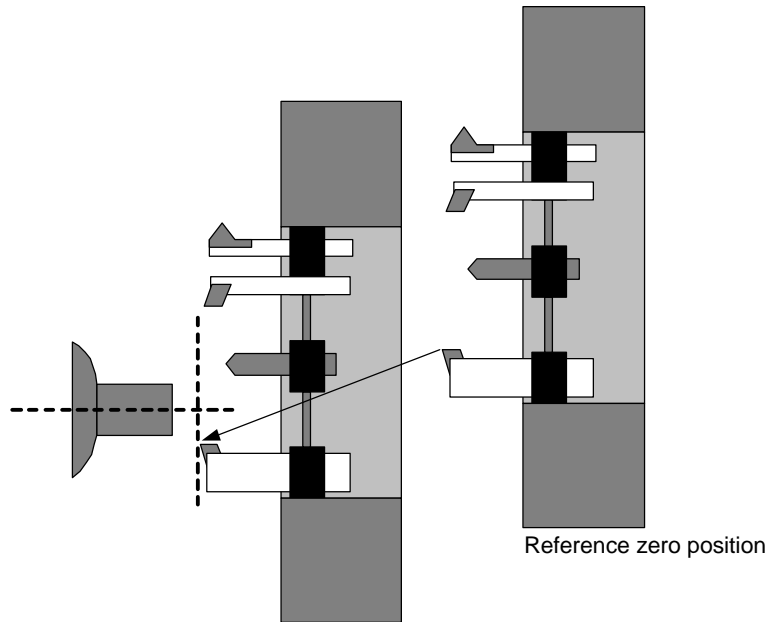
Tool Offsets

If however you call a tool that has not been set, there may be a collision. For the following examples we will assume:

- The material is approx. .5" diameter.
- The part will be programmed so that all of the tools will start at the center of the part in X, and .1" away from the face in Z. Follow this format for the first few programs you write. Later you can be more efficient with time and movements after you have more experience!

To Set Left Hand Turning Tools (Tool #3)

- Be sure that the slide has been HOMED.
- From the main screen go to the Jog mode by typing "J"
- Using the jog keys and joystick move the cutting tool until it is just off the material and slightly smaller.

Reference zero position

- Turn the spindle on, select the jog speed for slow, and take a skim pass of the material as shown next.

Move slide in and cut material

Move slide back in Z. do not move X!!!
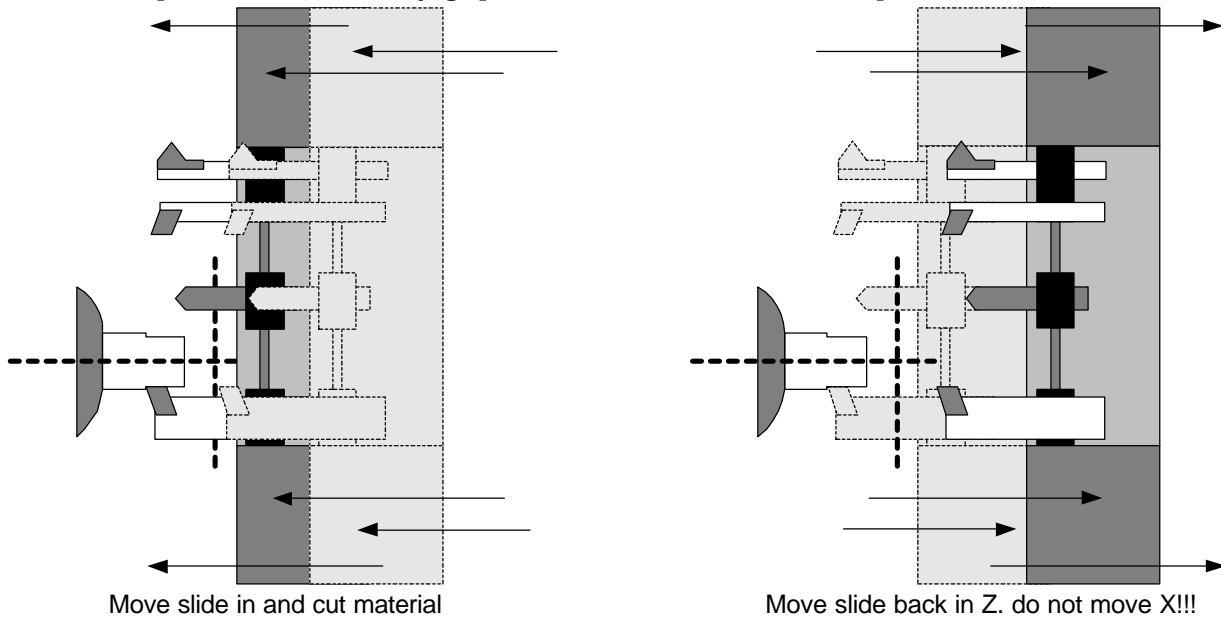
- Then, move the slide back in Z. **Do not move the slide in X.**
  This cut will be used to establish the offset.

3.4

Tool Offsets

To Set Left Hand Turning Tools continued

```
POSITION      : X +0.00000  Z +0.00000          FEED  10.0 IPM

COMMAND       : X +0.00000  Z +0.00000          PERCENT FEED:  100


MAKE JOG SELECTION


  Jog          Automatic        Single Block       Manual Data Input

   1. Slow         7. .1000                            - X
   2. Medium       8. 1.0000                            ↑
   3. Fast         9. Est Home              - Z ←    → +Z
   4. .00005       S. Set Zero
   5. .0010        H. Go Home               ↓
   6. .0100        T. Set Tool             + X



 PRESS 'ESC TO RETURN TO MENU
```

Select "T" to start entering to tool offset

At this point, you are still in the jog mode. Instead of selecting a new jog speed now, select "T". The control will now begin the sequence for entering a tool offset. See the next screen.

```
POSITION      : X +0.00000  Z +0.00000

COMMAND       : X +0.00000  Z +0.00000



OFFSET NUMBER


  Jog          Automatic        Single Block       Manual Data Input

   1. Slow         7. .1000                            - X
   2. Medium       8. 1.0000                            ↑
   3. Fast         9. Est Home              - Z ←    → +Z
   4. .00005       S. Set Zero
   5. .0010        H. Go Home               ↓
   6. .0100        T. Set Tool             + X
 O   IS NOT A VALID OFFSET NUMBER
  PRESS ESCAPE TO RETURN TO JOG MENU

 PRESS 'ESC TO RETURN TO MENU
```

Enter the tool number you are setting

After you have selected T, the control will ask what tool it is that you are about to enter. Type the number tool, in this case 3. Then hit the "RETURN" key.

Tool Offsets
To Set Left Hand Turning Tools -continued

```
POSITION      : X +0.00000 Z +0.00000

COMMAND       : X +0.00000 Z +0.00000


PRESS X or Z TO STORE PRESENT X or Z AXIS OFFSET

   Jog           Automatic          Single Block       Manual Data Input

   1. Slow           7. .1000                              - X
   2. Medium         8. 1.0000                              ↑
   3. Fast           9. Est Home              - Z ←   → +Z
   4. .00005         S. Set Zero                            ↓
   5. .0010          H. Go Home                            + X
   6. .0100          T. Set Tool

 O   IS NOT A VALID OFFSET NUMBER
   PRESS ESCAPE TO RETURN TO JOG MENU

   PRESS 'ESC TO RETURN TO MENU
```

Select the axis you are setting

After you have selected the #3 tool offset, the control will ask you whether you want to enter the Z or X offset. In this case we have set the tool on the diameter of the material and we are ready to enter the X offset, so hit X.

```
POSITION      : X +0.00000 Z +0.00000

COMMAND       : X +0.00000 Z +0.00000


WHAT WAS THE TURNED DIAMETER?

   Jog           Automatic          Single Block       Manual Data Input

   1. Slow           7. .1000                              - X
   2. Medium         8. 1.0000                              ↑
   3. Fast           9. Est Home              - Z ←   → +Z
   4. .00005         S. Set Zero                            ↓
   5. .0010          H. Go Home                            + X
   6. .0100          T. Set Tool

 O   IS NOT A VALID OFFSET NUMBER
   PRESS ESCAPE TO RETURN TO JOG MENU

   PRESS 'ESC TO RETURN TO MENU
```

Enter the diameter of the material just cut

Then measure the diameter of the material you just cut accurately with a micrometer. Enter this diameter when the screen asks for it. Remember that this will be a **diameter** measurement.

NOTE: If the tool was touched off on the back side of the part (-X), then enter the diameter as a negative.

Tool Offsets
To Set Left Hand Turning Tools -continued

Measured Ø = .4923"
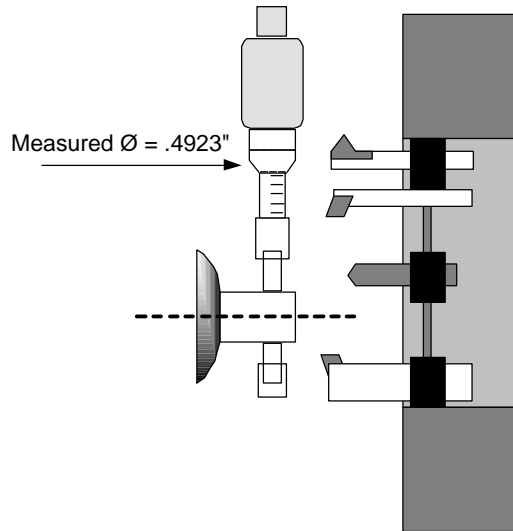
Take some care but do not be overly careful since any error made here can be easily corrected with the tool offset correction later when you are making the first piece. After typing .4923 hit "RETURN".

Now establish the Z offset, for tool #3

The setting of the Z offset is a little different.
- Touch the tool off in the Z axis and then press T like you did with the X axis.
- The control will now ask for a tool number. In this example you would press 3 and then enter.
- Then press Z when asked which axis you are setting.
- Then the control asks what the location of the tool is from absolute zero in Z.

POSITION   : X +0.00000 Z +0.00000

COMMAND   : X +0.00000 Z +0.00000

WHAT IS THE CURRENET Z LOCATION?

Jog	Automatic	Single Block	Manual Data Input

1. Slow   7. .1000
2. Medium   8. 1.0000
3. Fast   9. Est Home
4. .00005   S. Set Zero
5. .0010   H. Go Home
6. .0100   T. Set Tool

- X
- Z ← → +Z
+ X

O IS NOT A VALID OFFSET NUMBER
PRESS ESCAPE TO RETURN TO JOG MENU

PRESS 'ESC TO RETURN TO MENU

Enter where the tool is from absulute zero on the part

Using a finished part to touch off in Z:
If you put a finished part in the collet against a stop this would give you the absolute face of the part. The parts you machine should have the same location in Z when they are done. So you can jog the tool over the face of the part and touch off and when asked what the location is in Z you could enter 0.

Tool Offfsets

Using an unmachined part to set Z:

If you are using a rough part that has material left on the face to be removed you can still use it. Before you put the part in the collet measure how much you have to make the part to size. Then touch off on the known face.
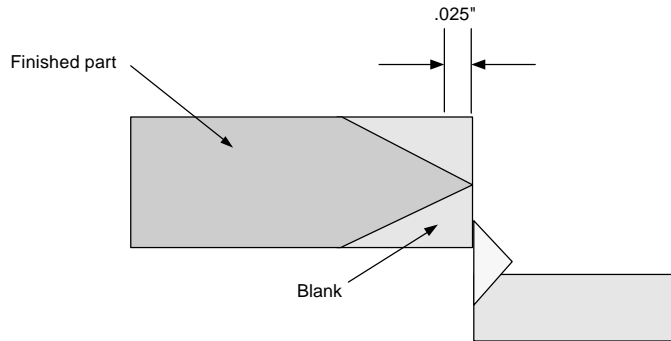
Then when asked what the location is you can enter in the known amount. In the above example you would enter in .025.

Working with bar stock:

If you are working with bar stock then the location of the Z start is only important in a relative way. Set the tool starting point far enough from the collet or chuck so there will not be a collision.

Tips - Tool Offset

A short cut for setting both X and Z offsets at the same time:

If you can set your tool in the required location in both X and Z before you go to set the tool with the "T" command it is possible to set both tools at one time without having to ESC and re-enter the T command.

To do this follow the procedure to get the tools into location, then press T and set the X tool, then before pressing ESC, press Z. This will set the Z tool and then exit back to Jog.

POSITION   : X +0.00000 Z +0.00000

COMMAND   : X +0.00000 Z +0.00000

Z OFFSET ENTERED

Jog	Automatic	Single Block	Manual Data Input
1. Slow	7. .1000		
2. Medium	8. 1.0000		
3. Fast	9. Est Home		
4. .00005	S. Set Zero		
5. .0010	H. Go Home		
6. .0100	T. Set Tool		

```
        - X
         ↑
- Z ←  →  +Z
         ↓
        +X
```

O   IS NOT A VALID OFFSET NUMBER
PRESS ESCAPE TO RETURN TO JOG MENU

PRESS 'ESC TO RETURN TO MENU

Press escape key to get back to the jog mode.

This shows that the Z offset is entered

3.8

Tool Offsets

- The screen will show "Z OFFSET ENTERED". You are now done with the Z offset for tool #3. Press "ESC" to return to the Jog menu.

Setting a Drill (Tool #1)

- Start the same as with the left hand turning tool, be sure that the slide has been HOMED
- If you are not in the jog mode, enter it now.
    - Put a 5/8" collet in the headstock.
    - Get a 5/8" pin and put it in the collet
    - Put a 5/8" bushing holder on the tool slide
    - Jog the slide until it is located where you want the drill to be.
    - With the 5/8" pin still in the collet, slide the bushing holder onto the pin.
    - Tighten the bushing holder to the slide.
    - Without moving the slide start the tool offset procedure
        - Start with the X offset
        - When asked, Set the diameter measured as 0
- Press "ESC" to go back to the Jog mode.

Postion the drill tip
just touching the face
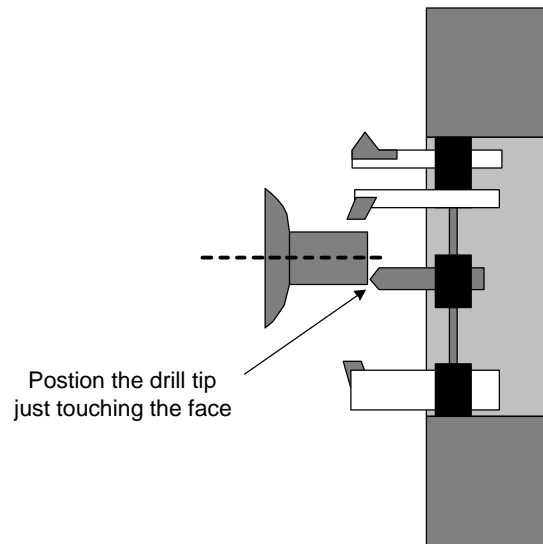
Setting a Drill (Tool #1) -continued

The Z Offset for the drill is done the same way the left hand tool was done:

- Touch the drill tip to the face of the part

- Select "T" from the jog menu

- Enter the tool number, in this case it is still # 1, then "RETURN"

- Select the axis being set, "Z".

- Now enter the location of the tool from zero. This could be 0 when touching off on a finished part. Then

-Press "ESC" to get back to the jog mode.

Tool Offsets

An alternative method of setting drill:

Position the tool as show below. Either side of the tool will do.

Setting up a drill on the + side of the spindle
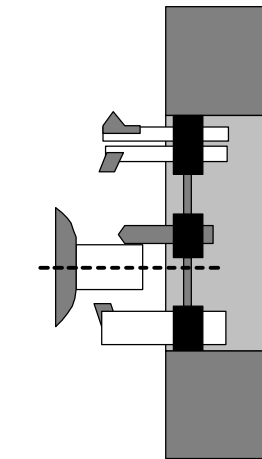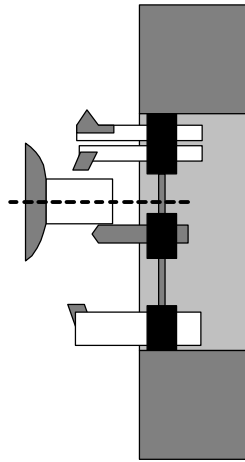
Setting up a drill on the - side of the spindle

Above there are two ways to setup a drill. In both cases the tool is not used to cut as we did in the first example with the left hand turning tool. "Touch" off the material with a feeler gauge or piece of paper. The sign of the diameter entered will tell the control which side of the spindle your tool is on.

   (+) side of spindle: enter diameter measured as a positive number
   (-) side of spindle: enter diameter measured as a negative number

With the tool just touching the material select "T" to start entering the tool offset.

• Next the control will ask "OFFSET NUMBER ? ". Type the Tool #, in this case it will be 1, and then "RETURN„

-The next question is "PRESS X OR Z TO STORE PRESENT X OR Z-AXIS OFFSET", type X

•"WHAT WAS THE TURNED DIAMETER?". This question can be answered two ways. The answer depends on what side of the spindle the tool is located. The sign (+) or (-) will change with the tool location. In either case the value of the offset will be the same.

Material diameter = .4923" from turning for left hand turning tool

.3" drill

Drill location in X = Ø material + Ø drill
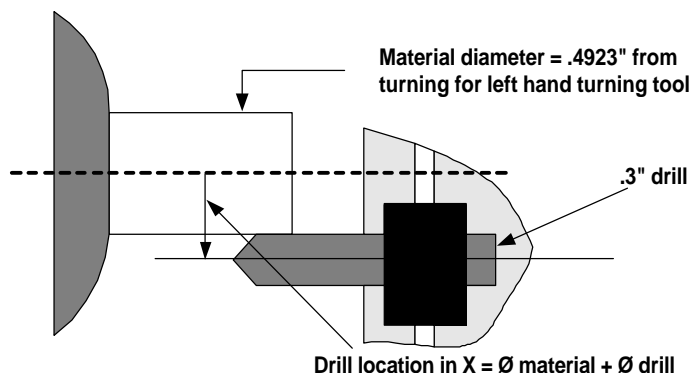
The value is equal to the diameter of the material plus the diameter of the drill. In the above figure you can see that the location of the center of the drill is the sum of these two numbers. .3" drill + .4923" material = .7923" location of drill center in X. For this example we are on the positive side so type .7923 and "RETURN".
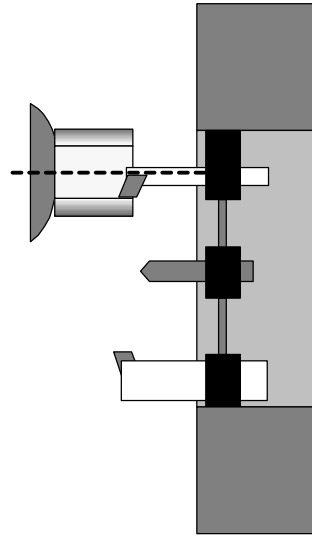
• Then set the Z as you have been shown before.

Tool Offsets

Setting ID Tools, ie Boring tools & Threading tools

The procedure for setting ID tools is similar to the two previous tools. The only difference is how you will touch off to determine the turned diameter.
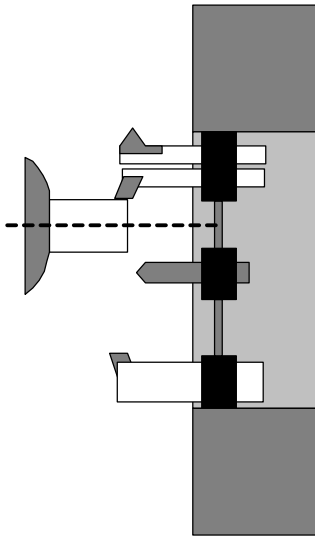
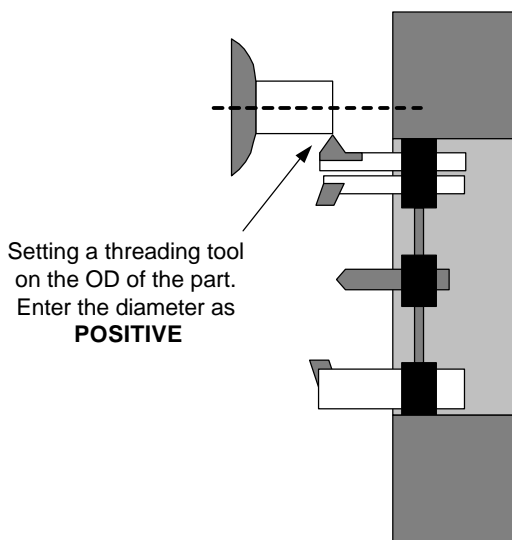Setting a boring tool on the OD.
The measured Ø will be entered as a
NEGATIVE

Setting a boring tool on the ID.
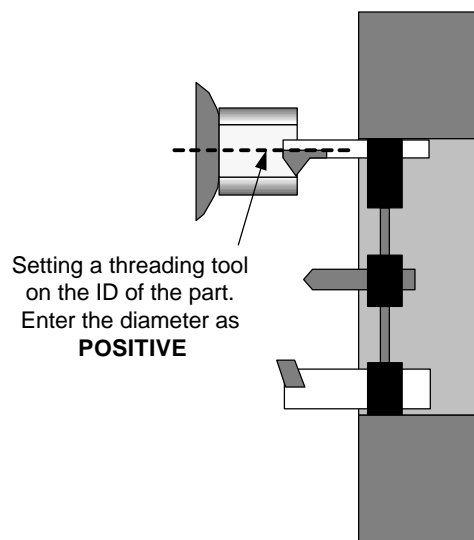The measured Ø will be entered as a
POSITIVE

Setting Threading tools

The threading tool is set similar to the other tools.

Setting X: The turned diameter is set like the other tools you have done. The offset can be set on the OD or ID.

Setting a threading tool
on the OD of the part.
Enter the diameter as
POSITIVE

Setting a threading tool
on the ID of the part.
Enter the diameter as
POSITIVE

Tool Offsets

Setting Threading tools, continued

Setting Z: The approximate location of the tool can be done by eye with the corner of the piece.

Estimate the face
of the part