Tool Radius Compensation

by

Dr. Sotiris Omirou

As the cutter moves around the shape you should be able to see that the point of contact between cutter and the shape changes.

Watch the simulation to see what we mean.
The fact that the centre of the cutter is not on a path that is simply the radius of the cutter from the edge of the work means that programming the actual tool path is very difficult.

Most control systems have a method of cutter radius compensation to make programming easier.

Cutter radius compensation means that the actual profile to be machined provides the dimensions for programming. The control system automatically calculates the position that the centre of the cutter needs to be at to machine the profile.
The G codes used for cutter/radius compensation are:

G41 - Compensation left
G42 - Compensation right

G40 cancels the codes above