

Educational CNC Milling Trainer

With 8 Automatic Tools Changer



Edu Mill II Vr.1



ATMI Surakarta

Jl. Adisucipto/Jl. Mojo No.1 Kotak Pos 215 Surakarta 57102, Jawa Tengah-Indonesia
Telp : 0271 714466 • Fax : 0271 714390 • Email : atmi@indo.net • Web : www.atmi.ac.id



SMK St. Mikael Surakarta

Jl. Adisucipto/Jl. Mojo No.1 Kotak Pos 215 Surakarta 57145, Jawa Tengah-Indonesia
Telp: 0271 712728 • Fax: 0271 714390 • Email: info@smkmikael.com • Web: www.smkmikael.com



Edu Mill II Vr1

With Automatic Tool changer

The Educational CNC Milling II Trainer Series was designed to meet the actual demands of qualified industrial Education for CNC-Standard with ATC. Based on a sophisticated machine tool concept at high Industrial Standard with most modern control system Siemens 802C is offered to be used for training in Din-Code and Conversational Programming as well. In addition to the CNC Machine we offer specialized PC-Based software applications for basic training and programming to be used for qualified education of students.



Mechanical Specifications	
Controller/Servo Motor	Siemens 802C /Siemens
Work table	450X200mm
Work table T slots	10X3mm
Max Work load	50kg
X travel	280mm
Y travel	200mm
Z travel	300mm
Spindle Speed	100—3000rpm
Power	1.1kw
Main Spindle Taper	ISO30 (7: 24)
Magazine Type	Horizontal Rotate
ATC	8 tools
Abors Std.	BT30
Tools Changes Time	8 secs
Min. Display Unit (X. Y. Z)	0.005mm
Feed X. Y. Z	0—500mm/min
Max. Traverse (X. Y. Z)	3000mm/min
Accuracy (X. Y. Z)	0.04mm
Resolution (X. Y. Z)	0.03mm
Nominal Dimension (LxWxH)	1650X1200X1800mm
Weight	Apprx. 900kg
Power Supply	3 Ph, 380V/50Hz/2KVA
Pressure	0.6MPa
No. of tool offset	T20, D30

Standard Accessories.

- Drill holder 1.
- Transparent cover 1.
- Tool Holders 4.
- Collets 4. ($\varnothing=6$ & $\varnothing=10$).
- Manual Vice 1.

Sets of Cutting Tools.

- End Mills – 4. ($\varnothing= 3 / 4 / 6$ & 8)
- Drills – 4 .



CNC Controller, Siemens 802C

Set-Up: Entering Tools and Tools Offsets, Creating

A New Tools, Tools Compensation Data, Entering Modifying and Determining Zero Offset and R Parameter.

Manual Operated Mode: Jog Mode, Hand wheel and MDA (Manual Data Input).

Automatic Mode: Selecting/Starting Part program, Block search, Stop / Abort Part program and Repositioning after interruption.

Part Programming: Enter/Edit New program, Programming Support, Vertical menu, Cycles, Contour and User Assignable Soft keys.

Services and Diagnosis: Data Transfer Via V24 Interface, Interface Parameters, Special Functions, Diagnosis and Start-Up.

Programming: Program Structure, Word structure and Block structure and Character Set.
Cycles: refer to Controller Features.

Pocket Calculator: To calculate the required value and can use the four basic arithmetic operations and functions Sine, Cosine, Squaring as well as square root function.

Standard Features:

- Number of Controlled Axis – 3.
- Monochrome Screen.
- Memory – 100 Kbytes.

Registerable Programs – 100

Siemens 802C Controller Features

- Absolute / Incremental Dimensions... G90 – G91
- Metric / Inch Dimension.....G71 – G70
- Plane Selection..... G17, G18, G19
- Programmable Zero..... G158, G258, G259
- Settable Zero Offset.....G54 to G57, G500, G53
- Linear Interpolation with Rapid Traverse..... G00
- Linear Interpolation with Fee..... G01
- Circular Interpolation G02-G03
- Circular Interpolation via Intermediate Point G05
- Thread Cutting with Constant Lead..... G33
- Fixed Point Approach..... G75
- Reference Point Approach..... G74
- Exact Stop / Continuous Path Control..... G09, G60, G64
- Feed F.
- Dwell Time.....G04.
- Spindle Movements.
- Spindle Speed S , Directions of Rotation.
- Spindle Speed Limitation..... G25-G26
- Tapping with Compensating Chuck.....G63
- Tool Radius Compensation G41 – G42
- Tool Radius Off. G40
- Behavior at Corners..... G450 – G451
- Cycles – Machining Cycles. (Dialog Text)
- Drilling Spot Facing.....LCYC82
- Deep Hole Drilling..... LCYC83
- Tapping with Compensating Chuck.....LCYC840
- Tapping without Compensating Chuck.....LCYC84
- Boring..... LCYC85
- Drilling a Row of Holes..... LCYC60
- Drilling a Holes of Circle..... LCYC61
- Cutting Square Pockets, Slots, and Circular Pockets..... LCYC75
- Program Branches – Unconditional / Conditional. Subroutine.
- Machine command – M code function

Off –line Simulation /Programming system (optional Items)

- MTS CNC simulation system Ver7.0
- MDI machine setup
- Off line program / cutting verification
- Machining quality control system
- Machining analysis
- 3-D simulation
- DNC transmission



Dialogte Programming

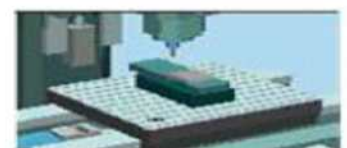
Courseware

CNC Basic – covered CNC history, Cartesian coordinate, machine structure, workshop mathematic, machining process, radius compensation, cutting force theory, rpm & feed calculation, machine components & device, tooling system, machine attachment, application and etc. CNC Exercises with Solution.



Tools Magazine

Modular chip



Magnetic table

