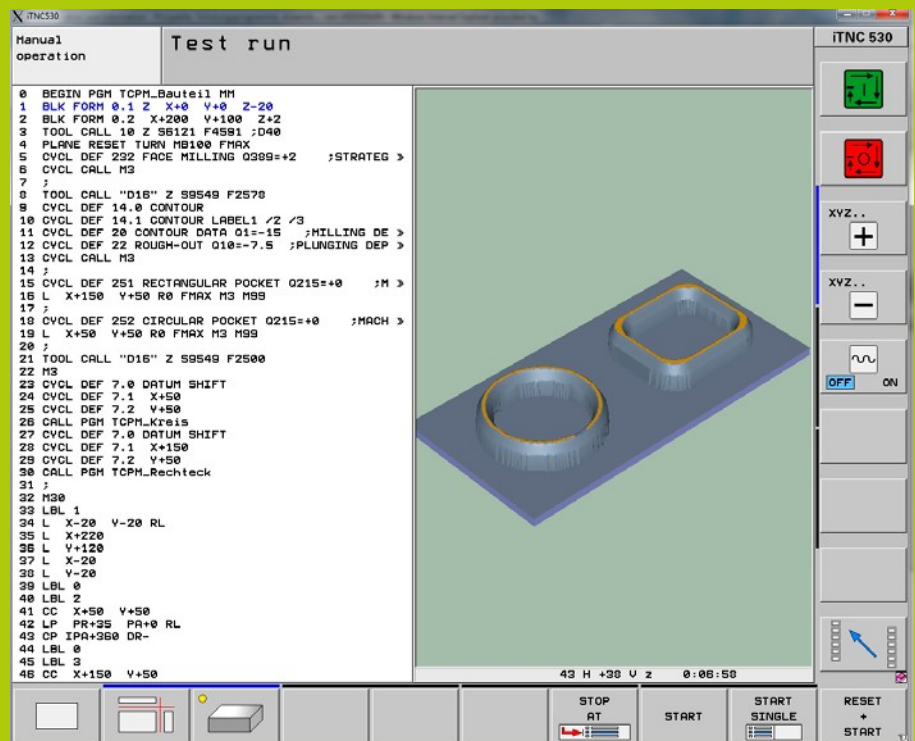


From the HEIDENHAIN NC program database

Chamfering like a champion

Machining of workpiece edges—can it be an easy task? Chamfers are encountered every day, at a certain angle on a circle or rectangle. KLAR-TEXT shows you a parameter program with which a chamfer is easily produced, using the lateral surface of an end mill. Of special interest: the program is machine-neutral, meaning that it is independent of the rotary axes actually present. This is possible with the TCPM command (Tool Center Point Management).



The TCPM function is an enhancement of the M114 and M128 functions. With TCPM the control calculates the inclination of the rotary axes. An advantage of TCPM is that you decide how the feed rate is defined in the program, i.e. whether it refers to the tool tip (F TCP) or the associated axes (F CONT).

Use our parameter program to easily mill a peripheral chamfer on studs or pockets. The coordinates of the rotary axes are important: AXIS SPAT is used to specify that the subsequent axis positions are interpreted as spatial angles, so that the program can be used on any machine.

In the last entry you define the type of interpolation between the starting and

end position. Since we want to use hob milling, we select PATHCTRL VECTOR. This way we ensure that the upper and lower course of the contour are parallel to each other.

NC database

In the NC program database under <http://applications.heidenhain.de/ncdb>, HEIDENHAIN presents programming solutions for everyday milling tasks. Download these prepared programs from the Internet at no charge, and easily adapt them to your needs by just a couple of changes in the programming. The program described here ("Chamfers with inclined tool") is in Chapter 5 Deburring.