



HANDY CAD MARK II

Mill Application

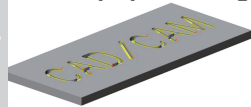
Easy-to-Use / Feature-Rich 2D/2.5D CAM System

The Milling application is a practical 2D/2.5D CAM system that allows you to create NC data efficiently with a sleek and user-friendly interface in a short amount of time.

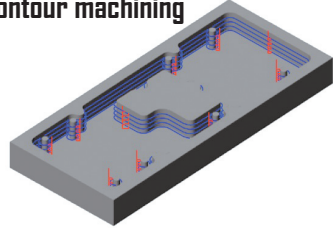
- ▶ Our system is equipped with a variety of machining definition commands, allowing for customizable cutting conditions for each command. Save and select (tool conditions/processing Z/approach, etc.) to reduce input operations.
- ▶ Detecting leftover areas and performing finishing operations, machining paths are defined to achieve optimal results.
- ▶ The process tree allows for easy verification of processes and definitions, while simulation via the tool path is also possible.
- ▶ By modifying cutting conditions and redefining paths, changes to the shape can be achieved.
- ▶ Additionally, operations such as machining definition, path modification/deletion are supported by UNDO (undo) and REDO (redo) functions.
- ▶ When generating NC data, you can select the machine type to create the desired data.

2D machining

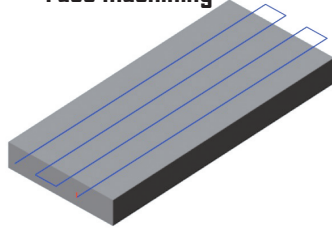
Shape processing



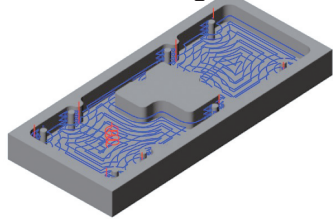
Contour machining



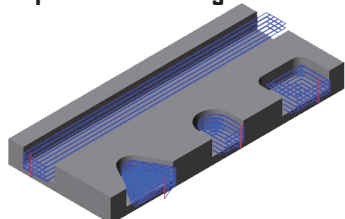
Face machining



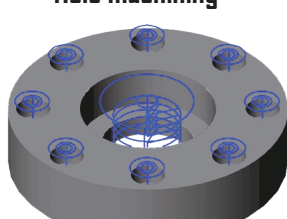
Pocket machining



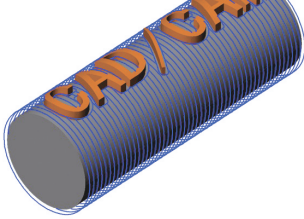
Open pocket machining



Hole machining

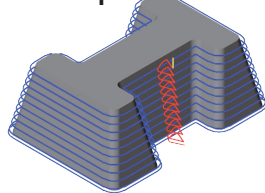


A-axis/B-axis rotation

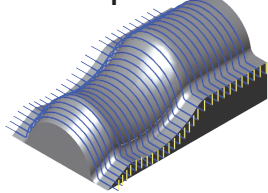


2.5D machining

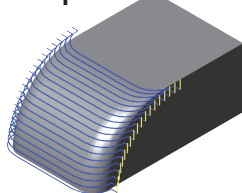
Taper



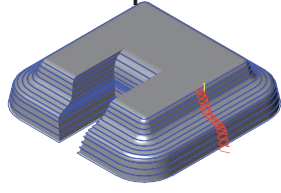
Slope Z



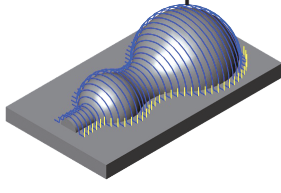
Slope XY



Contour XY plane



Contour XZ/YZ planes



Basic specifications

- 2D machining definition Shape, indentation, pocket, open pocket, hole, face, contour, taper, slope XY, slope Z
- 2.5D machining definition Shape, indentation, pocket, open pocket, hole, face, contour, taper, slope XY, slope Z
- Auxiliary functions Parallel, grid, rotation, symmetrical arrangement possible (with deletion function)
- Placement ※ When arranged symmetrically, the upcut/downcut will change.
- Post-related G-code, coordinate modal management, and various significant digit specifications are possible. Program expression (absolute/incremental values) can be specified for main/sub programs. Arc commands: I/J specification, R specification, and automatic I/J/R switching. Helical arc approximation function. Corner deceleration settings. Work coordinate settings are possible.

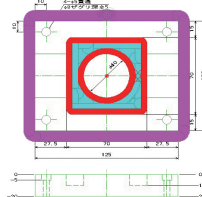
Easy verification and modification of processes/definitions.

The process tree displays the defined processes and definitions, where operations such as verification and modification can be performed.



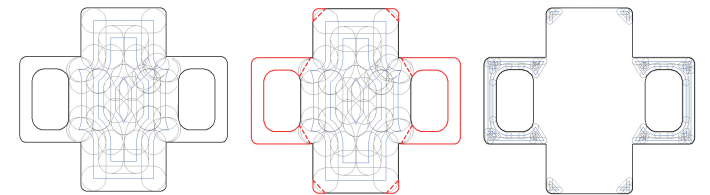
Processes and definitions can be reordered or deleted. Creating new processes is done, and definitions are added to the active process. Properties allow changes to process and definition settings. Redefining allows changes to the shape and settings of processes and definitions. Enabling or disabling checks for definitions and processes allows specifying screen display and NC generation.

The screen displays in color for each type of machining, with the selected definition (or entire process) highlighted (in red). The right image shows the state when fill mode display is enabled.



Detect remaining uncut areas and define machining for the necessary range.

Remaining uncut areas can be recognized in pocket machining and contour machining. These areas can then be defined for further pocket machining or contour machining using small-diameter tools.



Roughing with large-diameter tools Detecting areas with leftover Roughing with large-diameter tools

※ The above image shows the state when in contour line mode.

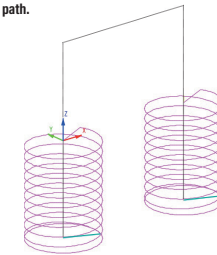
Significantly reducing leftover material with optimized offset calculations.

When the step over exceeds the tool radius, triangular leftover material appears at the edge. However, by using optimized offset calculations, the leftover areas can be automatically removed, leading to significant reduction in machining time.

Hole machining.

Machining is performed in the order defined in the hole cycle. By selecting the hole type in the cycle and utilizing features such as batch recognition of circles and optimization functions, it is easy to create an optimal machining path.

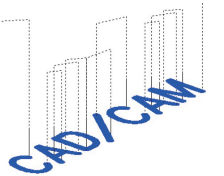
Hole position specification	Batch input through individual or group recognition. Recognition conditions can be set based on representative elements or diameter. Return to B point and return to initial point can be specified.
Optimization	Optimization of travel distance is possible with the sorting function. Tool change frequency can be optimized during NC generation.
Command	Fixed cycle, G01 drill, circular pocket, circular contour, circular helical, macro modal call.



Shape machining.

Without switching input modes, you can freely create NC data by specifying the elements or waypoints you want to machine.

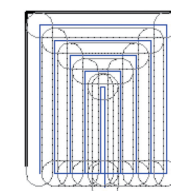
Path	Cutting movement can be selected from three types: linear, arc, and rapid movement. UP movement (rapid traverses) can be selected from two levels: escape Z and initial Z. Batch recognition automatically determines the path based on the recognized elements. Text elements automatically generate vector data.
Diameter Compensation	Diameter compensation can be specified.
Approach	It is possible to simply add an arc radius from the current position.



Pocket machining.

Pocket machining and island cutting are supported, and with offset calculations, efficient paths can be generated by separating/combining paths for complex shapes.

Diameter compensation	It is not possible to specify.
Approach	Fixed cycle, helical, zigzag, and 2-up are selectable. Entry coordinates can be specified.
Direction	Area: Inside/Outside, Cutting: Down-cut/Up-cut can be selected respectively.
Remaining cut	Both detection and definition are possible.

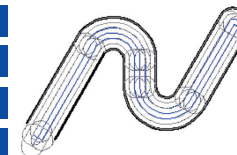


It is also possible to specify an open loop, enabling machining definitions for shapes that were previously undefined.

Open pocket machining.

For specific open shapes, a dedicated path is sought, different from pocket machining, to perform optimal processing.

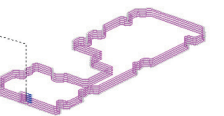
Diameter compensation	It is not possible to specify.
Approach	Entry coordinates can be specified.
Direction	Cutting: Down-cut/Up-cut can be selected.
Remaining cut	Detection/definition are both possible.



Contour machining.

A path is created along the contour shape. Interference checking between the path and contour shape is performed to prevent cutting interference.

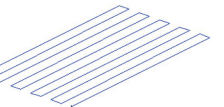
Diameter compensation	Diameter compensation can be specified.
Approach	Specifications such as approach, arc, diameter compensation distance, etc., are possible. Shape and size can be saved/selected. Approach position can be specified (only for closed loops). Entry coordinates can be specified.
Direction	Area: Inside/Outside, Cutting: Down-cut/Up-cut can be selected respectively.
Remaining cut	Detection/definition are both possible.



Face machining.

A dedicated command for surface milling has been prepared, allowing for the generation of the most optimal path for the work size.

Approach	Entry coordinates can be specified.
Direction	Cutting: Down-cut/Up-cut/Bidirectional can be selected. Path angle: Horizontal/Vertical/Automatic can be selected.

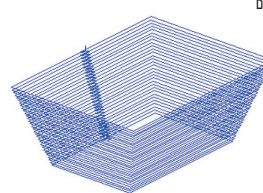


2.5D Processing

Equipped with commands corresponding to the machining surface, enabling easy definition of machining. Ball end mills, flat end mills, and radius end mills can be used (except for ball end mills with a slope of 2D).

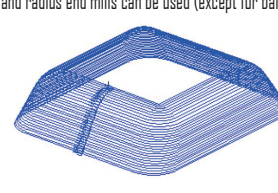
Taper

By specifying the top shape and bottom shape, the corresponding elements are automatically detected and machining is defined.



Contour lines

"Specify basic shapes from the XY plane, and slope shapes from the XZ plane. Finishing and roughing can be selected, with roughing options including spot drilling, helical, and other approaches available."



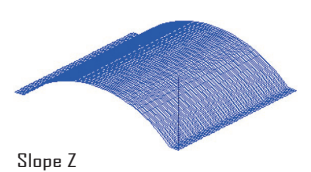
Slope XY

From the basic shape on the Y plane, processing can be defined for a single slope.



Slope Z

Processing can be defined along a slope on the YZ or ZX plane from the basic shape.



- Inspection functions Feed and rotation range setting and inspection functions. Setting and inspection of each tool's effective length. Duplicate inspection function for diameter correction values. Converting maximum arc radius exceedance and shortest arc length under minimal data to approximate data.
- Mouse recognition Simultaneous input of elements/coordinates with navigation, dragging, and rubber-band input assistance.
- Search loop recognition Automatic recognition of continuous elements, perimeter recognition with a single click, multiple inner perimeters recognition, and multiple outer perimeters recognition via diagonal point specification are possible. Recognition conditions can also be set using mask settings. There are no restrictions on the number of elements or loops for recognition and calculations (dependent on computer environment).
- NC Generation Assistance Work instructions can be created during NC generation. Optimization of tool exchange frequency during hole machining.