



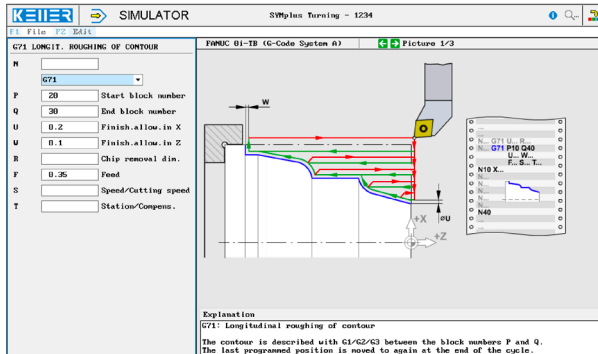
CONTROLS

Control simulator FANUC Oi (turning)

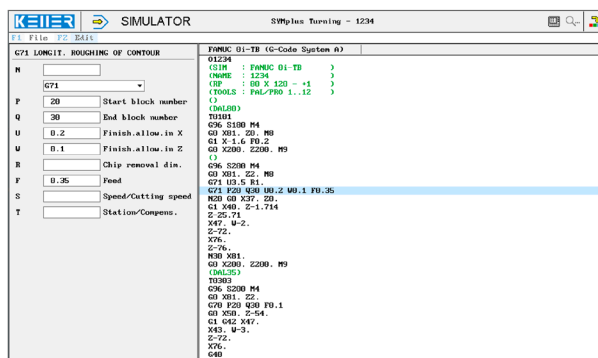


The control related simulator is an additional module for SYMplus. This simulator is used to learn the control system programming. A program generated by a postprocessor can also be edited and simulated.

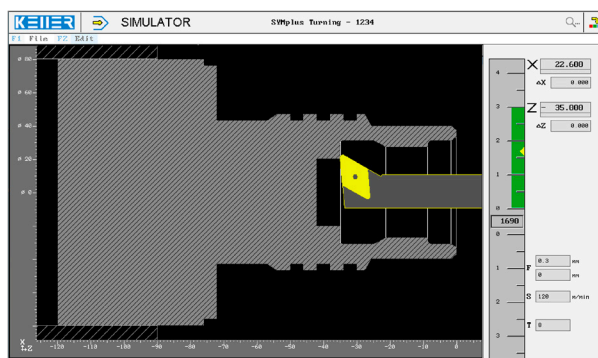
Help pictures and texts (for all input dialogues)



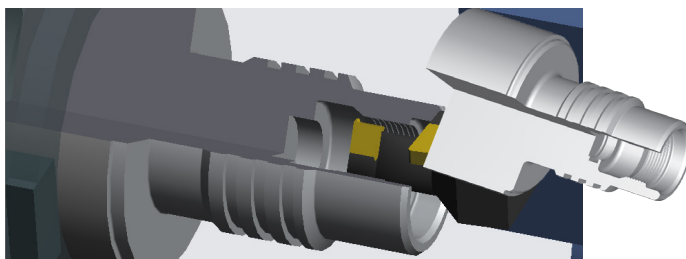
NC editor incl. input dialogue and syntax control



Optimal control through simulation with position display (real time, fast run, single block with path preview etc.)



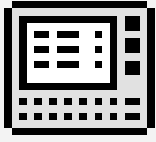
Especially vivid: the 3D simulation



COMMANDS/ FUNCTIONS/CYCLES

G0	Rapid traverse
G1	Line
G2	Arc, clockwise
G3	Arc, counter-clockwise
G4	Dwell time
G28	Approach reference point
G32	Threading
G40/G41/G42	Radius compensation
G50 S	Speed limitation
G50 Z	Set local coordinates
G52	Additive zero offset
G53	Machine coordinates
G54-G59	Zero offset
G65	Macro call
#	Parameter
GOTO/IF/WHILE	Jumps and loops
G70	Finishing contour
G71	Longitudinal roughing of contour
G72	Face roughing of contour
G73	Parallel roughing of contour
G74	Face recessing
G75	Longitudinal recessing
G76	Threading
G80	Finish cycle
G83	Deep hole drilling
G84	Tapping
G85	Boring
G90	Longitudinal roughing
G92	Threading cycle
G94	Face roughing
G96	Constant cutting speed
G97	Constant speed
G99	Feed per revolution
Comment	Explanations
T	Tool call
M98	Program call
X/Z/F/S/M	Modal commands

Attention: FANUC distinguishes between different G-code systems when it comes to turning. This simulator is based on system A.



CONTROLS

Interactive training



www.cnc-keller.de



CNC KELLER GmbH



keller.software



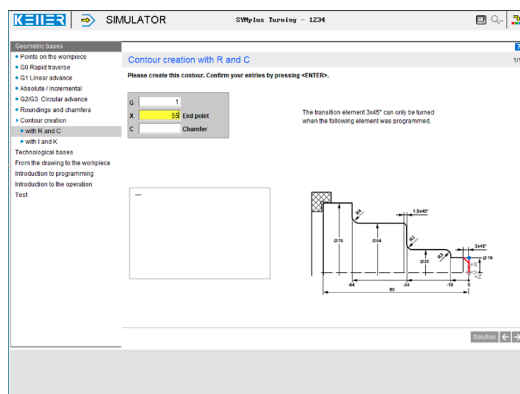
KELLER Software



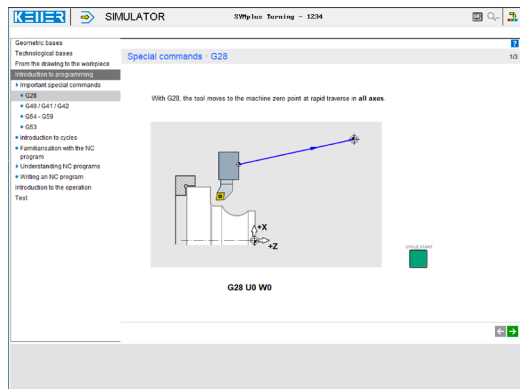
The FANUC simulator also includes a multimedia, interactive training module. Here, the learner can independently practice programming the FANUC controls and take a test in the end.

The contents are divided into 4 main chapters:

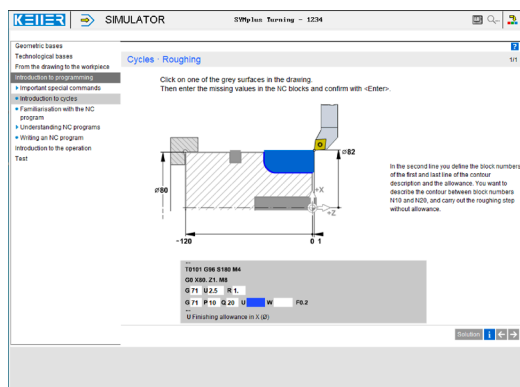
- Geometrical basics
- Technological basics
- From the drawing to the workpiece
- Introduction to programming
- Introduction to the operation



In this exercise you learn interactively how to use the command G28 to approach the zero point of the machine.



Cycles are introduced here.
Later you „write“ your own small programs interactively (guided by the software).



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

G0/G1/G2/G3

Contour creation with rounding and chamfer

G28

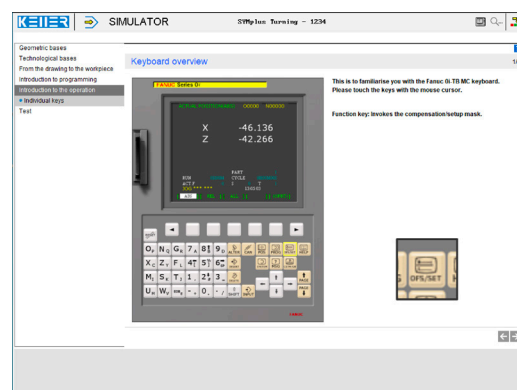
G40/G41/G42

G53/G54-G59

G71/G75/G83

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

You learn the meaning of each key ...



... and you can check yourself in a test.

Test · Operating the Fanuc controls · Key functions

Assign the correct meanings to the keys using Drag&Drop.

Shift key		
Softkey		
The content of the keyboard buffer memory is applied as final.		
Resetting the CNC, resetting an alarm, etc.		
Toggling softkey bars		
Invoking the help function		