# simCNC – the list of supported gcodes and mcodes

simCNC – the list of supported gcodes and mcodes

The software provides dynamics and precision of motion not seen before in other proprietary CNC control software. This was achieved by the **S-curve profile** as well as sophisticated algorithms for motion trajectory optimization. It means that a machine can be fast, dynamic and precise at the same time and a treatment process is smooth and steady. It highly affects processing time, extends the life of tools and of mechanics of a machine. SimCNC software is equipped with Python scripting language.
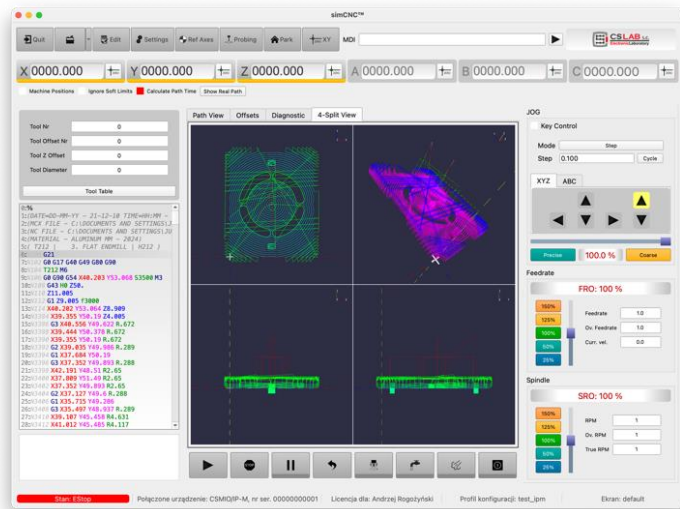
The program uses the popular, standard ISO G-Code trajectory description language, but when loading and working it analyzes every segment, arc or corner and practically recreates all shapes using advanced mathematical algorithms. Many variables are taken into account, such as given tolerance, speed, accelerations and type of curvature to generate data for the controller and axis drives in such a way that the movement is accurate, smooth and fast.

simCNC – the list of supported gcodes and mcodes:

G00 – Rapid Move
G01 – Linear Move
G02/03 – Arc Move
G04 – Dwell
G10 – Tool Offset and Work Offset Tables
G17/G18/G19 – Plane Selection
G20 / 21 – Unit Selection
G28 / G30 Return Home
G31 – Probing – Probe number 0
G31.1 – Probing – Probe number 1
G31.2 – Probing – Probe number 2
G31.3 – Probing – Probe number 3
G32 – Threading
G40 – Accepted but not used
G43 – Set Tool Length Offsets
G49 – Disable Tool Length Offsets
G50 – Accepted but currently not used
G53 – Move in ABS Coordinates
G54 – G59.3 Work Offsets 1- 9
G64 – Set precision value (CV)

G68 – Rotate Coordinates Command (version 3.300 and newer)
G80 – Cancel Canned Cycles
G81 – Drill Cycle
G82 – Drill Cycle with Dwell
G83 – Peck Drill Cycle
G90 – Distance Mode (absolute)
G91 – Distance Mode (incremental)
G92 – Temporary work offsets shift (version 3.300 and newer)
G94 – Accepted but currently not used
G98 – Canned Cycle Return to Z plane
G99 – Canned Cycle Return to R plane
M01 – Program stop
M03 – Optional program stop
M04 – Rotate spindle clockwise/counterclockwise
M05 – Stop spindle rotation
M06 – Tool change (tool change script activation)
M07 – Mist on
M08 – Flood on
M09 – Mist & flood off
M30 – Program end and rewind
M62 P0 – enable output synchronized with trajectory number 0
M63 P0 – disable output synchronized with trajectory number 0
M62 P1 – enable output synchronized with trajectory number 1
M63 P1 – disable output synchronized with trajectory number 1
M62 P2 – enable output synchronizedwith trajectory number 2
M63 P2 – disable output synchronized with trajectory number 2
M62 P3 – enable output synchronizedwith trajectory number 3
M63 P3 – disable output synchronized with trajectory number 3

Description of the gcodes will be released soon. The definition of G2 / 3 is in accordance with the ISO standard.