

## G72 - G73 Diameter and Radius Programming

G72 sets the control so that X moves will now be read as diameter values. This follows more closely to the Fanuc style format.

### G72 Diameter Programming Mode

The G72 command is modal. Once it is used this mode stays active until it is turned off with a G73 command. *The default mode is G73 - radius.* So be sure to use the G72 command in your program if you want to program in the diameter mode.

NOTE: Use the G72 command in the beginning of the program before you call any tools.

NOTE: If you plan on using SEARCHTO (F6) in the automatic page be sure to put a G72 after the tool call statement. The search to command will not pick up the call out for the diameter mode at the beginning of the program. If you have it on the first line after the tool call it will work. IE:

```
T1  
G72X.5Z.1
```

### Caution for those using older software !!!!

1996 and earlier

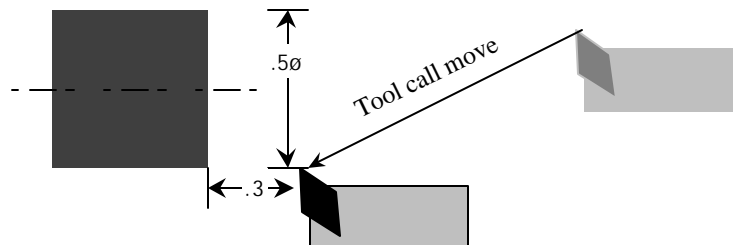
#### Setting tools for Diameter programming

*Note:* for those using old software (before 12/96) you will have to be careful! The SET TOOL command in the Jog mode will make the tool go to the *radius value* for the STARTING POSITION. As an example if you want your tool to start at a 1" diameter you will have to enter .5 when asked where you want the tool to start. Then when you call your tool you will have a X value of 1 in the G92 statement since in the program it will read the X values as diameter.

*Until you change the software we suggest that you start you tools as X=0. This will eliminate any problems with confusing diameter and radius moves.*

#### Tool Call statements - G92

After a G72 command has been used the value of X in the G92 statement is used to represent a diameter location. In the example below the tool is moved to the diameter value of X=.5 (see the section on *setting tools with diameter programming*)



The G92 statement would be: G92X.5Z.3