

CNC SWIFT



Low Cost,

Three Axis Simultaneous

Milling Controller with

User Friendly Graphic

Interface and International

Machine Tool Language

CNC SWIFT

STATE OF THE *Art* PC BASE CNC MILLING MACHINE CONTROLLER

INTRODUCTION

CNC SWIFT is a general purpose CNC milling machine controller. It comprises of a motion control card, I/O cards and a software, dedicated for the three-axis simultaneous milling. It is targeted at the CNC milling machine manufacturing and retrofitting market. The user friendly graphic and the standard M & G code are irresistible to the machine user - with the onboard graphic function, most of the simple part drawings need not to be programmed, it can be drawn directly on the computer screen and a CNC part program will be automatically generated by the software. Before the machining operation takes place the tool path can be verified on the PC monitor to avoid programming error. As it is compatible with FANUC CNC software, user may use FANUC O-M SERIES postprocessor for generating CNC part programs from any CAD/CAM system and further editing can be carried out under the **CNC SWIFT** environment if necessary.

CNC SWIFT PROGRAMMING COMMANDS

The first block of each CNC part program normally begins with G90(or G91), G20 (or G21) and G40. For safety purpose, always clear the controller from the previous program before setting to an appropriate mode. The basic programming commands, listed with their functions, are as follows:

- Axis motions (G00, G01, G02, G03)
- Dwell (G04)
- Exact stop (G09)
- Polar coordinate commands (G15, G16)
- Working plane selections (G17, G18, G19)
- Positioning systems (G90, G91)
- Units (G20, G21)
- Reference point returns (G27-G30)
- Tool selection and change (T___, M06)
- Feed rate change (F___, M48, M49)
- Spindle controls (S___, M03, M04, M05)
- Coolant controls (M07, M08, M09)
- Stop controls (M00, M01, M02, M30)
- Working coordinate selection (G52-G59)
- Tool radius compensations (G39-G42)
- Tool length compensations (G43, G44, G49)
- Offset parameter specifications (D___, H___, H99)
- Tool offset modifications (G45-G48)
- Scaling (G50, G51)
- Exact stop mode (G61)
- Tapping mode (G63)
- Continue cutting mode (G64)
- Canned cycle commands (G73, G74, G80-G84, G98, G99)
- Sub-program calls (M98, M99)
- MACRO instructions (G65)
 - Arithmetic (H01-H05)
 - Logic (H11-H13)
 - Functions (H21-H28)
 - Trigonometry (H31-H34)
 - Conditions (H80-H86)
- Rotary table motions (G00, G01, G200, G201, G202)
- Programmable logic inputs (M501, M502, M503 ... extensible)
- Customizable outputs (M201-M207, M301-M307, ... extensible)

FEATURES

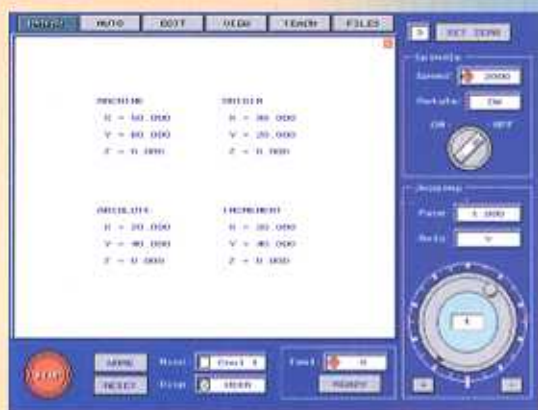
- 2 / 3 / 3+1 axes CNC software for milling and machining center.
- Simplest CNC configuration! PC motion control card and **CNC SWIFT**
- Support inverter controlled spindle and servo-controlled spindle.
- Connection with control panel is made easy.
- Electronics hand wheel is available.
- Support Auto-Tool Changer (ATC)
- Three cutting modes: Exact Stop Mode, Continue Cutting Mode and Tapping Mode.
- Feed rate can be overridden on the fly in two different ways (new feed rate setting or percentage).
- ISO Standard G codes and M codes. Compatible with all CAD/CAM systems.
- All system parameters, IO parameters, program control parameters, tool parameters are written in system files: System Configuration File, IO Configuration File, Error Compensation File, Directory File, Parameter File, Macro Table, Offset Table, Tool Data Table, ATC Table. That makes the **CNC SWIFT** flexible and easily customized.
- Unlimited CNC program size.
- Sub-programs and MACRO-instructions make programming work quick and neat.
- Sub-programs can be embedded up to 9 levels.
Primary MACRO-instructions were built in for arithmetic calculation, logical operation, mathematical functions and conditional divergence.
- Two inputs for special events (one for pause and other for stop).
- Programmable logical inputs and outputs are extensible and customizable.
- Canned cycle commands for drilling, tapping and boring.
- Cartesian coordinate and polar coordinate are available.
- Commands for tool path scaling and mirroring.
- Tool offset is controlled either by offset parameters in the tool data table or by the offset commands in a CNC program.
- Multi coordinate systems, such as the machine coordinate system, the user-defined coordinate system, pre-defined working coordinate systems and the local coordinate system.
- ATC function is controlled with ATC parameters table. However, manual tool change is also allowed.
- Backlash compensation and pitch compensation.
- Emergency stop and resume functions.
- Twin-software, **CNC SWIFT SIMULATOR**, including all the operation modes and functions in **CNC SWIFT** except servomotor controlling functions. It is developed for CNC education, operator training and off-line programming.
- Utility software, **CNC SWIFT TOOLBOX**, for installation and troubleshooting. The software consists of six modes: System test mode, Limit setup mode, Table shifting mode, System updating mode, Digital Read-Out mode, ATC Setup mode.
- A CNC machine could be set-up either fully automatic with the Automated Operation mode, or semi-automatic with the Manual Operation Mode, or manual with the DRO feature in the TOOLBOX.
- Graphic display is controllable. Less key-inputs are required in Graphic User Interface.
- More functions and automations are added for operator's convenience.
- More hot keys are added for high efficiency.

ELECTRIC HARDWARE

- Pentium system & above with 128 MB RAM running with DOS 6.22, Windows 95 and Windows 98
- Motion control card
- Digital I/O card (144 I/O)
- Analog I/O card (optional if automatic speed control for the spindle is not required)
- Encoder card (optional if electronic hand wheel is not required)

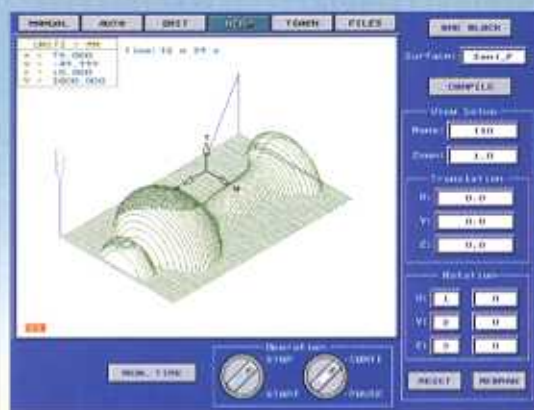
"WINDOW-LIKE" DATA DISPLAY AND DATA INPUT OPERATION MODES IN THE CNC SWIFT ENVIRONMENT

There are six operation modes in CNC SWIFT. User may activate any mode by means of the mode control menu bar in the control window. The modes are :



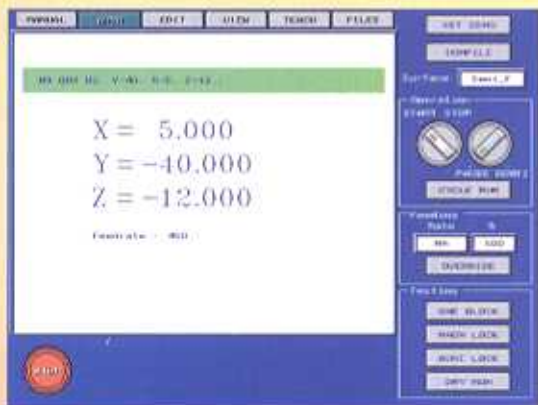
Manual Operation Mode

User may manually return the machine sliding table to its reference point by either using the hand wheel or rotating the "manual handle" displayed on the monitor; adjust step resolution; control spindle speed; turn coolant on/off, etc.



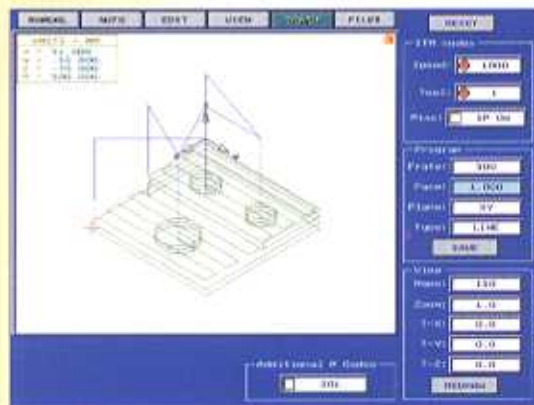
View Mode

The tool location and tool path of a CNC part program could be monitored without operating the machine. When the tool path created by a CNC program is displayed on the screen, different views, zoom-in/out views and every command block could be specified in real time if desired.



Automated Operation Mode

CNC part program runs automatically in this mode. The process can be Stopped by pressing ESC key or Emergency Stop button or Paused by pressing Spacebar or Pause button. The current executing command and coordinate system will be displayed on the screen as well as the current status of the machine.



Teach-in Mode

The key points of a tool path could be graphically input in this mode. The feed rate and step could be selected respectively from one point to another. Once the graphic input is saved the CNC SWIFT will automatically create the CNC part program in the memory.



Editing Mode

CNC part program and system files in the memory can be edited in this mode. It is a full-screen editor. The text can be inserted, deleted, marked, cut, copied, pasted and saved in this mode.



File Management Mode

CNC part program may be upload, download, viewed and saved in this mode.



SDI Technologies Pte Ltd

10, Kaki Bukit Road 1, #01-36 Kaki Bukit Industrial Park, Singapore 416175

Tel: (65) 6841 4290 (Main line) Fax: (65) 6297 3682

URL: <http://www.sdltech.com.sg> Email: sdltech@sdltech.com.sg