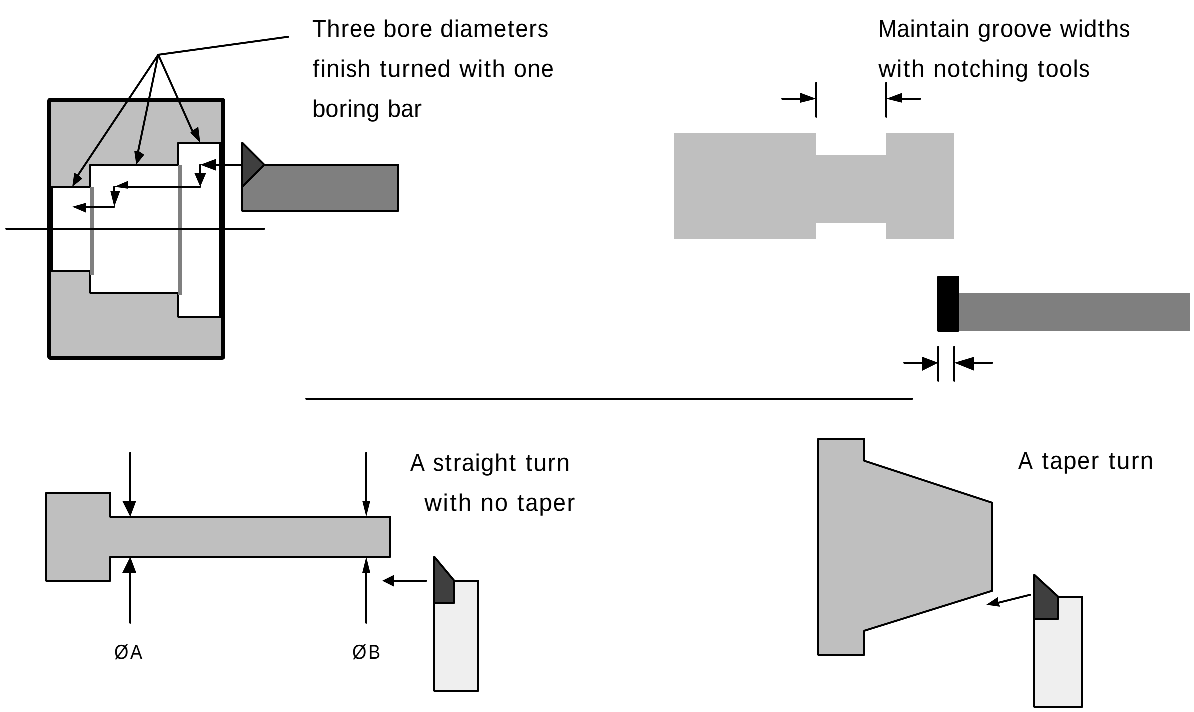
Secondary Offsets

What are secondary offsets?

Secondary offsets are corrections that you can put into your program that the operator can adjust when running the program without having to go into the program to edit it. Once the program has been written with the secondary offsets incorporated, these corrections are made by pressing **F9 while in the Automatic mode** and inputting the amounts. This procedure is very similar to adjusting tool offsets. The big difference with secondary offsets is that there can be more than one correction to a tool.

There are a number of ways that they can be used. Below are a few examples of typical applications.



In all three cases it would be very advantageous to be able to have the operator make corrections to the parts that entail more than just moving the tool by changing the tool offset (T). If you made a change to the tool offset, the overall size of the part would change in each of the above examples.

If, however, you had a taper in the long thin part (lower left sample) and had to correct it to get the part straight, offset changes would not help. The secondary offset allows you to add or subtract a little, to any move, at any point in the program. So the correction of the taper can easily be taken care of.

NOTE: Clear secondary offsets before using them!

Before you run a program that uses secondary offsets be sure that you have reset the secondary offsets that you are using to zero! This can be done by pressing C when asked to make a correction to the offset table (See F9 in the Automatic section)

Using Secondary Offsets

Secondary offsets are used with D commands. The format is the same as a T command. Add the D with a number, ie: “D2” for #2 secondary offset, to the line of code to be corrected. This command will call up the value located in the secondary offset table and add it to the move. The secondary table looks like this:

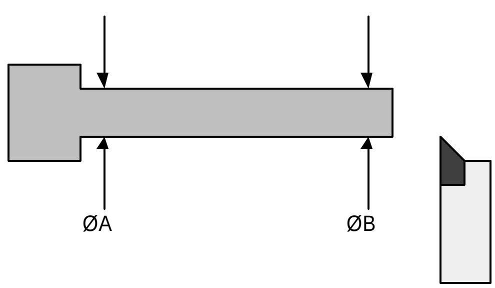
1	X: +0.00000	Z: +0.00000	17	X: +0.00000	Z: +0.00000
2	X: +0.00000	Z: +0.00000	18	X: +0.00000	Z: +0.00000
3	X: +0.00000	Z: +0.00000	19	X: +0.00000	Z: +0.00000
4	X: +0.00000	Z: +0.00000	20	X: +0.00000	Z: +0.00000
5	X: +0.00000	Z: +0.00000	21	X: +0.00000	Z: +0.00000
6	X: +0.00000	Z: +0.00000	22	X: +0.00000	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

Secondary offset number:

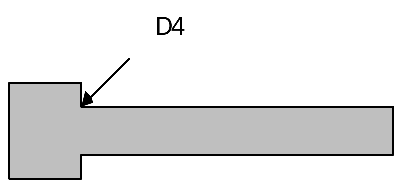
Press C to clear all offsets:

Press Esc to exit offset adjustment screen

Look at the following example:



With this shaft, ØA = ØB. In order to do this, put an offset command at the end of the turn.



Secondary Offsets

The code to face and turn the shaft is:

Without correction	With correction	
X0Z0	X0Z0	move to the center and face
X.125	X.125	move to the diameter
Z-.75	Z-.75**D4**	turn the diameter, corrected
X.2	X.2**D0**	move to the major diameter, turn offset off

If there is no problem with a taper on the part, the X and Z values of D4 are set to zero. If, however, there is a taper, say .001" oversize at the base of the turn, this can now be corrected. Call up the secondary offset table "F9". The control will ask "Offset Number?" Type 4 and return. Then it will ask "X DIAMETER CORRECTION?". Type -.001 and Return. You will notice that the value of #4 - X will now be -.0005. This is because the offsets affect the radius, not the diameter. Then the control will ask "Z CORRECTION?". If there is no correction just hit Return. Then to return to the automatic mode press ESC. This will bring you back to Automatic mode ready to run the program again. To start the program again, press "CYCLE START".

You will notice that the operator does not have to go into the program to adjust for the taper. With this feature you can have personnel make corrections without having to understand programming.

The correction of -.0005 will be added to the third line move and it will behave like the line was:

X.1245Z.75

Since there was no correction in X, there is no X value.

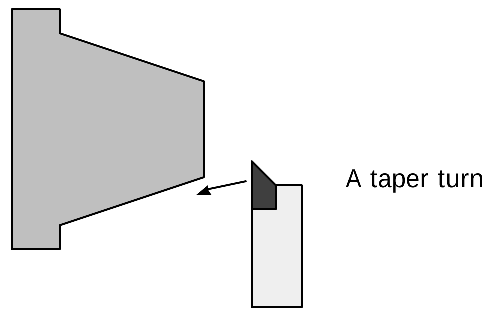
Canceling a secondary offset

The secondary offset correction stays in effect until:

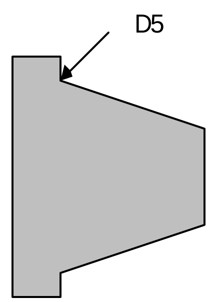
- There is a tool change. This cancels an offset
- A secondary offset D0 will turn off the offset.
- Calling up another offset will cancel the original offset and enact the new offset

Secondary offset examples:

A taper that has to be single point turned and then maintain the major diameter.



The actual taper that a tool cuts will depend on the toolnose radius. If this is not an easy tool to maintain then the taper will vary as the tool changes. Normally this would be a big problem. However with the secondary offsets this is very simple. We could put a secondary offset at the end of the taper and then turn the offset off. This would look like:



The commands might look like this:

X0Z0	Move to the center and face of the part
X.2	Move to the minor diameter of the taper
X.3Z-.5**D5**	Mover to the major diameter of the taper, corrected with D5
X.4**D0**	Mover to the major diameter of the part, correction off, D0
Z-.7	Turn the major diameter of the part

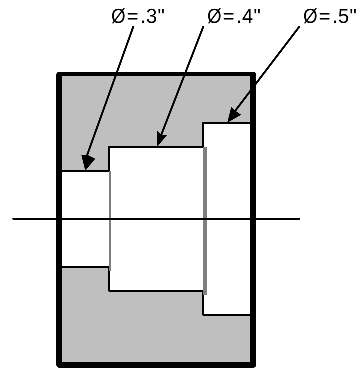
In this case D5 can be used to adjust two features:

- The diameter at the end of the taper - D5 X value
- The location in Z of the taper. - D5 Z value

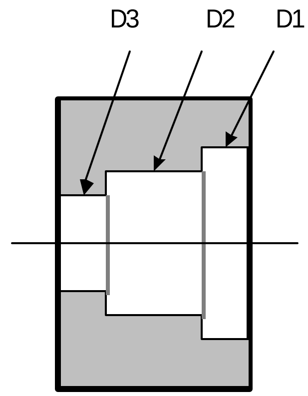
Neither, either, or both values can be entered. Each will effect the angle generated.

Secondary Offsets

Here is another example:



For this example, three diameters have to be turned with only one tool. Each of the diameters can have an individual offset.



The coding for this is:

X-.35Z0	move to the face of the part at -.7 diameter
X.25**D1**	move to .5”Ø, corrected
Z-.2	Turn the first bore
X.2**D2**	Move to the second bore diameter, corrected
Z-.5	Bore the second diameter
X.15**D3**	Move to the last bore diameter, corrected
Z-.7	Bore the last diameter

Here we have used the secondary offsets to help position the tool at the beginning of each of the bores. The offset correction will be added to the positioning move and then be maintained until a new offset is changed. For the .5”Ø we have added D1 to the move that brings the tool to the .5” diameter. Then, when we bore the first diameter, the correction is continued and the hole is on size. If we have no correction on the remaining two bores they are not corrected since there are zero values in each of the offsets. The next move to .4”Ø and .3”Ø will not be corrected. When the next tool is called, the D3 offset will be canceled.