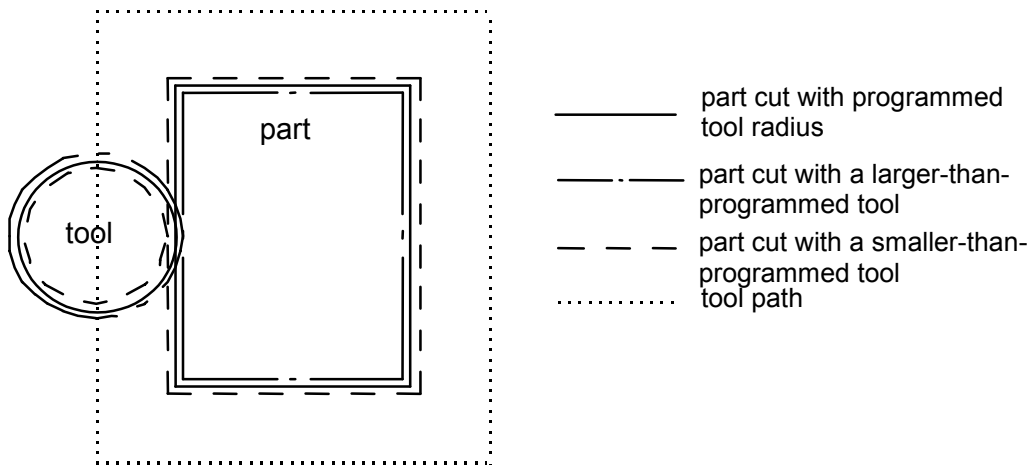


## Understanding Cutter Compensation

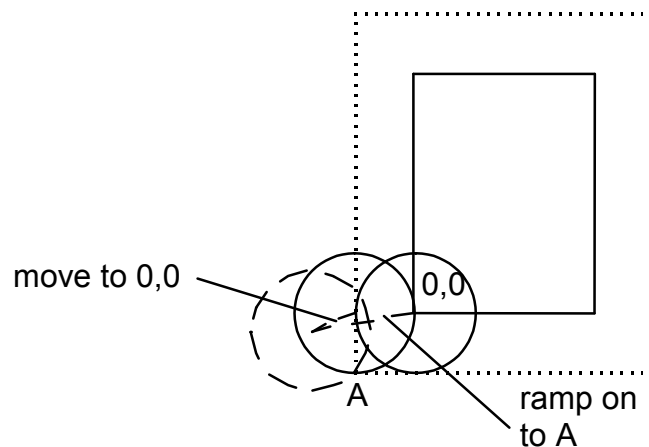
Cutter compensation is one of the most useful things to know when CNC machining. Cutter compensation allows you to program the geometry and not worry about the toolpath. It also allows you to adjust the size of your part, based on the tool radius used to cut your part. This is useful when you can't find a cutter of the proper diameter. This is best explained in the graphic below.



The solid circle is the nominal sized tool. The dashed circle is an undersized tool, and the dash-dot circle is the oversized tool. With a little imagination, you can see all the possibilities for tweaking your part, or getting your part made with any size endmill.

## TURNING CUTTER COMPENSATION ON AND OFF

It is important to note that cutter compensation becomes active after the next line move or rapid that is at least the length of the tool radius. Failure to account for this will give a funny part. A good method around this is to zero your part and program a move away from the part in the X and Y direction equal to the tool radius. Then move back to 0,0, and then continue cutting your profile. See the graphic below. Note the tool center is now perpendicular and to the left of point A.

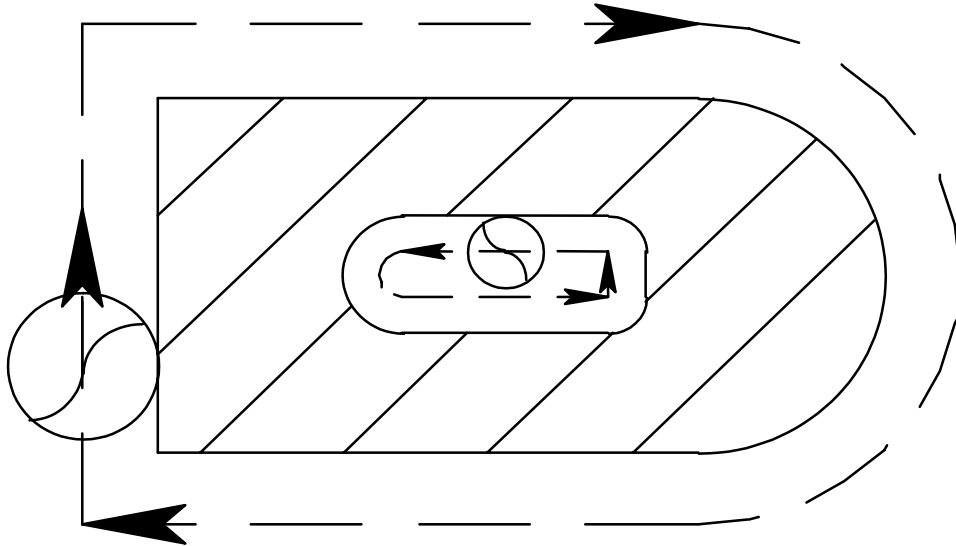


To turn cutter compensation off, you must do a ramp off move similar to the ramp on move. Again, send the tool off in the X and Y direction a distance equal to the tool radius. For the graphic above, after reaching 0,0 turn off cutter compensation and ramp off to A. Depending on the shape, you may have to go beyond 0,0 to eliminate any 'nurkies' (a nurkie is an unintentional over or under cut left by the tool). This terminates cutter compensation, and you can go on to something else.

There are three G-Codes involved in using cutter comp : G41 initiates cutter comp to the left of the path; G42 initiates cutter comp to the right of the path; and G40 cancels cutter compensation.

## CLIMB & CONVENTIONAL CUTTING WITH CUTTER COMPENSATION.

This refers to the direction of cut relative to the direction the spindle is turning. In CNC milling it is common practice to climb cut. With the spindle turning clockwise, as it usually does, cut a contour in a clockwise direction. If cutting a pocket or hole cut counter clockwise. Your parts will come out okay if you go the wrong way, but surface



This part is shown with two tools climb cutting  
The big tool is set cutter comp left  
The small tool is also set cutter comp left

finish and accuracy is usually better when you climb cut.

### SUMMARY

#### Cutter Compensation:

- allows you to program the geometry not the tool path
- is useful when you don't have the right endmill
- is helpful in tweaking your part size
- allows you to compensate for tool wear
- is generally a neat and powerful thing to know about

#### revision history :

Ver 1.0	4/94	John Wadsworth	original text
Ver 1.1	8/01	Katherine Kuchenbecker	minor revisions