

Radius compensated tool path

Cutter Radius Compensation

With the cutter radius compensation the control calculates automatically a path parallel to the programmed contour and compensates so the cutter radius.

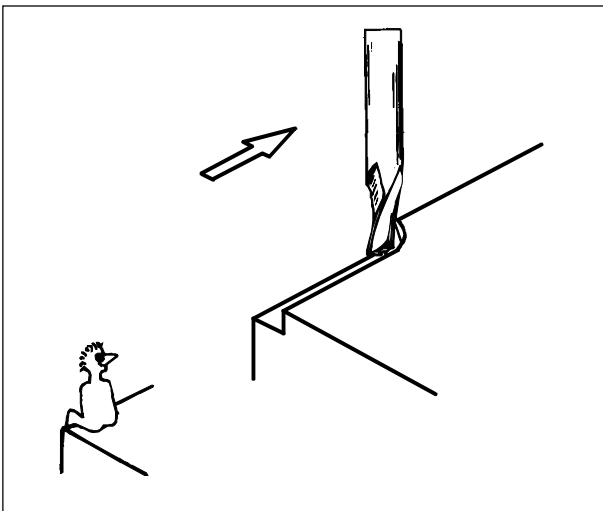
G40 Cancel Cutter Radius Compensation

The cutter radius compensation will be cancelled by G40.

Cancellation is only permitted in combination with a linear traversing movement (G00, G01).

G40 can be programmed in the same block like G00 or G01 or in the previous block.

Usually G40 will be programmed with the retraction to the tool change point.



Definition of G41 cutter radius compensation left

G41 Cutter Radius Compensation left

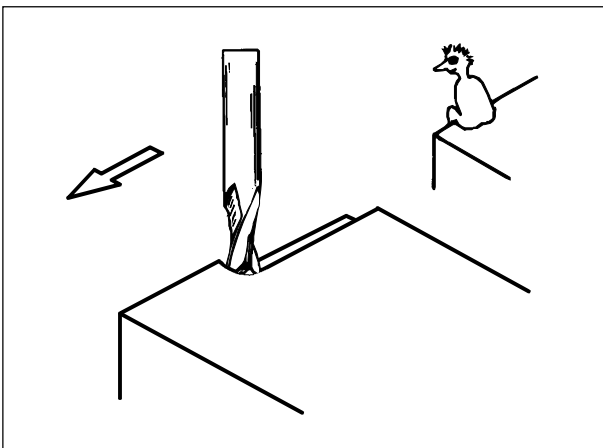
If the tool is (viewed in feed direction) at the **left** side of the contour to be worked, G41 has to be programmed.

For calculating a radius, an H parameter in the offset register (OFFSET) which represents the cutter radius must be programmed and called up with G41 e.g.:

N... G41 H..

Notes

- Direct change between G41 and G42 is not allowed - previous cancellation with G40.
- Selection in combination with G00 or G01 necessary
- Programming an H parameter is necessary unconditionally, the H parameter is effective modally.



Definition of G42 cutter radius compensation right

G42 Cutter Radius Compensation right

If the tool is (viewed in feed direction) at the **right** side of the contour to be worked, G42 has to be programmed.

Notes see G41!