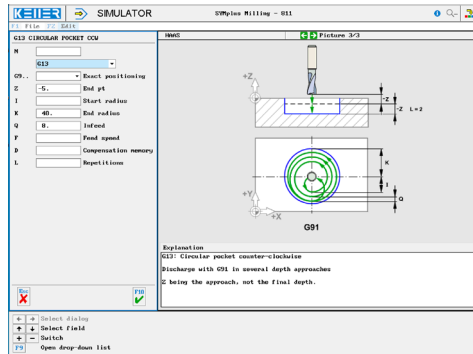# CONTROLS
Control simulator HAAS (milling)
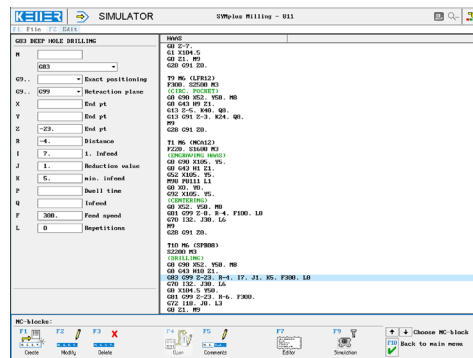
KELLER SOFTWARE

The control related simulator is an additional module for SYMplus. This simulator is used to learn the control system programming. A program generated by a postprocessor can also be edited and simulated.
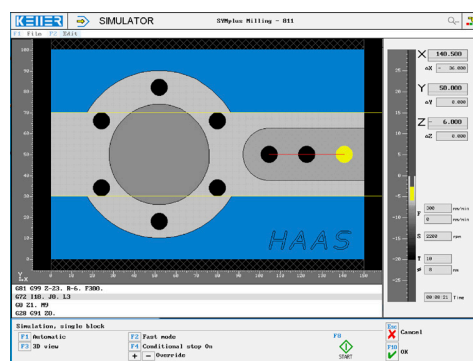
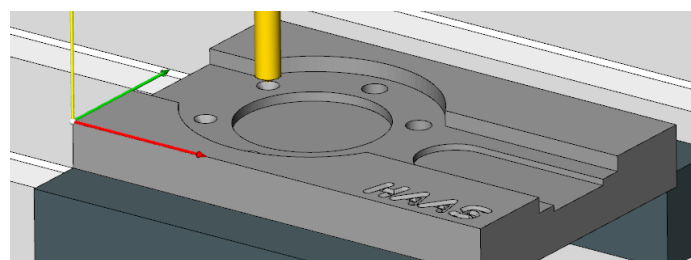## Help pictures and texts (for all input dialogues)

NC editor incl. input dialogue and syntax control

## Perfect control due to simulation with position indicator (real time, fast mode, single block with path preview, …)

Particularly vivid depiction: 3D simulation

COMMANDS/ FUNCTIONS/CYCLES

G00	Rapid traverse
G00 A	Rapid traverse a axis
G01	Line
G02	Arc, clockwise
G03	Arc, counter-clockwise
G04	Dwell time
G12	Circular pocket cw
G13	Circular pocket ccw
G28	Approach reference point
G29	Exit from reference point
G40/G41/G42	Radius compensation
G43/G44/G45	Length compensation
G50/G51	Scaling
G52	Local coordinate system
G53	Machine coordinates
G54-G59	Workpiece coordinate system
G65	Macro call
#	Paramater
GOTO/IF/WHILE	Jumps and loops
G68/G69	Orientation
G70	Bolt hole circle
G71	Bolt hole arc
G72	Bolt hole along a line
G73	Deep hole drilling
G74	Left tapping
G76	Fine boring
G77	Back drilling
G80	Finish drilling cycle
G81	Drilling
G82	Spot drill
G83	Deep hole drilling
G84	Tapping
G85…G89	Drilling
G90/G91	Exact positioning
G92	Set local coordinates
G94	Feed mm/ min
G100-G101	Mirror image
G110-G129	Workpiece coordinate system
G150	Pocket milling
T	Tool call
Comment	Explanations
X/Y/Z	Modal commands
M97	Local sub-program call
M98	Sub-program call
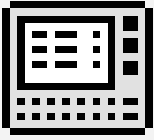
CONTROLS
Interactive training

The HAAS simulator also includes a multimedia, interactive training module. Here the learner can independently practice programming and operating the HAAS control system using original key sequences and then take a test at the end.

The contents are divided into 4 main chapters:

- Geometrical basics
- Technological basics
- From the drawing to the workpiece
- Introduction to the operation

The circular pocket cycle G13 is introduced here.

Guided by the software, you learn the key sequences for creating, editing, simulating, saving and transferring programs.
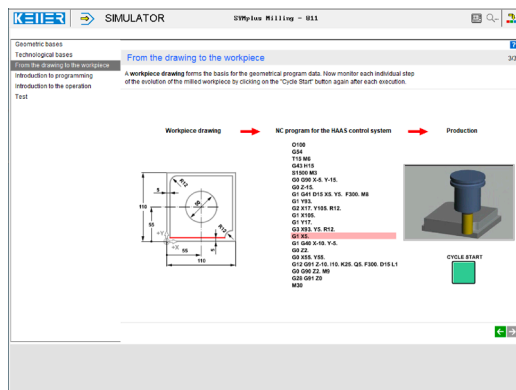

Exercises are included for the following functions/topics, among others:

G0/G1/G2/G3
Contour creation with rounding and chamfer
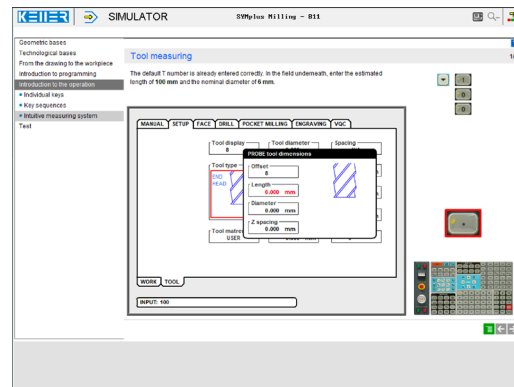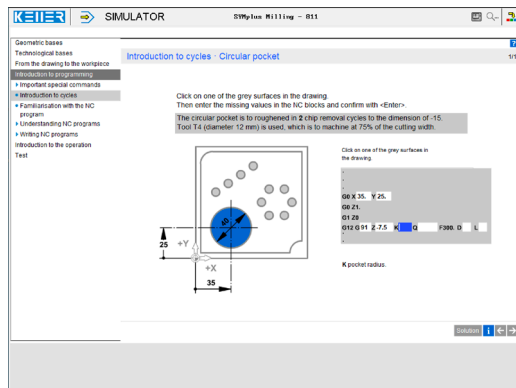
G28
G40/G41/G42
G43
G53/G54

G12/G13
G70/G72

In addition, there is general learning content on cutting data with numerous practical examples and a final test with randomized tasks for all chapters.

In a further sequence the measuring of tool and workpiece by means of a measuring system is practiced.