



To creates
[processing]

HANDY CAD MARK II

Lathe Application

Lathe Application

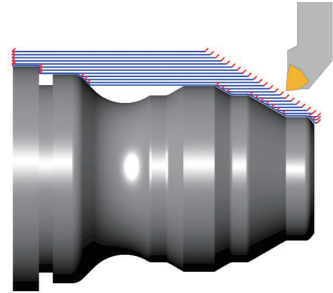
The Ultimate Solution for 2-Axis Lathes: Simple and High-Performance Lathe CAM System

Lathe applications have a simple command structure, achieving easy operation that automatically determines the path and processing direction based on the tool used.

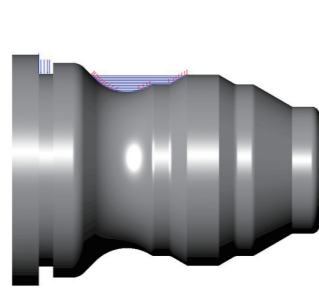
- ▶ With the process tree, it is easy to verify the process and definitions, and simulation using tool paths is possible
- ▶ You can check the interference conditions of chips and bytes during definition or toolpath.
- ▶ By entering the parameters, you can register chips and holders and combine them to create a bite.
- ▶ Each machining definition determines the type of operation—external diameter, internal diameter, or end-face machining—based on the tool used and establishes the path direction.
- ▶ The sentence "During NC generation, you can choose the machine type and generate the required data.

Machining Type

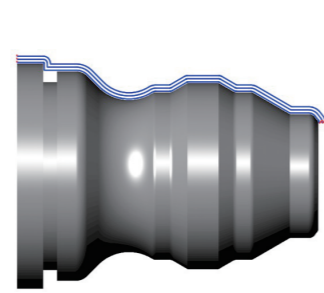
Contour (Outer Diameter) Machining



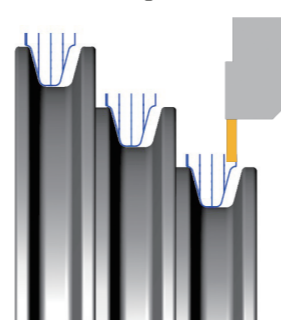
Contour (Stack Remaining) Machining



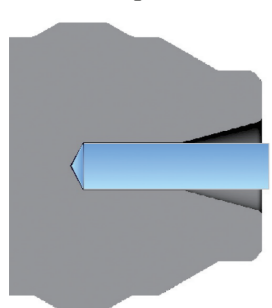
Contour (copying) Machining



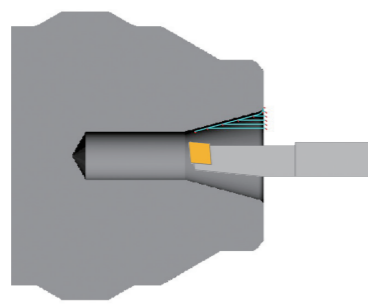
Groove Machining



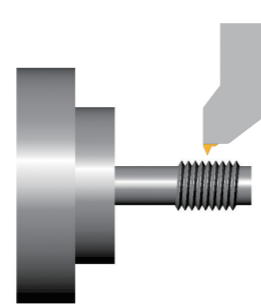
Hole Machining



Contour (Inner Diameter) Machining



Thread Cutting Machining

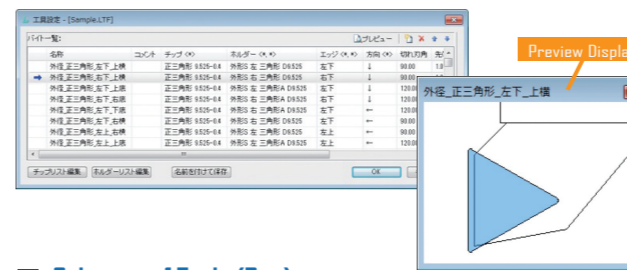


Through-Cutting



Tool (byte) Registration

Registration can be done by just entering the chip and holder parameters. You can combine them and register them as a byte



Selection of Tools (Bits)

In each machining definition command, tool selection, machining conditions, and various settings are made from the command properties. The filter function allows you to narrow down the tool selection based on conditions such as chip shape and edge direction. Additionally, tools can be selected from the thumbnail display.

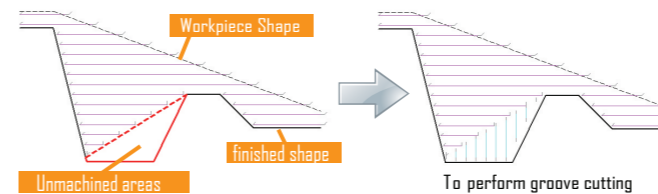


Command Properties

Tool Thumbnail List

Specification of Workpiece (Material) Shape and Detection of Unmachined Areas

In contour and rough groove machining, you can define the shape of the workpiece, allowing you to restrict machining to only the required areas. Additionally, it can identify areas of leftover material caused by the tool's cutting angle, making it possible to machine those areas using a different tool.

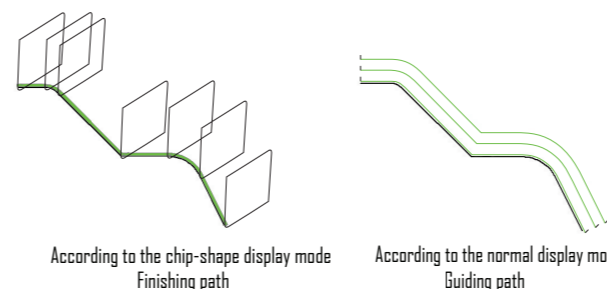


Leftover areas of contour milling (workpiece shape specification)

Contour Processing

Create a path parallel to the shape. Can define standard finishing paths, pitch, and the number of passes for contouring operations.

Workpiece Shape	No Specification	Tool Cutting Depth	Select from None, Yes, or Ignore
Tool Radius Compensation	compensation specification is possible	Cutting Residue	Both Detection and Definition are Possible



According to the chip-shape display mode Finishing path

According to the normal display mode Guiding path

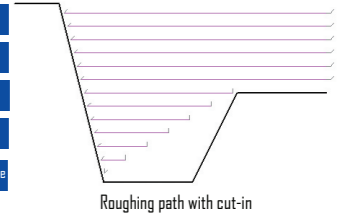
General Specification

- Inspection Function Exceeds Maximum Arc Radius. Convert data with an arc length shorter than the minimum into approximate data.
- Mouse Recognition
- Search Loop Recognition

Contour Milling

It defines the roughing paths with contour machining. You can also set the finishing allowance and simultaneously create the finishing paths using the same tool.

Workpiece Geometry	Unspecified
Correction	Radius correction specification is possible
Mimetic Action	Possible in Rough Cutting
Tool Feed	Choose from None, Yes, Ignore
Cutting Residue	Both Detection and Definition are Possible

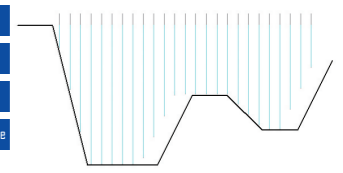


Roughing path with cut-in

Groove Machining

The machining is defined using cutting tools such as groove tips and round tips. The roughing and finishing paths can be defined, and in roughing, the contouring operation can be specified.

Workpiece Shape	Specified
Radius Compensation	Not Specified
Contouring Operation	Possible In Roughing
Stock Remaining	Detection and Definition Are Both Feasible

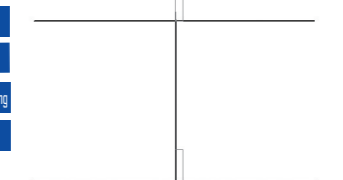


Roughing path without contouring operation

Piercing Cutting

The piercing cut path can be defined by selecting two points.

Workpiece Shape	No specification
Radius Compensation	No Specification
Cutting	Rotational speed can be specified during cutting
Unfinished Cutting	None

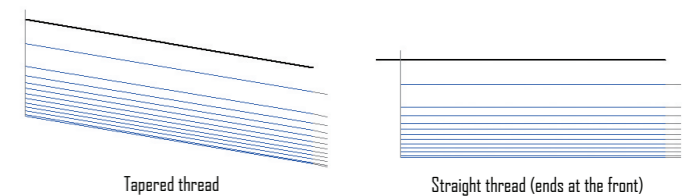


Standard piercing cutting path

Threading

The straight and tapered thread-cutting processes can be configured. A pitch list is provided, with up to 30 cutting cycles allowed, and an automatic calculation feature is included.

Beveled Edge	Specifiable
Cutting Method	Select from Right Angle, Single edge, or Staggered



Tapered thread

Straight thread (ends at the front)

Hole Processing

Select and define from cycle patterns for center drill, drill, and chamfer.

Command	Fixed Cycle, G01 Drill
---------	------------------------



Key Specifications

- Processing Definition shape, contour, contour line, groove, cutter or cutting edge, threading, hole, and corner.
- Processing Conditions Material can be specified as desired / Maximum number of threading cuts: 30. There is no limit to the number of registered tips, holders, and bites.
- Tool Editing definitions through properties and redefinition is possible.
- Auxiliary functions Function for detecting leftover areas to be trimmed.
- Post-related supports G-code, modal management of coordinates, and the specification of various significant digits. Program expressions (absolute values and incremental values) can be specified for the main program. Arc command: IK specification, R specification, IK/R Auto-Switching. No work coordinate set.

- Simultaneous input of elements/coordinates. Input assistance through navigation, dragging, and rubber banding. The features include automatic recognition of continuous elements, one-click detection of outer contours, recognition of both outer contours and multiple inner contours, as well as the ability to identify multiple outer contours by specifying diagonal points. It is also possible to set recognition conditions using mask settings. There are no limits on the number of elements or loops for recognition and calculations (dependent on the computer environment).