*cam-tool*V3

Text Book for Intensive Course - CAM -

T31-09-02

Before you read this book…,

In this text book, minimum contents required for operating the application are described, in order to emphasize the workflow of user's practical operation.

About this book

■ Version information

cam-tool V3 Version 3.1.4.1
cam-tool MX Version 1.1.4.1

■ Files used in this book

Use the files in the CD-ROM supplied with this book.
Refer to [File name], [Layer number] written in each chapter.

■ About spot information

If  mark is found in this text book, [spot information] is described.

■ About restrictions/precautions

If  mark is found in this text book, [restrictions/precautions] are described.

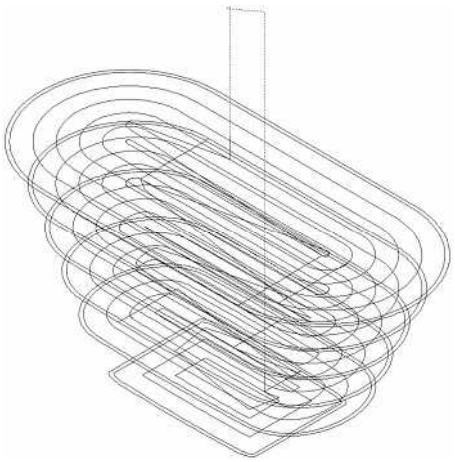
Chapter 1

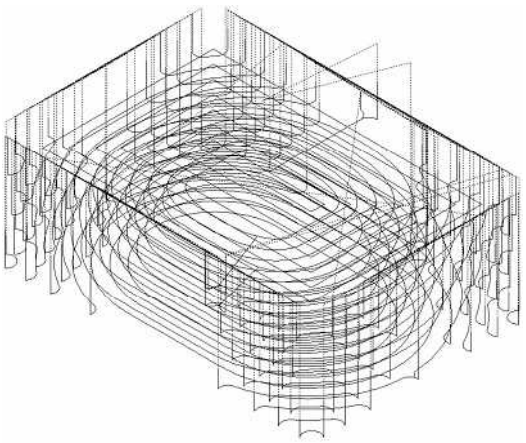
Cutting strategies in *cam-tool* V3

■ Rough cutting

○ Z-level Rough Cutting

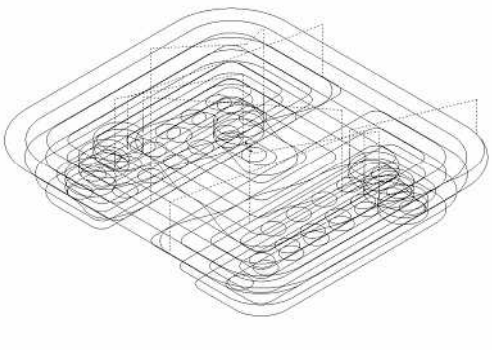
This is the function that tool travels around in Z direction with the fixed tool center (Z-level traveling) to perform rough cutting.

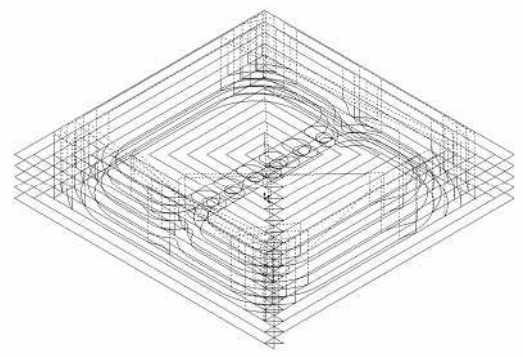




○ Z-level High Efficiency Rough cutting

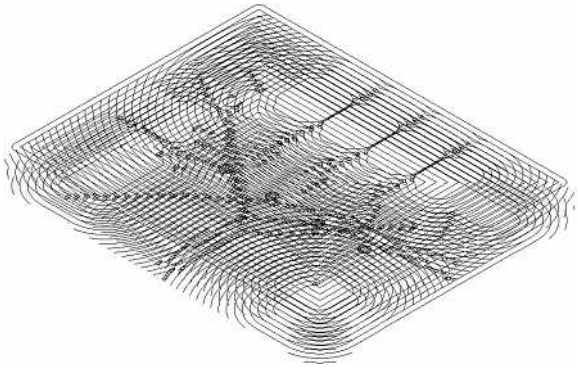
Tool travels around in Z direction with the fixed tool center, and tool loads can be reduced by performing efficient rough cutting.

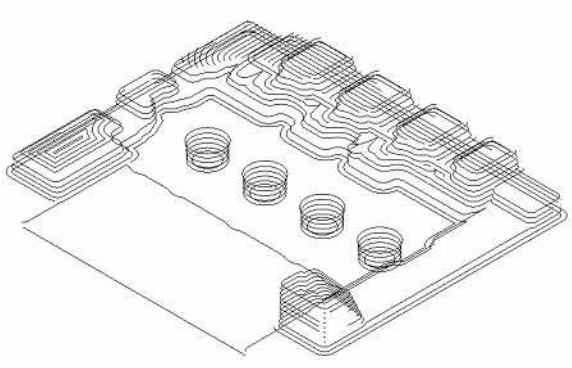




○ Z-level Rough cutting with Multiple Tools

Z-level offset cutting can be performed in various processes including first rough cutting to semi-rough cutting, which stock is left from the previous process. Air-cut can be reduced by registering multiple tools, in first rough cutting to semi-rough cutting.

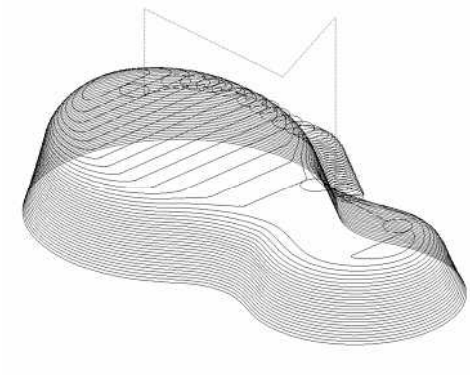


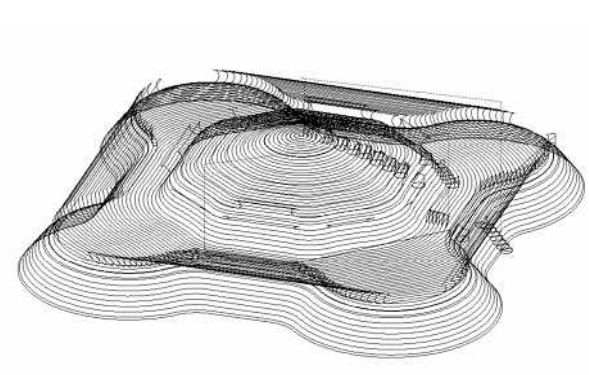


■ Finishing

○ Z-level Finishing

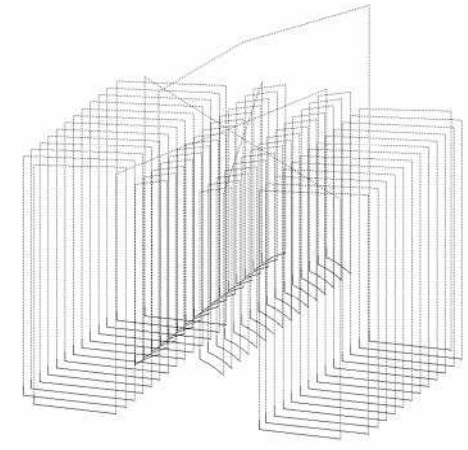
This is the cutting strategy that tool travels around in the Z direction with the fixed tool center Z value.

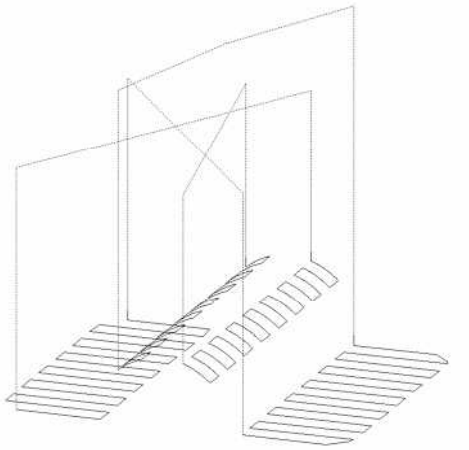




○ Scanning-line Area

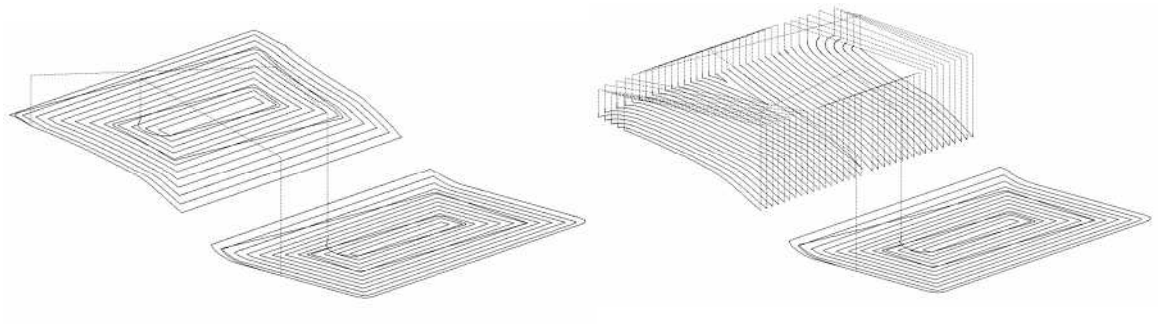
This cutting strategy performs Finishing on near-horizontal area of the surface in one direction.





○ Flat Part Circumference

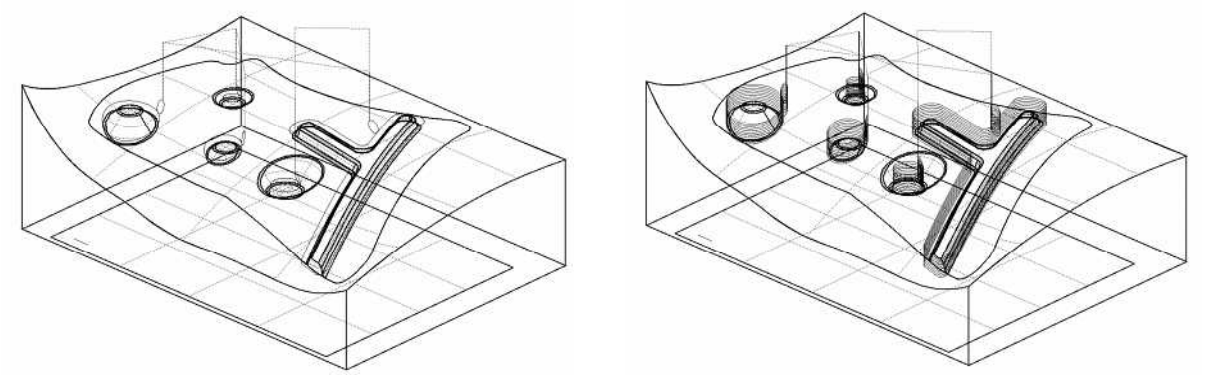
Tool travels around on near-horizontal area of the surface to perform finishing.



■ Area cutting

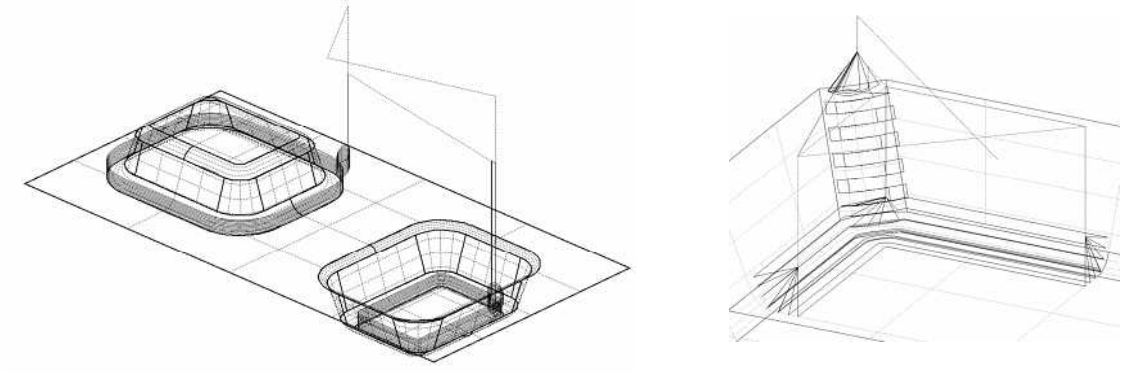
○ Pencil cutting

Performs cutting at edge between surfaces.



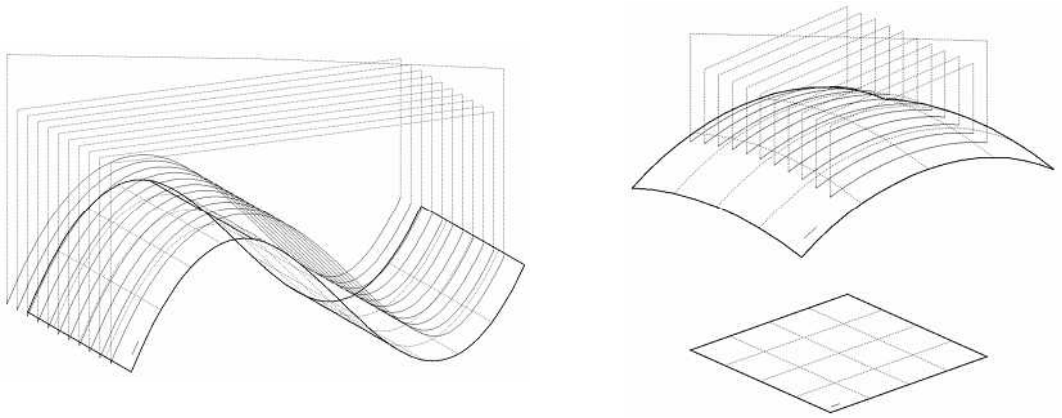
○ Re-machining

It is performed on area, which stock from the previous process is automatically detected by the system.



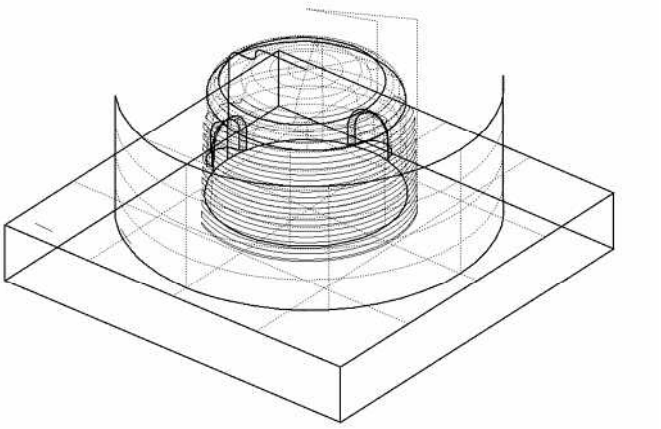
○ Surface Finishing

Tool travels along the surface with the fixed pitch to perform finishing.



○ Aiming Check

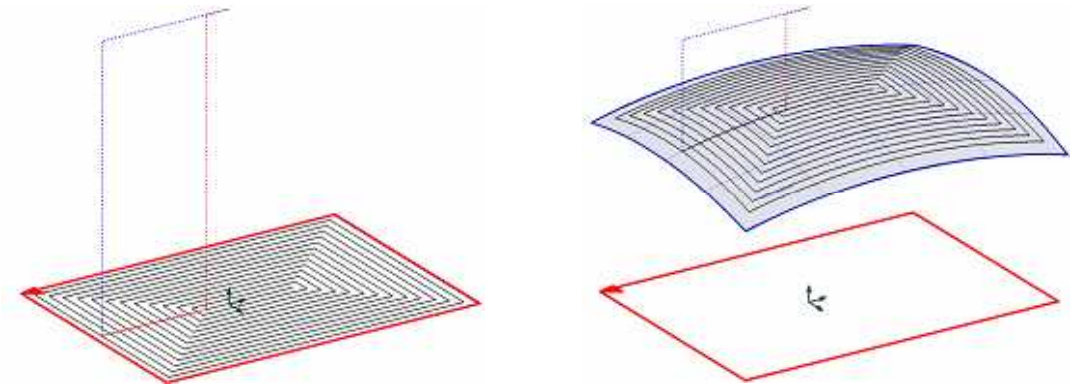
Project cutter paths in the normal line direction of path surface to perform finishing.



■ Other cutting

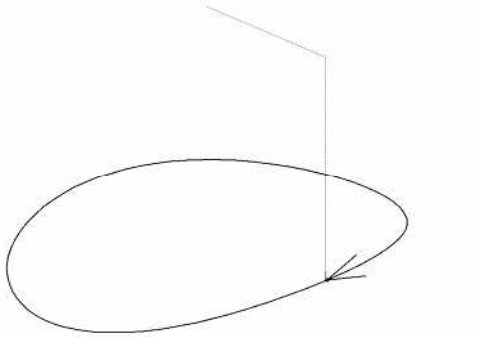
○ CL+ Check

Project the created CL to surface to output new CL data.



○ Curve Cutting

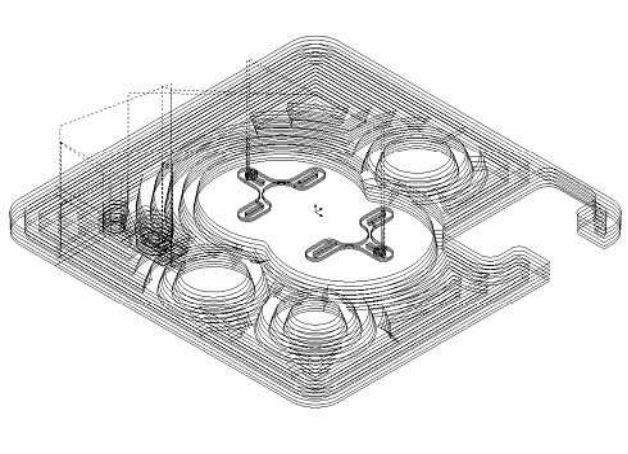
Create compcurve (contour) instead of surface to perform 3-axis cutting.



■ 2.5D Rough Cutting

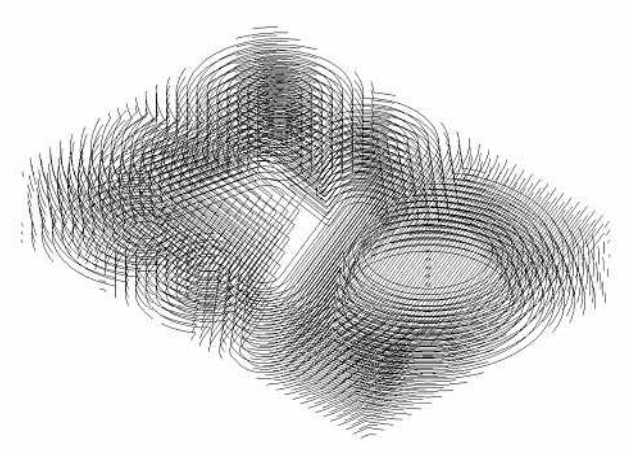
○ Round of Cavity

Tool travels around in Z direction with the fixed tool center (Z-level traveling) to perform rough cutting of 2.5-axis on cavity shape.



○ Round of Core

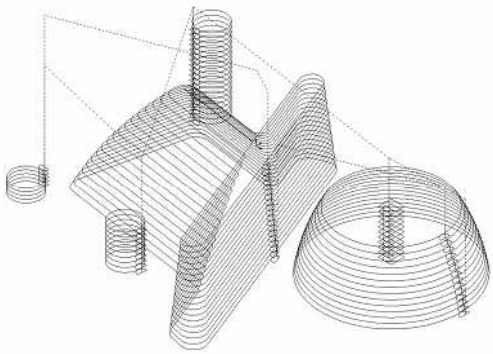
Tool travels around in Z direction with the fixed tool center (Z-level traveling) to perform rough cutting of 2.5-axis on core shape.

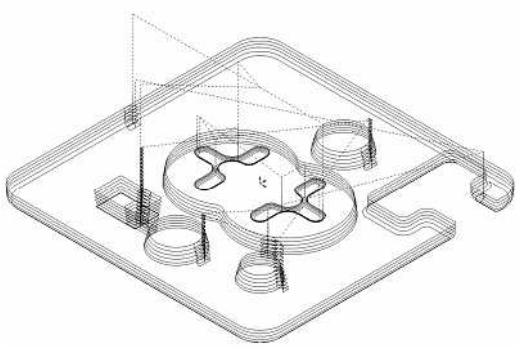


■ 2.5D Finishing

○ Contour Cutting

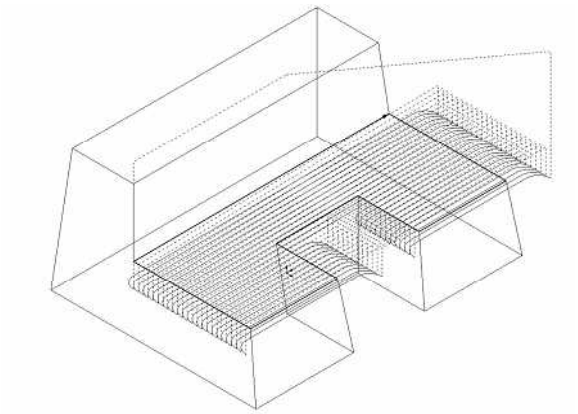
Performs finishing of 2.5-axis along contour.

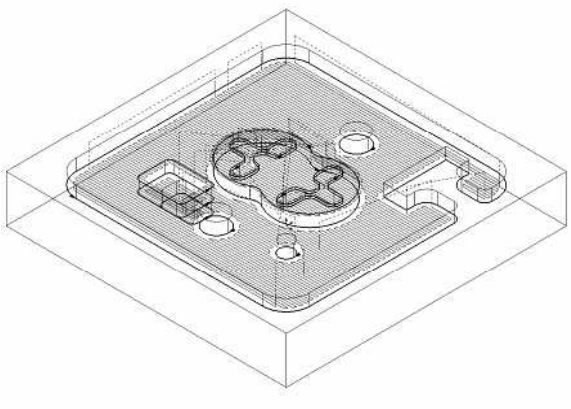




○ Flat Cutting

Tool travels around in Z direction with the fixed tool center (Z-level traveling) to perform Finishing of 2.5-axis on top/bottom surface.





Chapter 2

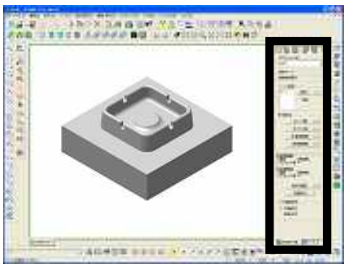
Operating flow of CAM processing

Operating flow of CAM processing (Flow chart)

(1) Create [Cutting shape] in a [model file].

When you save a model file, create a folder first, and save the model file in the folder.

(2) Create [Profile(cutting process)], and set [Calculation conditions] / [Cutting shape].



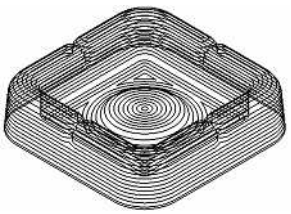
Ex.)
Process1：Rough cutting (Tool D10 R5)
Process2：Semi Finishing (Tool D 8 R4)
Process3：Finishing (Tool D 6 R3)

(3) Create [Calculation process list (File for CAM calculation)].

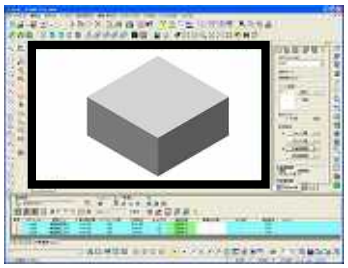
Create a calculation process list file in the folder that the model file is in.

(4) Register the profile (2) to the [Calculation process list(3)].

(5) Create [CL data] by executing CAM calculation.

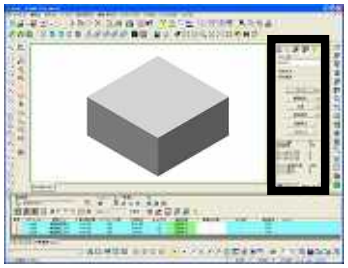


(6) Create [Material shape] to the [Model file].



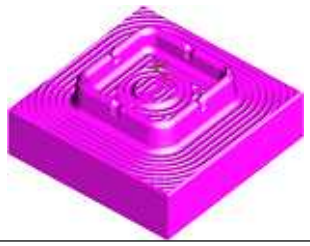
When you do not execute [Simulation/ Optimization], omit the operations of (6)-(9) .

(7) Create [Work (Material)].



(8) Register [Work (7)] to the [Calculation process list (3)].

(9) Execute [Simulation/ Optimization] by executing CAM calculation.



(10) Create [NC data] by executing CAM calculation.

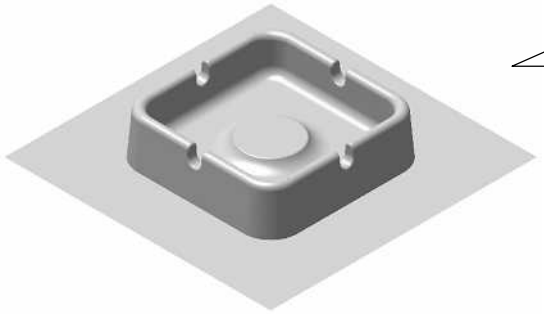
```
%
G00G40G49G80
G91G28Z0.
G28X0.Y0.
T01
M06
S1000M03
G91
G17G00X-80.Y75.
G18Z-95.
M08
G01Z-5.F80
G17X5.F110
```

<<Chapter2 Operating flow of CAM processing>>

What are needed for model shape?

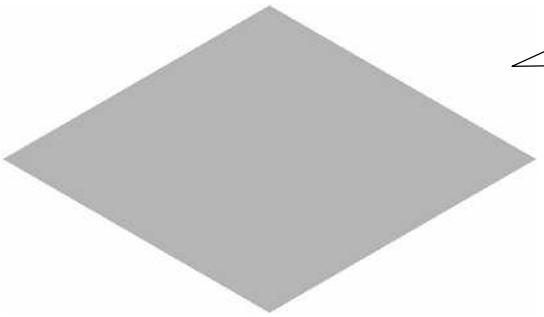
Open the model file of sample shape, and confirm.
CD-ROM/Intensive Course/CAMmodel1.gmd

(1) Cutting shape (Check surface)



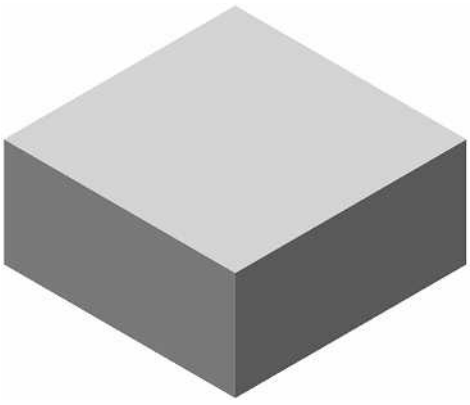
Confirm the shape in layer1.

(2) Surface to define calculation area in plane direction(Calculation area surface)



Confirm the shape in layer1.

(3) Material shape (Work) * Only for executing cutting simulation



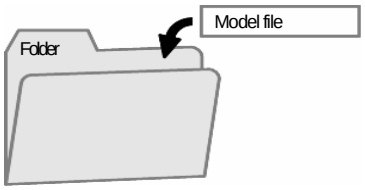
Confirm the shape in layer10.

[Model shape to be prepared] or [Precautions for creating shape] varies depending on each cutting strategy. Confirm them in the description of each cutting strategy.

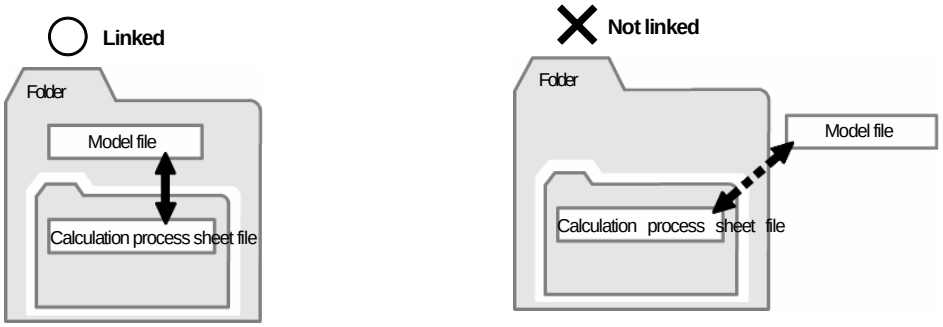
What is the precautions for saving a model file?

- Recommended file configuration -

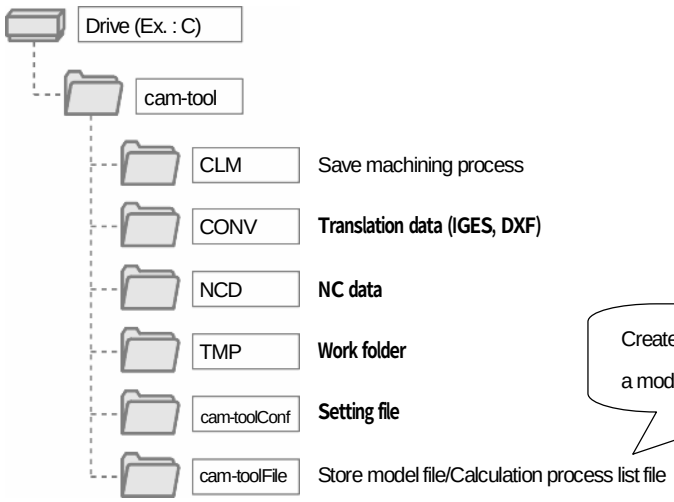
When you save a new [model file], create a [folder] first, and save the model file in the folder.



CAM calculation is executed with the [Model file] that is linked to [Calculation process list file].
[Calculation process list file] is linked to [Model file] relatively. Therefore, if link between [Model file] and [Calculation process list file] is destroyed by operations such as moving a file, CAM calculation cannot be executed.
It is recommended to save those two files in the same folder. When you work with any file, operate on the [folder] basis.



File configuration of *cam-tool* V3, *cam-tool* MX - Recommended file configuration -

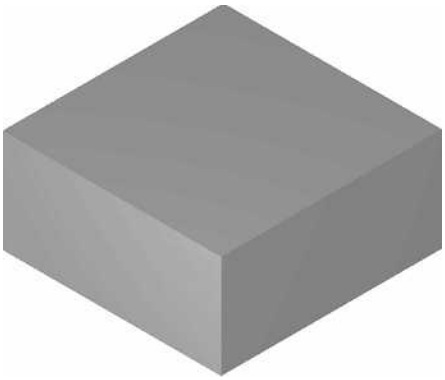


<<Exercise>> Save the model file with the name [CAM model1.gmd].
<Destination folder > D:/ cam-tool / cam-toolFile / Exercise1 / CAMmodel1.gmd

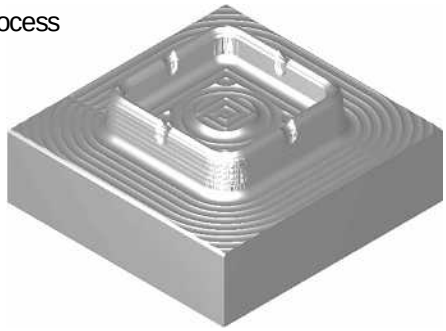
What is a profile (cutting process)?

A profile is a process for cutting material. Create profile in a model file, and set [Calculation conditions] / [Cutting shape]. Let's practice cutting the sample shape in the three processes such as [Rough cutting process] / [Semi Finishing process] / [Finishing process] !

• Material (Work)



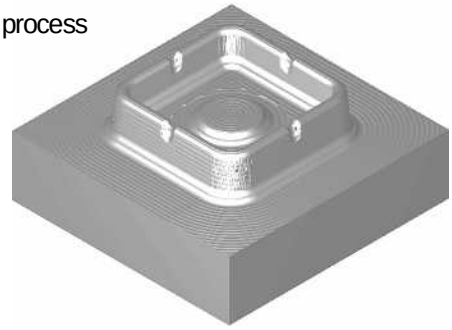
•Rough cutting process



<Calculation condition>

tool : D10 R5
Z step down : 3 mm
XY step over : 5 mm
Stock : 1 mm

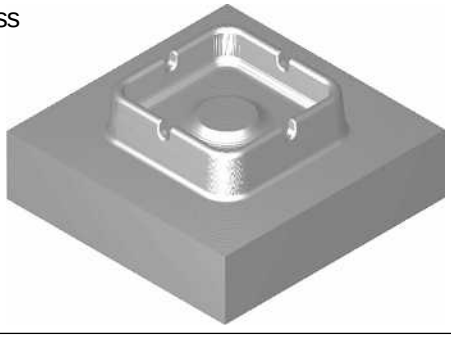
• Semi Finishing process



<Calculation condition>

tool : D8 R4
Z step down : 2 mm
XY step over : 2 mm
Stock : 0.5 mm

• Finishing process

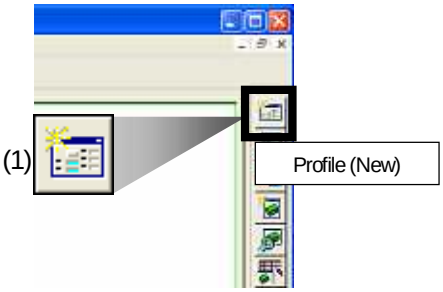


<Calculation condition>

tool : D6 R3
Z step down : 1 mm
XY step over : 1 mm
Stock : 0 mm

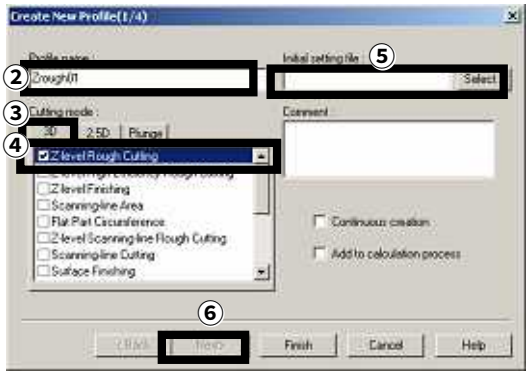
How to create a profile

Create a new [profile] and set [Calculation conditions].
[Profile (New)] is consisted of four pages (1/4) – (4/4).



Click the [Profile(New)] button.

[Profile (New)] (1/4)



(2) Input [Profile name].
* Input name within twenty three alphabetic /numerical letters (alphabetic for first letter)

(3) Select a category for [Cutting strategy].

(4) Select a [Cutting strategy].

(5) Select [Initial setting file].

(6) Click the [Next] button.

• What is [Initial setting file]?

Save settings of profile and use it for other process as a template file.
[Initial setting file] is not described in this text.

• Add to calculation process : ON

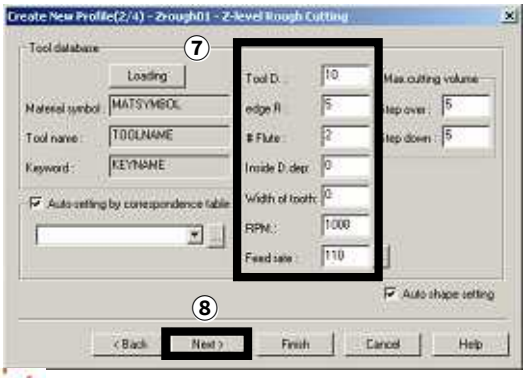
When it is set ON, if you click [Finish] button, the [Profile] is added(registered) to [Calculation process list].

* If you click [Finish] button with [Add to calculation process] set ON, before you create calculation process
list, the message of [Unable to add the calculation process.] is displayed. However, [Profile] is created in the model file. After you create a [Calculation process list], add the profile to the [Calculation process list].

• Continuous creation : ON

When you click [Finish] button, dialog [Profile(New)(1/4)] is opened, so that you can create another profile continuously.

• [Profile (New)](2/4)



(7) Input [Tool parameter].

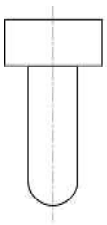
* Specify tool for this cutting process.

(8) Click [Next] button.

Tool type

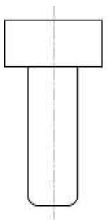
Ball end mill

Cutting edge R=Tool diameter1/2



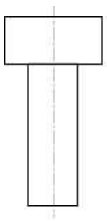
Radius end mill

Cutting edge R < Tool diameter1/2

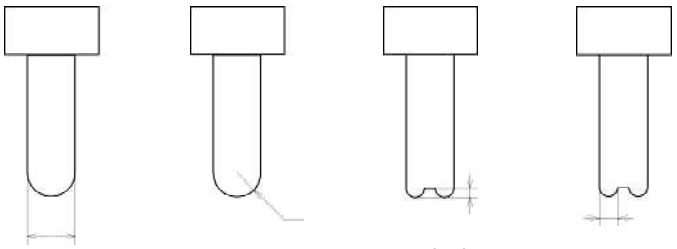


Flat end mill

Cutting edge R=0

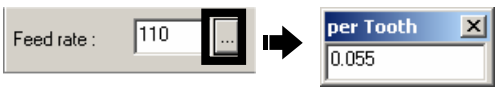


Tool parameter

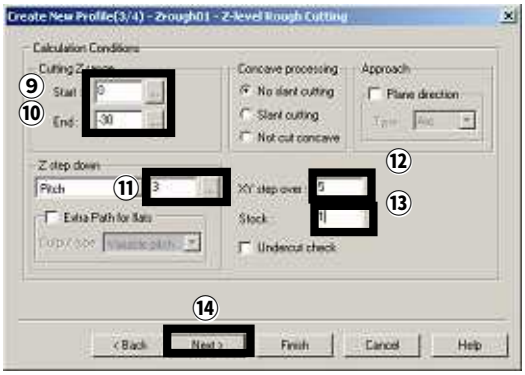


• [Inner depth] and [Edge width] is the parameter for throw-away tool.
If you use tool other than throw-away tool, input [0].

Click the button on the right of input area of [Feed rate] to check feed rate for [per Tooth].
Or, input feed rate for [per Tooth] to calculate [Feed rate].



• [Profile (New)](3/4)



(9) Input [Cutting range: Start].

(10) Input [Cutting range: End].

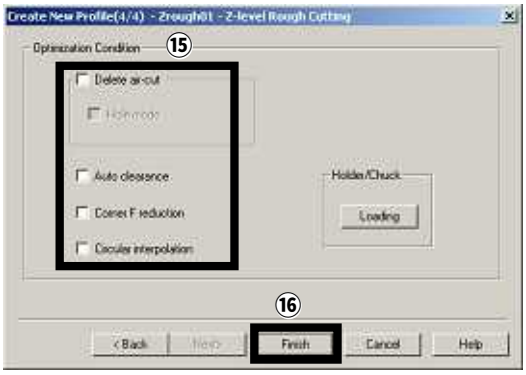
(11) Input [Z step down].

(12) Input [XY step over].

(13) Input [Stock].

(14) Click the [Next] button.

• [Profile (New)](4/4)



(15) Set [Optimization condition]. (All OFF this time)

(16) Click [Finish] button.

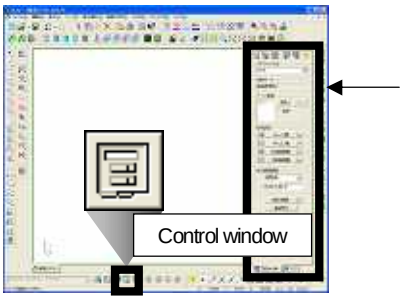
* When [Finish] button is clicked, [Control window] is displayed.

What is Optimization?

When cutting simulation is executed, the [Calculated CL data] is recalculated to create [Optimized CL data for cutting]. When all of functions for [Optimization] are set OFF, only cutting simulation is executed. [Optimization] functions are not described in this text book.

Control window

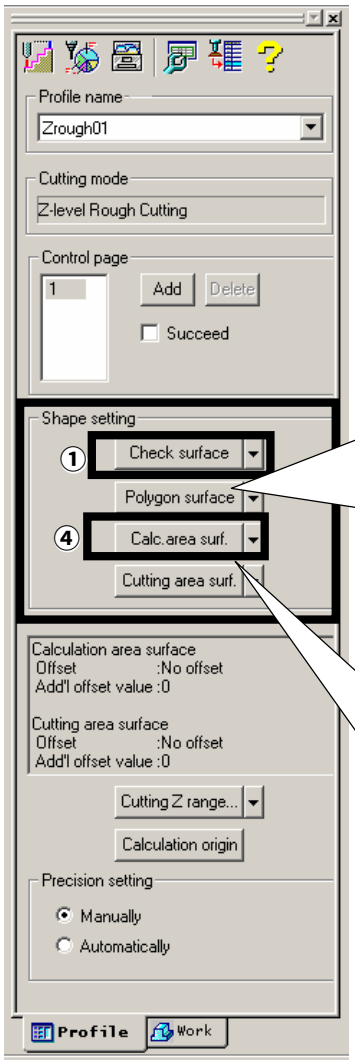
Set or edit [profile] or [Work] that is created in a model file on this widow.
Click [Control window] button to set [ON/OFF] of [showing/hiding] the window.



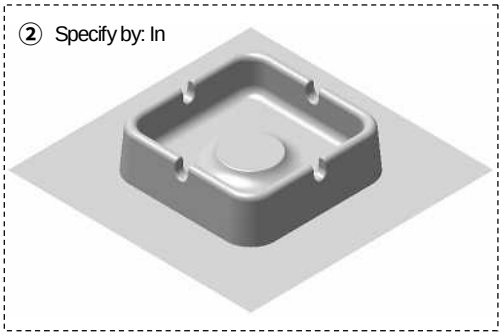
How to set cutting shape

Set [Shape setting] in a profile on the Control window.
Set cutting shape as [Check surface], and set surface for defining calculation area in plane direction as [Calculation area surface].

<Control window>

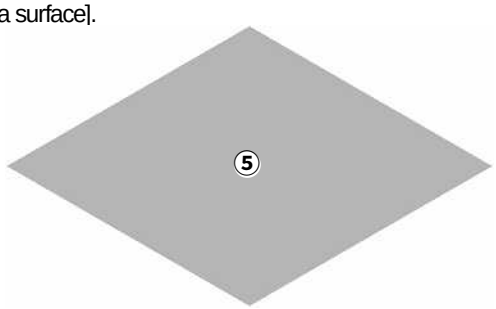


① Click [Check surface] button.
② Specify surface to be cut.



③ Press the center mouse button to determine the specifying.

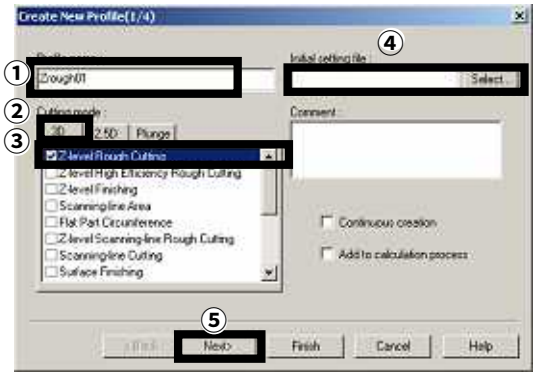
④ Click [Calculation area surface] button.
⑤ Specify a surface for defining an area that cutter paths are output.
* Only a single surface can be specified as [Calculation area surface].



<<Exercise>>

Create profiles of three processes, such as [Rough Cutting], [Semi Finishing], [Finishing] for the sample shape.
First of all, create a profile of [Rough Cutting] and set [Calculation conditions].

• Profile(New)(1/4)



(1) Profile name： Zrough01

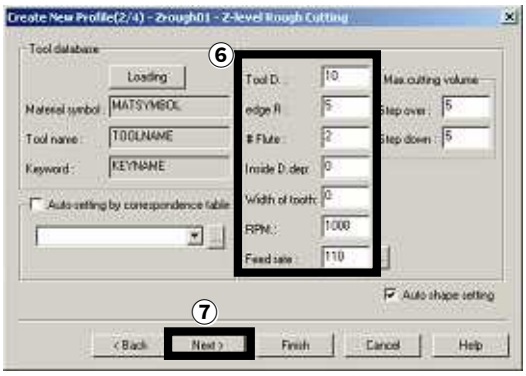
(2) Cutting mode： 3D

(3) Cutting strategy：Z-level Rough cutting

(4) Initial setting： (Blank)

(5) Click the [Next] button.

• Profile(New)(2/4))

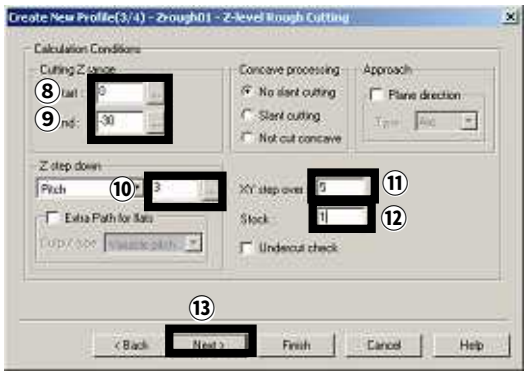


(6) Tool parameter (Ball end mill D10)

- Tool diameter：10
- Edge R：5

(7) Click the [Next] button.

• Profile(New)(3/4)



(8) Cutting Z range/Start：0

(9) Cutting Z range/End：-30

(10) Z step down：3(mm)

(11) Z step over：5(mm)

(12) Stock：1 (mm)

(13) Click the [Next] button.

• Profile(New)(4/4)

(14) Optimization Condition：(All OFF)

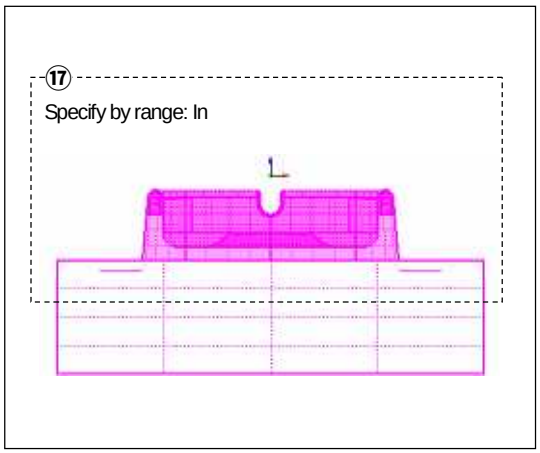
(15) Click the [Finish] button.

<<Chapter2 Operating flow of CAM processing>>

≪Exercise≫

Set [Shape setting] for the created profile.

• Shape setting [Check surface]

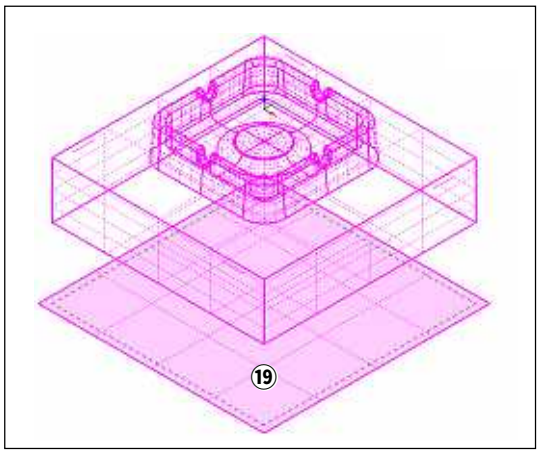


(16) Click the [Check surface] button.

(17) Specify a surface to be cut. (Specify by range: In)

(18) Press the center mouse button to determine the specifying.

• Shape setting[Calculation area surface]



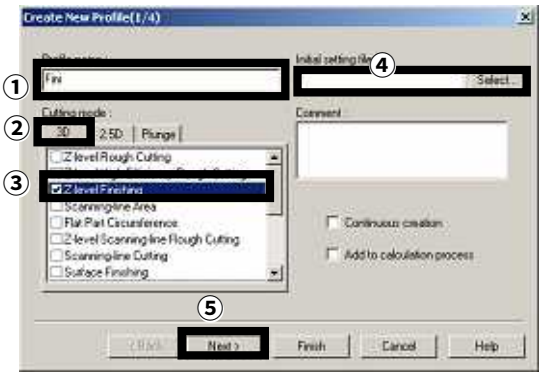
(19) Click the [Calculation area surface] button.

(20) Specify a surface that cutter paths are output.
* Only one surface can be specified as [Calculation area surface].

≪Exercise≫

Next, create a profile of [Semi Finishing process] and set [Calculation condition] and [Shape setting].

• Profile(New)(1/4)



(1) Profile name : Fini01

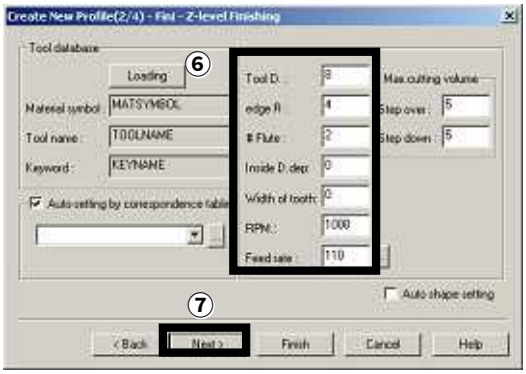
(2) Cutting mode : 3D

(3) Cutting strategy : Z-level Rough cutting

(4) Initial setting : (Blank))

(5) Click the [Next] button.

• Profile(New)(2/4))

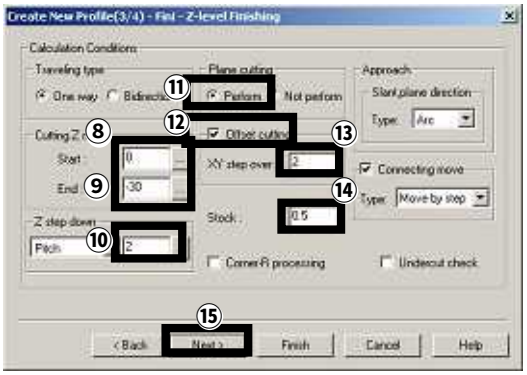


(6) Tool parameter (Ball end mill D8)

- Tool diameter : 8
- Edge R : 4

(7) Click the [Next] button.

• Profile(New)(3/4)



(8) Cutting Z range/Start : 0

(9) Cutting Z range/End : -30

(10) Z step down : 2(mm)

(11) Plane cutting: Perform

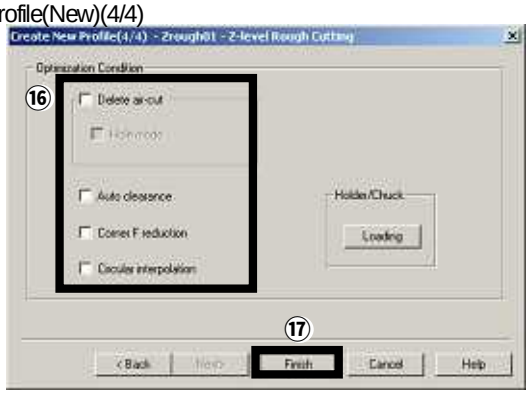
(12) Offset cutting : ON

(13) XY step over : 2 (mm)

(11) Stock : 0.5 (mm)

(15) Click the [Next] button.

• Profile(New)(4/4)



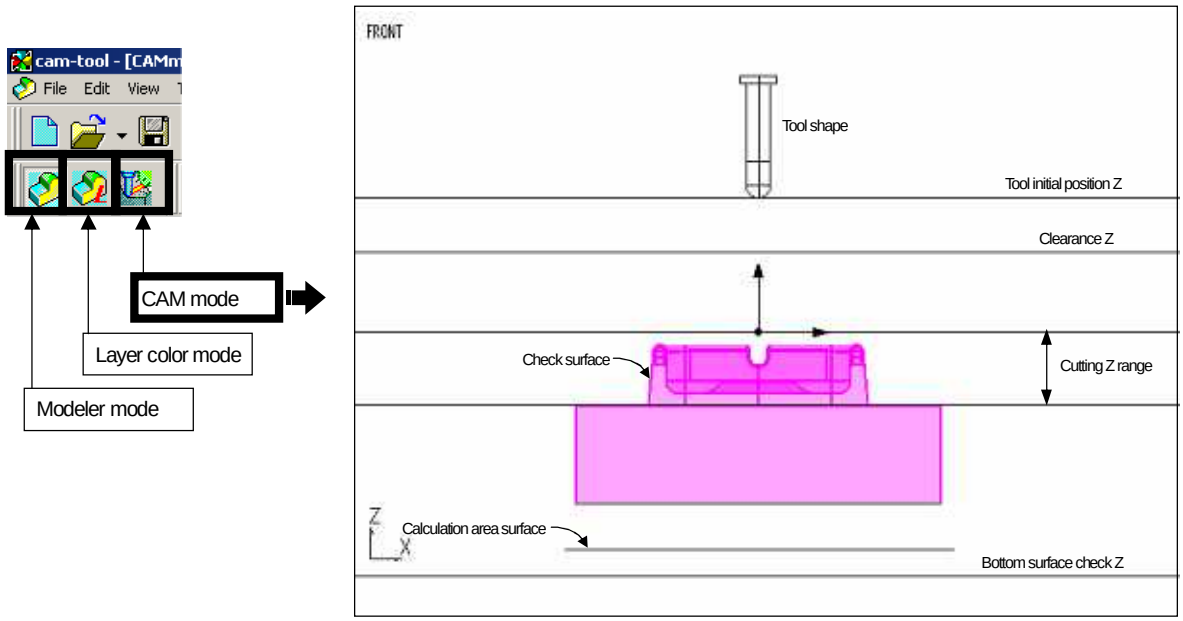
(16) Optimization condition : (All OFF)

(17) Click the [Finish] button.

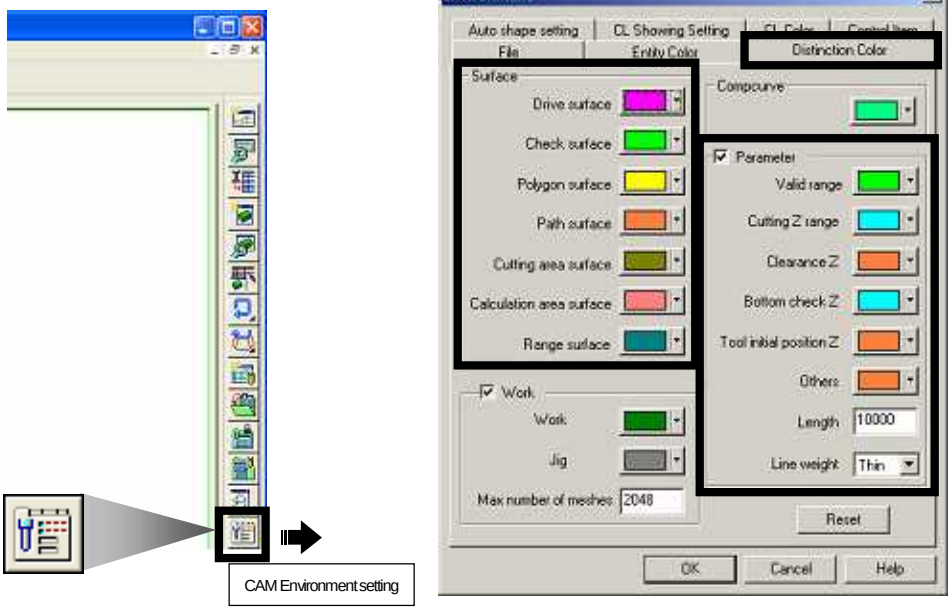
* Specify [Check surface] and [Calculation area surface] with the same operation you did in [Rough cutting].

How to confirm the settings of profile

It is available to confirm the settings of profile in CAM mode visually.
Settings such as [Color of surface (Check surface/Calculation area surface)], [Parameter (Tool initial position Z value, Clearance Z /Cutting Z range)] are displayed by the color, which is set in [CAM environment/Distinction color].
Confirm each parameter with [View direction(FRONT) or [View direction(RIGHT)] function in horizontal direction.

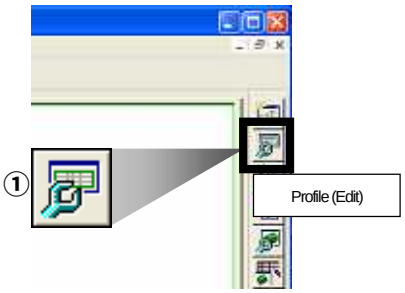


CAM Environment setting/Distinction Color

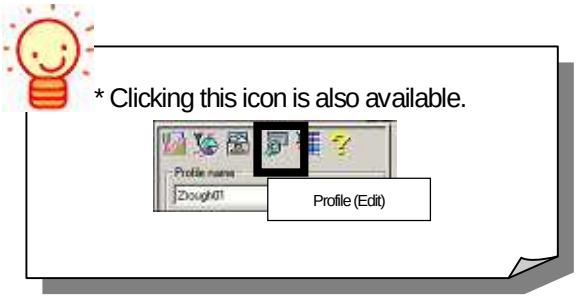


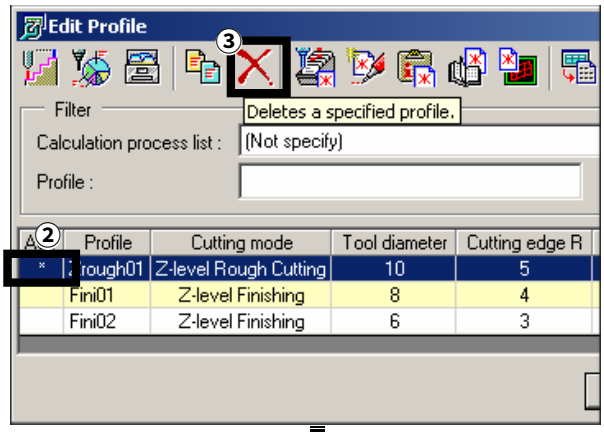
How to delete a profile

To delete an unnecessary profile, operate on the [Profile(Edit)] window.



(1) Click the [Profile(Edit)] button.

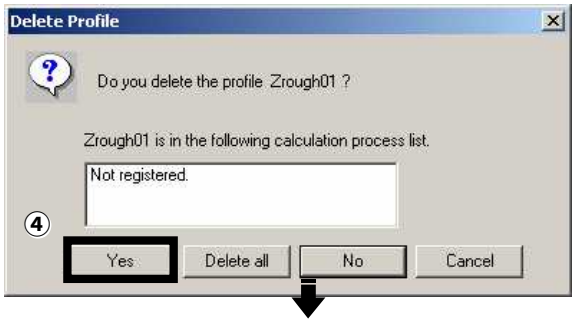




(2) Select a profile to be deleted.

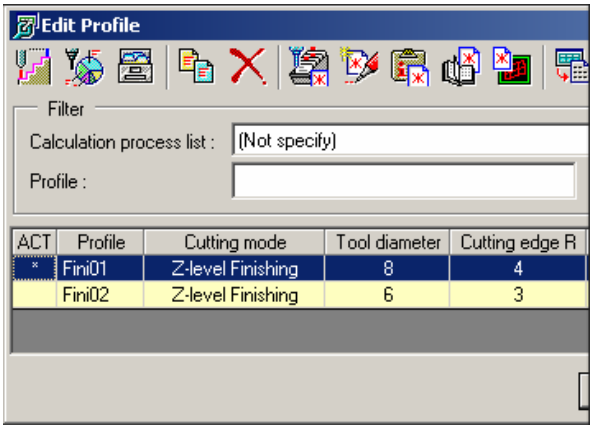
(3) Click the [Deletes a specified profile] button. 

(4) When the confirmation message appears, click [Yes] button. The specified profile will be deleted.



* Do not delete the created profile this time.

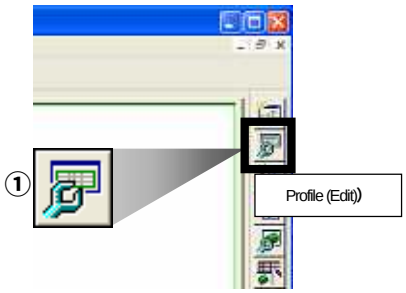
(5) Click the [Close] button to finish editing profile.



<<Chapter2 Operating flow of CAM processing>>

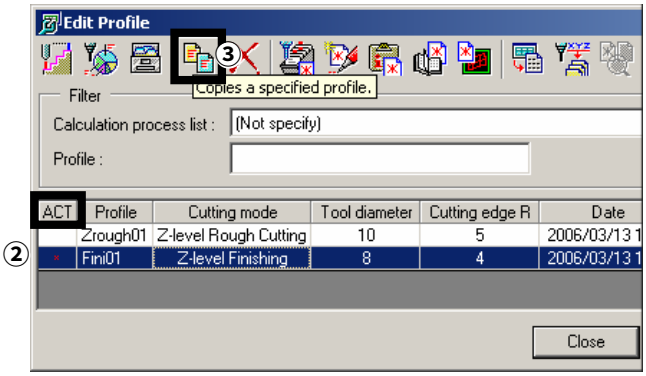
How to copy a profile

To copy a profile, operate on the [Profile(Edit)] window.



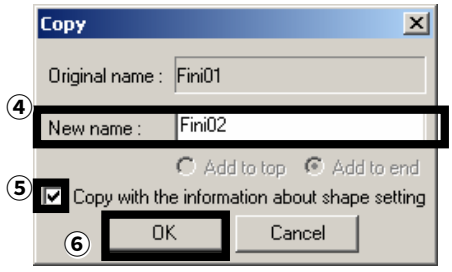
(1) Click the [Profile(Edit)] button.

* Clicking this icon is also available.



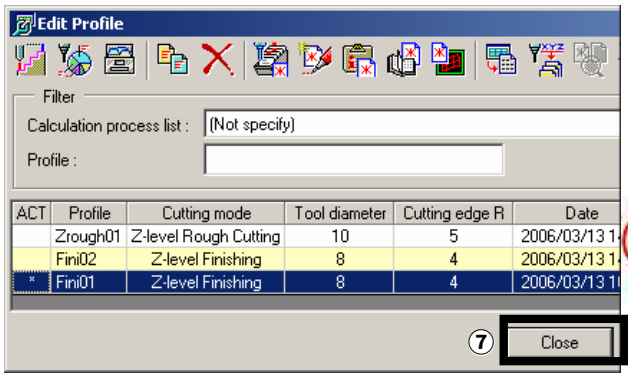
(2) Select a profile you wish to copy.

(3) Click the [Copies a specified profile.] button.



(4) Input name for a new file.

(5) If [Copy with the information about shape setting] is set ON, [Shape setting] such as [Check surface] /[Calculation area surface] will also be copied.



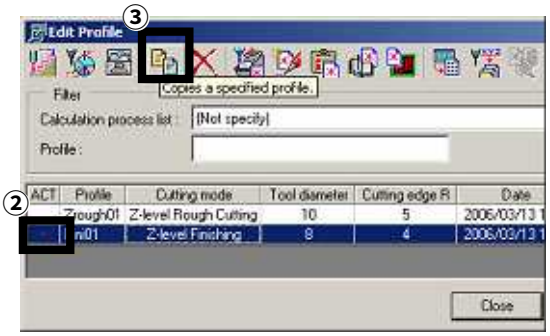
(6) Click the [OK] button, and the profile is copied.

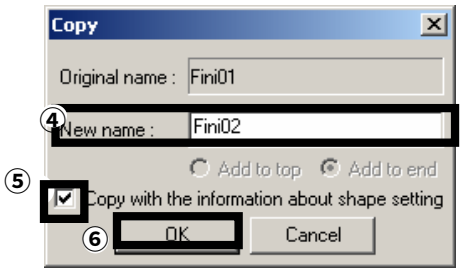
(7) Click [Close] button to finish editing profile.

If you select multiple profiles, they are copied simultaneously.

<<Exercise>>

Copy a profile of [Semi Finishing process] and create a profile of [Finishing process].





(1)  Click the [Profile(Edit)] button.

(2) Select [Fini01].

(3) Click the [Copies a specified profile.] button. 

(4) New profile name : Fini02

(5) [Copy with the information about shape setting]: ON

(6) Click the [OK] button.

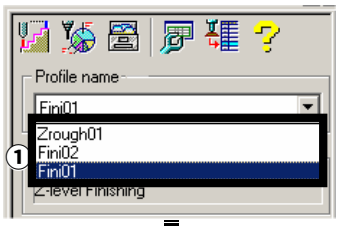
(7) Click [Close] button.

On the [Profile(Edit)] window, you can [Change profile name] / [Input comment]. Double-click on each item, and input name/comment directly.

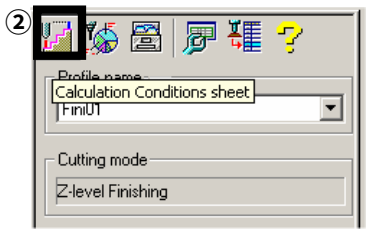
How to edit settings of profile

If you change [Calculation conditions] that you set when you created a new profile, or if you set calculation conditions in more details, click the [Calculation condition sheet] button to open the window.

<Control window>



(1) Activate a profile you wish to change.

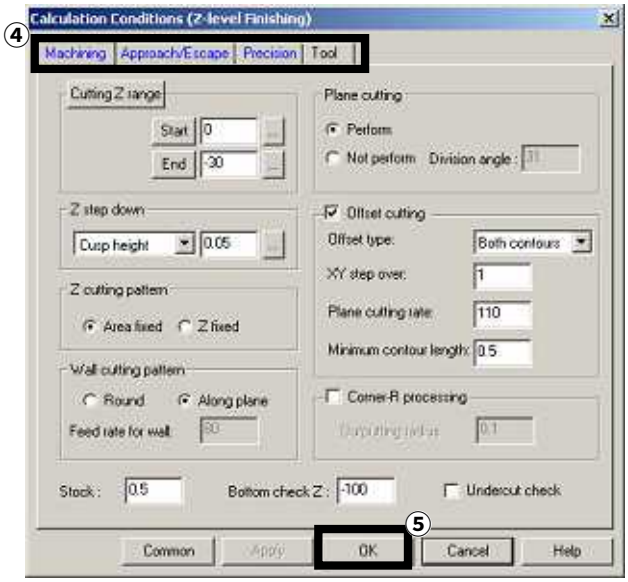


(2) Click the [Calculation condition sheet] button.

(3) [Calculation condition sheet] window is displayed.

(4) Activate a tab of item you wish to edit.

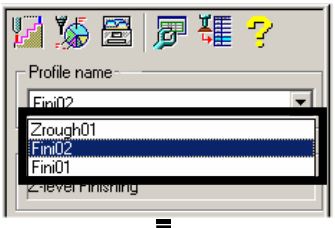
(5) After you set conditions, click the [OK] button.



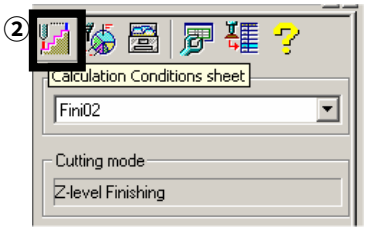
[Calculation condition sheet] page is categorized by [Machining]/[Approach/Escape]/[Precision]/[Tool]. Click each tab to set calculation conditions.

<<Exercise>>

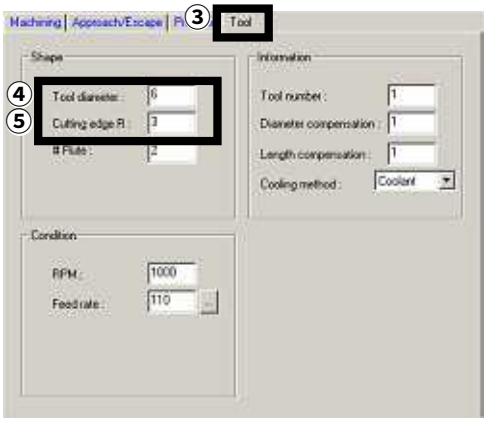
Change the setting of profile [Fini02] to create profile for [Finishing process].



(1) Activate profile [Fini02] on the Control window.



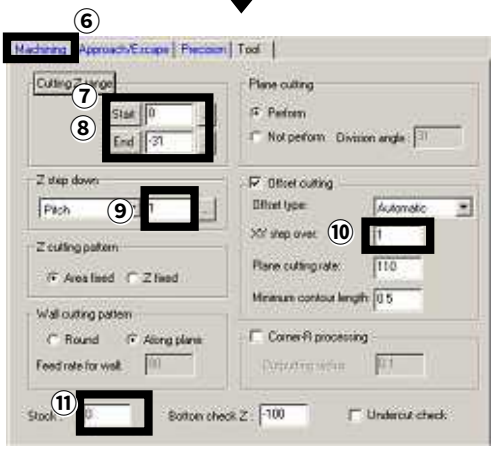
(2) Click the [Calculation Conditions sheet] button.



(3) Click the [Tool] tab.

(4) Tool diameter：6

(5) Cutting edge R：3



(6) Click [Machining] tab.

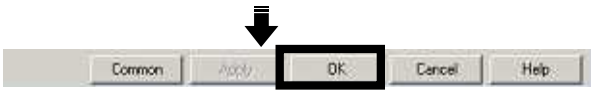
(7) Cutting range/Start: 0

(8) Cutting range/End: -31

(9) Z step down: 1 (mm)

(10) XY step over：1 (mm)

(11) Stock：0 (mm)

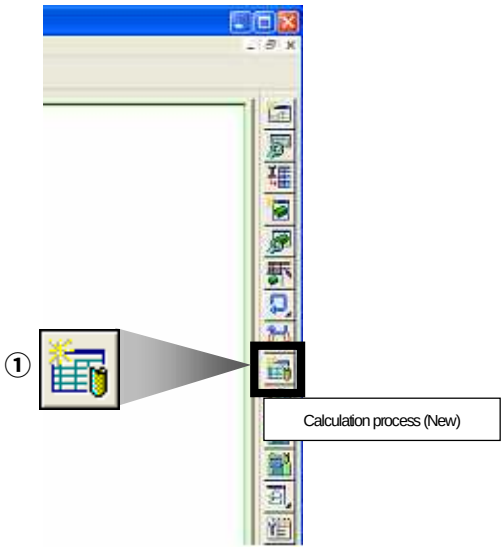


(12) Click [OK] button to finish editing.

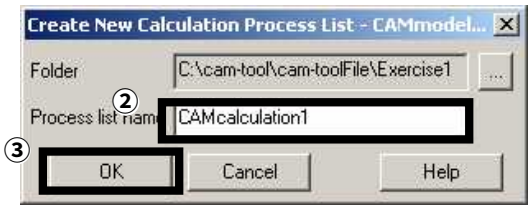
<<Chapter2 Operating flow of CAM processing>>

Before you execute CAM calculation…

Register the [profile] you set in a model file to [Calculation process list], which is a file for CAM calculation, and create [CL data]/[NC data]. On the [Calculation process list], you can also open other applications such as [CL Editor] that you edit the created CL, or [Machining process] that you output NC data.



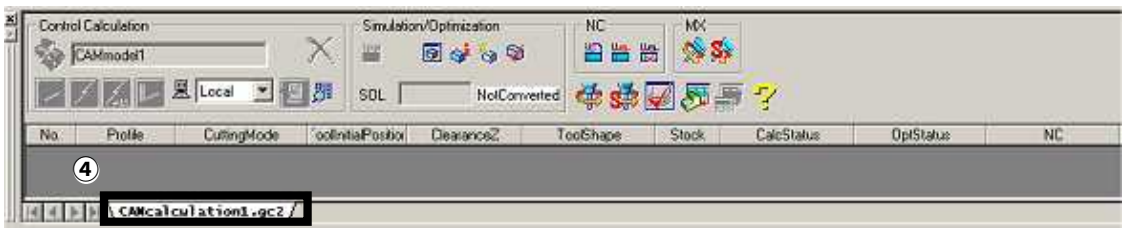
(1) Click [Calculation process (New)].



(2) Input [Process list name].

(3) Click [OK] button.

(4) [Calculation process sheet] is created.



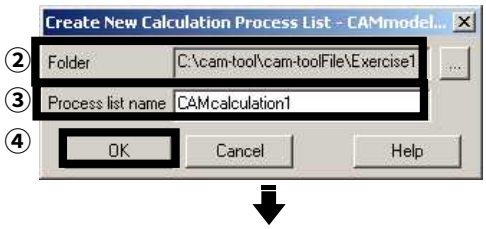
Create [Calculation process list] in the same folder, which the model file is saved in. [Calculation process list file], which [Profile] is registered to, is linked to [Model file], which [Profile] is saved in. Therefore, if link between [Model file] and [Calculation process list file] is destroyed by operations such as moving a file, CAM calculation cannot be executed normally.

Therefore, it is recommended to save those files in the same folder. When you work with any file, operate on the [folder] basis.

<<Exercise>>

Create Calculation process list [CAM calculation1.gc2].

(1) Click the [Calculation process(New)] button.

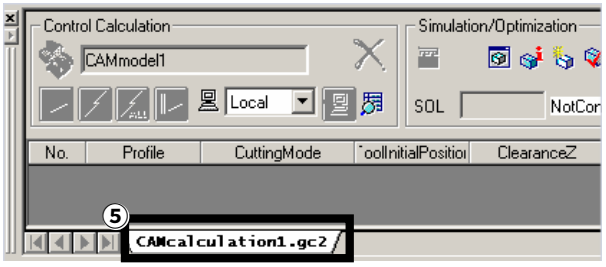


(2) Folder : Destination folder of the current active model file is selected.

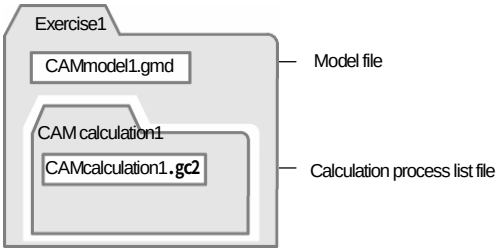
* Manage files on the folder basis.

(3) Calculation process list name : CAM calculation1

(4) Click the [OK] button.



(5) Calculation process list name is displayed.



(6) A folder which has the same name as the one of [Calculation process list file] is created in the folder that [Model file] is saved in, and [Calculation process list file] is created in the folder.

*CAM related files are created in [Calculation process list] folder.

[Calculation process sheet] related command

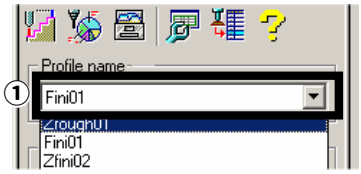
Calculation process (New)

Calculation process (Close)

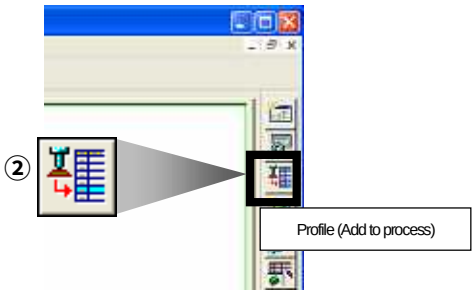
Calculation process (Recent)

How to register/change order of /delete file

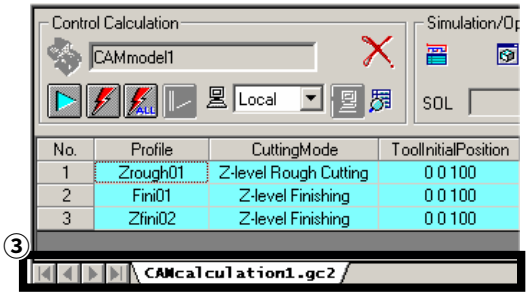
Before you execute CAM calculation, register the [Profile] that is created in the model file, to a [Calculation process list].
[Profile] that is currently activated on the [Control window] is registered to [Calculation process list].



(1) Activate the profile you wish to register.



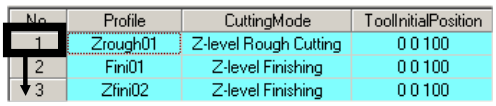
Clicking this icon is also available.



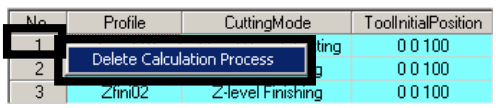
(2) Click the [Profile(Add to Process)] button.

(3) The profile is registered.

- To [change the cutting order], drag the number of process with the left mouse button.



- To [delete process], click the number of process with the right mouse button.



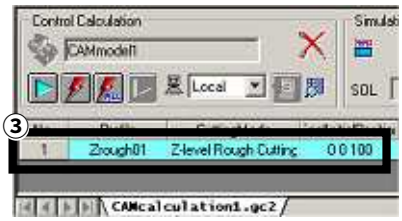
- If a process, which has the different too initial position from other processes, is registered, a warning message appears.

<<Exercise>>

Register profiles, [Zrough01]/[Fini01]/[Fini02] to a Calculation process list.

(1) Set [Zrough01] as active layer.

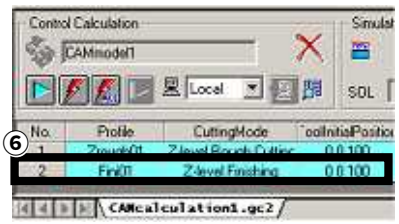
(2) Click the [Profile(Add to process)] button. 



(3) [Zrough01] is registered to the calculation process list.

(4) Set [Fini01] as active layer.

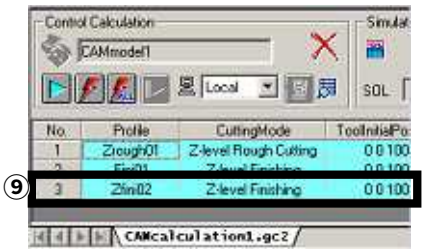
(5) Click the [Profile(Add to process)] button. 



(6) [Fini01] is registered to the calculation process list.

(7) Set [Fini02] as active layer.

(8) Click the [Profile(Add to process)] button. 

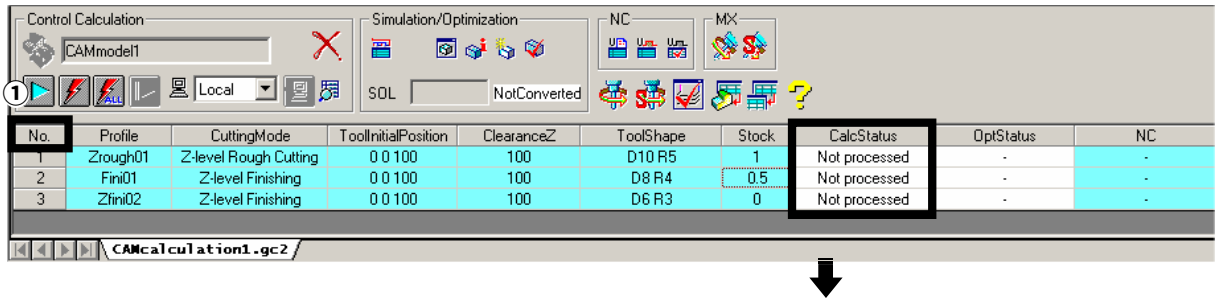


(9) [Fini02] is registered to the calculation process list.

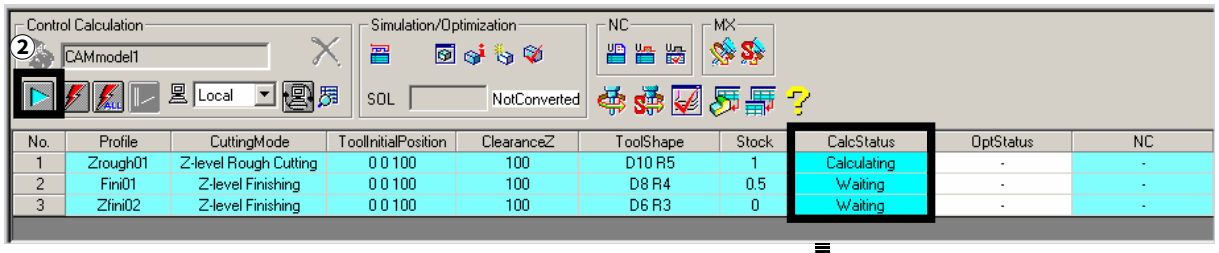
How to start CAM calculation

After you finish preparation for executing CAM calculation (Create Calculation process list/Register profile), execute calculation of [Profile] to create [CL data].

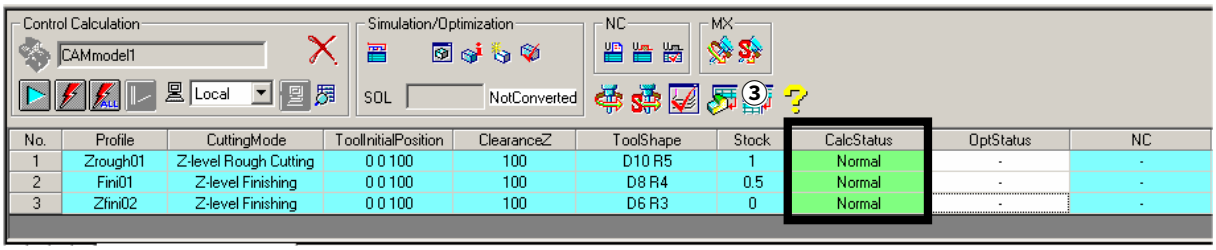
(1) Select a process that you wish to execute calculation.(To select all of processes, click[Number].)



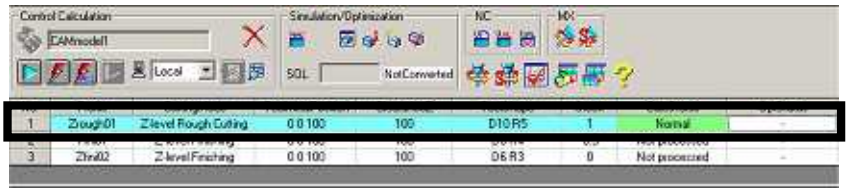
(2) Click the [Start] button.



(3) When calculation ends normally, [Normal] is displayed. ([CL data] is created.)



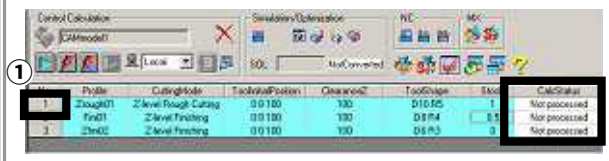
The selected process can be calculated separately.



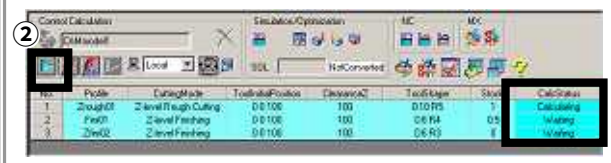
When calculation is started, [Model file] is overwritten. Please note that if multiple [Model files] are opened, all of processes are overwritten simultaneously.

<<Exercise>>

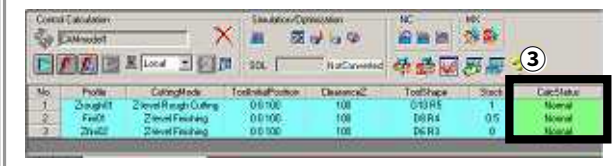
Execute [CAM calculation].



(1) Click the [number] button to select all of processes.

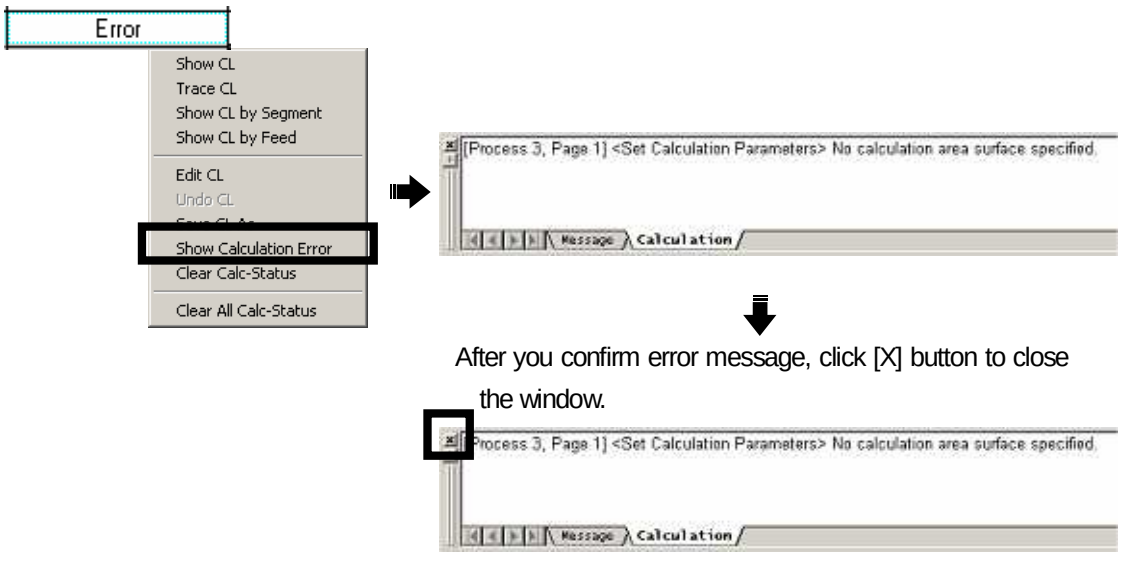


(2) Click the [Start] button.



(3) When calculation ends, [Normal] is displayed.

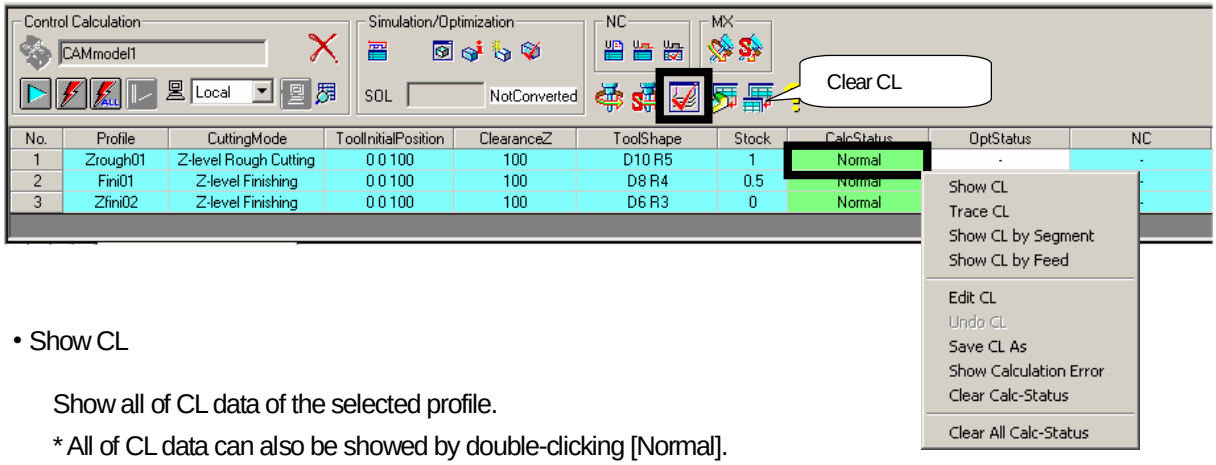
If [Abnormal] is displayed instead of [Normal], right-click on [Abnormal], and select [Show calculation Error] to confirm the cause of error.



After you confirm error message, click [X] button to close the window.

How to confirm calculation result

Right-click on [Calculation result(Normal)], and confirm [CL data] that is the calculation result.
CL data can be displayed using [Show CL]/[Trace CL]/[Show CL by Segment]/[Show CL by Feed] function.



• Show CL

Show all of CL data of the selected profile.

* All of CL data can also be showed by double-clicking [Normal].

• Trace CL

Show CL data trace of the selected profile.

 Frame step function in backward direction

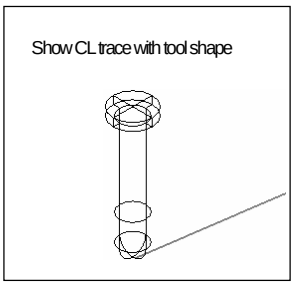
 Pause

 Start/Restart

 Frame step function forward direction

 Stop

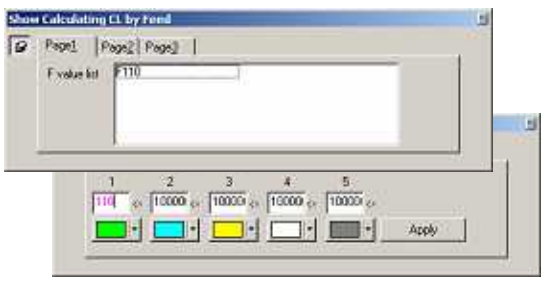
Show tool shape : ON ➡



• Show CL by Segment

Show CL data of the selected profile by the segment.

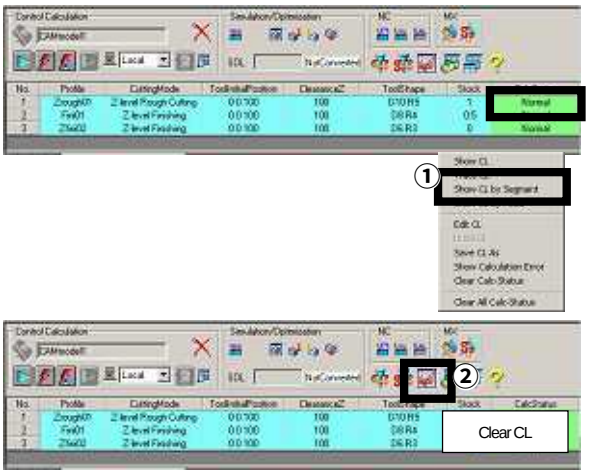
• Show CL by Feed



Show CL data of the selected profile by the Feed rate in color. (This function is not used in this text book.)

<<Exercise>>

Confirm the created [CL data].



(1) Right-click on [Normal] of [Zrough01], and select [Show CL by Segment].

* In [Z-level Rough Cutting]/[Z-level Finishing], if [Show CL by Segment] command is executed, CL is showed at each Z level.

Click the [Next] button to confirm CL data at each Z level from the start Z level.

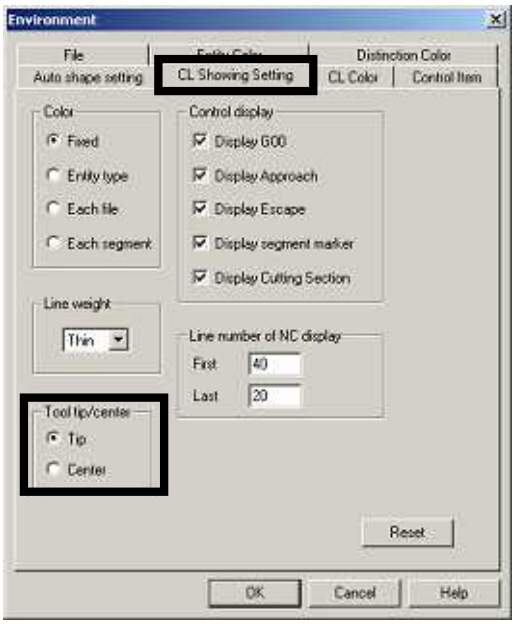
(2) Click the [Clear CL] button to clear the displayed CL.

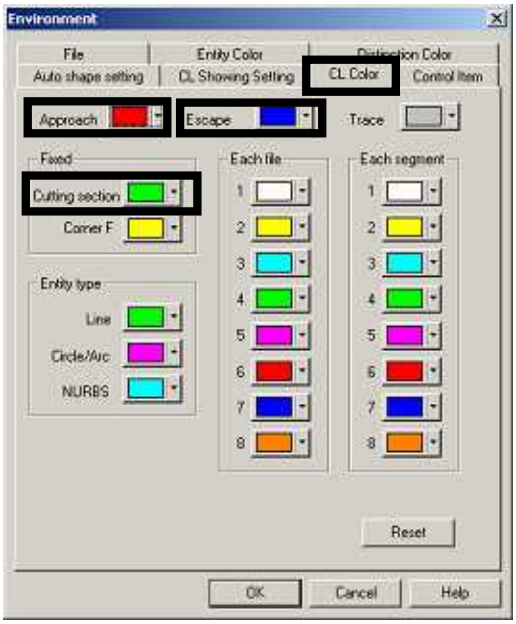
- Note -

You can set color or method of showing CL on the [CAM Environment] window.
To confirm [CL data], at least the items marked with [ ] below, are necessary.

CAM Environment: Distinction Color/CL Color

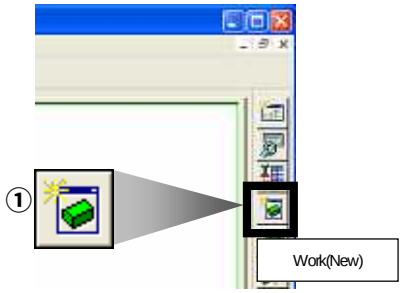
(Fourteenth icon on the right toolbar)



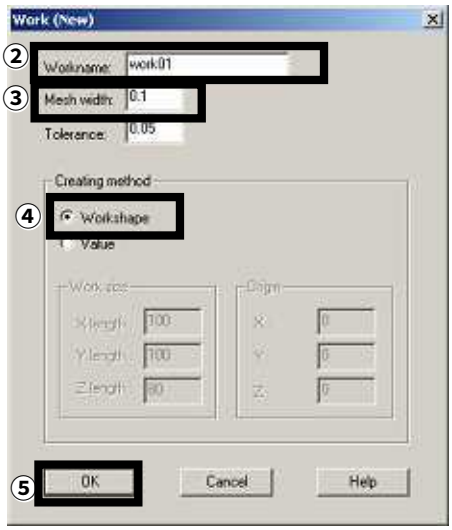


How to create work (For executing cutting simulation)

When calculation ends, if you wish to confirm solid shape after cutting, execute [Cutting simulation].
(If you do not need to confirm solid shape, omit the operations in page 48-57.)



(1) Click the [Work(New)] button.

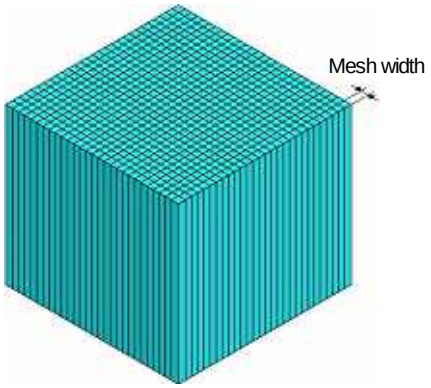


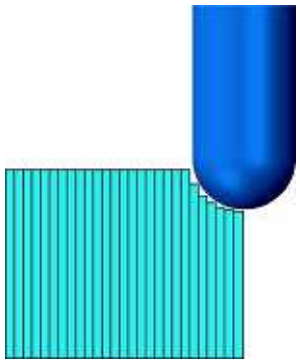
(2) Input [Work name].
* Input alphanumeric characters within twenty three letters (Alphabetic for the first letter.)
(3) Input [Mesh width].

(4) Select [Work shape] for [Creating method].

(5) Click the [OK] button.

What is [Mesh width]?
Collective entity, which is a cuboid of [One side = mesh width], is defined as solid shape for [Cutting simulation].
The smaller [Mesh width] you input, the higher precision of [Cutting simulation] is set.
However, calculation time increases at the same time. Therefore, specify proper [Mesh width] depending on the cutting shape.

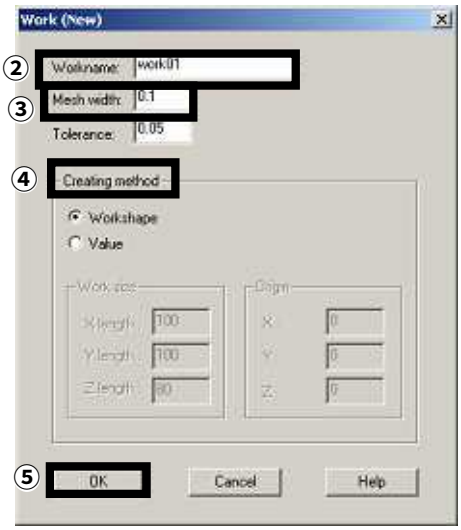




<<Exercise>>

Create a new work file [work01]. Set [Layer 10] for active layer to show material shape.

(1) Click the [Work(New)] button. 

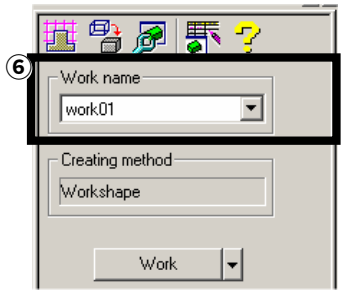


(2) Work name： work01

(3) Mesh width： 0.1

(4) Creating method： Work shape

(5) Click the [OK] button.



(6) [Work] is created.

[Work] related command


Work(New)


Work(Edit)

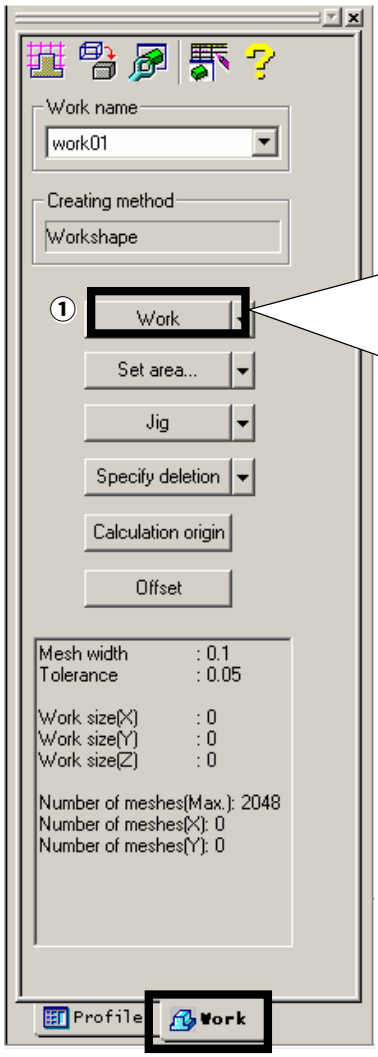

Work(Register in process)

<<Chapter2 Operating flow of CAM processing>>

How to set material shape for work (For executing cutting simulation)

Set material shape for work in [Shape setting] on the Control window.

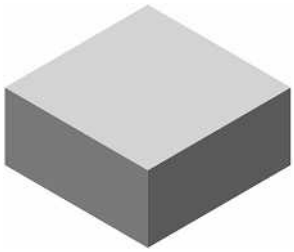
<Control window>



(1) Click the [Work] button.

(2) Specify surfaces for material shape.

② Specify by area:In

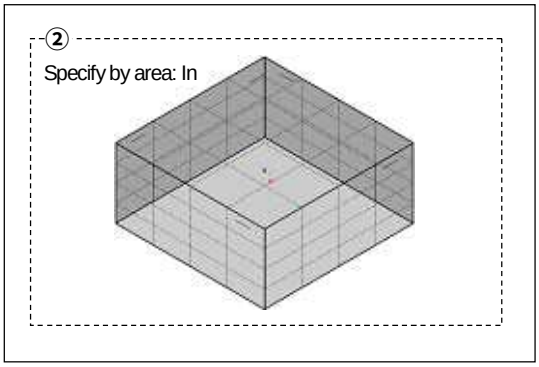


(3) Press the center mouse button to determine the specifying.

<<Exercise>>

Set [Layer10] for active layer.

Next, set [Shape setting] for the created [work01].



(1) Click the [Work] button.

(2) Specify surfaces as material shape. (Specify by area: In)

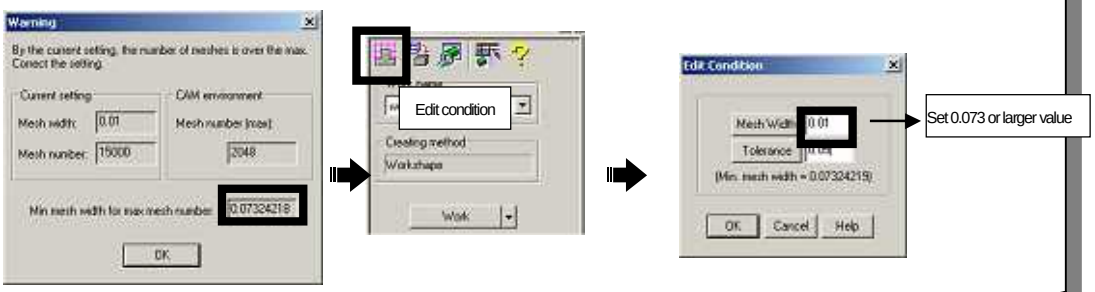
(3) Press the center mouse button to determine the specifying.

After you set shape, confirm in [CAM mode]. (It is displayed by the color specified in [CAM Environment/Distinction Color/Work].

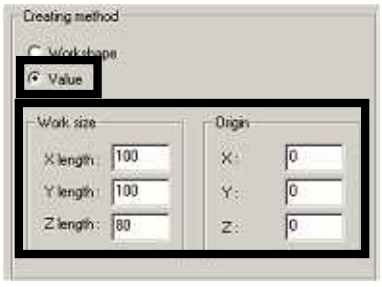
Then, set [Layer1] for active layer.

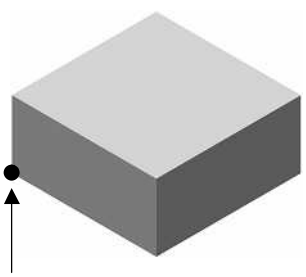
If small [Mesh width] is set for [Work], an warning message appears.

Click [OK] to close the dialog, and change the value on the [Edit Condition] dialog.



When [Value] is selected, [Work] of cuboid shape is created by the specified value.



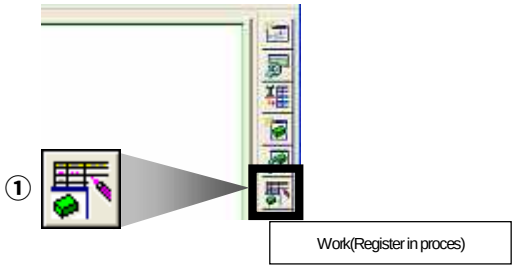


Work is created at the smallest value of XYZ coordinate value as the base point .

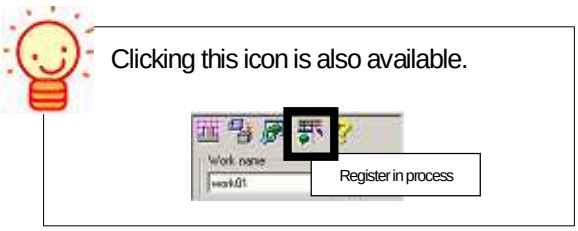
<<Chapter2 Operating flow of CAM processing>>

How to start cutting simulation (For executing cutting simulation)

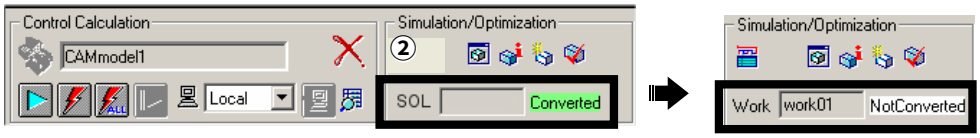
Register [Work] to [Calculation process list], and execute [Cutting simulation].
Before you execute [Cutting simulation], change the [Optimization Status] from [-] to [Not processed].



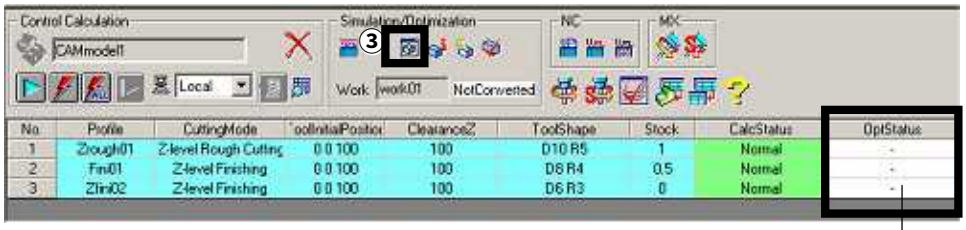
(1) Click the [Work(Register in process)].



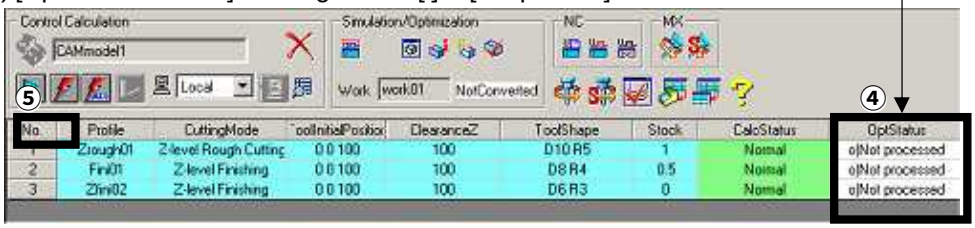
(2) When you register work, [Work name] is displayed in the [Simulation/Optimization] area.



(3) Click the [Optimize All] button.

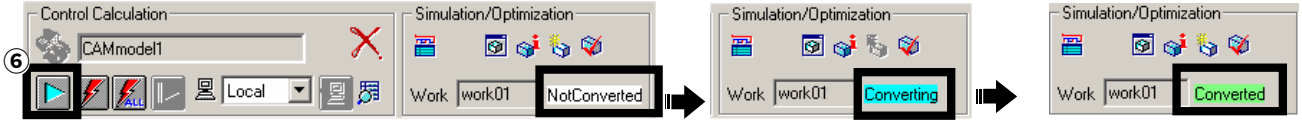


(4) [Optimization Status] is changed from [-] to [Not process].

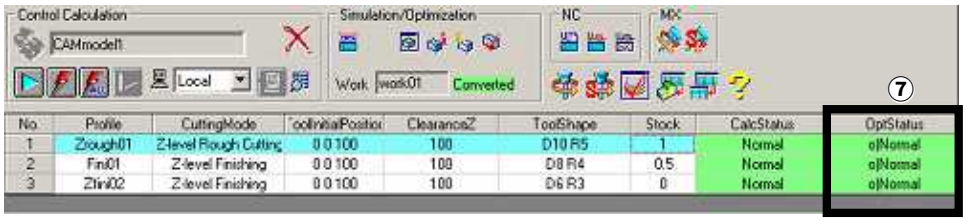


(5) Select process you wish to execute calculation. (If you wish to select all of processes, click the [Number] button.)

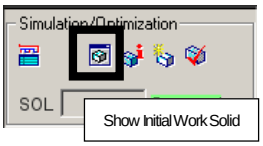
(6) Click the [Start] button to convert work to solid shape. (Not Converted >> Converting >> Converted)



(7) After work is converted to solid shape, execute [Optimization]. When calculation for optimization ends normally, [Normal] is displayed.



• How to display work that has been converted to solid

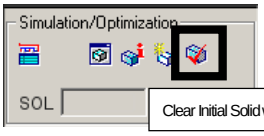


Confirm initial work shape by [Show Initial Work Solid] function.

• How to confirm information of work that has been converted to solid

Confirm information of initial work, such as mesh width.

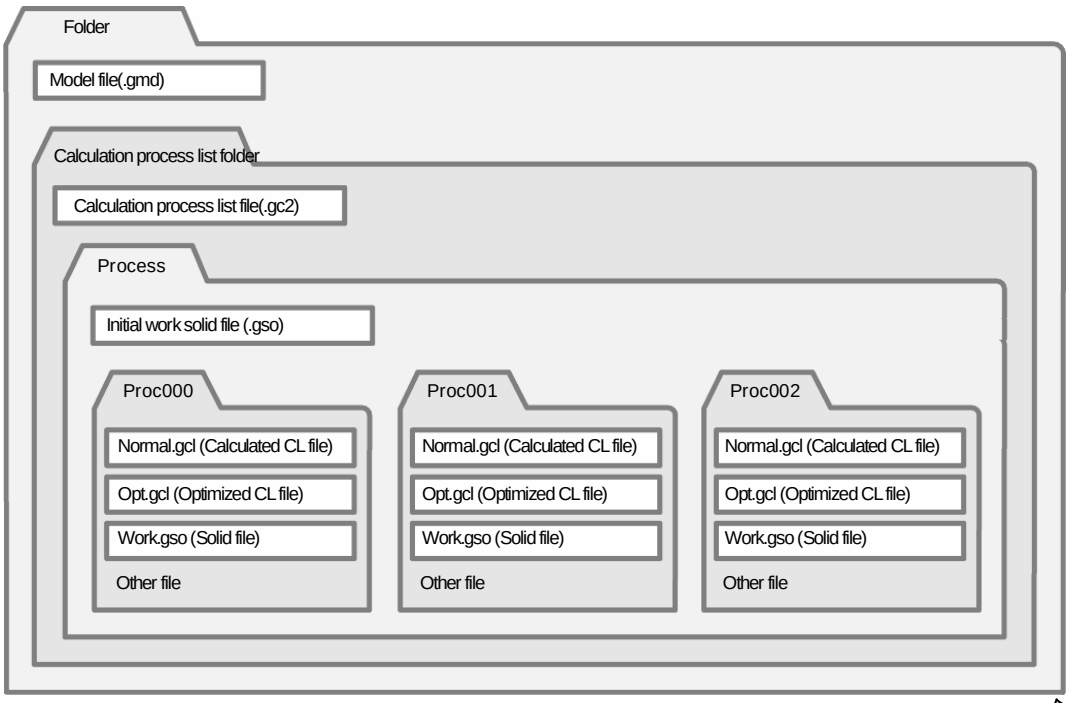
• What if you register work by mistake?



Cancel the registration, and register work to calculation process list again.

Where is [CL data] file saved in?

Each calculated data files are automatically controlled as in the figure below. If you create [CL data] by the basic CAM processing, you do not need to specify files directly.



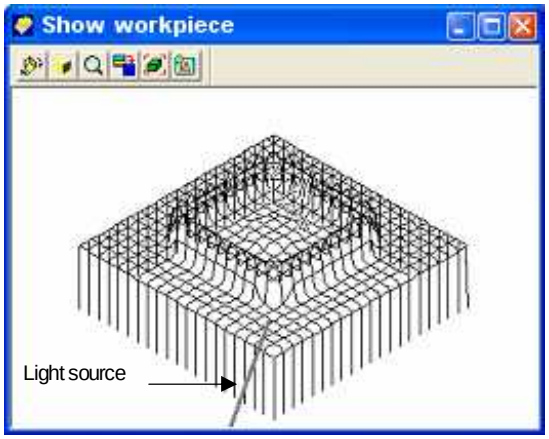
* Only when you execute [Cutting simulation (Optimization), [Opt.gcl] [Work.gso] files are created.

<<Chapter2 Operating flow of CAM processing>>

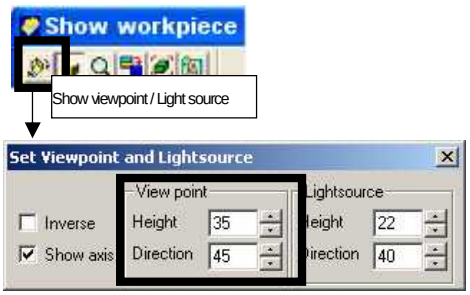
How to confirm simulation result (For executing cutting simulation)

Right-click on [Normal] of calculation result of [Optimization], and confirm the optimized solid work piece.

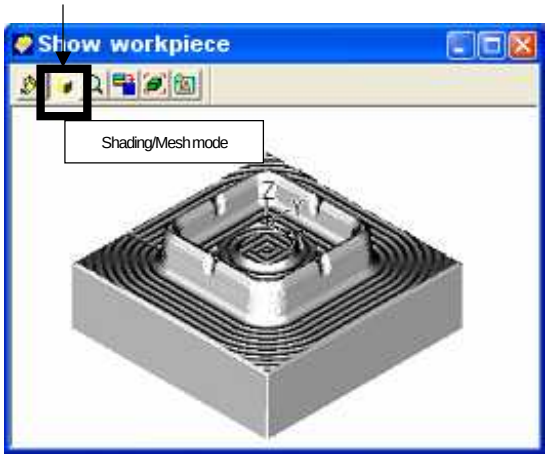
• Show Solid



Display solid work piece of the selected profile, which cutting has been performed.* [Show viewpoint] can be edited.

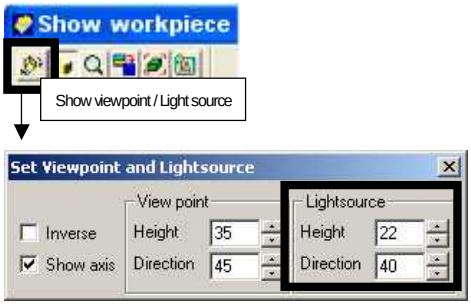


• Show Solid (Shading)



Display solid work piece of the selected profile, which cutting has been performed, by shading.

* [Light source] can be edited.



In showing solid, area that is cut in each process is displayed by different color.

- Initial work solid : Blue
- The first process solid : Dark purple
- The second process solid : Yellow
- The third process solid : White (The fourth process and later process are displayed in the same color order such as, Dark purple, Yellow, White. This color order is repeated.)


Zoom In/Out: Zoom in/out to display solid. (For executing shading function, it is recalculated.)


Reset color: Reset the solid work piece of each process that is displayed by different color to initial work solid(Blue). * Once this function is executed, solid cannot be restored.

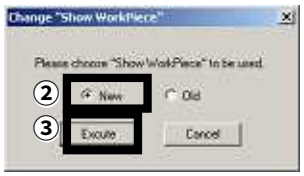

Zoom Extent: Zoom up to enlarge entities to fill the window.

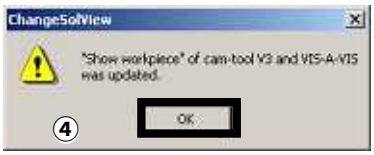

Savecurrentview: Save the current view to apply it to the next showing solid.

If you wish to rotate solid work piece by pressing the right mouse button?

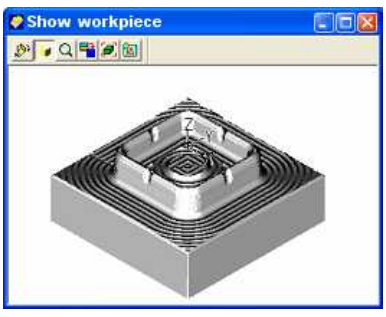
[Show solid] function can be used more efficient with the mouse operation.
Close cam-tool V3 or cam-tool MX , and execute [ChangeSolView.exe] file that is stored in the [Installation CD-ROM].


ChangeSolView.exe
(1) Double-click





- Show solid/New function



* [ChangeSolView.exe] file can be copied such as on the desktop.

* To restore the previous function, execute the file and select [Old].

<<Chapter2 Operating flow of CAM processing>>

<<Exercise>>

Register [Work] to Calculation process list, and execute [Cutting simulation]. And confirm [Solid work piece] that has been cut.

(1) Click the [Work (Register in process)] button. 

(2) [Work01] is registered to [Calculation process list].

(3) Click the [Optimize All] button.

(4) [Optimization status] is changed to [Not process].

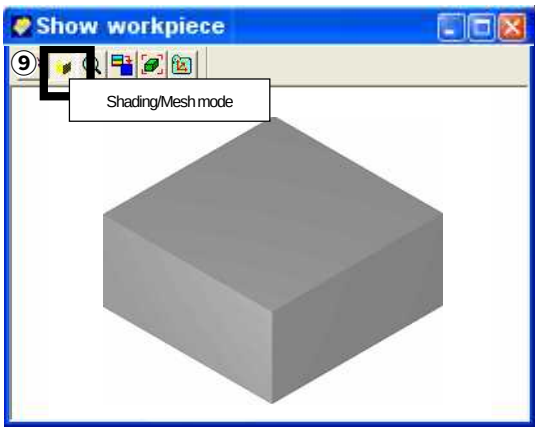
(5) Click the [Number] button.

(6) Click the [Start] button.

(7) When calculation ends normally, [Normal] is displayed.

(8) Fist, confirm the solid after cutting.

Click the [Show initial solid work] button. 



(9) [Show work piece] window is displayed.

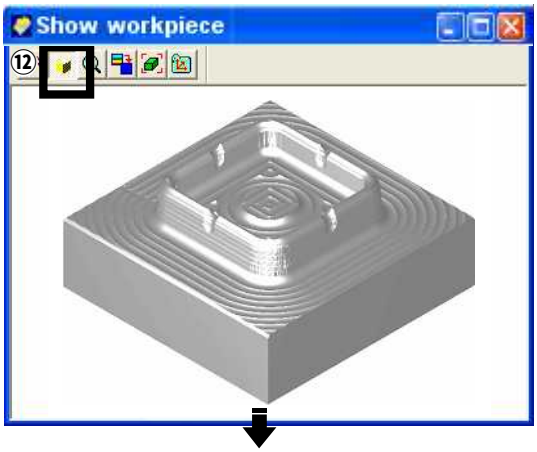
Click the [Shading/Mesh mode] button and confirm the initial work solid.

* Initial work solid is displayed in blue.

(10) After you confirm, close the [Show solid work] window.

(11) Confirm the solid after the cutting of the [first process] has been finished.

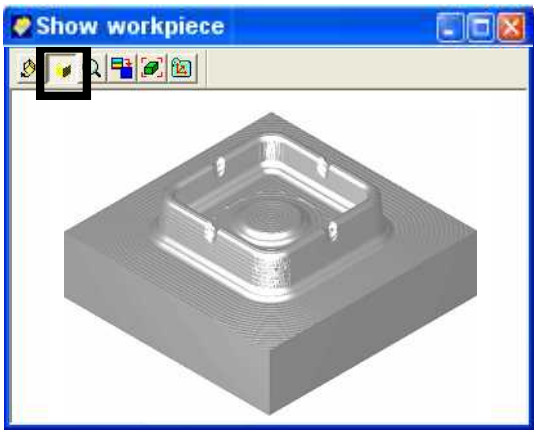
Right-click on [Normal] of [Optimization Status] of [Zrough01] , and select[Show Solid work].



(12) [Show work piece] window is displayed.
Click the [Shading/Mesh mode] button and confirm the work solid.

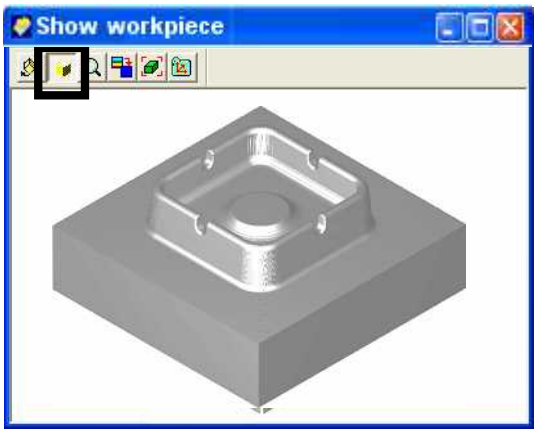
*Area that has been cut in the first process is displayed in dark purple.

(13) After you confirm, close the [Show work piece] window.



(14) Confirm the solid after the cutting of the [second process] has been finished. *Area that has been cut in the second process is displayed in yellow.

(15) After you confirm close the [Show work piece] window.



(16) Confirm the solid after the cutting of the [third process] has been finished.

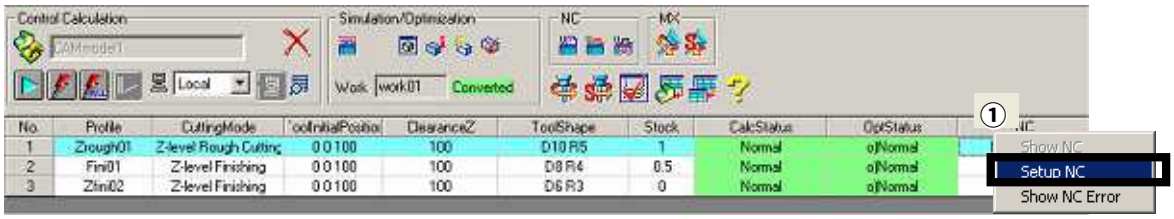
* Area that has been cut in the second process is displayed in white.

(17) After you confirm, close the [Show work piece] window.

How to create NC data (1) – Create NC data on Calculation process list -

Create NC data on [Calculation process list]. NC data is output with [Tool tip] fixed.
Before you create NC data, change [NC] status from [-] to [Not created].

(1) Right-click on [NC] of the selected process, and select [Setup NC].



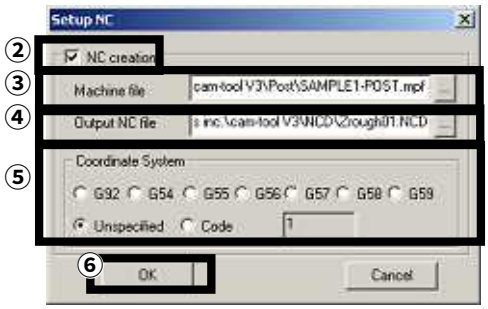
(2) Set [NC creation] ON.

(3) Specify [Machine file].

(4) Specify [Output NC file].

(5) Specify [Coordinate System].

(6) Click the [OK] button.

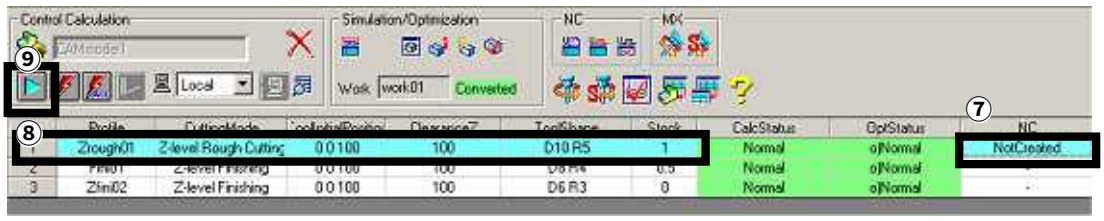


* If NC data which has the same name already exists, confirmation dialog is displayed. If it can be overwritten, click [OK].

(7) [NC] status is changed to [-] to [Not created].

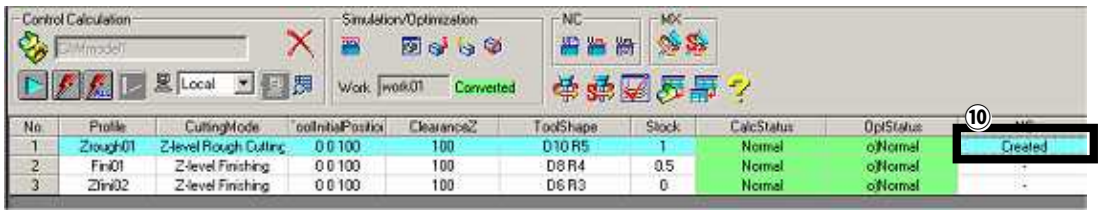
(8) Select process you wish to create NC data.

(9) Click the [Start] button, and execute [NC creation].



* If NC data which has the same name, can be overwritten, click [Yes] or [All OK].

(10) When NC data has been created, [Created] is displayed.



What is machine file?

Machine file is a format to output NC data.

Since format for outputting NC data is need to be changed depending on machine tool, it is used.

(Recommended file management) C:/ cam-tool/ cam-tool Conf/ Post / [ ]. mpf

What is Output NC file?

Since NC data is not controlled by [Calculation process list], specify a folder to output NC data.

(Recommended file management) D:/ cam-tool /NCD

Environment setting of [NC creation]

Machine file : Specify machine file.

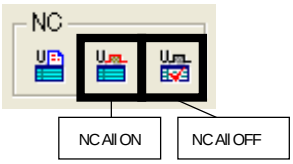
Output in: Specify a folder to output NC data.

Update : Specify [Machine file]/[Output in] for all of processes on calculation process list.

OK : Specify [Machine file]/[Output in] for a processes that is newly registered to calculation process list.

* [Coordinate System] cannot be specified. To specify it, do the operations described in [Page.58 (5)].

Set [NC creation] ON/OFF by one operation

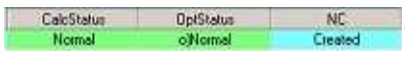


NC All ON : Set [Not created] (NC creation target) to all of processes of calculation process list.

NC All OFF : Set [-] (not NC creation target) to all of processes of calculation process list.

NC data is created differently depending on the status of [Optimization status] on calculation process list.

Create NC data with the calculated CL data (Normal.gcl).



Create NC data with the optimized CL data (Opt.gcl).

<<Chapter2 Operating flow of CAM processing>>

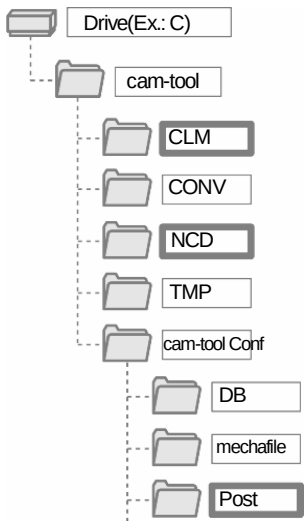
<<Exercise>>

Create [NC data] on [Calculation process list].

(1) Set Environment.

Click the [Setup NC Condition] button.

Recommended file management



(2) Machine file:

C:/cam-tool/ cam-toolConf / Post / [ ]. mpf

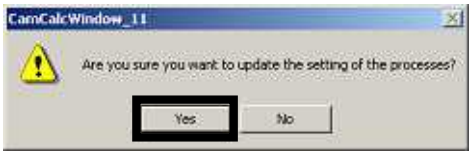
* Select a machine file for your company.

(3) Output in：

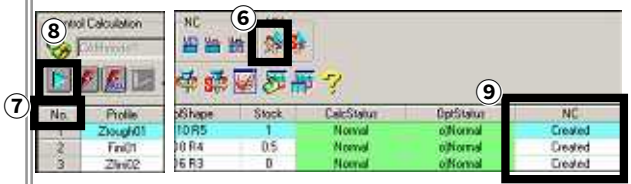
C:/ cam-tool/NCD

(4) Click the [Update] button.

When confirmation dialog is displayed, click [OK].



(5) Click the [OK] button.



(6) Set the status for creating NC data.

Click the [NC All ON] button.

(7) Select a process you wish to create NC data.

* If you wish to select all of processes, click the [Number] button.

(8) Click the [Start] button.

(9) When [NC data] has been created, [Created] is displayed.

Ex. created NC data

```
%
G00G40G49G80
G91G28Z0.
G28X0.Y0.
T01
M06
S1000M03
G91
G17G00X-80.Y75.
G18Z-95.
M08
G01Z-5.F80
G17X5.F110
```

(10) Confirm the created NC data.

Open a [NC data file] created in the folder that is specified for [Output in], by double-clicking.

D:/cam-tool /NCD /Zrough01.NCD

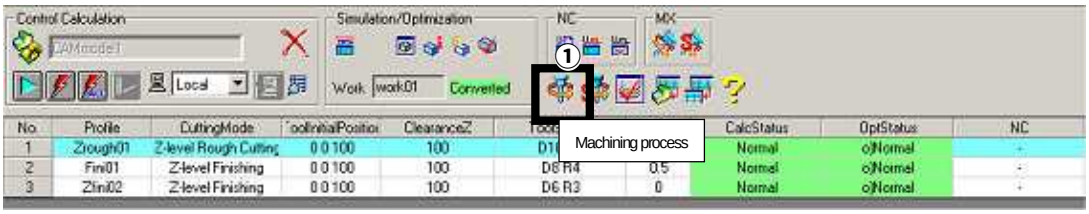
D:/cam-tool /NCD /Fini01.NCD

D:/cam-tool /NCD /Fini02.NCD

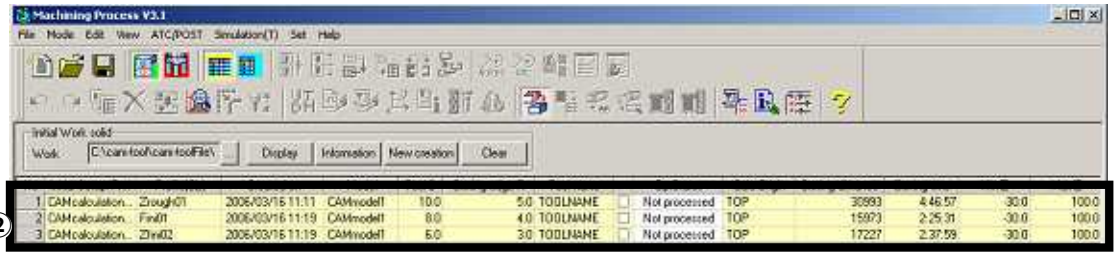
How to create NC data (2) – Create NC data on Machining process -

Crete NC data on [Machining process]. When you create NC data, you can set conditions such as [Program number], [Tool number], [Feed rate], [Create NC data of multiple processes as a single NC data].
And select [Tool tip] or [Tool center] for outputting NC data.

(1) Click the [Machining process] button.

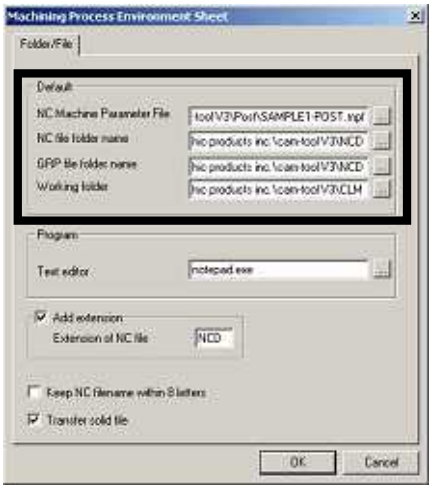


(2) Application of [Machining process] is opened, and processes(profiles) that calculation have ended normally on [Calculation process list] are registered.

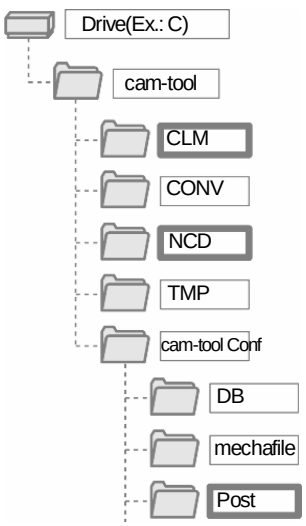


Before you create [NC data],

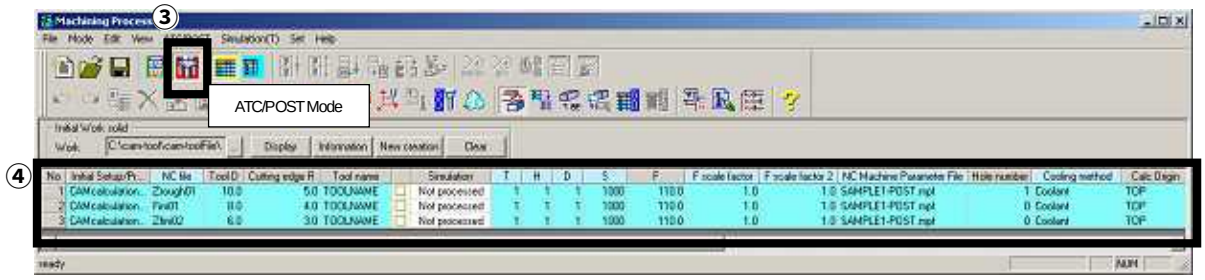
Before you create NC data, [Environment setting] such as [Machine file], [Folder for outputting data] is required.



Recommended file management



(3) Click the [ATC/POST] button.



(4) If necessary, edit parameters.

How to edit each parameter

To edit [NC file]/[T number]/[H number]/[D number]/[S(Spindle speed)/[F scale factor]/
[F scale factor2], double-click on each item on machining process, and input value directly.
To edit [NC machine file]/[Coolant method]/[Coordinate system], right-click on each item of the selected process, and select a parameter you wish to edit. When dialog is displayed, input value.

* F (Feed rate) cannot be edited directly. To edit F value, input value for [F scale factor] or [F scale factor2].
  F scale factor : [F x F scale factor] = Feed rate at cutting area
  F scale factor2 : [F x F scale factor2] = Feed rate at Approach/Escape, Connecting move

How to edit parameters of multiple processes simultaneously (Select processes before you operate)

Set THD

Set THD number for the selected processes.

- Sequential : Value for the first process is defined as the initial value, and sequential number is set for later processes.

Skip. sequential : For processes which have the same Tool name/Tool diameter/Tool radius, the same number is set.

Set NC file

Set NC data file name for the selected processes. If multiple processes are selected, [-1] [-2] [-3] … are added to NC data file name.

Set NC Machine file

Set N NC Machine file name for the selected processes.

Set coordinate system

Set coordinate system for the selected processes.

Set Coolant method

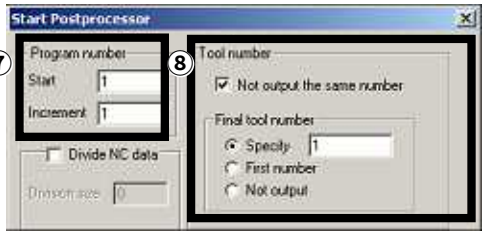
Set Coolant method for the selected processes.

(5) Select process you wish to create NC data. (If you select all of processes, click the [Number] button.)
To select multiple processes, use [Shift key] or [Ctrl key].
If you create NC data of all of processes simultaneously, select all of processes before operation.

(6) Click the [Start Postprocessor] button.

(7) Set [Program number].

(8) Set [Tool number].



• Program number

Start : Specify program number. If you create multiple NC data, program number of the first NC data is specified.

Increment : This function is valid when [Multiple] is set for [Create file mode]. For the second NC data and later processes, the value specified here is added to each program number.

* To output program number in NC data, open the [Set Machine Parameter File] dialog, and click [Edit Macro]. (Operation of [Edit Macro] is not described in this text book. Refer to On-Line HELP of Machining process.)

• Tool number

Not output the same number :

Set not to output tool number when sequential processes have the same tool number. Next tool number is output in NC data for preparing tool change in the next process.

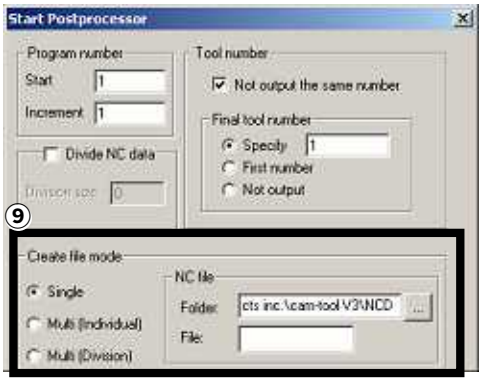
Final tool number :

Specify the tool number for the final process.

- Specify: Specify number for the next tool number of the last process.
- First number: Specify the first tool number for the next tool number of the last process.
- Not output : Not output the next tool number of the last process in NC data.

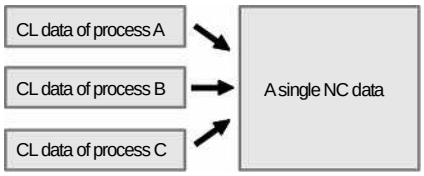
<<Chapter2 Operating flow of CAM processing>>

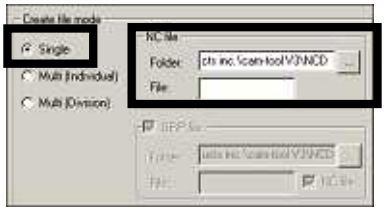
(9) Set [Create file mode].



Single： Create a single [NC data] with [CL data] of multiple processes.

It is available to output command of tool change for each process, depending on the setting on [Set Machine Parameter File].

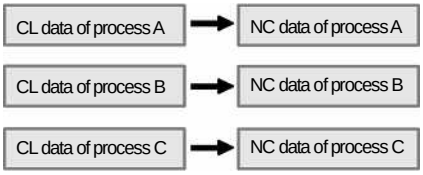




* When you select [Single], input [NC file name].

Multiple (Individual): With multiple [CL data], create [NC data] separately.

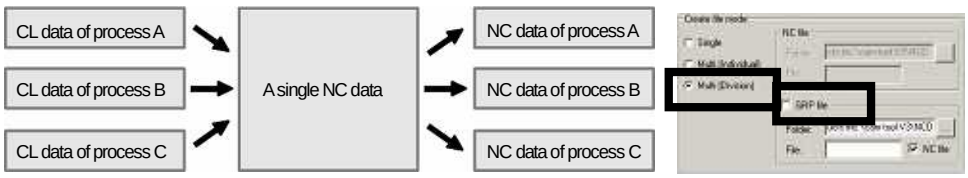
For each NC data, [START] is added to the start, and [END] is added to the end.



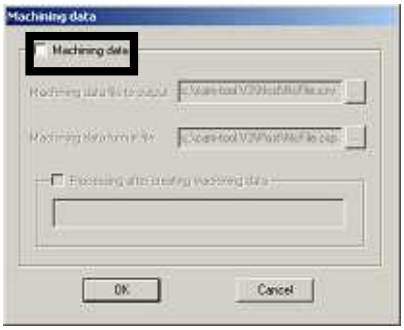
* When [Multiple] is selected, profile name is specified for [NC file name].

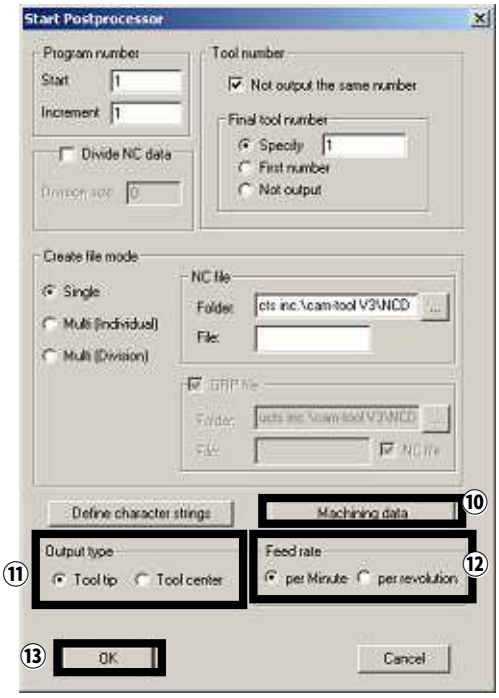
* Since [GRP file] is for our DNC device, basically set it OFF.

Multiple (Division): After a single [NC data] is created with [CL data] of multiple processes, NC data is divide by process. [START] is added to the first NC data, and [END] is added to the last NC data. Nothing is added to the other NC data.



(10) Click the [Machining data] button and set [Machining data] OFF, and click the [OK] button.





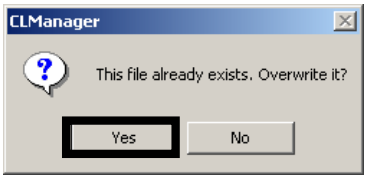
(11) Specify [Output type].

(12) Specify [Feed rate].

(13) Click the [OK] button.

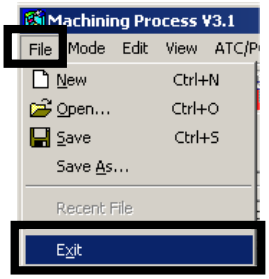
(14) [NC data] is created.

If NC data with the same name already exist, confirmation dialog appears. If it can be overwritten, click [Yes].



(15) Click the [OK] button.

(16) Close the [Machining process].



(17) Confirm the created [NC data].

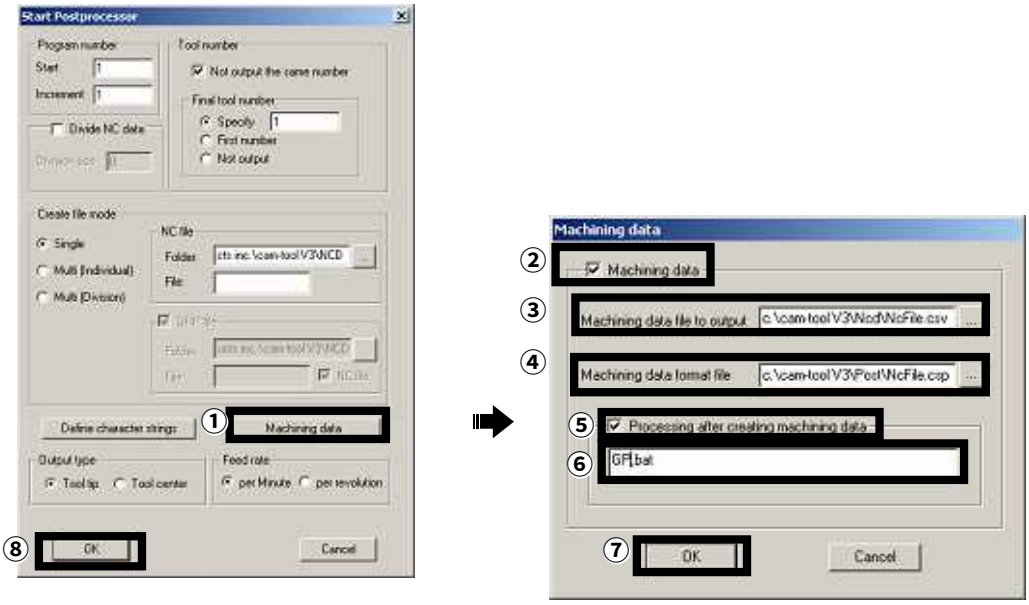
<<Chapter2 Operating flow of CAM processing>>

How to create process sheet

When you create NC data, you can create [Process sheet] using [Machining data]. Read [Machining data file(CSV file) that is output on Machining process to the file such as Excel with macro file.

* CSV(Comma separated value) file is a comma-deliminated text file.

(1) To create [Process sheet], click the [Machining data] button on [Start Postprocessor] dialog.
(Click the [Machining data] button with the procedure written in [P.65 ⑩], and set files as below.)



(2) Set [Machining data] ON.

(3) Specify [Machining data file]. (Refer to the following note)

(4) Specify [Machining data format file]. (Refer to the following note)

(5) Set [Processing after creating machining data] ON

(6) Input a [Macro file]. (Refer to the following note)

(7) Click the [OK] button on [Machining data] window.

(8) Click the [OK] button on [Start Postprocessor] window.

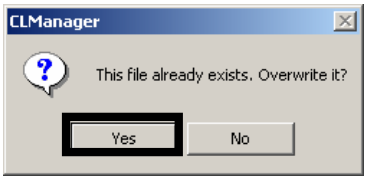
As a sample file, the following [Machining data format file], [Machining data file], [Macro file] are available for [Process sheet]. Specify the following settings for creating [Process sheet], and confirm.
To use a sample file, open Excel, and read [Machining data file] of CSV format.

Machining data format file : C: /cam-tool/ cam-tool Conf /Post /NcFile.csv

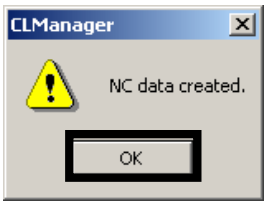
Machining data file : C: /cam-tool/ cam-tool Conf /Post /NcFile.csp

Macro file : GP.bat

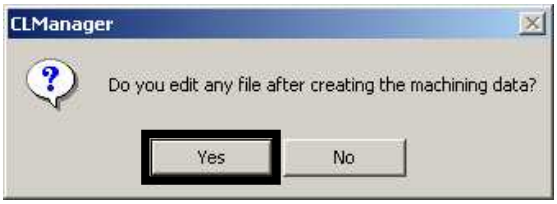
(9) [This file already exists. Overwrite it?] >> [Yes]



(10) [NC data created] >> [OK]



(11) [Do you edit any file after creating the machining data?] >> [Yes]



(12) When Excel is opened, click [Enable Macros].

(13) [Would you like to create the process list?] >> [Yes]

(14) Calculation process list is created. (If necessary, save the file with a name.)

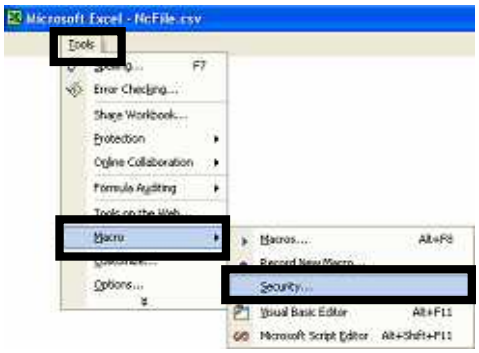
GP-PROCESS LIST

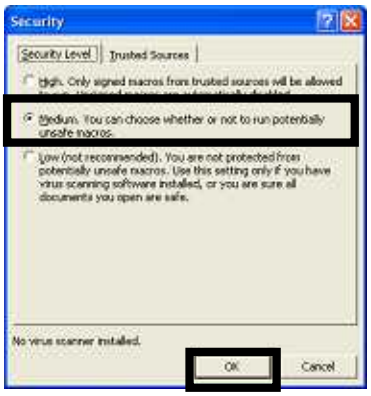
Product No.	Product name	Parts name	Drawing No.	Create by	Mochi@GP	Approve	Confirm	Create
MR8767	Sample-Unit	Sample-Model	15380107-5	Date at	18/APR/2006			
Machine name	Material	Base Position	Tool origin	Remarks	This is a sample List.			
GP-Machine-A	S55C	0,0,0	0,0,100					

No.	NC file name	Cutting mode	Tool name	Tool type	Tool Dia.	T	H	S	F	W	Z-min	Z-max	Cutting dist.	Cutting time	Data size
1	RGH-01	Z-level Rough Cutting	TM_GP_10RB	BALL	10.0	1	1	3200	4000	–	-30.0	100.0	45559.62	7:01:29	151,089
2	FIN-01	Z-level Finishing	TM_NA_4RB	BALL	8.0	2	2	3800	3200	–	-30.3	100.0	21834.51	3:19:08	262,064
3	FIN-02	Z-level Finishing	TM_AC_3RB	BALL	6.0	3	3	4800	2800	–	-32.1	100.0	55238.02	8:23:32	693,060
													Total	122657.15	18:44:07	1,106,133

If macro file is not started, set the security level of [Excel] to [Medium].

[Menu bar] >> [Tool] >> [Macro] >> [Security] >> [Security level: Medium



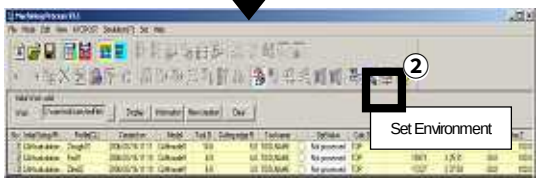


<<Exercise>>

Create [NC data], [Process sheet] on [Machining process].

■ Before start Machining process,

① Start [Machining process].
Click the [Machining process] button.

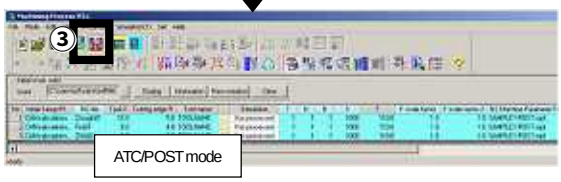


② Set environment in the [Machining process].
Click the [Set Environment] button.
Set as in the description in [page 61].

③ Close [Machining process].
* Once you set environment one time, the settings will be applied to the machining process when you open next time.

■ Create [NC data], [Process sheet]

① Start [Machining process].
Click the [Machining process] button.



② [Machining process] is started, and profiles, which calculations have ended normally, are registered in the calculation process list.

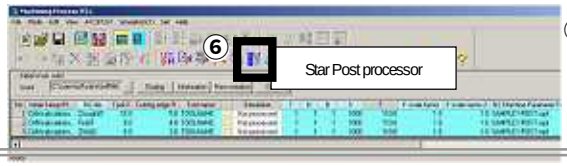
③ Click the [ATC/POST] button.
Items displayed on the [Machining process] are changed.

④ Confirm that [Machine parameter file] specified in the [Set Environment], is set.

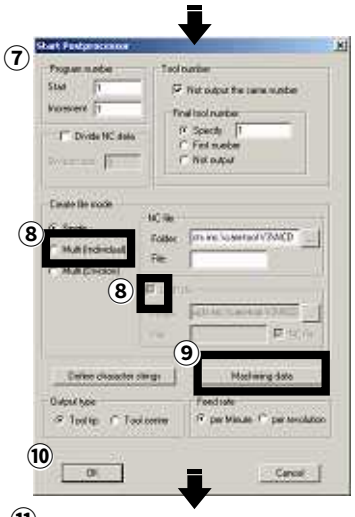
⑤ Select processes to create [NC data].

Ex.) Click [Number] to select all processes.

* To select sequential processes, press [Shift key] when selecting.



⑥ Create [NC data].
Click the [Start post processor] button.



⑦ [Start Post processor] dialog is displayed.

⑧ Select [Create File mode].

Ex.) Multi [Individual]/ GRP file : OFF

⑨ Set environment for [Machining data].

Refer to the procedure in [page 66 ②-⑦].

⑩ Click the [OK] button on the [Start Post processor] dialog.

⑪ This file already exists. Overwrite it?

>> [OK]

⑫ [NC data created] >> [OK]

⑬ Do you edit any file after creating the machining data?

>> [OK]

⑭ Excel is started. >> [Enable Macro]

⑮ Would you like to create the process list? >> [OK]

⑯ Process list is created.

⑰ Confirm the created [NC data], [Process sheet].

Double-click to open the [NC data] created in the [Output folder].

D:/TOOLS /NCD /Zrough01.NCD

D:/TOOLS /NCD /Fini01.NCD

D:/TOOLS /NCD /Fini02.NCD

Ex.) Created NC data

```
%
G00G40G49G80
G91G28Z0.
G28X0.Y0.
T01
M06
S1000M03
G91
G17G00X-80.Y75.
G18Z-95.
M08
G01Z-5.F80
G17X5.F110
```

Chapter 3

Rough Cutting

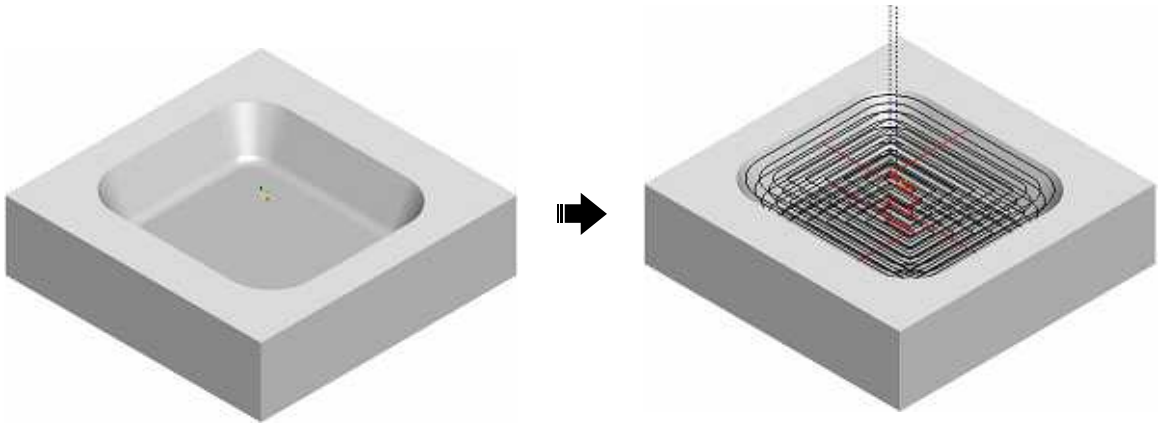
－ Z-level Rough Cutting －

Overview of Z-level Rough Cutting

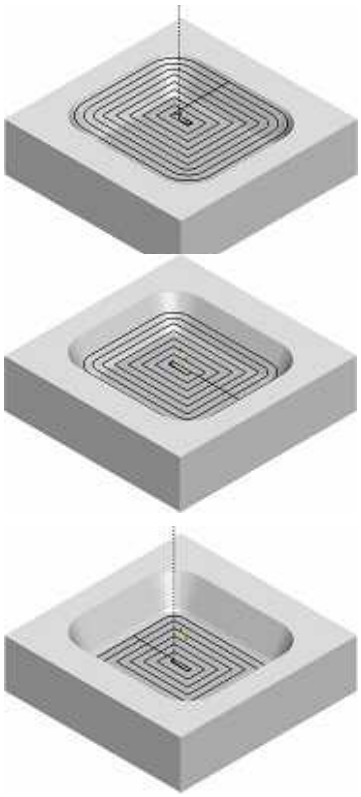
This is the function that tool travels around in Z direction with the fixed tool center (Z-level traveling) to perform rough cutting.
This strategy always performs down cut cutting, and output NC data consists of G01 with simultaneous two axes.
（Simultaneous three axes for slant cutting）
Cavity area and core area are automatically recognized by the system, and a shape with both areas can also be cut.

This function is used on the premise that slant cutting is always performed by Ball end mill. Therefore, if you use Flat end mill or Radius end mill, select the one with end cutting edge.

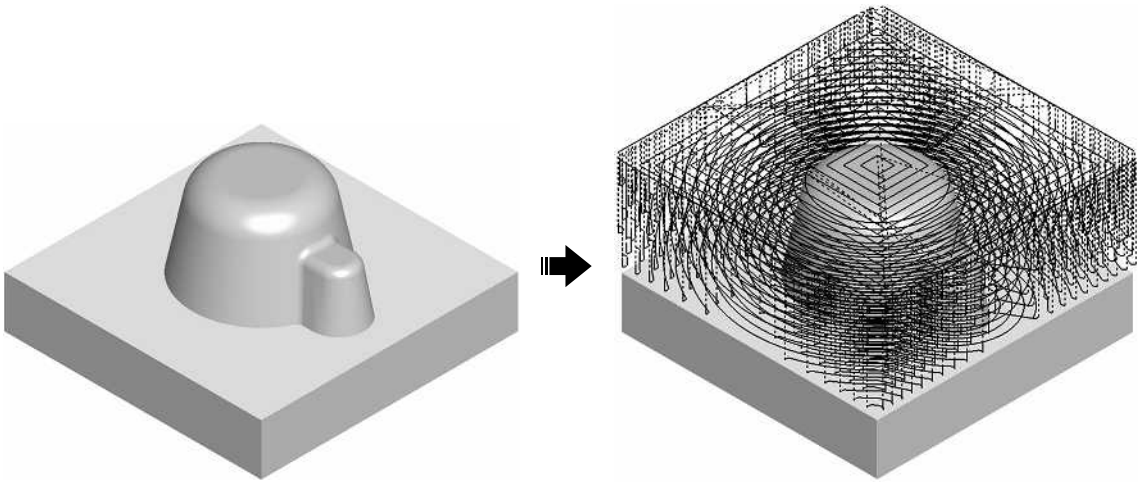
■ Cutter path for cavity shape



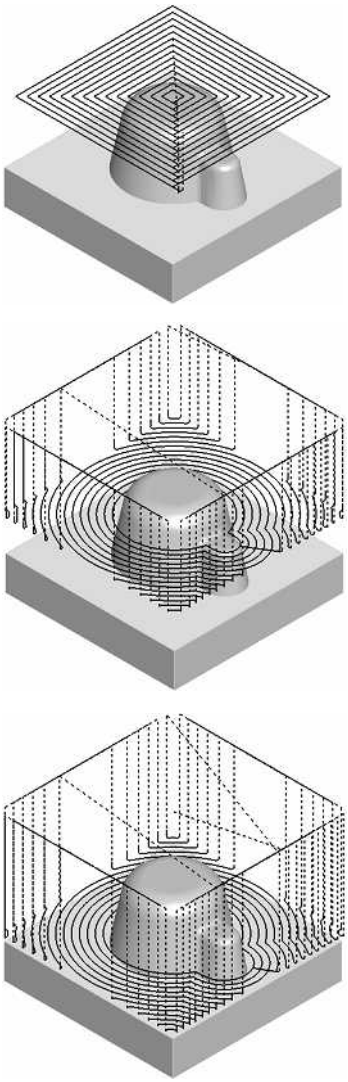
(Z-level cutter path at each Z value)



■ Cutter path for core shape



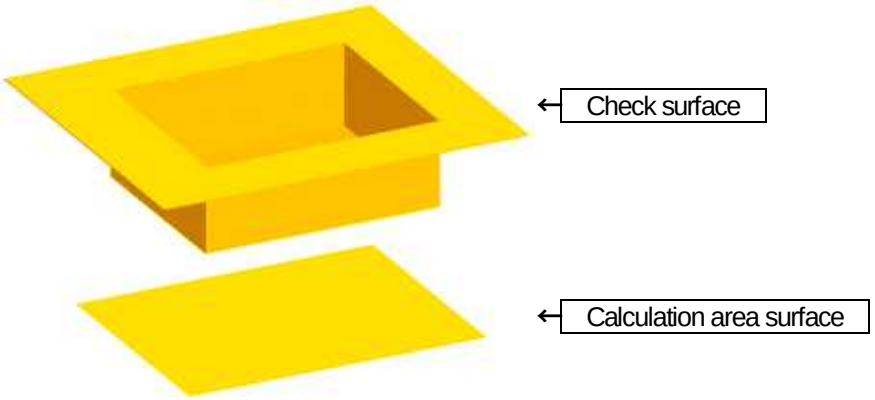
(Z-level cutter path at each Z value)



What are needed for model shape?（Precautions/Restrictions）

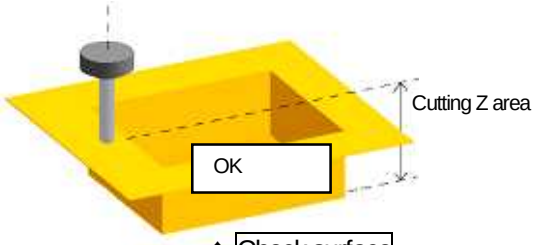
To cut cavity shape, the followings need to be prepared as model shape.

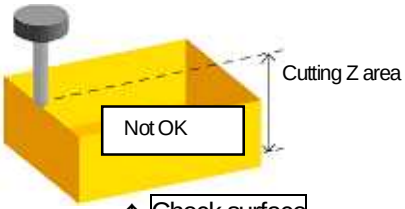
① Cutting shape ＋ Parting surface（Check surface）
② Surface that defines calculation area in plane direction （Calculation area surface ）



Precautions for creating [Check surface]

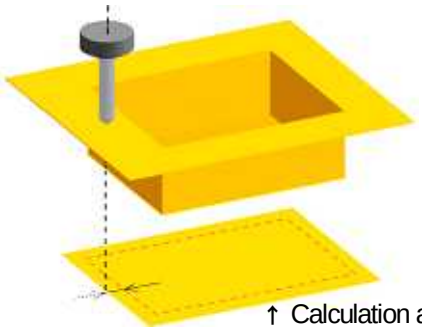
Create a [Parting surface] at a starting position in cutting Z area, and specify both parting surface and the cutting shape as [Check surface]. If [Parting surface] does not exist in [Check surface], cutter path will be output outside of the shape. And if a hole or a gap is found in [Parting surface], necessary cutter path may not be output normally or invalid cutter path may be output.

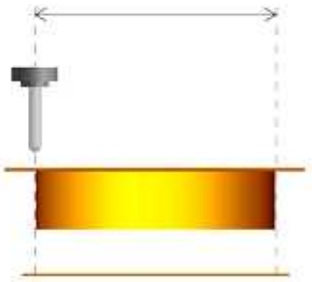




Precautions for creating [Calculation area surface]

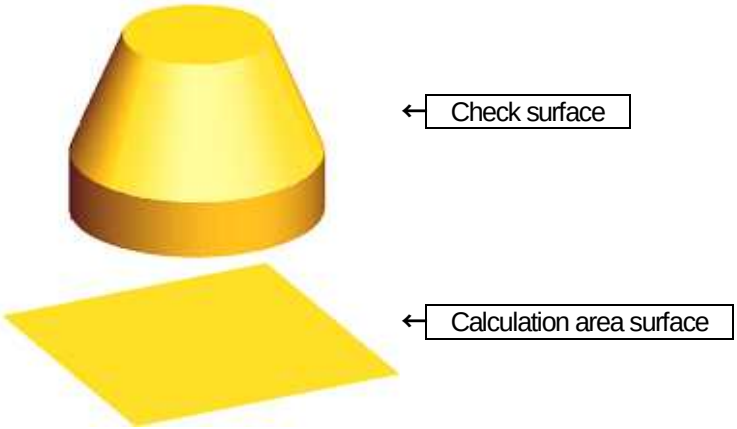
Create [Calculation area surface] , so as to be larger than [cutting shape] and smaller than [Parting surface], viewing from the top.





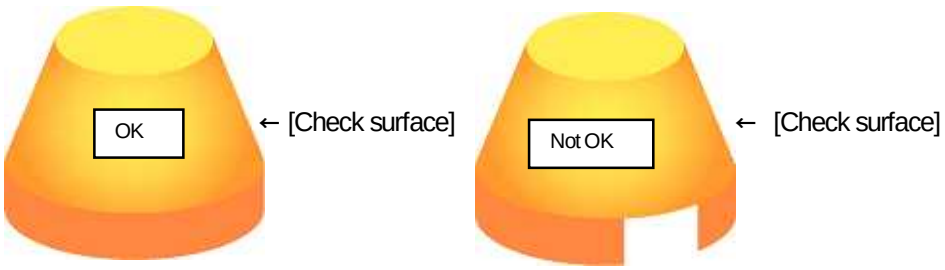
To cut cavity shape, the followings need to be prepared as model shape.

① Cutting shape（Check surface）
② Surface that defines calculation area in plane direction Calculation area surface



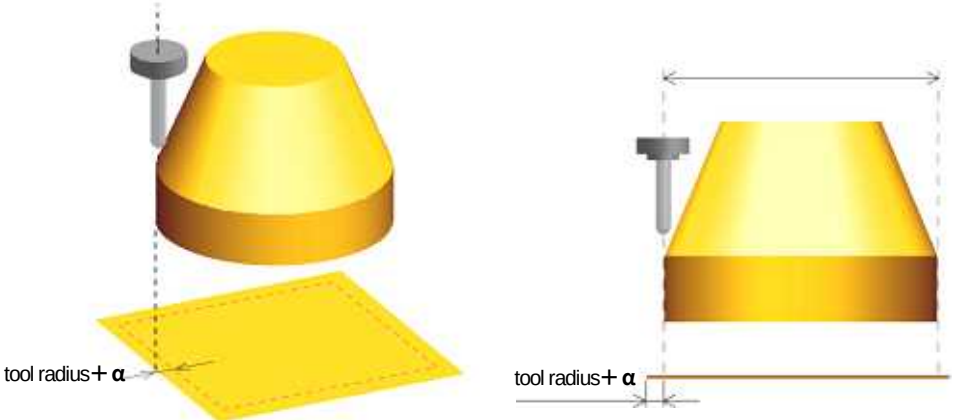
Precautions for creating [Check surface]

Create cutting shape with closed shape, and specify it as [Check surface].
（Creating bottom surface is not required.）



Precautions for creating [Calculation area surface]

Create [Calculation area surface], so as to be [tool radius + a] larger than cutting shape, viewing from the top.
If the size of [Calculation area surface] is exactly the same as the one of cutting shape, tool center can move only up to the contour of the cutting shape.

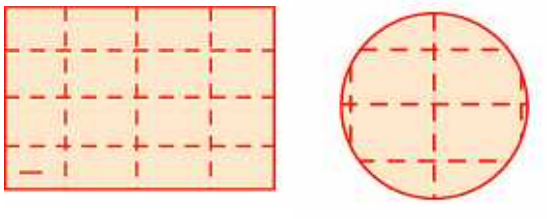


■ Restrictions for [Calculation area surface]（For both cavity shape and core shape）

① Create [Calculation area surface] whose mesh (UV parameter line) cross in a direction perpendicular to each other.

Available

- Original surface whose mesh (UV parameter line) cross in a direction perpendicular to each other
- Trim surface whose mesh (UV parameter line) cross in a direction perpendicular to each other



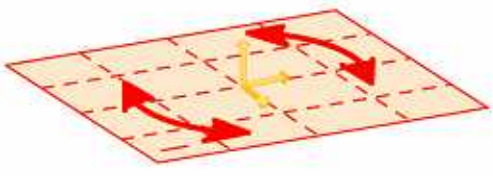
Not available

- Original surface whose mesh (UV parameter line) do not cross in a direction perpendicular to each other

② Create [Calculation area surface] that is parallel to [XY plane of work coordinate(Work plane)].

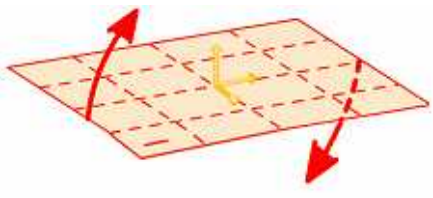
Available

• Tilt in XY direction (parallel to [XY plane of work coordinate(Work plane)]



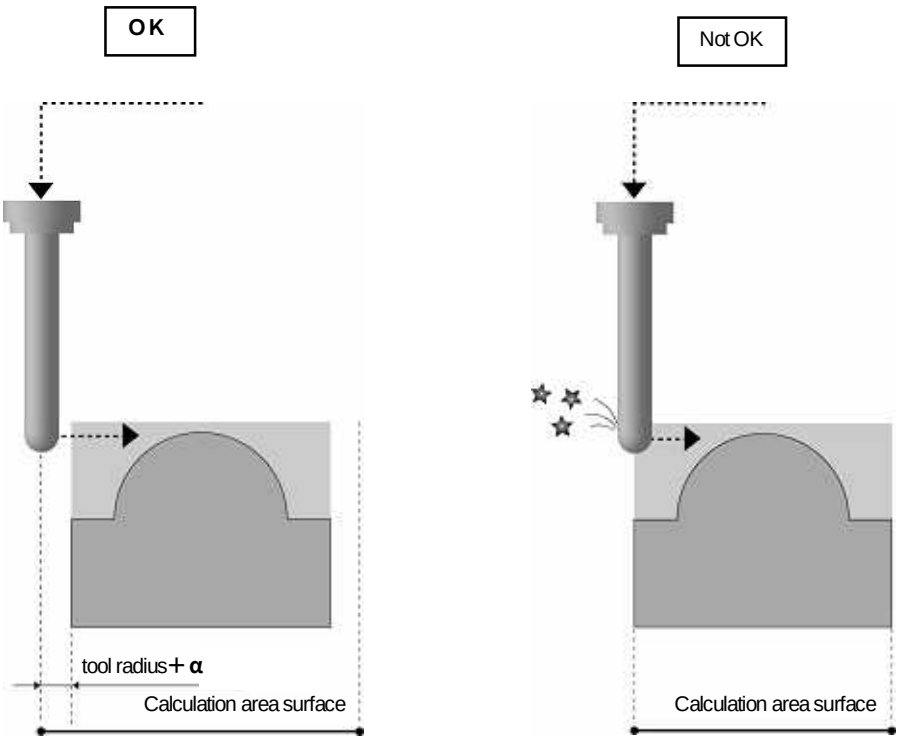
Not available

• Rotate in the direction other than XY direction



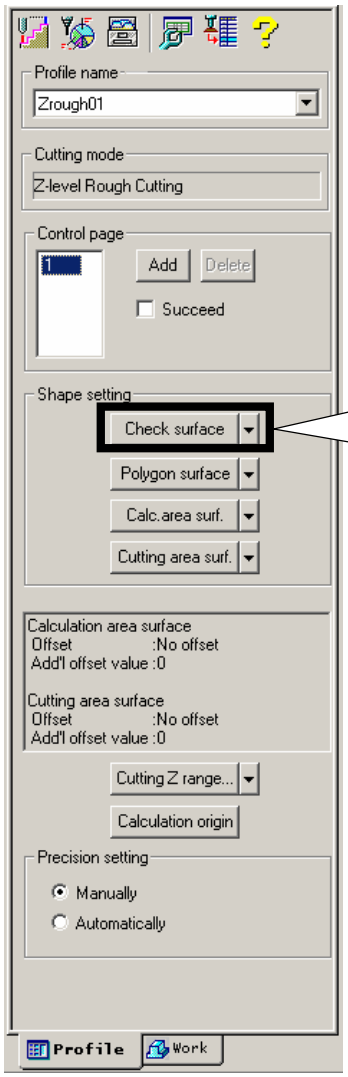
Work(Material)/Calculation area surface

For core shape, create [Calculation area surface] so as to be [tool radius + a] larger than work (material). If the size of [Calculation area surface] is exactly the same as the one of work (material), and calculation is executed, tool may collide with work in Z direction. However, performing approach in plane direction prevents tool from the collision.



How to set cutting model shape → Control window

Set [Check surface] in [Shape setting] on the Control Window.



① Click [Check surface] button.

② Specify surface to be cut.

* For cavity shape, specify [Parting surface] as well.

③ Press the center mouse button to determine the setting.

To set [Setting shape] again, press▼ button on the right of [Check surface] button, and click [Clear Part] or [Clear All]. When you execute [Clear All], confirmation dialog appears. Click [OK] button to execute the command.

≪Exercise≫

Set shape using the model file of ashtray that you created in the [Text Book for Intensive Course/CAD].

1. Open a model file. CD-ROM /Intensive Course/CAMmodel2.gmd
   Set [Layer1] for active layer

2. Save the file. <Destination folder/File name>
   D:/cam-tool /cam-toolFile /Exercise2 / CAMmodel2.gmd

3. Create a new profile
   ① Profile name: Zrough01
   ② Cutting strategy: Z-level Rough Cutting
   ③ Initial setting file: (blank)
   ④ Continuous creation : OFF
   ⑤ Add to calculation process : OFF >> [Finish] button

4. Create a new calculation process list <Destination folder/File name>
   D:/cam-tool /cam-toolFile /Exercise2 / CAMcalculation2

5. Add the profile to the calculation process list

6. Set [Check surface] on a profile.

① Click [Check surface] button.

② Specify surface for cutting shape (Specify by area : In)

③ Press the center mouse button to determine the specifying.

* Confirm the setting with [CAM mode].

Function to operate [Shape setting] efficiently

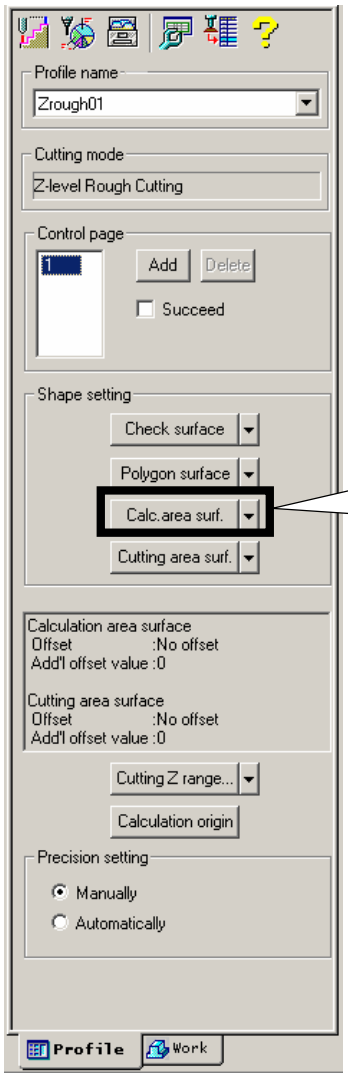
When you click [Check surface]/[Calculation area surface] button, [Set check surface] dialog appears.

- Specify layer : In case that you control shapes by layer, you can set shape by entering the layer number.
- Set all pages : In case that you create plural pages, you can set shapes in all of pages.

How to set cutting area in plane direction

→ Control window

Set [Calculation area surface] in shape setting on the Control window.



① Click [Calculation area surface] button.

② Specify a surface for defining an area that cutter paths are output.

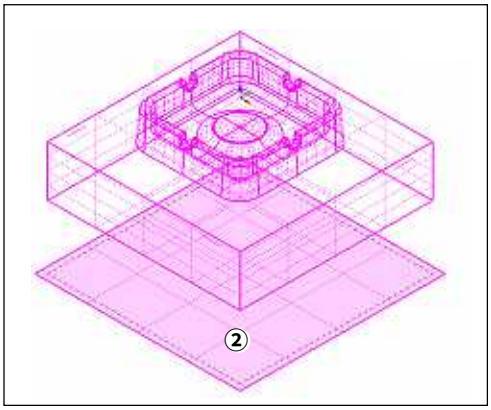
* Only a single surface can be specified as [Calculation area surface].

To set [Setting shape] again, press▼ button on the right of [Calculation area surface] button, and click [Clear] . When you execute [Clear], confirmation dialog appears. Click [OK] button to execute the command.

≪Exercise≫

Next, set [Calculation area surface].

1. Set [Calculation area surface] on the created profile.



① Click [Calculation area surface] button.

② Specify a surface for defining area that cutter path are output.
* Only a single surface can be specified as [Calculation area surface].

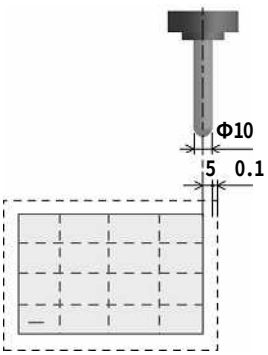
2. When shape setting is completed, confirm with [CAM mode] function.

Convenient function to adjust size of calculation area surface

When you click [Calculation area surface] button, [Set calculation area surface] dialog appears.
Size of surface, which you specified as [Calculation area surface], can be changed by the value you specify on this dialog.
With this function, you do not need to recreate calculation area surface every time you change tool radius.

- Offset method：Select one from [No offset], [Offset outside], [Offset inside].
- Additional offset value：Adjust area with the entered [Additional offset value] by the specified [Offset method],.

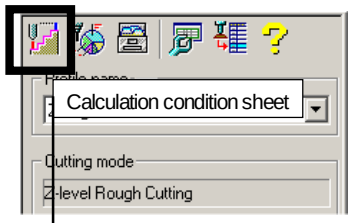
Ex.) [Tool diameter=D10]
- Offset method：Offset outside
- Additional offset value：0.1



>> [5.1mm] will be added to the set calculation area surface

How to set cutting area in Z direction → Calculation condition sheet : Machining

Set calculation conditions of the selected profile on the calculation condition sheet.

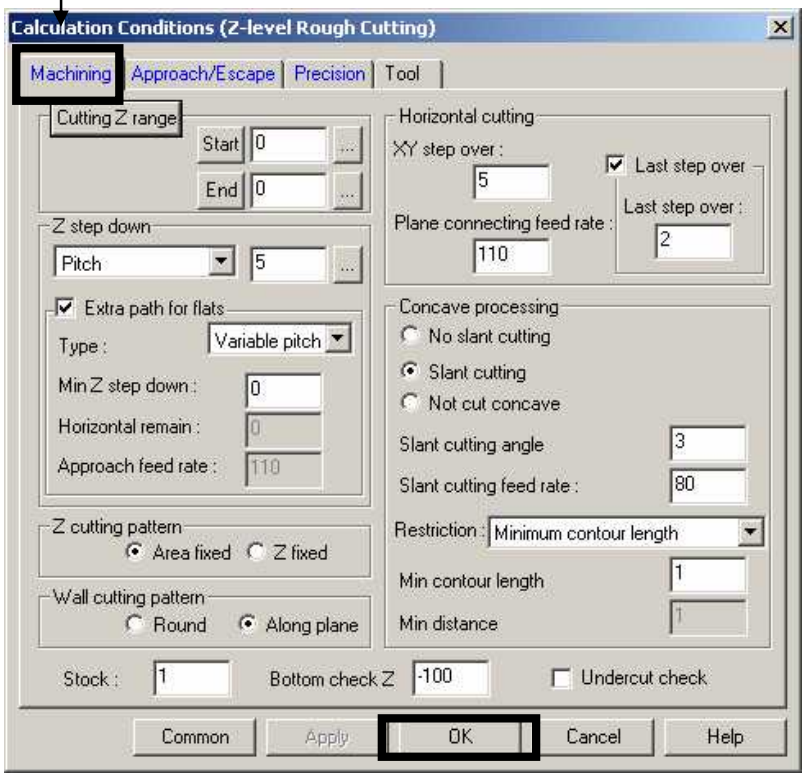


① Click [calculation condition sheet] button.

② [Calculation condition] dialog appears.

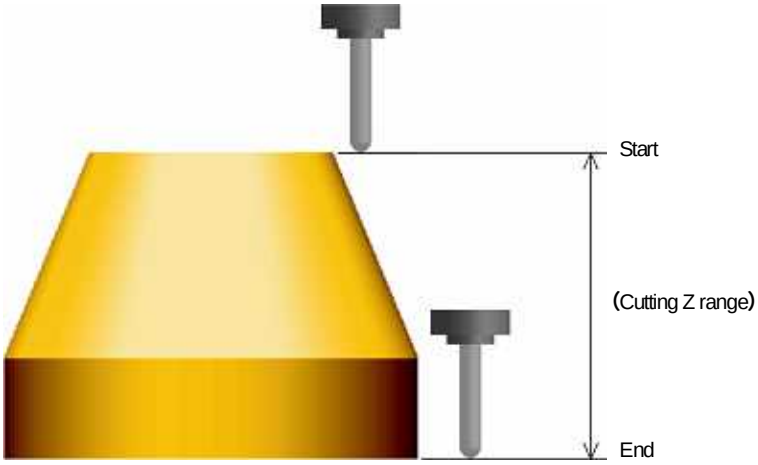
③ Click tab of the item that you wish to edit.

④ Click [OK] button after setting conditions.

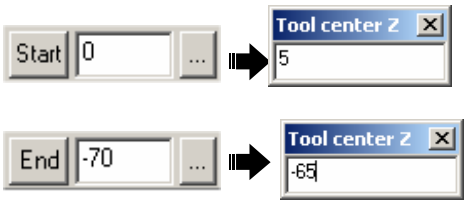


On the calculation condition sheet, calculation conditions are categorized by [Machining], [Approach/Escape], [Precision] and [Tool]. Click tab to set each conditions.

Set [Cutting Z range] by entering [Start], [End] position of tool end.

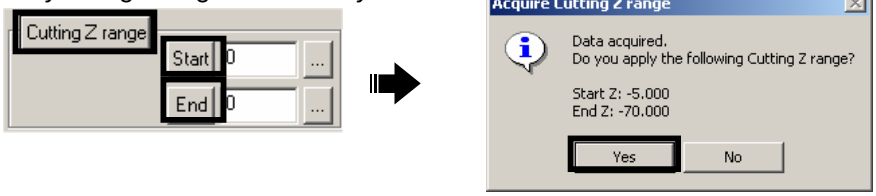


Specify tool center Z



Basically enter tool end Z value.
If you wish to specify tool center Z value, click the button on the right of [input numerical value] area.

Specify cutting Z range automatically



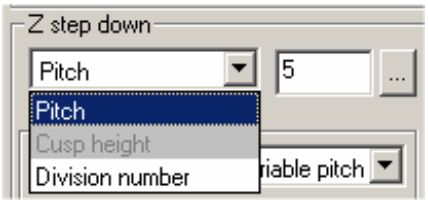
Cutting Z range (Start, End) is automatically specified with the information of the shape that is set as [Check surface].
(When automatically specified, the system truncates a number to three decimal places)

* When [Check surface] has not set yet, the error message appears.

How to set Z step down

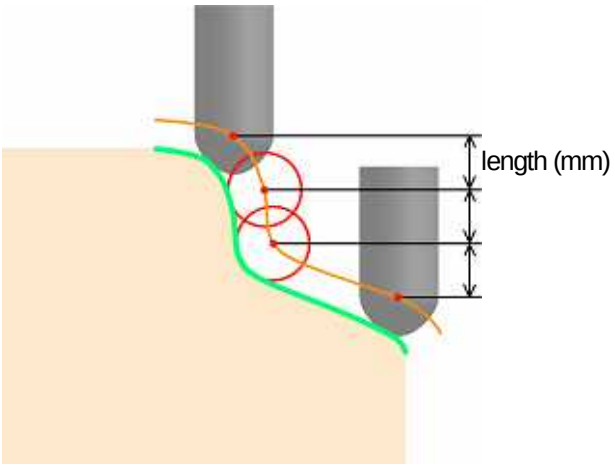
→ Calculation condition sheet : Machining

Specify type of [Z step down], and enter value.



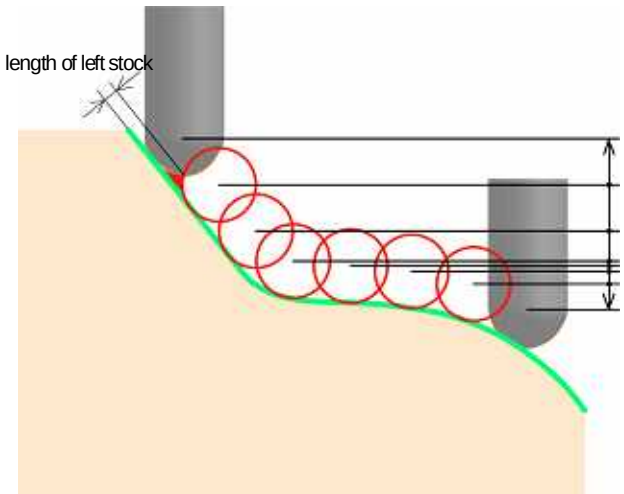
• Pitch

Specify [length (mm)] that tool cuts in Z direction to determine Z step down.



• Cusp height

Specify [height of material (mm)] to determine Z step down so as to keep the length of stock that is left after tool cuts previous Z area.



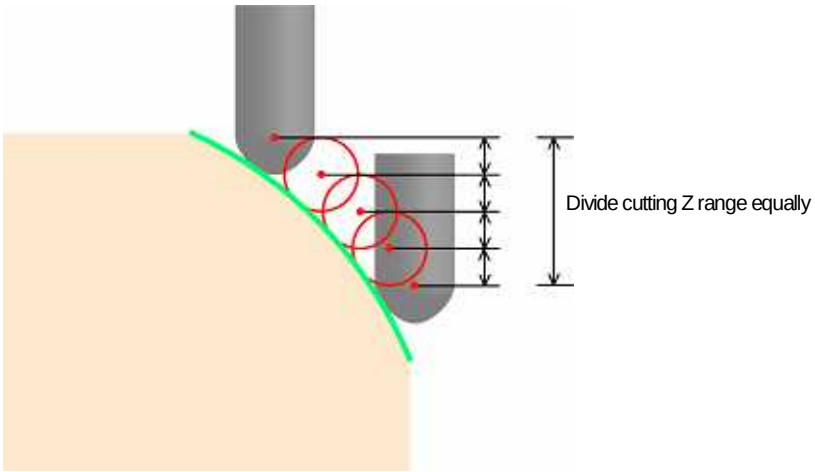
• Pitch between each Z value varies.

• Cusp height is determined based on the nearest horizontal area of the target surface

• Do not use this function when using a flat tool.

• Division number

Z step down is determined by dividing [Cutting Z range], which is specified by [Start], [End] value, equally.



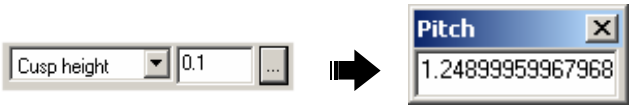
Effective method for using Ball end mill/Radius end mill.

• When specifying [Pitch]

When you enter value for [Pitch], value for cusp height is automatically calculated. Click the button on the right of numerical value input area to confirm the calculated value.

• When specifying [Cusp height]

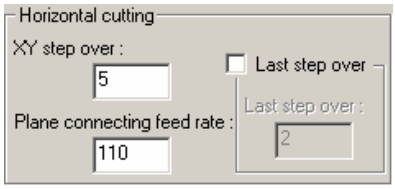
When you enter value for [Cusp height], value for [Pitch] is automatically calculated. Click the button on the right of numerical value input area to confirm the calculated value.

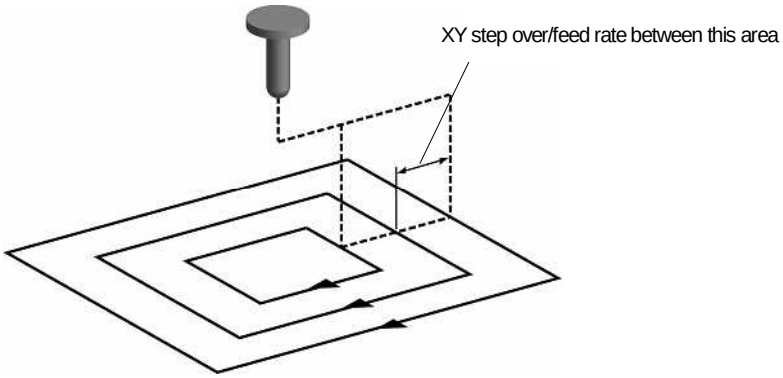


How to set XY step over

→ Calculation condition sheet: Machining

In Z-level Rough Cutting, the system calculates cutter path that travels around cutting shape, and offsets the calculated cutter path in plane direction.
For [XY step over], enter [Pitch (mm)] in plane direction.
And set [Feed rate] for connecting movement between pitch.



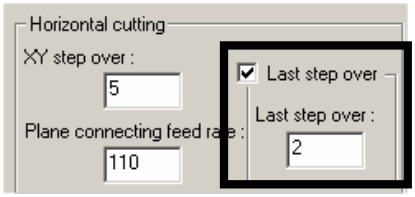


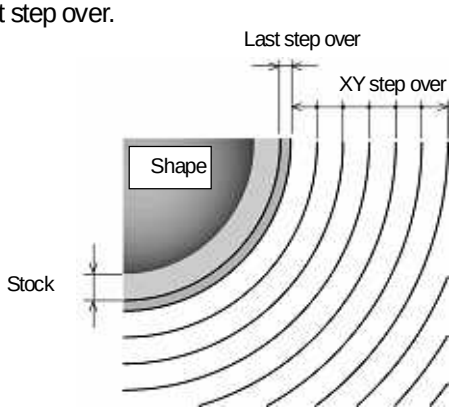
For [XY step over], enter larger value than [0]. There is no limit on the maximum value.
Please note that stock may be left depending on cutting shape.

How do I specify XY step over that is closest to the shape?

[Last step over] can be specified for each cutting Z value.

When [0] is entered, tool travels twice at the last step over.





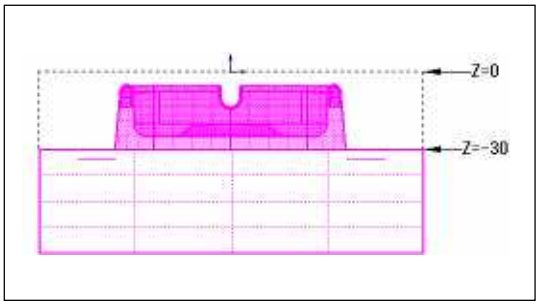
≪Exercise≫

Set [Cutting Z range]/[Z step down]/[XY step over] and execute calculation.

1. Specify profile　　Profile: Zrough01

2. Click [Calculation Conditions/Machining]

3. Set [Calculation conditions (1)-(4)]



(1) Cutting Z range：Start[0]，End [-30]

(2) Z step down：Pitch [3(mm)]

(3) XY step over：5(mm)

(4) Last step over：OFF

6. Set conditions, and click [OK] button.

7. Click [Start] button in Control Calculation.

8. When [Normal] is displayed, calculation has ended and CL data has been created.

9. Right-click on [Normal], and select [Show CL by segment]. And confirm [CL data].

10. After you confirm [CL data], click [Clear CL] button to clear the displayed CL.

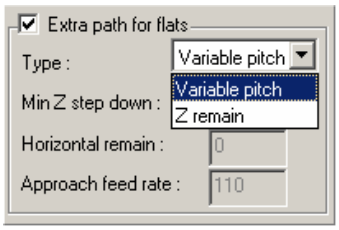
What is CL data?

It is [Cutter location (tool trace) data].
In this textbook, it is described with terms such as [CL data], [Cutter path], [Path].

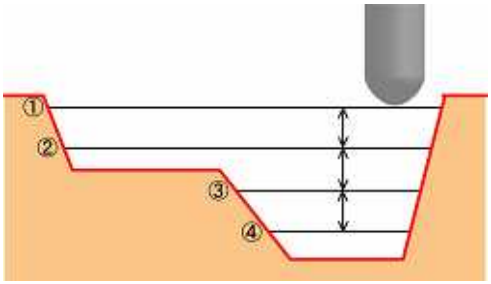
How to add cutter path on flat area

→ Calculation condition sheet : Machining

[Flat area] of cutting shape (Check surface) is automatically detected by the system, and Z-level cutter paths are created for the detected area.

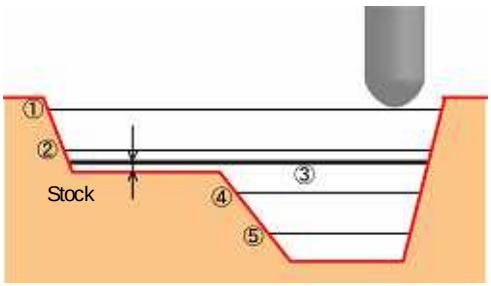


When [Extra path for flats] is not used, only Z-level cutter path that is specified by [Z step down] is output.



• Extra path for flats : Variable pitch

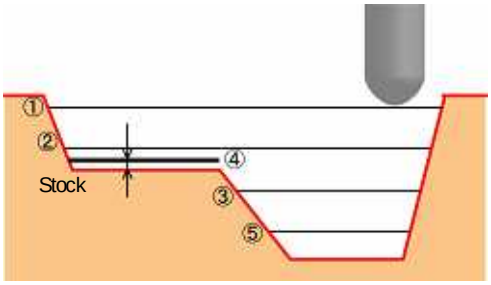
In addition to paths generated by the specified [Z step down], extra paths are created at Z level of flat area.



* If [Stock] is specified, cutter paths are created at [Z level of flat area + stock].

• Extra path for flats : Z remain

In addition to paths generated by the specified [Z step down], extra paths are created only at Z level of flat area, where stock has been left.



* If [Stock] is specified, cutter path are created at [Z level of flat area + stock].

* Approach feed rate

Specify feed rate for Arc approach added to cutter paths that are created by setting [Z remain].

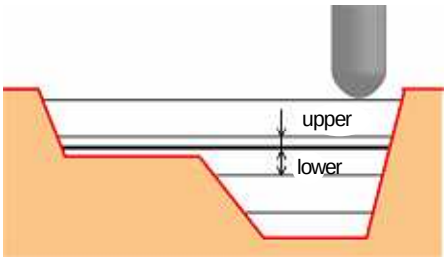
How do I control cutter paths that are created for flat area?

Cutter path that are created for flat area can be controlled by specifying numerical value.

• Min Z step down

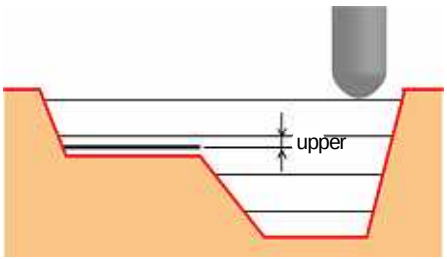
This function is effective when [Extra path for flats] is set ON.
No path is created for flat area, if the distance between the Z- level cutter path and extra path for flat area, is shorter than the specified [Min Z step down].

When [Variable pitch] is specified



No path is created for flat area, if the distance between the upper/lower Z- level cutter path and extra path for flat area, is shorter than the specified [Min Z step down].

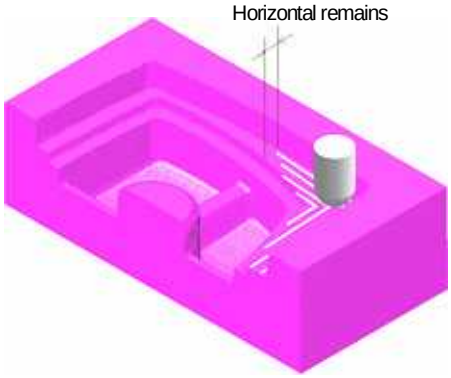
When [Z remain] is specified



No path is created for flat area, if the distance between the upper Z- level cutter path and extra path for flat area, is shorter than the specified [Min Z step down].

• Horizontal remains

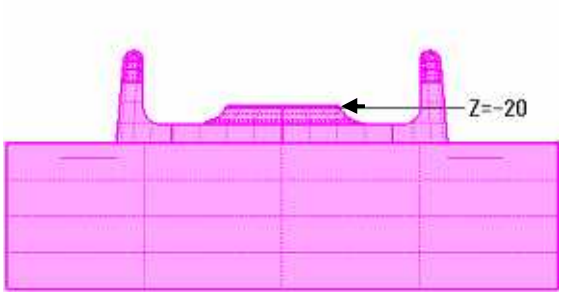
This function is effective when [Extra path for flats] is set ON, and [Z remain] is specified.
Specify minimum width of area where cutter path is output. No path is created at area that is narrower than the specified value.



≪Exercise≫

In [CAMmodel2.gmd], there is a flat area at [Z = -20].

Change the setting for [Extra path for flats], and execute calculation. Confirm how differently cutter paths are created depending on the setting.

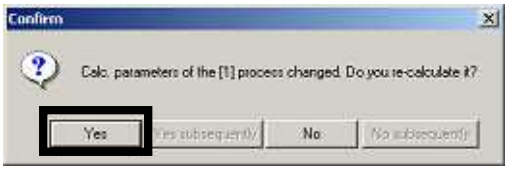


＜Extra path for flats：OFF＞

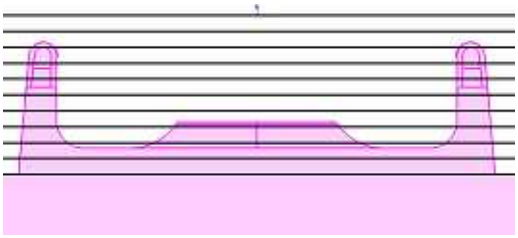
1. Set the conditions (1)-(4) in [Calculation conditions sheet/Machining], and click [OK] button.

   (1) Cutting Z range：Start[0], [End] [-30]
   (2) Z step down：Pitch [3(mm)]
   (3) Extra path for flats：OFF
   (4) Stock：1(mm)

2. Click [Start] button in Control Calculation.
   If the calculation of the specified condition sheet has already been executed, confirmation dialog appears.
   Click [OK] button.



3. Right-click on [Normal], and select [Show CL by segment], and confirm the created CL data.
4. Confirm that no path has been created at [-20+1mm（stock）].

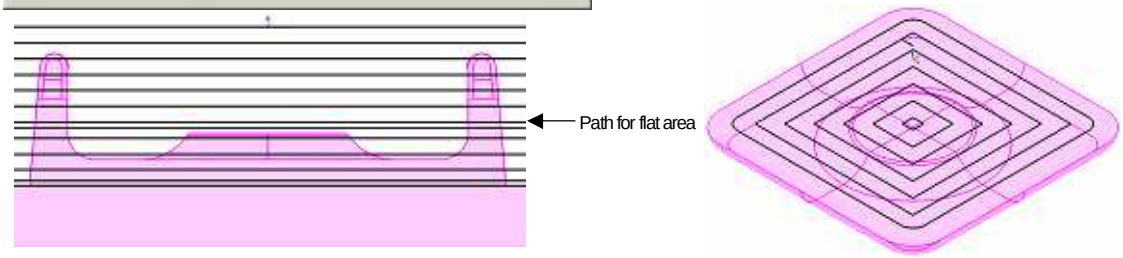


← Only paths of specified [Z step down (3mm)] have been created.

5. After you confirm the CL, click the [Clear CL] button to clear displayed CL.

＜Extra path for flats：Variable pitch＞

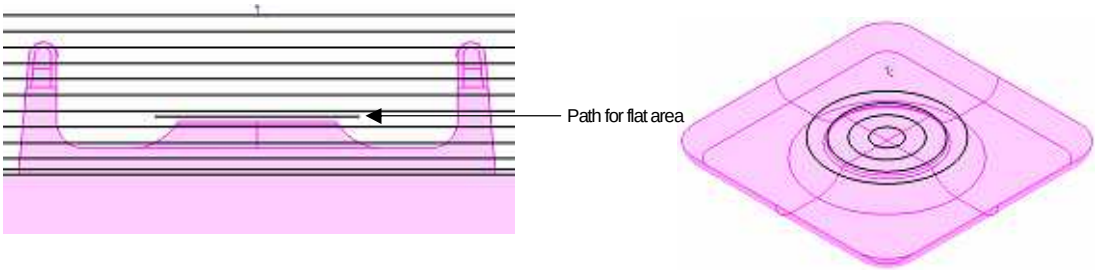
1. Change the setting (5) in [Calculation condition sheet/Machining], click [OK] button.
   (5) Extra path for flats：ON　[Variable pitch]
2. Click [Start] button in Control Calculation.
3. Right-click on [Normal], and select [Show CL by segment], and confirm the created CL data.
4. Confirm that a path has been created at [-20+1mm（stock）].



5. After you confirm the CL, click the [Clear CL] button to clear the displayed CL.

＜Extra path for flats：Z remain＞

1. Change the setting (6) in [Calculation condition sheet/Machining], click [OK] button.
   (6) Extra path for flats：ON　[Z remain]
2. Click [Start] button in Control Calculation.
3. Right-click on [Normal], and select [Show CL by segment], and confirm the created CL data.
4. Confirm that a path has been created at [-20+1mm（stock）].

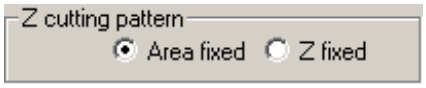


5. After you confirm the CL, click the [Clear CL] button to clear the displayed CL.

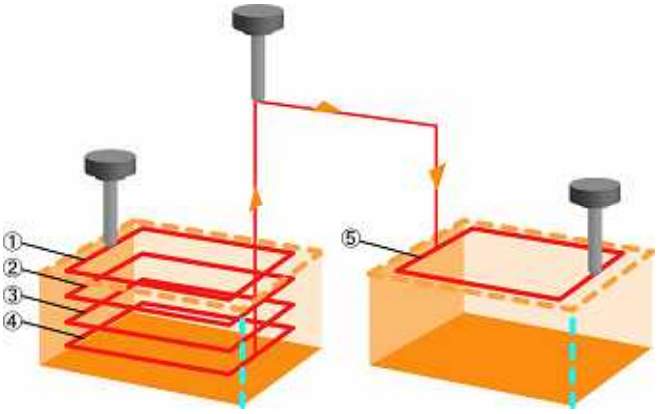
How to set cutting order if plural concave areas are included in a shape

→ Calculation Condition Sheet:/Machining

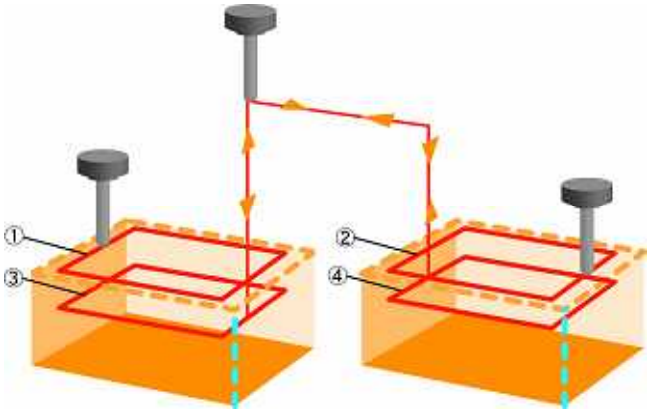
Select [cutting order] if plural concave areas are included in a shape.



• Area fixed: After cutting a concave area is completed, tools moves to the next area.



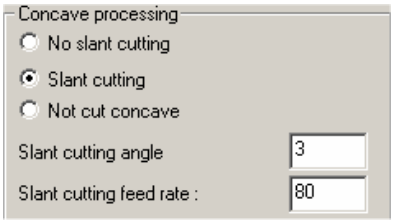
• Z fixed: Cuts the same Z level of all of concave shapes first, and then tool returns to the first concave area and cuts the next Z level.



This function is basically invalid for a core shape. However, if cutting area of a core shape is divided into plural areas, area cutting is performed with [Area fixed] ON.

How to set Approach for concave shape → Calculation Condition Sheet: /Machining

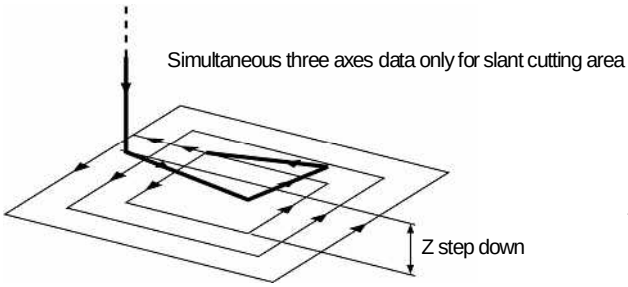
Specify cutting method of concave area of cavity/core shape.

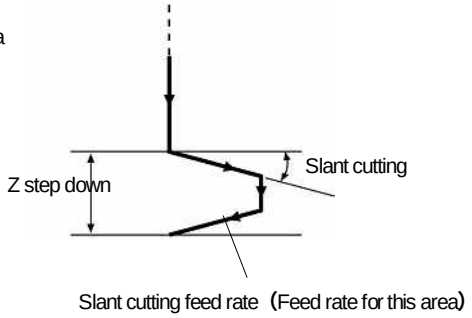


• No slant cutting: Cutter paths are created on the premise that a hole has already been created. The hole position is determined by the system. You can check it in Machining process (ATC-Post mode - Detailed Information)

- Slant cutting: When tool starts cutting work, it moves to work with an angle.
- Slant cutting angle： Angle from work plane << [0.0001° ≦ Slant angle ≦ 45°]

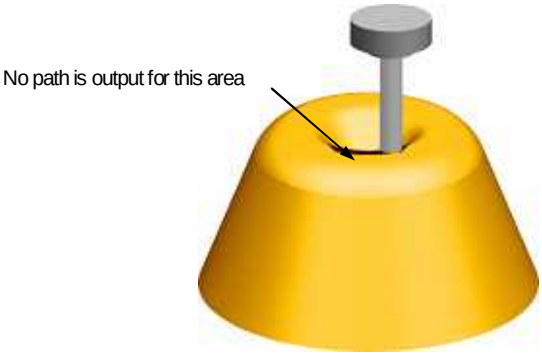
Slant cutting feed rate： Feed rate for slant cutting area





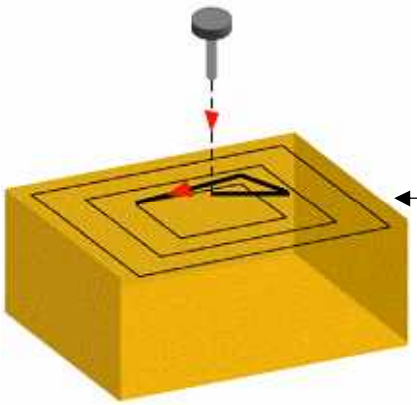
• Slant cutting height = [Start Z value + Z step down]
Specify higher Z value than the value above for [Clearance height]

• Not cut concave: When a concave area, which tool cannot reach outside of the area, exists in a core shape, no path is output for that area. It prevents tool without end cutting edge, from colliding with work.



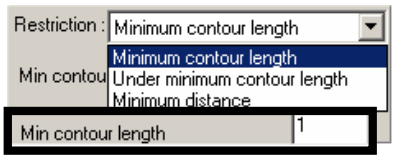
How to control slant cutting that may cause trouble

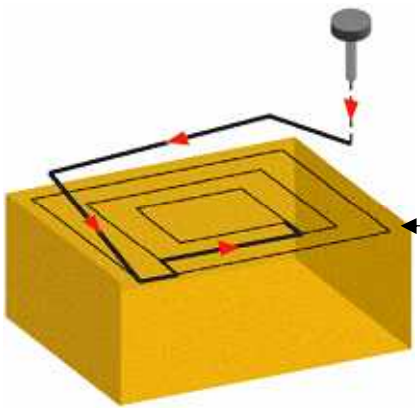
Basically, tool cuts along the [most inner contour]. However, when the length of the [most inner contour] is short, slant cutting can be controlled by the system.



Basically, tool cuts along [most inner contour].

When you execute slant cutting, choose either [Minimum contour length] or [Under minimum contour length] and enter value for [Min. contour length] (shortest length of contour that slant cutting is performed).





Basically, slant cutting is performed along the most inner contour.

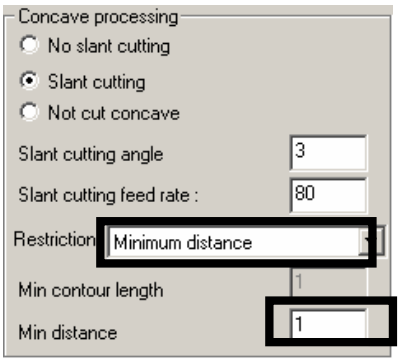
If the length of the most inner contour is shorter than the specified [Min. contour length], next inner contour is detected by the system. If longer contour than [Min. contour length] is detected, slant cutting is performed on the detected contour.

After slant cutting is performed, tool moves to the most inner contour and start cutting.

- [Minimum contour length]: If length of the most outer contour is shorter than the specified [Min. contour length], slant cutting is not performed and no path is output.
- [Under minimum contour length]：Even if the length of the most outer contour is shorter than the specified [Min. contour length], slant cutting is performed along the most outer contour.

Slant cutting for using a tool with round tip

Since tool with round tip has no end cutting edge, stock may be left because the length that tool moves in XY direction is short, or tool shank or holder may interfere work.
Therefore, if you use a tool with round tip, input [Inside diameter] for [Min distance]. If the length that tool moves in XY direction is shorter than the [inside diameter], no path is output at the Z level or at lower Z level.



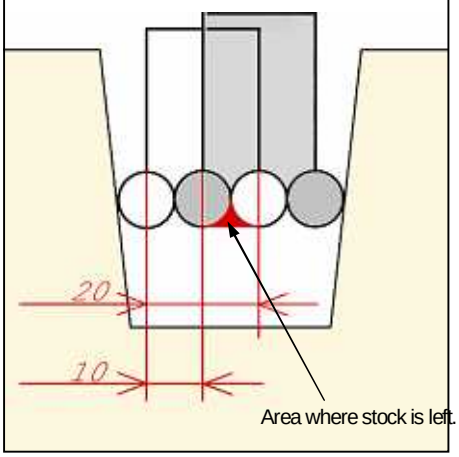
← Input inside diameter

< stock is left >

Inside diameter : <20>

Length in XY direction : <10>

Area between the arrows below is not cut.

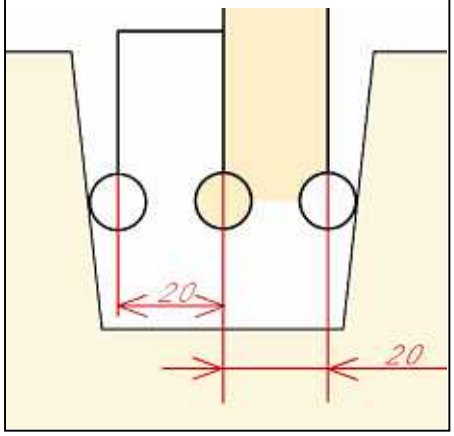


< stock is not left >

Inside diameter : <20>

Length in XY direction : <20 (inside diameter)>

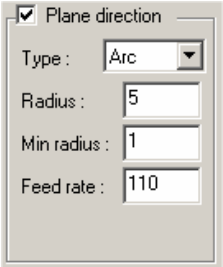
stock is not left.



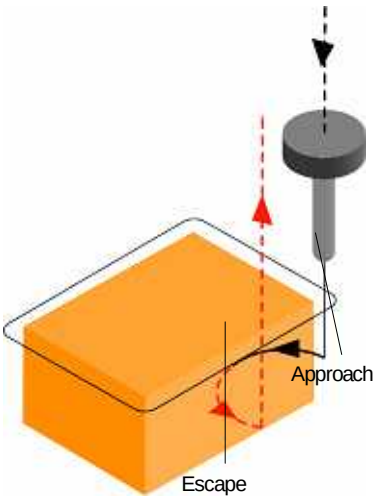
How to set Approach for core shape → Calculation Condition Sheet: Approach/Escape

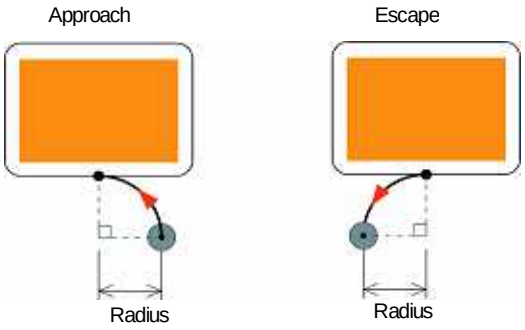
If Approach/Escape is performed from outside of a shape, [one-fourth arc] approach/escape is added.

* Although Approach/Escape are described as [Arc] here, all of arc data is automatically decomposed into G01 by the system.

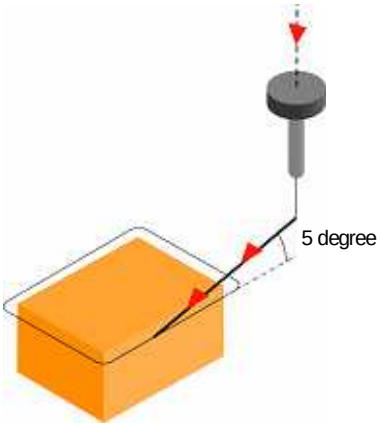


- Radius: Input radius value for Approach/Escape.
- Min radius：When the specified arc Approach/Escape interferes the check surface, the system makes arc radius smaller to the value of the specified [Min radius].
- Feed rate：Specify feed rate for Approach/Escape.





If Approach/Escape of [Min Radius] arc still interfere a shape, Approach with five degree is performed instead of arc Approach/Escape.



<<Exercise>>

Execute calculation without outputting [1/4 arc Approach/Escape].

1. Specify the profile.

   Profile：Zrough01

2. Open [Calculation condition sheet/Approach/Escape].

3. Set calculation condition (1).

   (1) Approach Plane direction: OFF

4. After you set condition, click [OK].

5. Click [Start] button in Control Calculation.

6. When [Normal] is displayed, select [Show CL by segment] to confirm [CL data].

   → Path of [one-fourth arc Approach/Escape] is not output.
   Approach: Red, Escape: Blue

7. After you confirm CL, click [Clear CL] button to clear the displayed CL.

Execute calculation with outputting [1/4 arc Approach/Escape].!

1. Open [Calculation condition sheet/Approach/Escape].

2. Set calculation condition (1)-(3).

   (1) Approach Plane direction：ON（Arc）
   (2) Radius：8
   (3) Min radius：1

3. After you set condition, click [OK].

4. Click [Start] button in Control Calculation.

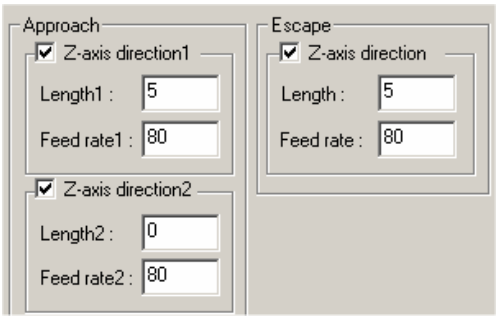
5. When [Normal] is displayed, select [Show CL by segment] to confirm [CL data].

   → Path of [one-fourth arc Approach/Escape] is output.
   Approach: Red, Escape: Blue

7. After you confirm CL, click [Clear CL] button to clear the displayed CL.

How to set Approach in Z direction → Calculation Condition Sheet: Approach/Escape

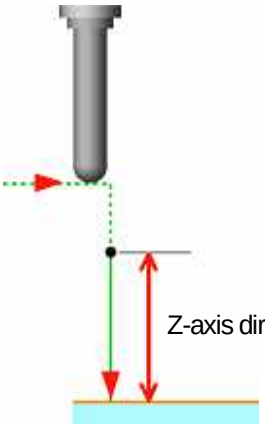
Set Approach/Escape in Z direction.（Common setting for both concave/convex shape）
If necessary, you can choose either [One step] or [Two steps] for Approach.
For Escape, you can choose either [Not set] or [One step].



Length: Input length that tool moves（Unit: mm）
Feed rate: Input feed rate for the length specified by [length].

• When Approach is set with a single step

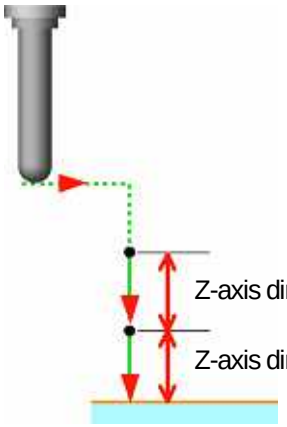
Z-axis direction1：ON
Z-axis direction2：OFF



Z-axis direction1：(Length1, Feed rate1)

• When Approach is set with two steps

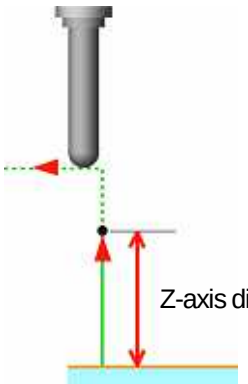
Z-axis direction1：ON
Z-axis direction2：ON



Z-axis direction1：(Length1, Feed rate1)
Z-axis direction2：(Length2, Feed rate2)

• When Escape is set

Z-axis direction：ON



Z-axis direction：(Length, Feed rate)

<<Exercise>>

Execute calculation with one Approach and without Escape.

1. Specify the profile.

   Profile：Zrough01

2. Open [Calculation condition sheet/Approach/Escape].

3. Set calculation conditions (1)-(2).

   (1) Approach　Z-axis direction1：ON / Length　5（mm）

   (2) Escape　Z-axis direction：OFF

4. After you set conditions, click [OK].

5. Click [Start] button in Control Calculation.

6. When [Normal] is displayed, select [Show CL by segment] to confirm [CL data].

   → Z - axis Approach is output with one step(5mm). * Approach = Red solid line (Dashed line: G01CL)

   → Z - axis Escape is not output.　　* Escape = Blue solid line (Dashed line: G01CL)

7. After you confirm CL, click [Clear CL] button to clear the displayed CL.

Execute calculation with two Approach and with one Escape.

2. Open [Calculation condition sheet/Approach/Escape].

3. Set calculation condition (1)-(2).

   (1) Approach　Z-axis direction1：ON / Length　5（mm）

   (1) Approach　Z-axis direction2：ON / Length　3（mm）

   (1) Escape　Z-axis direction：ON / Length　5（mm）

4. After you set condition, click [OK].

5. Click [Start] button in Control Calculation.

6. When [Normal] is displayed, select [Show CL by segment] to confirm [CL data].

   → Z-axis Approach is output with two step(5mm+3mm).

   → Z-axis Escape is output with one step(5mm).

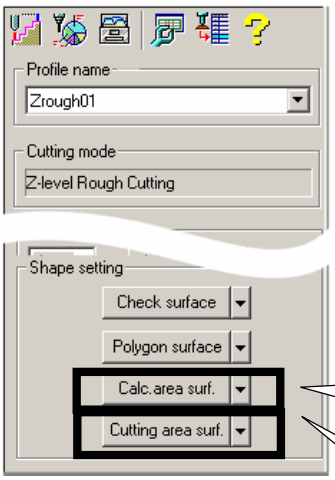
6. After you confirm CL, click [Clear CL] button to clear the displayed CL.

How to set for area cutting

→ Control window

There are two methods to perform area cutting.

■ Method1: Specify calculation area with [Calculation area surface], and specify area to be cut with [Cutting area surface].



After calculation is executed on entire shape, necessary area is cut out. Therefore, interference does not occur.

(1) Click [Calculation area surface] button.

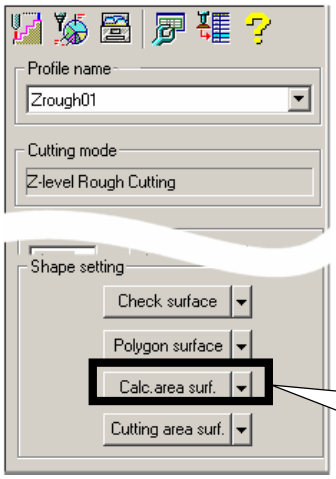
(2) Specify a surface for calculation area surface that cutter path are output.

* Only a single surface can be specified as [Calculation area surface].

(3) Click [Cutting area surface] button.

(4) Specify a surface that cutter path are output.
* Only a single surface can be specified as [Cutting area surface].

■ Method2: Specify area to be cut as [Calculation area surface].



Since specify a small area for calculation area, calculation time can be shortened.

(1) Click [Calculation area surface] button.

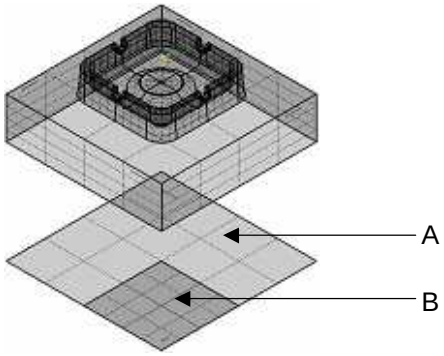
(2) Specify a surface that cutter path are output.
* Only a single surface can be specified as [Cutting area surface].

<<Exercise>>

Set [Layer2] for active layer, and perform [Area cutting] using two methods below. And, compare the result.

Specify calculation area with [Calculation area surface], and limit area for area cutting with [Cutting area surface].

1. Specify a profile Profile name : Zrough01
2. Click▼ button next to the [Calculation area surface], and clear the setting.
3. Click the [Calculation area surface] button, and specify the surface [A].
4. Click the [Cutting area surface] button, and specify the surface [B].



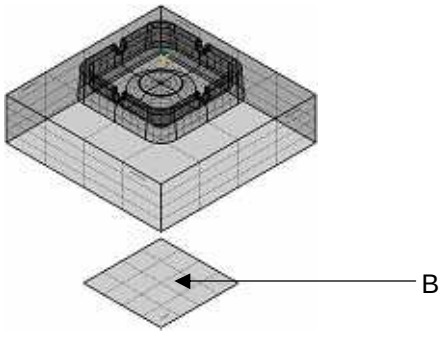
5. Click [Start] button in Control Calculation.
6. When [Normal] is displayed, select [Show CL by segment] to confirm [CL data].

   → After calculation is executed in the area [Surface A], CL data only in the area [Surface B] is left. Interference does not occur.

7. After you confirm CL, click [Clear CL] button to clear the displayed CL.

Specify area as [Calculation area surface] for area cutting.

1. Click▼ button next to the [Calculation area surface], and clear the setting.
2. Click▼ button next to the [Cutting area surface], and clear the setting.
3. Click the [Calculation area surface] button, and specify surface [B].



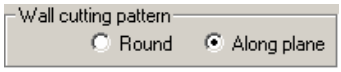
4. Click [Start] button in Control Calculation.
5. When [Normal] is displayed, select [Show CL by segment] to confirm [CL data].

   → Calculation is executed in the area [Surface B], calculation time can be shortened.

6. After you confirm CL, click [Clear CL] button to clear the displayed CL.

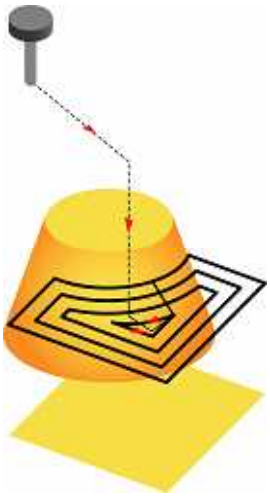
How to set cutting type for area cutting → Calculation condition sheet: Machining

Select cutting pattern for area cutting.
You can select the cutting type only when you specify area to be cut with [Calculation area surface].

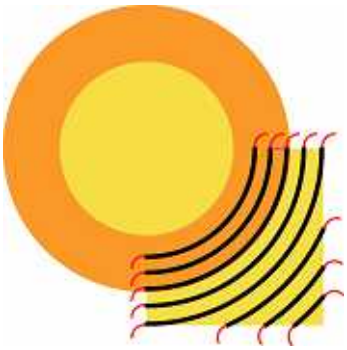


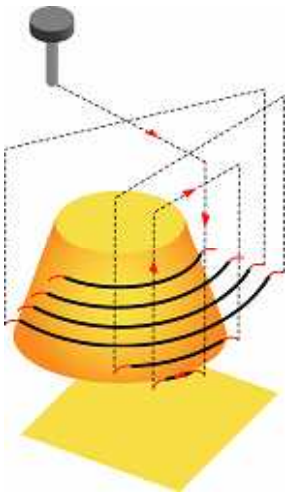
Wall cutting pattern

•Round：The system assumes that outer contour of [Calculation area surface] is wall, and tool travels around in the [Calculation area surface].



• Along plane: Tool cuts along the surface specified as [Check surface] within the area specified as [Calculation area surface] .

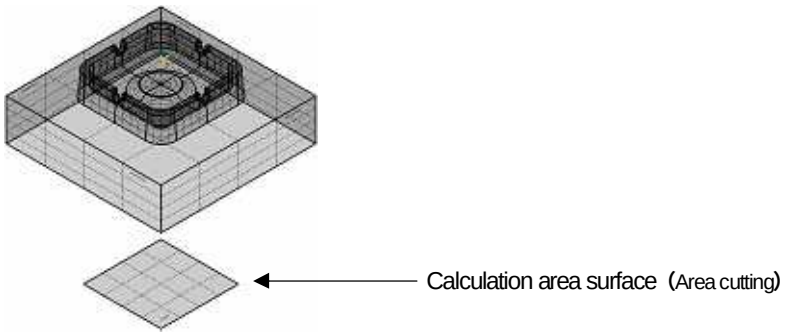




When you specify area for area cutting with [Cutting area surface], tool cuts along the surface.

<<Exercise>>

Set range for [Area cutting] with [Calculation area surface], and compare the result.



Calculate with the setting [Wall cutting pattern: Round].

1. Specify a profile Profile name : Zrough01
2. Set [Wall cutting pattern: Round] in [Calculation condition sheet/Machining].
3. After you set the condition, click [OK].
4. Click [Start] button in Control Calculation.
5. When [Normal] is displayed, select [Show CL by segment] to confirm [CL data].

→ Cutter paths, which tool travels around on outer contour of [Calculation area surface], are output.

6. After you confirm CL, click [Clear CL] button to clear the displayed CL.

Calculate with the setting [Wall cutting pattern: Along plane].

1. Set [Wall cutting pattern: Along plane] in [Calculation condition sheet/Machining].
2. After you set the condition, click [OK].
3. Click [Start] button in Control Calculation.
4. When [Normal] is displayed, select [Show CL by segment] to confirm [CL data].

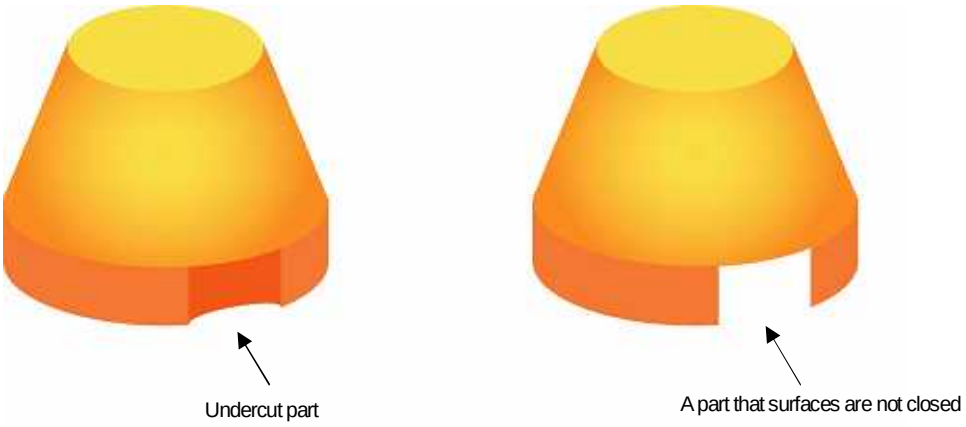
→ Cutter paths that [Check surface] is offset are output.

5. After you confirm CL, click [Clear CL] button to clear the displayed CL.

What if undercut part exists in a shape?

→ Calculation Condition Sheet: /Machining

Basically, it is preferable that any [Undercut part] or [Part that side surfaces are closed] does not exist in a shape. However, a shape may have invalid surfaces, because it is a converted data. In that case, the system check if any undercut part exists in the shape.



• Under cut check：OFF

No path is output at a undercut part.

• Under cut check：ON

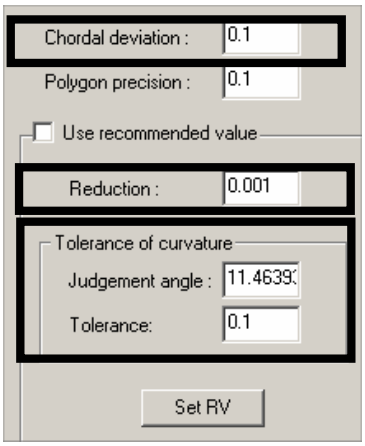
Execute undercut check. If an undercut part exists in a shape, [Wall] is generated by the system, and cutter paths are output.

When [Under cut check] is ON, calculation time will be longer than when it is OFF.

How to set machining precision

→ Calculation Condition Sheet: / Precision

In cam-tool V3 , cutting precision is determined by [Chordal deviation], [Reduction] and [Tolerance of curvature].

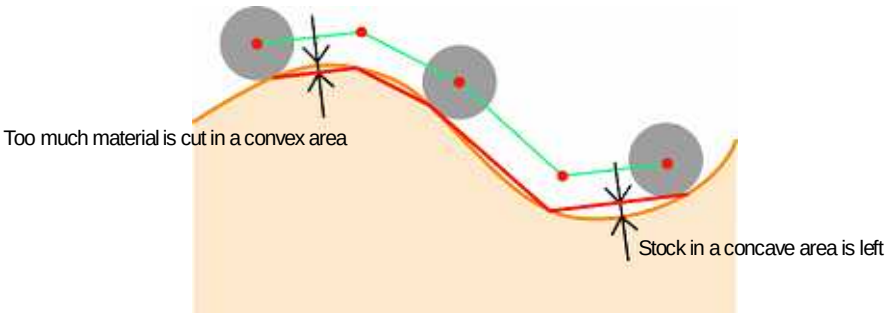


• Chordal deviation

Since NC data is output with only G01, material in a convex area is cut too much, and stock in a concave area is left, depending on the specified precision.

If you specify small value for chordal deviation, cutting precision will be raised. However, calculation time and amount of data will increase at the same time.

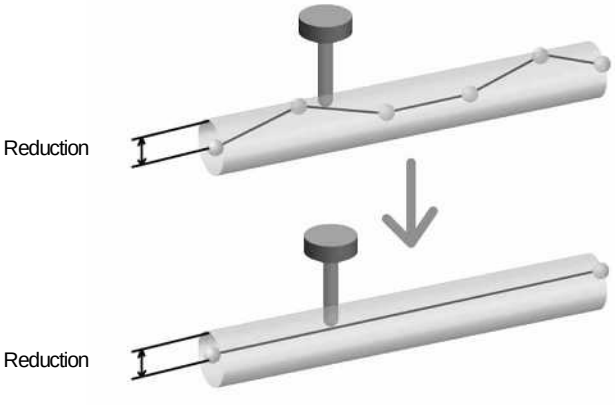
For [Z-level Rough Cutting] / [Z-level Finishing], input value of [Cutting precision x 4] for [Chordal deviation].



• Reduction

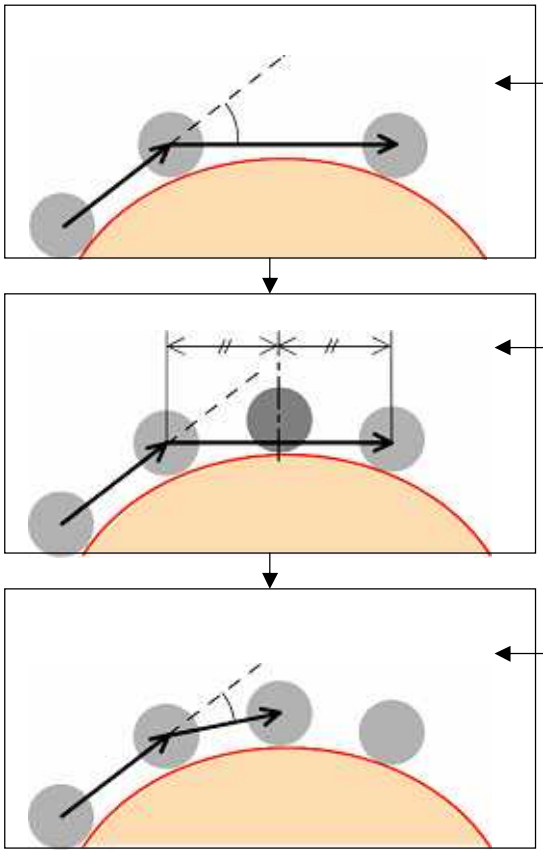
After calculation is executed with the specified [Chordal deviation] and [Tolerance of curvature], if plural paths exist in the specified range of [Reduction], those paths will become G01 of a single block. This function is automatically performed by the system.

Set smaller value for [Reduction] than [Chordal deviation]. (Minimum value = 0.0001)



• Tolerance of curvature - Judgement angle

If a surface has an area, whose [Vector direction] of cutter paths drastically changes partially, calculation needs to be executed with higher precision than the specified value for [Chordal deviation]. When angle that is the change between the two path's vector is larger than the specified [Judgement angle], the path is divided into two paths.



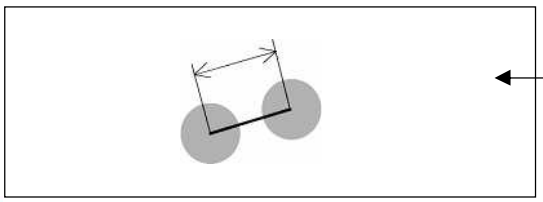
When the angle that is the change between the two path's vector is larger than the specified [Judgement angle]…,

A middle point is positioned, and the path is divided into two paths.

If the angle that is the change between the first path and the divided path is still larger than the specified [Judgement angle], the same process is repeated until the angle becomes smaller.

• Tolerance of curvature - Tolerance

Specify minimum length of path for [Tolerance of curvature] processing.



Paths are divided by the specified [Tolerance] value.

Use recommended value / Set RV

[Reduction] / [Tolerance of curvature - Judgement angle] / [Tolerance of curvature - Tolerance] are automatically determined by the specified [Tool shape] and [Chordal deviation], as to be [Cutting precision = Chordal deviation/4].

Reduction: [0.001] fixed

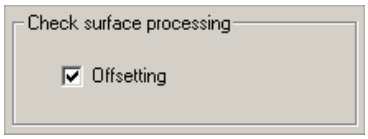
Tolerance of curvature - Judgement angle : automatically determined by the specified [Tool shape] and [Chordal deviation]

Tolerance of curvature - Tolerance : [0.1] fixed

How to change calculation method

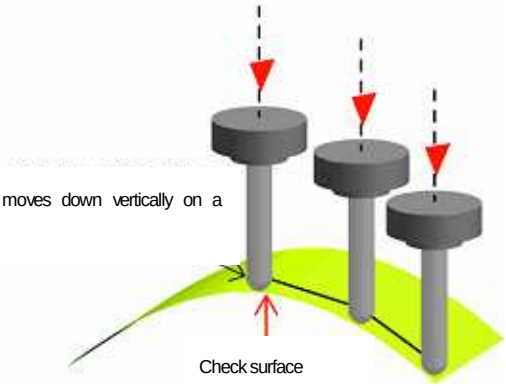
→ Calculation Condition Sheet: / Precision

Calculation method can be changed by setting [Offsetting] ON/OFF.



• Offsetting : OFF

Cutter paths are output on the position, where a tool moves down and hit the surface.

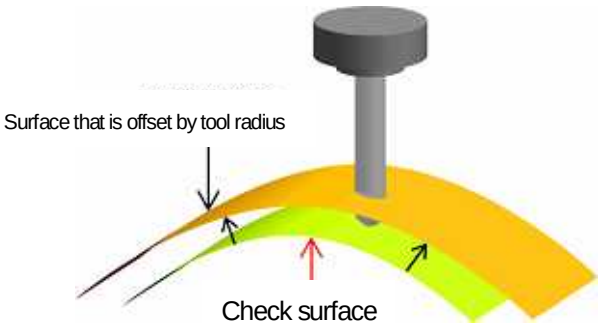


Advantage : Even if surfaces are not smooth, it is highly possible that cutter paths are output with accuracy.

Disadvantage : It takes long to calculate.

• Offsetting : ON

Cutter paths are output on the position, where is the intersection of Z level of tool center and the offset surface.



Advantage : High speed calculation.

Disadvantage : It is not effective when surfaces are not smooth.

Basically, set [Offsetting: ON]. For a complex shape, set [Offsetting: OFF].

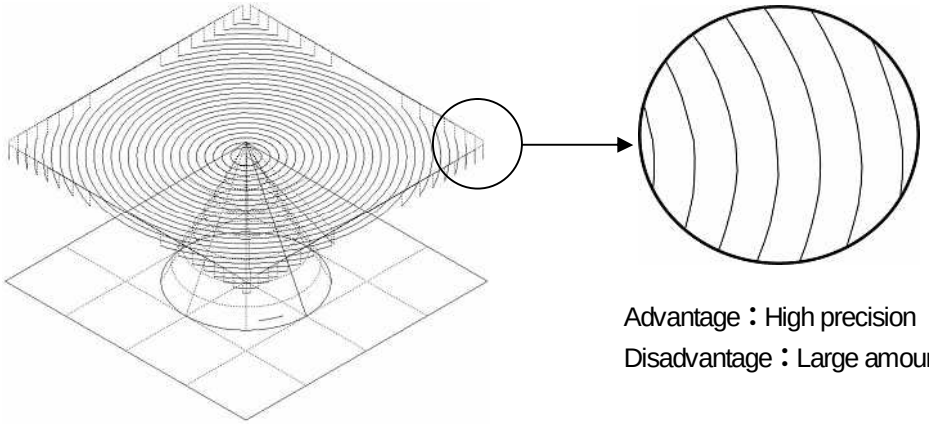
How to set method of processing at corner area → Calculation Condition Sheet: / Precision

In Z-level Rough Cutting, the system outputs cutter paths that travel around cutting shape, and offsets the path in plane direction.
Select processing method for [Corner area (Edge area)] of cutter paths, which are offset after linear interpolation is performed.

Corner offsetting

• Round

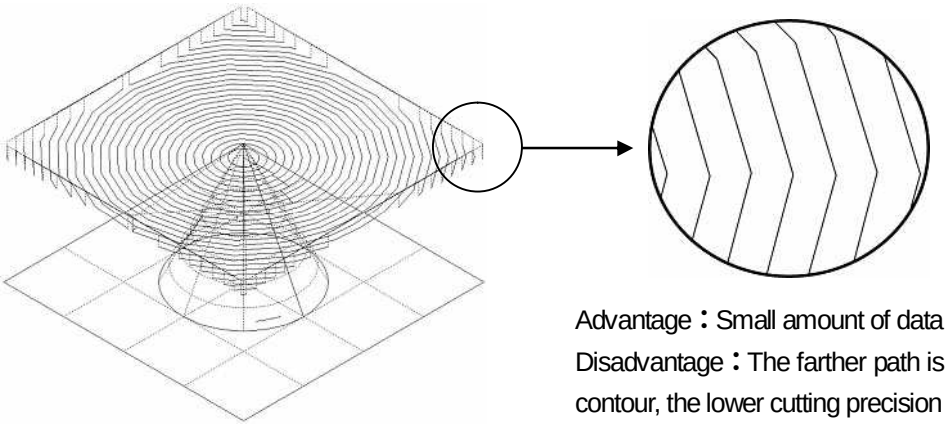
Interpolate cutter path at corner with short lines as to become arc (G01).



Advantage：High precision
Disadvantage：Large amount of data

• Sharp

Lengthen cutter path at corner.



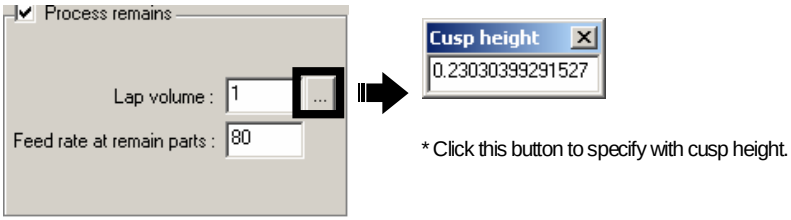
Advantage：Small amount of data
Disadvantage：The farther path is from original contour, the lower cutting precision becomes.

Basically, set [Sharp].

How to add path on area where stock is left

→ Calculation Condition Sheet: / Precision

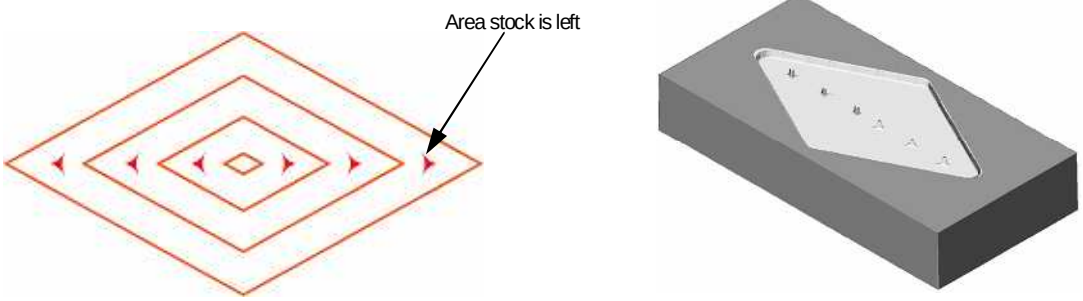
When stock is left because the specified XY step over is large, another cutter path can be added by the system. Use this function when you use a tool with round-tip and specify larger value than 80 % for XY step over. Specify [Remain volume] by [Lap volume] or [Cusp height(height of stock left)].



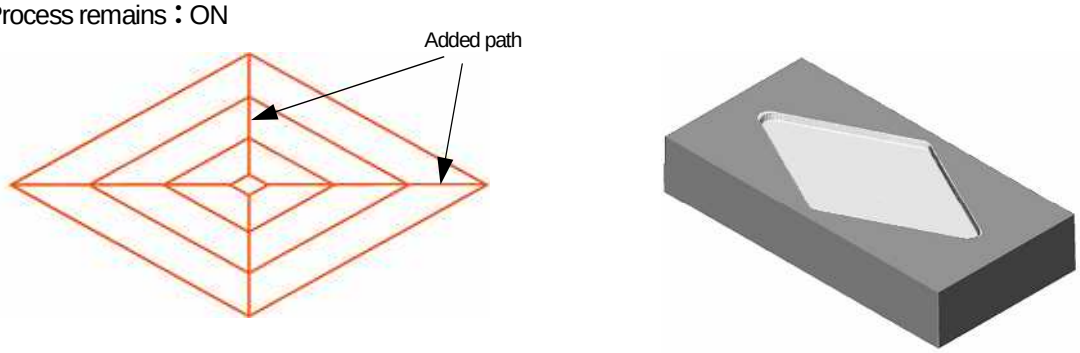
* Click this button to specify with cusp height.

Ex. Case of using tool with round-tip

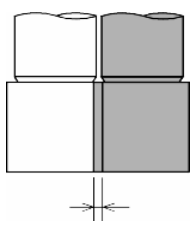
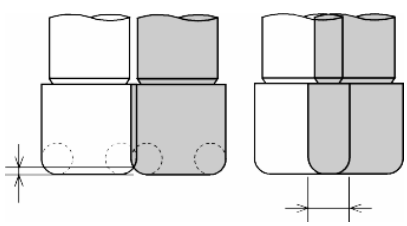
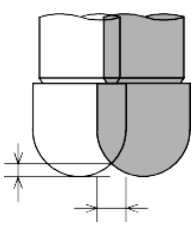
• Process remains : OFF



• Process remains : ON



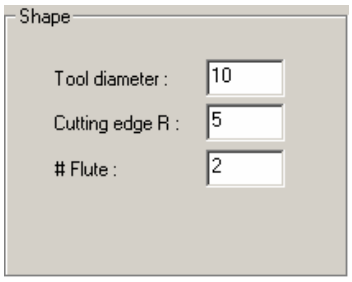
• Range for specifying Cusp height/Lap volume for each tool type

Tool type	Flat	Radius	Ball
Cusp height	×	0 - (Cutting edge radius)	0 - (Tool radius)
Lap volume	0 - (Tool diameter)	0 - (Tool diameter)	0 - (Tool diameter)
			

How to set tool shape

→ Calculation Condition Sheet: / Tool

Set tool shape by specifying [Tool diameter] / [Cutting edge R] / [Number of flute].

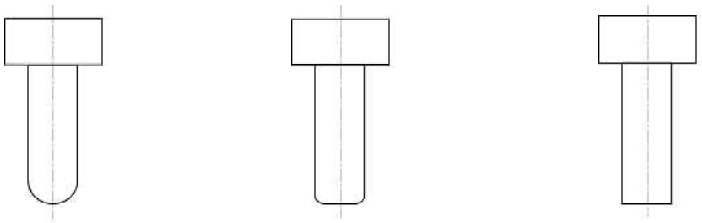


• Tool type

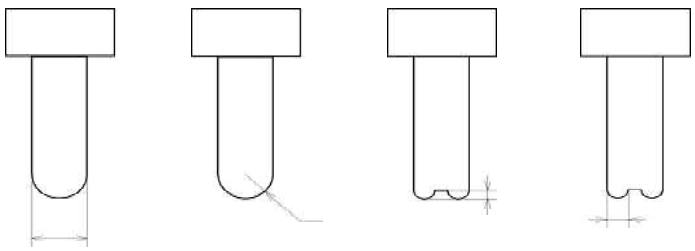
Ball end mill
Cutting edge R=Tool diameter1/2

Radius end mill
Cutting edge R < Tool diameter1/2

Flat end mill
Cutting edge R=0



• Tool parameter

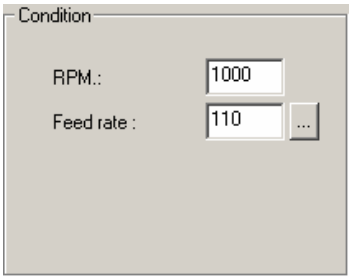


• [Inner depth] and [Edge width] is the parameter for throw-away tool.
If you use tool other than throw-away tool, input [0].

How to set RPM / Feed rate

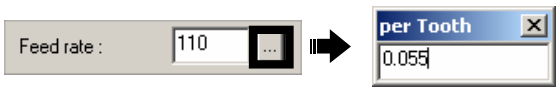
→ Calculation Condition Sheet: / Tool

Set [RPM] / [Feed rate].



- RPM : Specify spindle speed per minute.
- Feed rate : Specify feed rate per minute.

Click the button on the right of input area of [Feed rate] to check feed rate for [per Tooth].
Or, input feed rate for [per Tooth] to calculate [Feed rate].

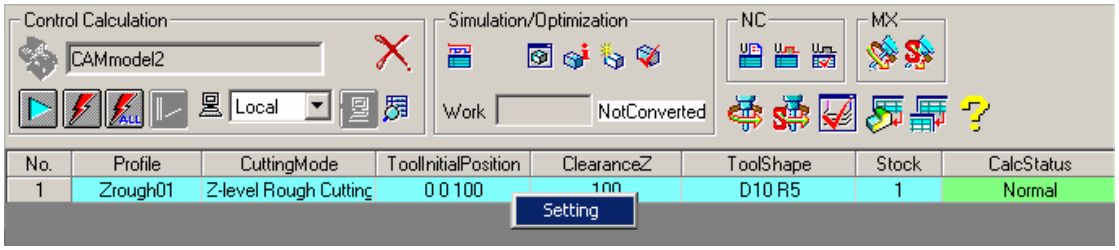


How to set Tool initial position/Clearance Z

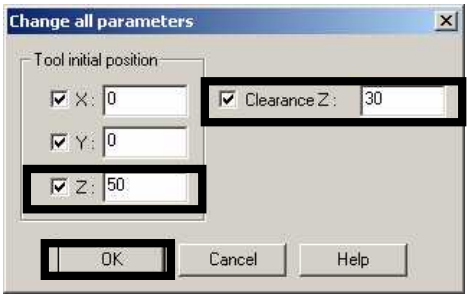
→ Calculation process sheet

Edit the [setting] of the selected profile on [Calculation process list].

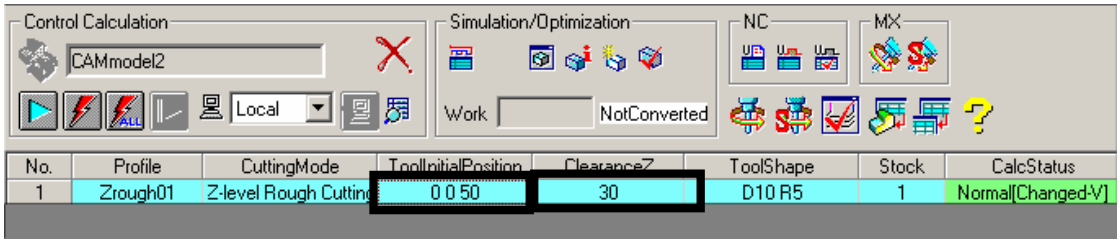
(1) Right-click on [Tool initial position] or [Clearance Z] and click [Setting].



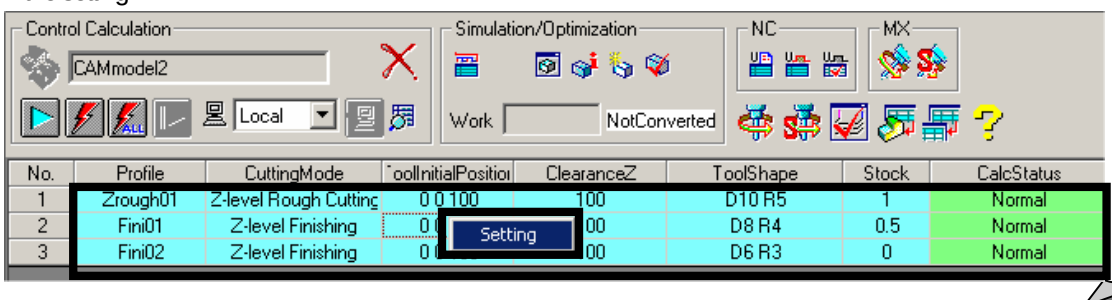
(2) Edit the setting of [Tool initial position] or [Clearance Z] on the displayed [Change all parameters] dialog.



(3) After you edit the setting, click [OK], and the setting will be applied to Calculation condition sheet.

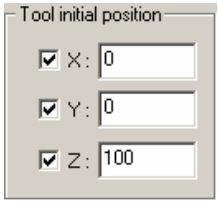


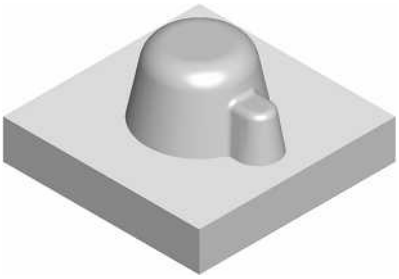
[Setting] of plural processes can be changed simultaneously by selecting plural processes when editing the setting.



○ Tool initial position

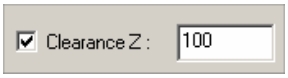
Set [Tool initial position] by X/Y/Z value of tool end point.

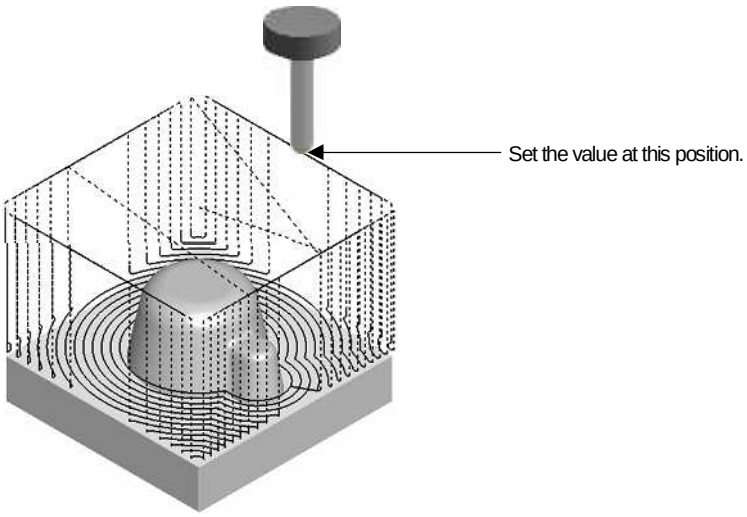




○ Clearance Z

Set Z value for [G00] motion that is generated when tool repeats traveling on a shape, by Z value of tool end point.





Chapter 4

Finishing （Side surface/Flat area）

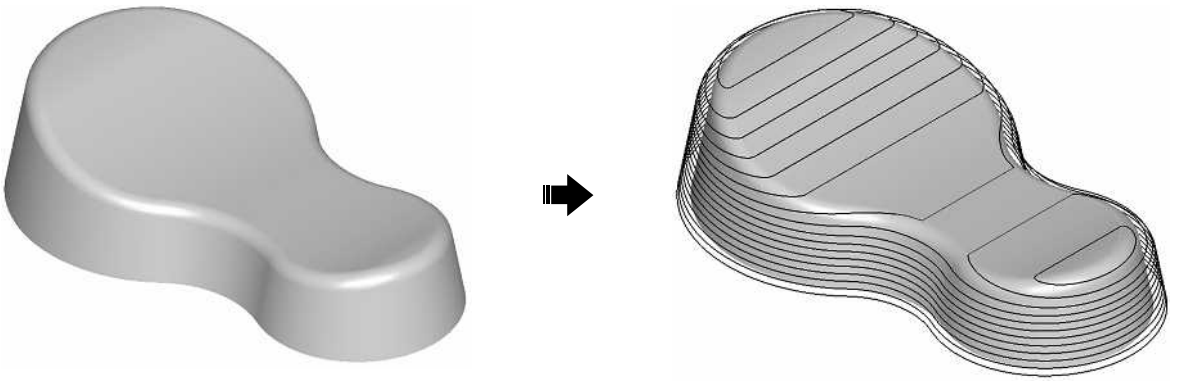
- Z-level Finishing -

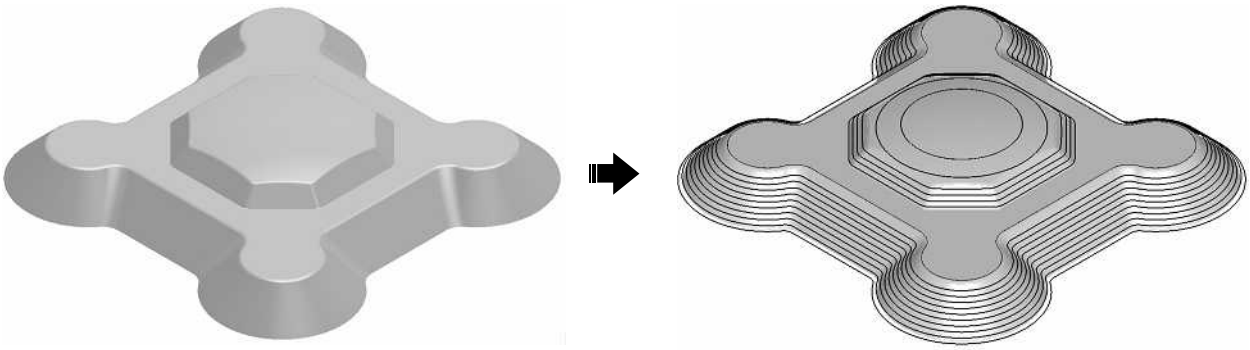
Z-level Finishing - Overview -

This is the cutting strategy that tool travels around with a fixed tool center Z value to perform finishing.
And using the combination of [Z-level Finishing] and [Offset cutting] enables to perform efficient Finishing on near horizontal area.

■Only [Z-level Finishing]

Output cutter paths that tool travels around with a fixed tool center Z value, at each Z level.
You can choose either perform [Plane cutting] or not perform.

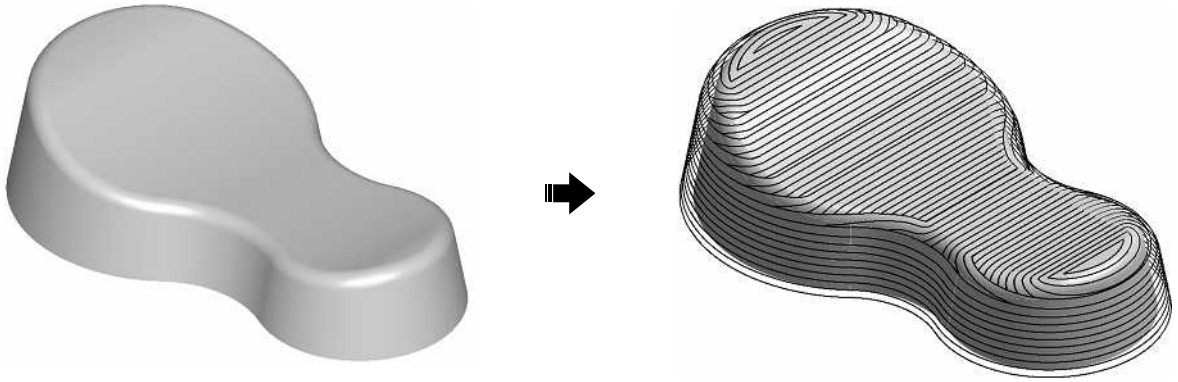


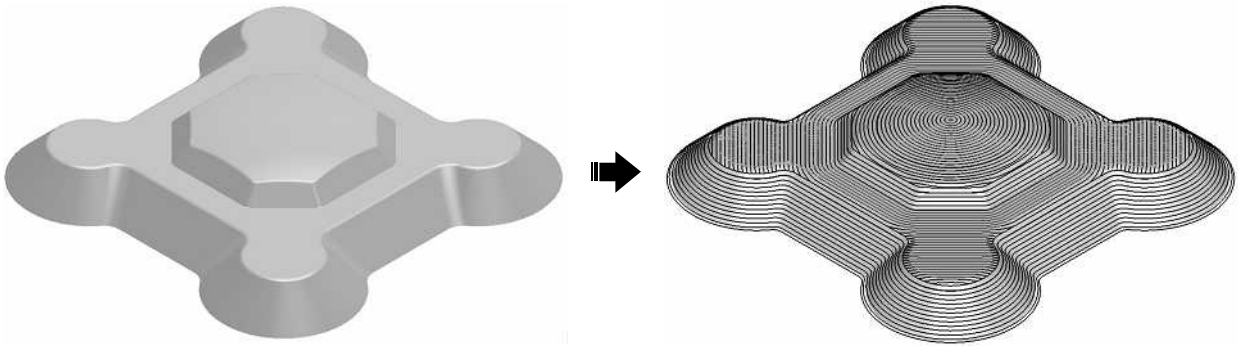


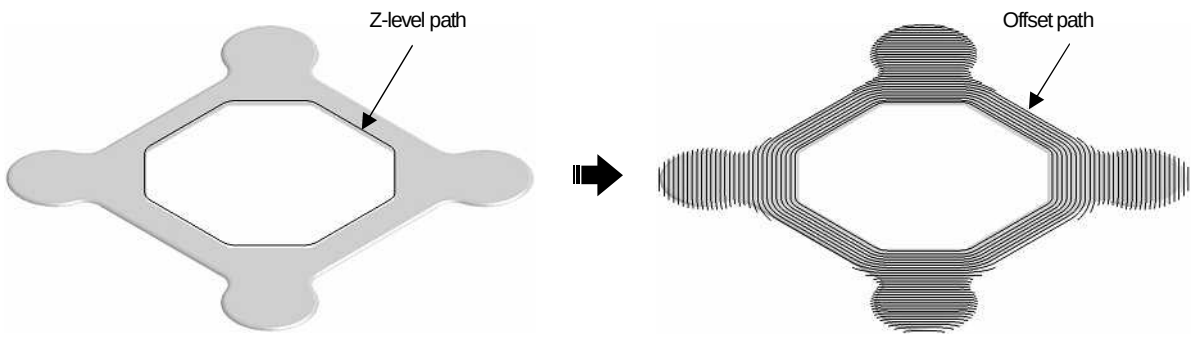
When you perform other cutting strategy such as Scanning-line Area on flat area or near horizontal area, set [Plane cutting] OFF.

■ [Z-level Finishing] + [Offset cutting]

When distance between the paths, which are output by [Z-level Finishing]on [flat area] or [near-horizontal area], is long, paths of [Offset cutting] are added to interpolate the distance.
To output paths of [Offset cutting], the system offsets Z-level path in the plane direction first, and project them to the surface. Therefore, paths of [Offset cutting] are not output at the fixed Z level.



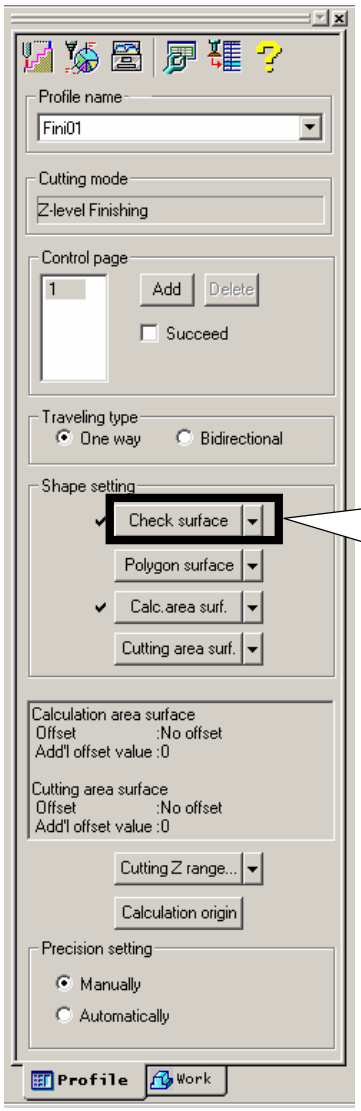




How to set cutting model shape

→ Control window

Specify [Check surface] in [Shape setting] on the control window.



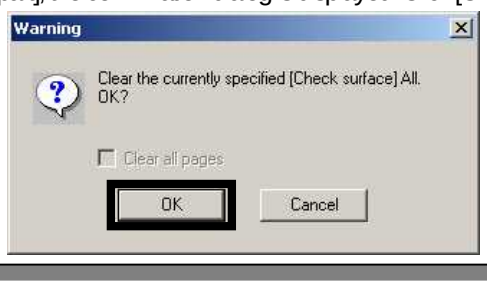
(1) Click [Check surface] button.

(2) Specify a surface for cutting shape.

* For cavity shape, specify [Parting surface] as well.

(3) Press the center mouse button to determine the setting.

To edit the setting, click ▼ button on the right of [Check surface], and select [Clear part] or [Clear All].
When you select [Clear part], the confirmation dialog is displayed. Click [OK] to execute the command.



<<Exercise>>

Set as follows in the model file of ashtray shape that you created in the [Text book for Intensive Course - CAD -].
If you operate with the model file that you performed Rough cutting, omit the following operations 1, 2, 4, and set [Layer3] for active layer.

1. Open the model file. CD-ROM / Intensive Course /CAMmodel2.gmd [Layer3]

2. Save the model file. <Destination folder/File name>
D:/cam-tool /cam-toolFile /Exercise2 / CAMmodel2.gmd

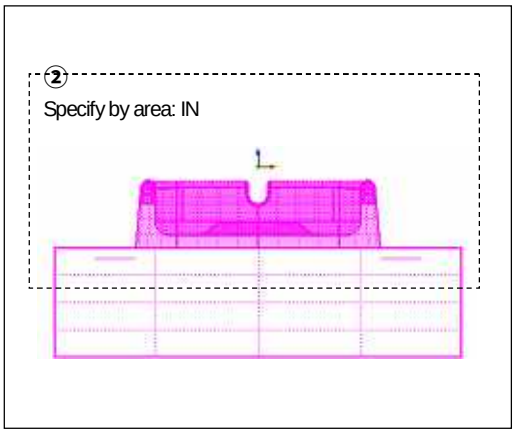
3.Create a new profile (1) Profile name : Fini01
(2) Cutting strategy: Z-level Finishing
(3) Initial setting file : (blank)
(4) Continuous creation : OFF
(5) Add to calculation process: OFF >> [Finish] button

4. Create a new calculation process sheet. <Destination folder/File name>

D:/ cam-tool /cam-toolFile /Exercise2 / CAMcalculation2

5. Add the profile to calculation process sheet.

6. Specify [Check surface] in the profile.

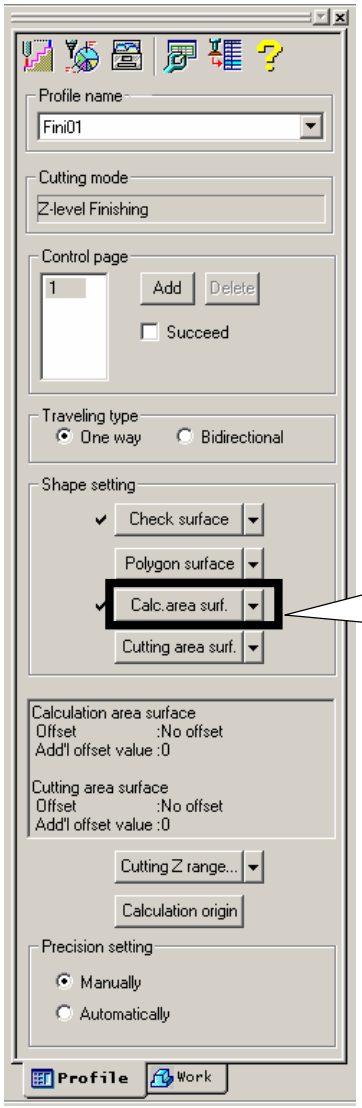


(1) Click [Check surface] button.
(2) Specify surfaces for cutting shape. (Specify by area: IN)
(3) Press the center mouse button to determine the specifying.

7. After you set, confirm with [CAM mode] function.

How to set cutting range in plane direction → Control window

Specify [Calculation area surface] in [Shape setting] on the control window.

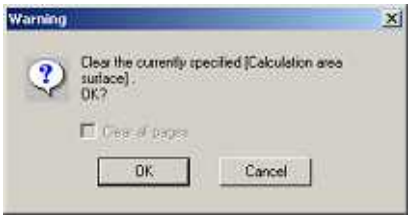


(1)Click [Calculation area surface] button.

(2) Specify a surface that cutter paths are output.

* Only a single surface can be specified.

To edit the setting, click ▼ button on the right of [Calculation area surface], and press [Clear]. When the confirmation dialog is displayed, click [OK] to execute the command.

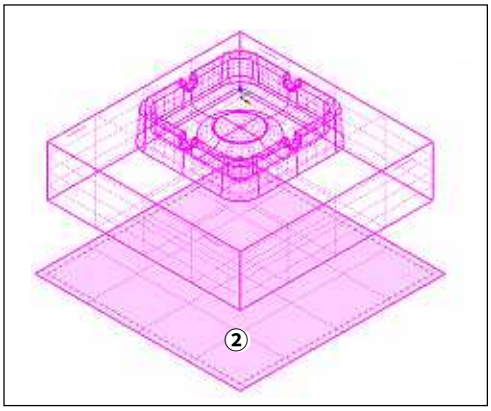


Calculation for plane direction is executed within the range of original surface of [Calculation area surface]. When you use a trim surface for [Calculation area surface], create a original surface as small as possible. It leads to reduce calculation time.

<<Exercise>>

Next, set [Calculation area surface].

1. Set [Calculation area surface] in the created profile.



(1) Click [Calculation area surface] button.

(2) Specify a surface that cutter paths are output.
* Only a single surface can be specified as [Calculation area surface].

2. After you set, confirm with [CAM mode] function.

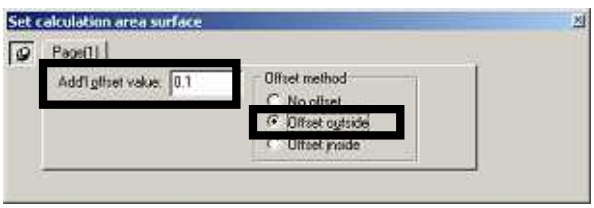
Cautions of [Set cutting area]

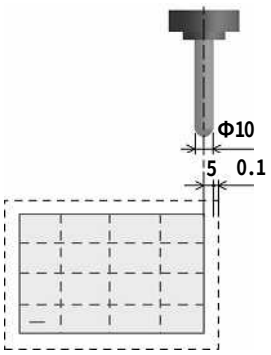
With [Set cutting area] function, [Calculation area surface] can be adjusted by the specified value.

Since contour of [Calculation area surface] is offset internally by the system, this function may not be used for a trim surface that has complex contours.
Use this function in case such as [Work shape = Calculation area surface] in [Rough cutting process].

- Offset method：Select one from [No offset] , [Offset outside] or [Offset inside].
- Additional offset value: Add the specified offset value to the area that is adjusted by the specified [Offset method].

Ex.) [Tool diameter =D10]
- Offset method：Offset outside
- Additional offset value: 0.1



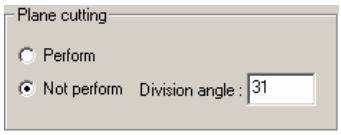


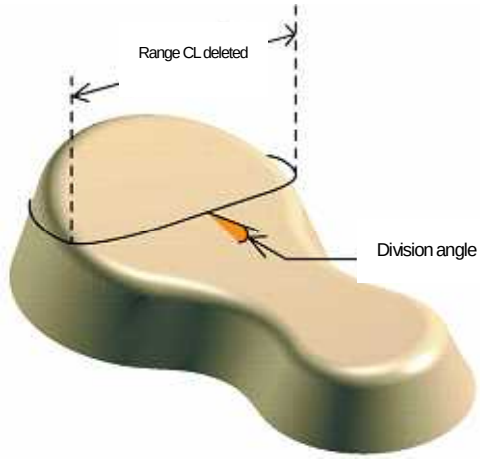
Calculation area surface is adjusted by [5.1mm].

If not perform Finishing on flat area?

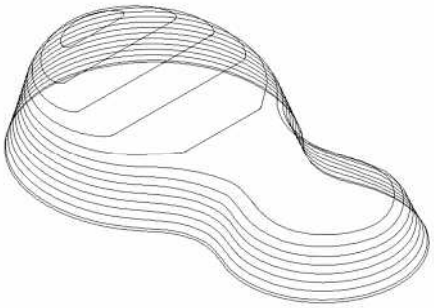
→ Calculation condition sheet: Machining

Select whether perform finishing or not perform on [near horizontal area].



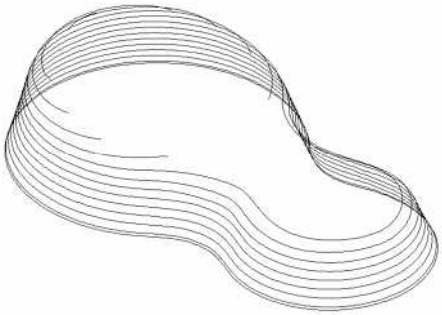


• Plane cutting: ON



Paths are output on [flat area] as well.

• Plane cutting: OFF



Input [Division angle].
If the angle of the surface slope is smaller than the specified division angle, the area is detected as [flat area] by the system, and cutter paths on the area are deleted.

When you perform other cutting strategy such as Scanning-line Area for near-horizontal area, set [Plane cutting] OFF.

<<Exercise>>

Set [Z cutting range] and [Z step down], and execute calculation.

Execute calculation with [Plane cutting] ON.

1. Specify a profile Profile name：Fini01
2. Open the [Calculation condition sheet/Machining] and set the following calculation conditions (1)-(5).

   (1) Cutting Z range：Start [0]，End [-30]
   (2) Z step down： Cusp height [0.05] * Cusp height = Height of left stock
   (3) Plane cutting：Perform
   (4) Stock ：0.5（mm）
   (5) Offset cutting：OFF

3. Open the [Calculation condition sheet/Tool] and set the following calculation conditions (6)- (7).

   (6) Tool diameter： 6
   (7) Cutting edge R： 3

4. After you set the condition, click [OK].
5. Click [Start] button in Control Calculation.
6. When [Normal] is displayed, calculation has ended, and CL data has been created.
7. Right-click on [Normal], and select [Show CL by segment] to confirm [CL data].

→ Z-level cutter paths are output on the near-horizontal area.

8. After you confirm CL, click [Clear CL] button to clear the displayed CL.

Execute calculation with [Plane cutting] OFF.

1. Open the [Calculation condition sheet/Tool] and set the following calculation conditions (8)- (9).

   (8) Plane cutting: OFF
   (9) Judgement angle: 32 degree

2. After you set the condition, click [OK].
3. Click [Start] button in Control Calculation.
4. When [Normal] is displayed, calculation has ended, and CL data has been created.
5. Right-click on [Normal], and select [Show CL by segment] to confirm [CL data].

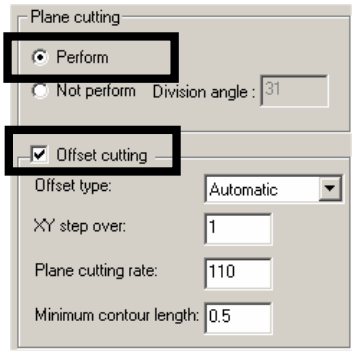
→ Z-level cutter paths are not output on the area whose slope angle is smaller than the specified judgement angle 32.

6. After you confirm CL, click [Clear CL] button to clear the displayed CL.

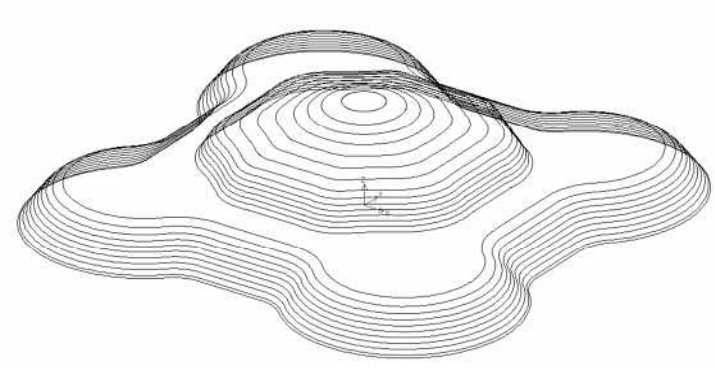
If perform Finishing on flat area with offset paths?

→ Calculation condition sheet: Machining

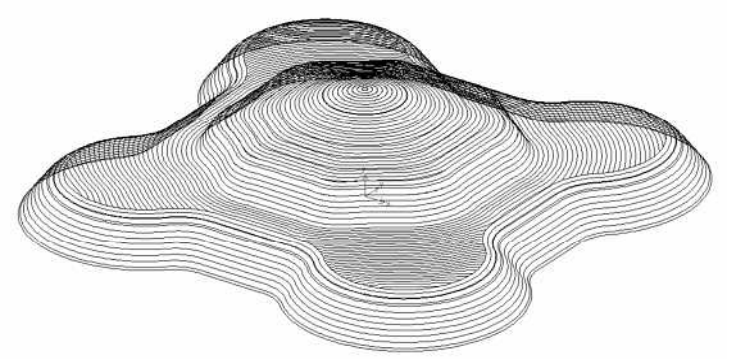
When distance between the paths, which are output by [Z-level Finishing]on [flat area] or [near-horizontal area], is long, paths of [Offset cutting] are added to interpolate the distance.
This function is available when setting [Plane cutting] ON.



• Offset cutting：OFF（only Z-level cutting）

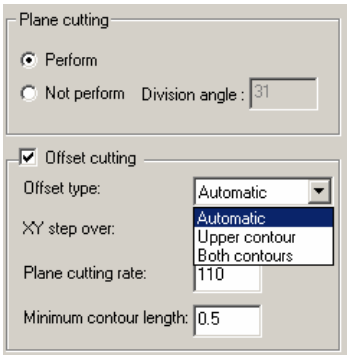


• Offset cutting：ON（Z-level cutting + Offset cutting）



• Offset type

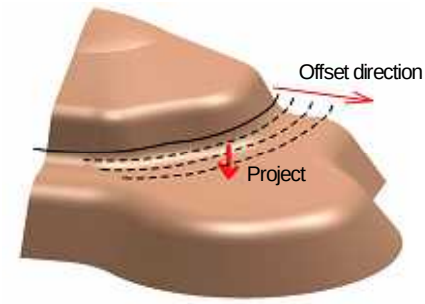
Specify the contour (Z-level path) so as to be prioritized for offsetting.



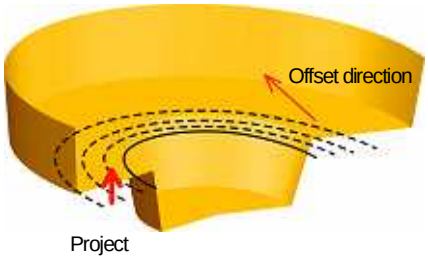
<Automatic>

• For convex shape, offsets upper contour, and project it to the check surface.
• For concave shape, offsets lower contour, and project it to the check surface.

Convex shape



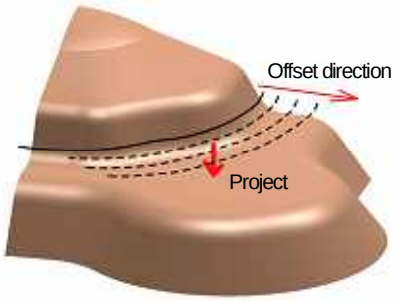
Concave shape



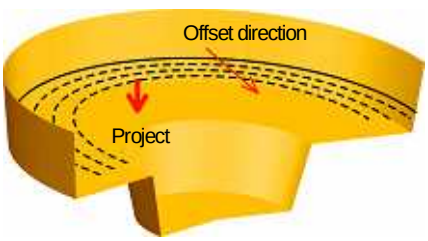
<Upper contour>

• Regardless of whether the shape is convex or concave shape, always offsets upper contour, and project it to the check surface.

Convex shape



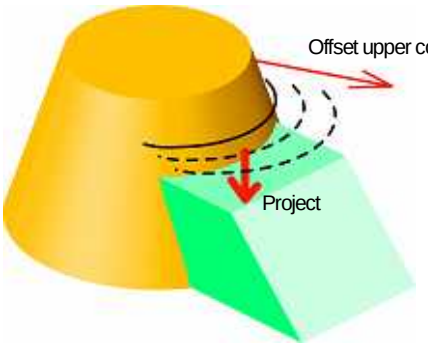
Concave shape

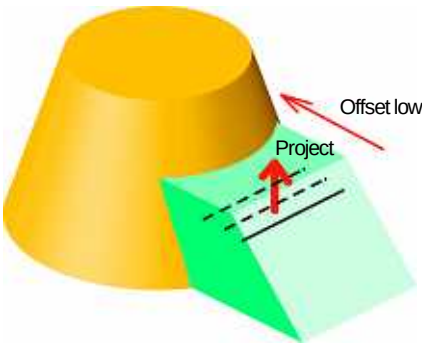


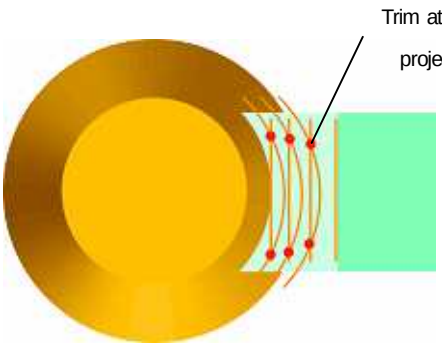
* For convex shape, the same processing is performed as [Automatic].

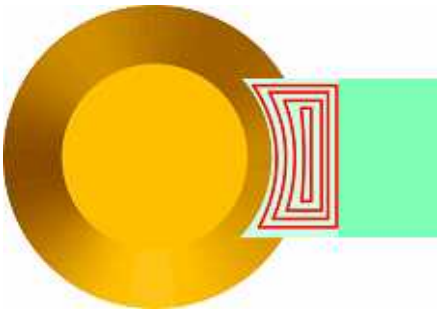
<Both contours>

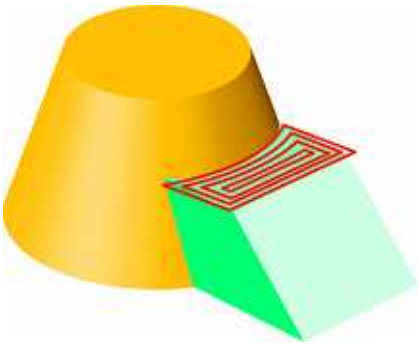
• Regardless of whether the shape is convex or concave shape, offsets both upper and lower contours, and project them to the check surface. The system connects intersections of the projected both contours, and create cutter paths.







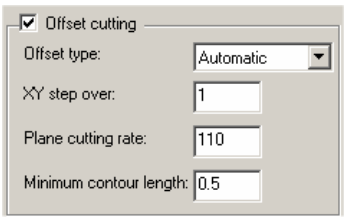




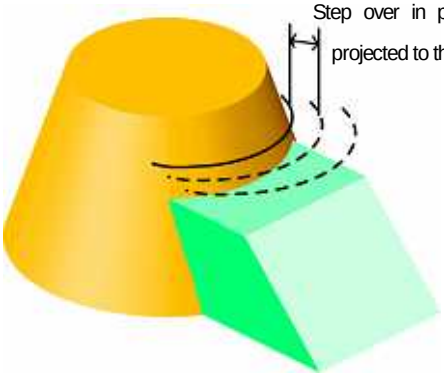
• Specify proper [Offset type] depending on the cutting shape.

• If you use [Offset cutting] function, the contour that is specified [Offset type] is offset in plane direction, and projected to the check surface. Therefore, Z step down will not be the same.

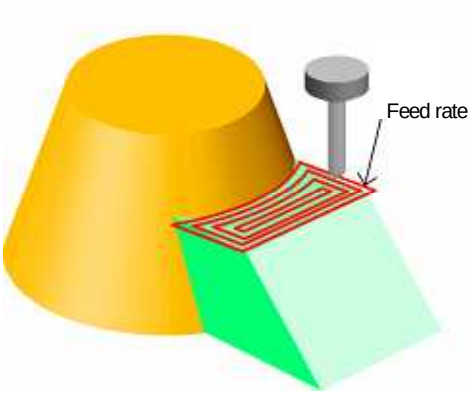
Specify [XY step over] / [Plane cutting rate].
(Specify XY step over/feed rate for Z-level cutting area and Offset cutting area separately.)



• XY step over

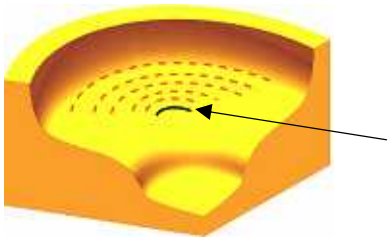


• Plane cutting rate



Method of not outputting micro path

Specify minimum length of cutter path on [Offset cutting] area by entering value for [Minimum contour length]. Shorter cutter path than the specified value is not output.



<<Exercise>>

Execute calculation with combination of [Plane cutting] and [Offset cutting].

Execute calculation with [Offset cutting :Automatic].

1. Specify a profile. Profile name：Fini01
2. Click [Calculation condition sheet/Machining], and set the cutting conditions (1)-(5).

   (1) Plane cutting : Perform
   (2) Offset cutting：ON
   (3) Offset type：Automatic
   (4) XY step over：1 (mm)
   (2) Minimum contour length：0.5 (mm)

3. After you set the condition, click [OK].
4. Click [Start] button in Control Calculation.
5. When [Normal] is displayed, calculation has ended, and CL data has been created.
6. Right-click on [Normal], and select [Show CL by segment] to confirm [CL data].

→For the convex area, the upper contour, and for the concave area, the lower contour is offset and projected to the check surface.

7. After you confirm CL, click [Clear CL] button to clear the displayed CL.

Execute calculation with [Offset cutting :Both contours].

1. Click [Calculation condition sheet/Machining], and set the cutting condition (6).

   (6) Offset type: Both contours

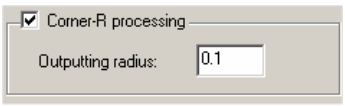
2. After you set the condition, click [OK].
3. Click [Start] button in Control Calculation.
4. When [Normal] is displayed, calculation has ended, and CL data has been created.
5. Right-click on [Normal], and select [Show CL by segment] to confirm [CL data].

→Both the upper contour and the lower contour are offset and projected to the check surface.

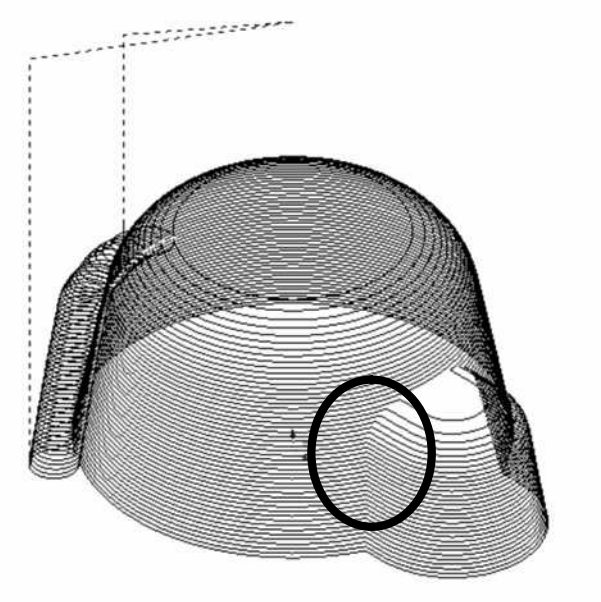
6. After you confirm CL, click [Clear CL] button to clear the displayed CL.

How to reduce tool load at corner area → Calculation condition sheet: Machining

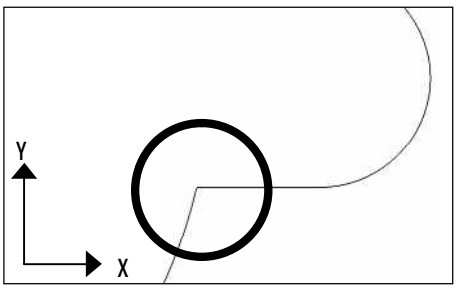
By inserting arc-shaped path at corner, tool load at corner can be reduced.
And this function also enables to perform cutting without dropping the feed rate at corner. (Arc is artificial G01 arc.)



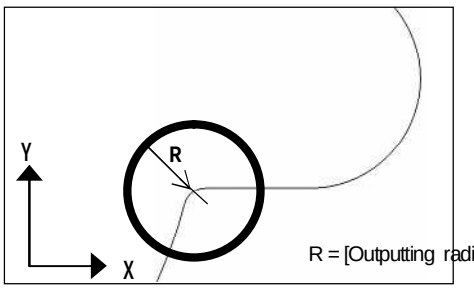
0 ≦ outputting radius ≦ tool diameter×0.05
And 1 or smaller



<Corner R processing: OFF>



<Corner R processing: ON>



• When arc of the specified [outputting radius] cannot be inserted, Z-level path is not output. When you execute [Show CL], a warning mark [■ (Red)] is displayed on the position where path is not output.

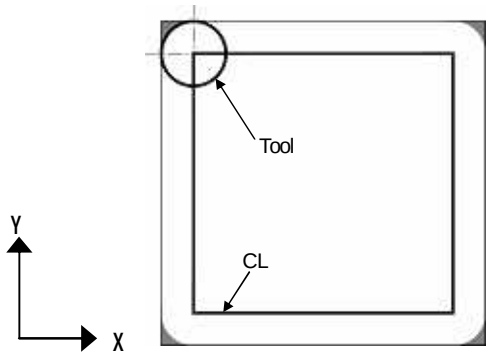
• When [Offset cutting] is ON, [Corner R processing] cannot be executed.

• When [Corner R processing] is performed, [Move by step] is selected for [Connecting move].

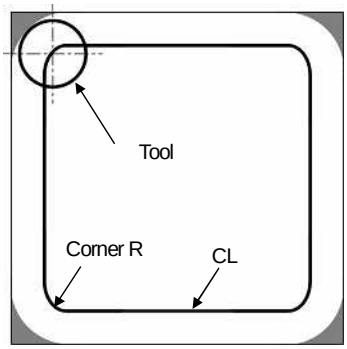
Precautions for [Corner R processing]

(1) If [Corner R processing] is performed, much more stock is left.

<[Corner R processing] OFF>

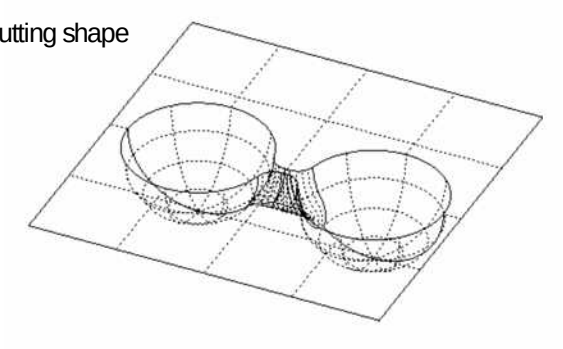


<[Corner R processing] ON>

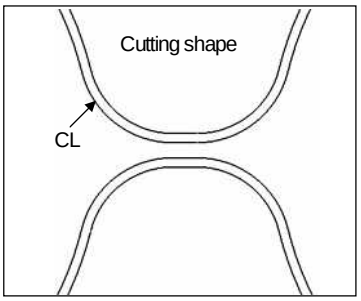


(2) When [Corner R processing] is performed, CL may be divided into plural areas depending on shape.

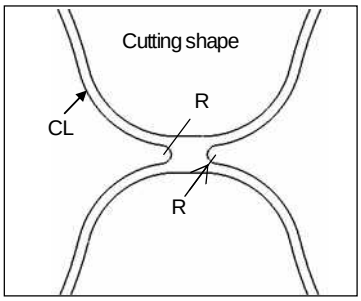
• Cutting shape



<[Corner R processing] OFF>

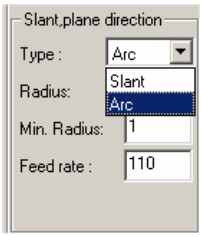


<[Corner R processing] ON>



How to set Approach in plane direction → Calculation condition sheet : Approach/Escape

Choose either [Arc] or [Slant] for plane direction Approach type.



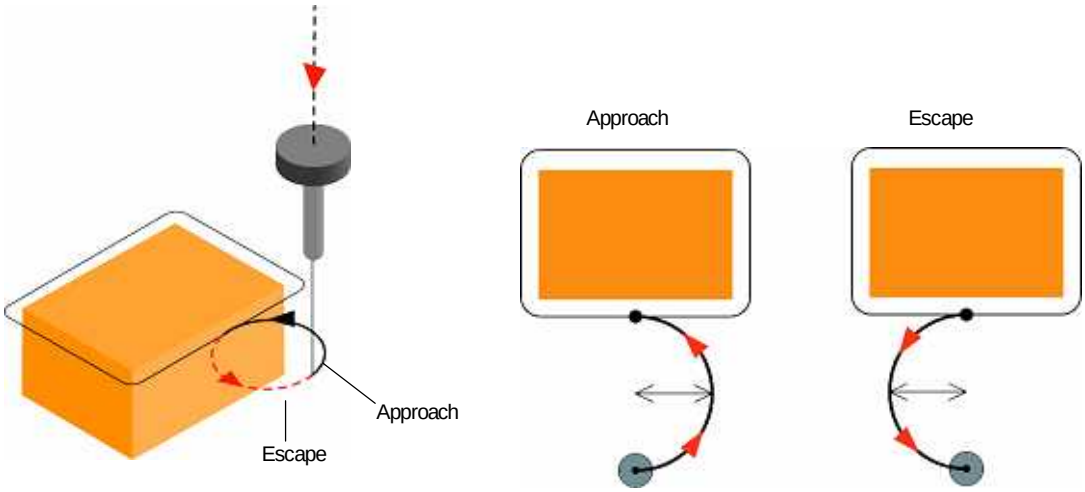
• Arc

Perform Arc Approach/Escape. If Approach/Escape of the specified arc interferes the shape, the arc radius will become smaller to the value that interference does not occur.

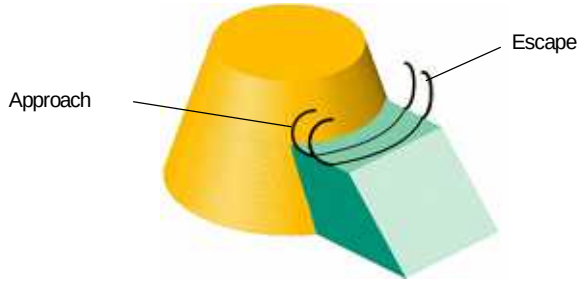
Radius : Input radius value for arc Approach/Escape.

Min. Radius : When Approach/Escape of the specified arc interferes the shape, the arc radius will become smaller to [Min. Radius].

Feed rate: Input feed rate for Approach/Escape.

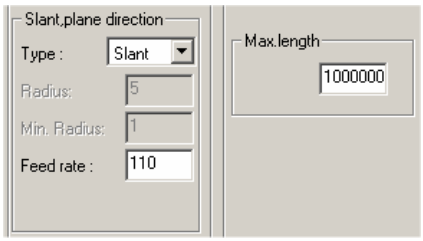


For the area that [Offset cutting] is performed, Arc Approach/Escape is performed from the normal line direction of the surface.



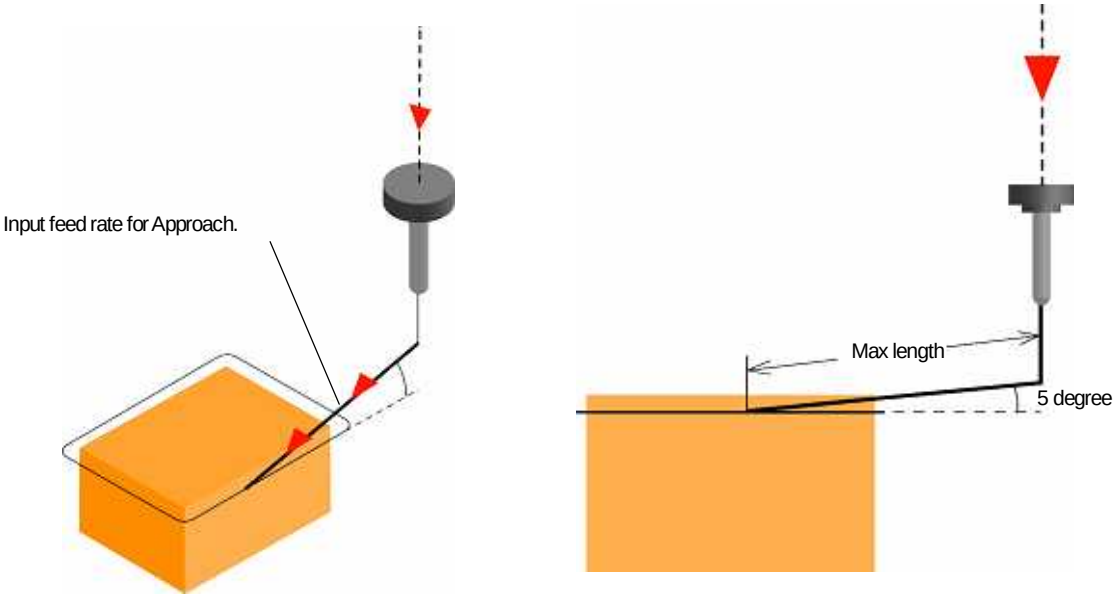
• Slant

Approach with the angle is performed. 5 degree is fixed for Slant angle.



Feed rate : Input feed rate for Approach.

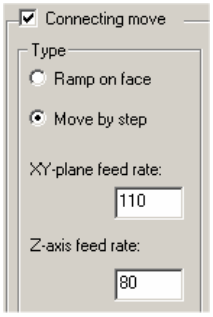
Max length： Specify maximum length of Slant Approach.
Length of Slant Approach is determined by the system within the range of the specified [Max length].



For cutter paths for [Offset cutting] area, Slant Approach is not output.

• Connecting move

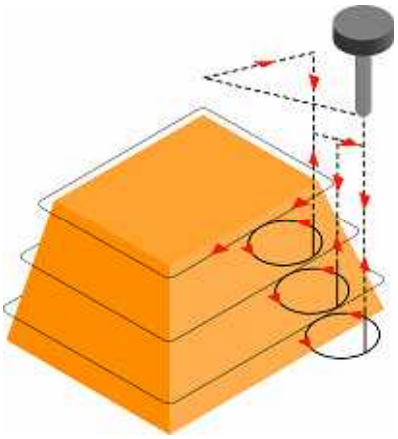
Select whether perform or not perform [Connecting move to the next Z level] between each Z level, after tool travels at each Z level.



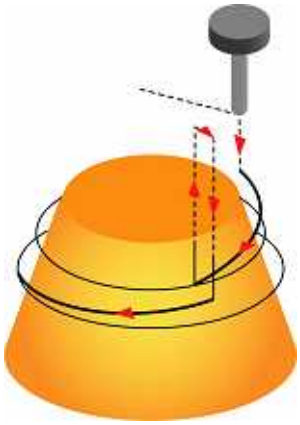
• Connecting move : OFF

Every time tool travels at Z level, it moves up to [Clearance Z], and moves to the next Z level as G01 motion.

<Arc Approach>



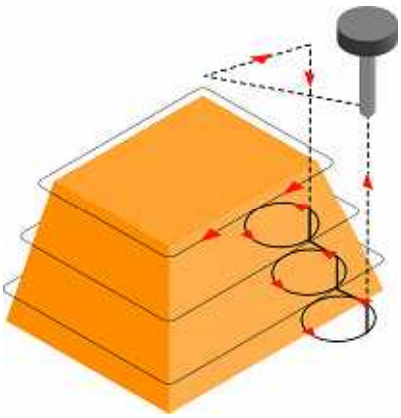
<Slant Approach>



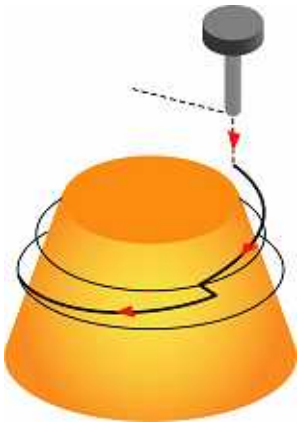
• Connecting move : [Move by step]

Tool moves to the next Z level by step without moving up to Clearance Z.
Input feed rate for both [XY direction] and [Z direction].

<Arc Approach>



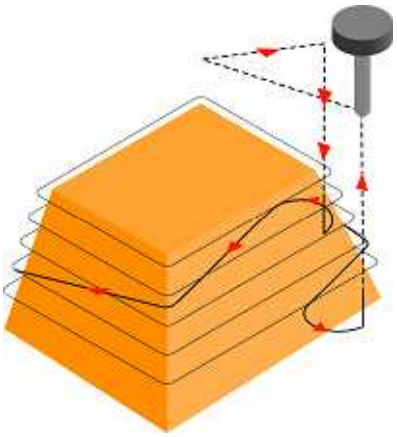
<Slant Approach>



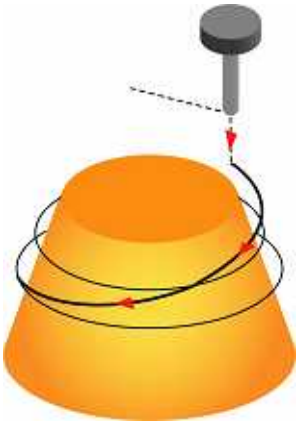
• Connecting move：[Ramp on face]

Tool cuts shape when it moves to the next Z level, without moving up to Clearance Z. For [Connecting move], tool moves at the cutting feed rate.

<Arc Approach>



<Slant Approach>



<<Exercise>>

Change the setting for [Approach/Escape] / [Connecting move] and confirm the calculation result.

Execute calculation with [Connecting move] OFF.

1. Specify a profile. Profile name：Fini01

2. Click [Calculation condition sheet/Machining], and set the cutting conditions (1).

(1) Offset cutting ：ON / Offset type: Automatic

3. Click [Calculation condition sheet/Approach/Escape], and set the cutting conditions (2)-(5).

Slant plane direction

(2) Type : Arc

(3) Radius：5 (mm)

(4) Min. Radius：1 (mm)

(5) Connecting move：OFF

4. After you set the condition, click [OK].
5. Click [Start] button in Control Calculation.
6. When [Normal] is displayed, calculation has ended, and CL data has been created.
7. Right-click on [Normal], and select [Show CL by segment] to confirm [CL data].

→ Tool moves up to Clearance Z every time it travels at Z level, and moves to the next Z level as G01 motion.

8. After you confirm CL, click [Clear CL] button to clear the displayed CL.

Change the setting for [Approach/Escape] / [Connecting move] and confirm the calculation result.

Execute calculation with [Arc approach]/[Connecting move: Move by step].

1. Click [Calculation condition sheet/Approach/Escape], and set the cutting conditions (5)-(9).

Slant plane direction

(5) Type : Arc
(6) Radius：5 (mm)
(7) Min. Radius：1 (mm)
(8) Connecting move：ON
(9) Type：Move by step

2. After you set the condition, click [OK].
3. Click [Start] button in Control Calculation.
4. When [Normal] is displayed, calculation has ended, and CL data has been created.
5. Right-click on [Normal], and select [Show CL by segment] to confirm [CL data].

→ Tool moves to the next Z level by step without moving up to Clearance Z, and perform arc Approach.

6. After you confirm CL, click [Clear CL] button to clear the displayed CL.

Execute calculation with [Slant approach]/[Connecting move: Ramp on face].

1. Click [Calculation condition sheet/Approach/Escape], and set the cutting conditions (10)-(13).

Slant plane direction

(5) Type : Slant
(6) Max length：100 (mm)
(8) Connecting move：ON
(9) Type：Ramp on face

2. After you set the condition, click [OK].
3. Click [Start] button in Control Calculation.
4. When [Normal] is displayed, calculation has ended, and CL data has been created.
5. Right-click on [Normal], and select [Show CL by segment] to confirm [CL data].

→ Tool moves to the next Z level by 5 degree without moving up to Clearance Z, and perform cutting.

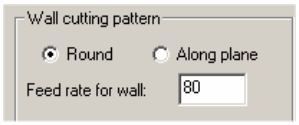
6. After you confirm CL, click [Clear CL] button to clear the displayed CL.

How to set cutting method for area cutting

→ Calculation condition sheet: Machining

Select Wall cutting pattern for area cutting.

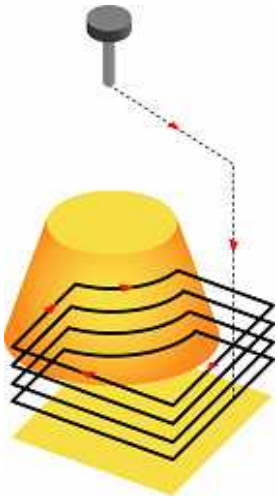
Selecting cutting pattern is valid only when you specify area for area cutting with [Calculation area surface].



• Round

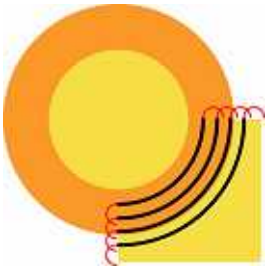
It is assumed that outer contour of [Calculation area surface] is wall by the system. Tool travels around the wall, and perform connecting move along the wall.

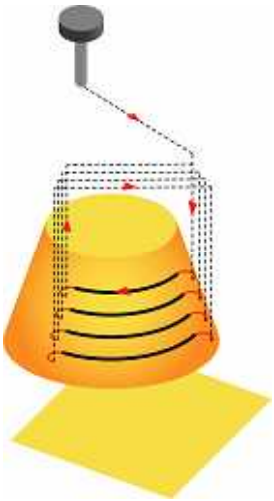
When you select [Round], specify [Feed rate for wall].



• Along plane

Tool travels along the check surface within the specified range of calculation area surface.





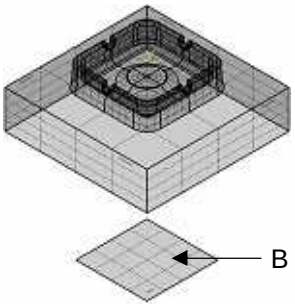
When you specify area for area cutting with [Cutting area surface], tool always travels along the check surface.

<<Exercise>>

Set [Layer 4] for active layer. Execute [Area cutting] with the following two methods, and compare the result.

Specify area for [Area cutting] as [Calculation area surface].

1. Specify a profile. Profile name: Fini01
2. Click ▼ button on the right of [Calculation area surface] and press [Clear].
3. Click [Calculation area surface] button, and specify surface B.



Execute calculation with [Wall cutting pattern: Round].

1. Click [Calculation condition sheet/Machining], and set the cutting conditions (1)-(3).
   (1) Wall cutting pattern : Round
   (2) Plane cutting: Perform
   (3) Offset cutting: Automatic
2. Click [Calculation condition sheet/ Approach/Escape], and set the cutting conditions (4)-(5).
   (4) Type: Arc
   (5) Connecting move: On / Type: Move by step
3. After you set the condition, click [OK].
4. Click [Start] button in Control Calculation.
5. When [Normal] is displayed, right-click on [Normal], and select [Show CL by segment] to confirm [CL data].
   → Cutter paths, which travel around on the outer contour of [Calculation area surface], are output.
6. After you confirm CL, click [Clear CL] button to clear the displayed CL.

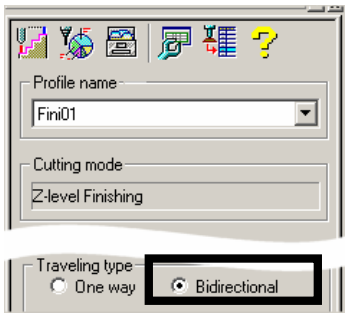
Execute calculation with [Wall cutting pattern: Along plane].

1. Click [Calculation condition sheet/Machining], and set the cutting conditions (6).
   (6) Wall cutting pattern : Along plane
2. After you set the condition, click [OK].
3. Click [Start] button in Control Calculation.
4. When [Normal] is displayed, right-click on [Normal], and select [Show CL by segment] to confirm [CL data].
   → Cutter paths, which are created by offsetting [Check surface] are output.
5. After you confirm CL, click [Clear CL] button to clear the displayed CL.

How to set Bi-directional traveling for area cutting

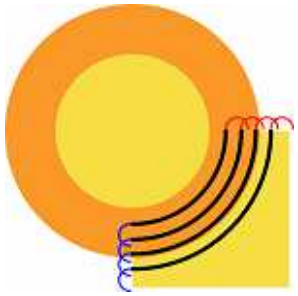
→ Control window

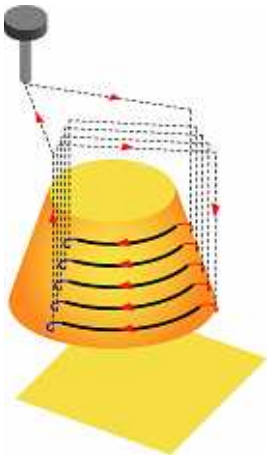
For area cutting, [Bi-directional] can be specified for [Traveling type].



• One way

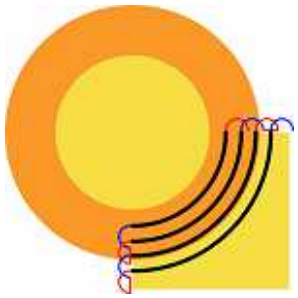
Tool travels in only one direction. It moves up to Clearance Z, and moves to the next Z level.

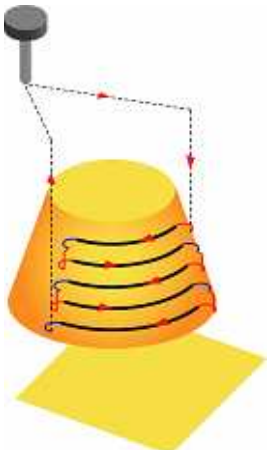




• Bi-directional

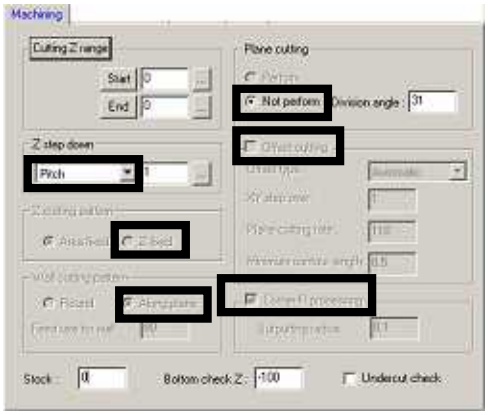
Connects the two closest points of each open paths.





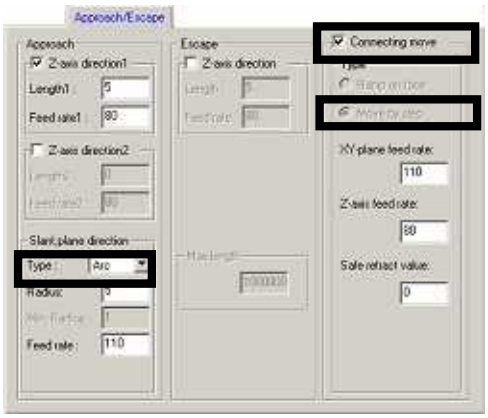
When [Bi-directional] is selected, the following items are automatically determined by the system.

<Machining>



(1) Z step down: Pitch
(2) Z cutting pattern: Z fixed
(3) Wall cutting pattern: Along plane
(4) Plane cutting: Not perform
(5) Offset cutting: OFF
(6) Corner R processing: OFF

<Approach/Escape>



(7) Slant/Plane direction: Arc
(8) Connecting move: ON [Move by step]

If you set [Bi-directional cutting], confirm the descriptions of the next page [How to set safe distance for Bi-directional cutting], and set [Connecting move].

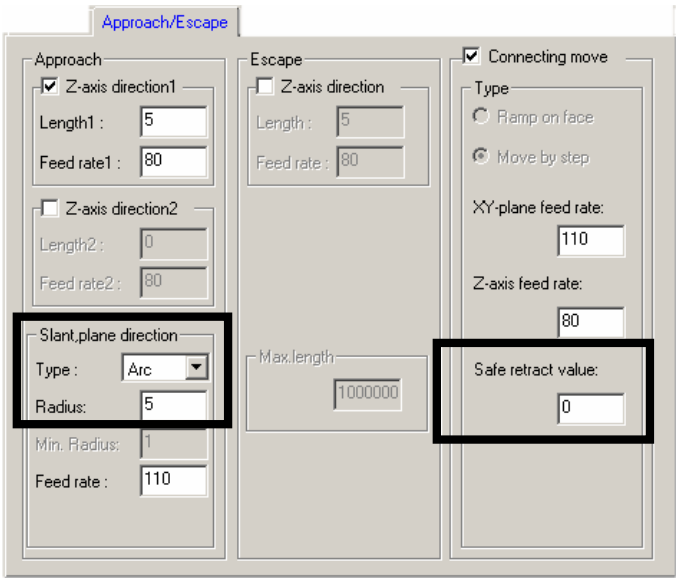
How to set safe distance for Bi-directional cutting

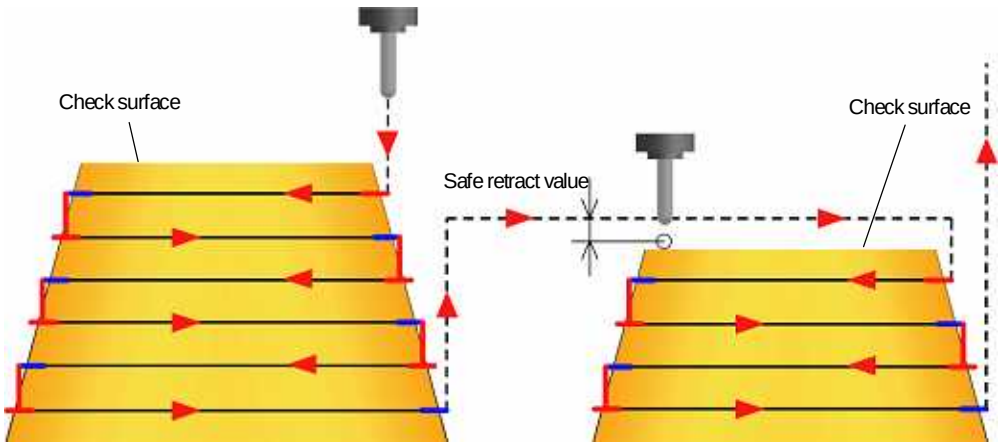
→ Calculation process sheet: Approach/Escape

When you execute [Bi-directional traveling] for plural cutting areas, specify [Safe retract value (stock)] to prevent tool's interference with work from occurring when tool moves between areas.

* When you specify [Connecting move], set radius of Arc Approach keeping the following relation.

Condition for inserting Arc : Connecting move < Radius of Arc Approach x 2]





Specifying [Safe retract value] is available only when [Bi-directional cutting] is performed.

When [Safe retract value] is specified, interference check for Approach is executed.

(1) If the distance between the start point of Approach and the check surface is longer than the specified [Safe retract value], Arc Approach is performed horizontally to the check surface.

(2) If the distance between the start point of Approach and the check surface is shorter than the specified [Safe retract value], Arc Approach is performed at the position that is [Safe retract value] far from the check surface.

(3) Even if Arc Approach is performed at the position that is [Safe retract value] far from the check surface, and tool still interferes the shape, linear Approach is performed at the position that is [Safe retract value] far from the check surface.

Chapter 5

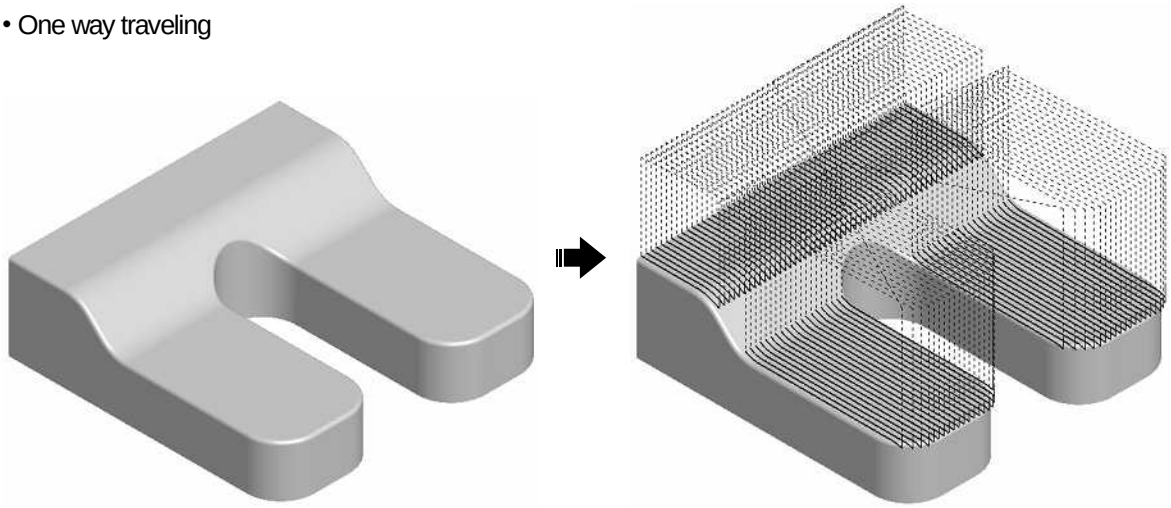
Finishing (Flat part)

- (Scanning-line Area) -

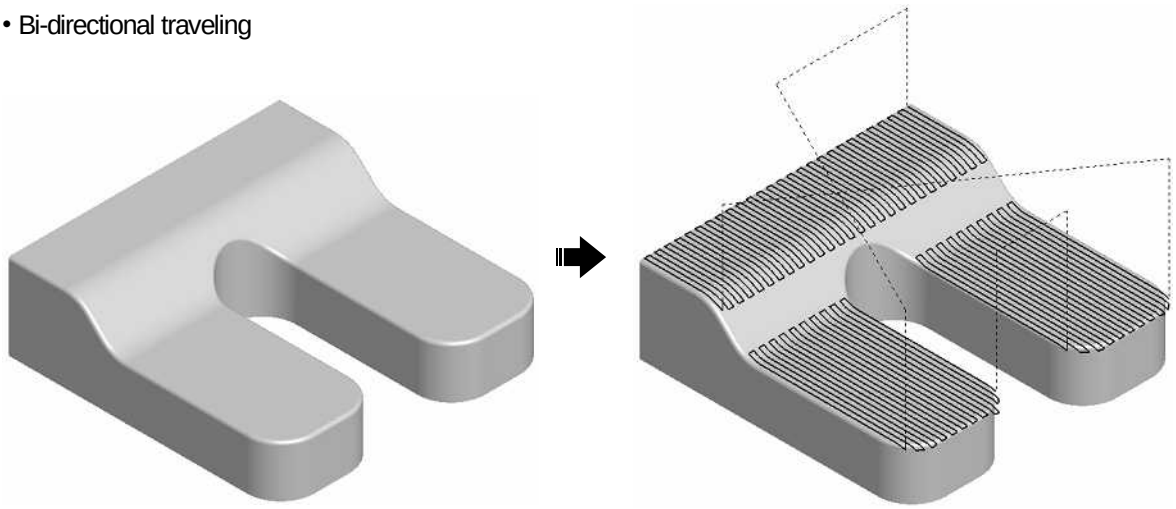
Overview: Scanning-line Area

This cutting strategy performs finishing on near-horizontal area in one direction.
Cutter paths are output on the near-horizontal area that is determined by specifying angle.
With this function, paths for [flat area] or [Near-horizontal area] that cannot be cut by [Z-level Finishing] are output.

• One way traveling



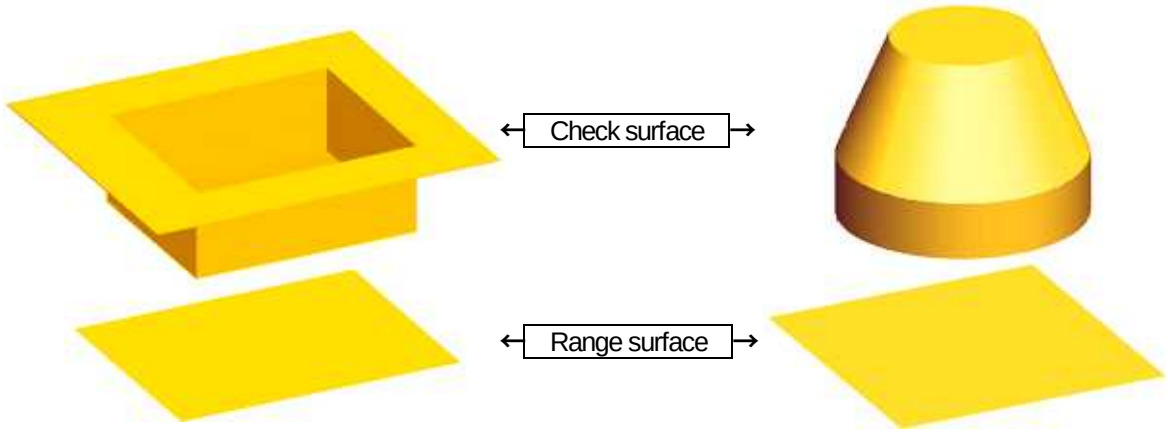
• Bi-directional traveling



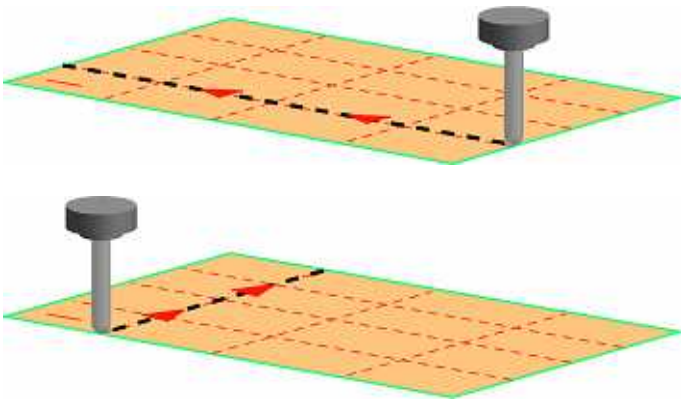
What are needed for model shape? (Precautions/Restrictions)

Prepare the followings as model shape.

(1) Cutting shape + Parting line surface (Check surface)
(2) Calculation area surface in plane direction / Surface for determining traveling direction(Range surface)



In Scanning-line Area, traveling direction of cutter paths is determined by [U/V direction (mesh direction) of the Range surface]. When you create a [Range surface], make the mesh direction to be same as the traveling direction.

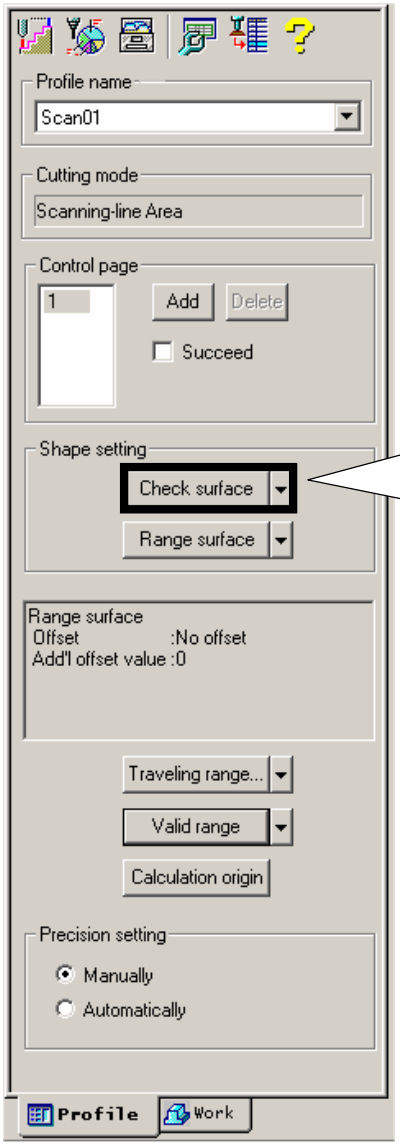


- Precautions for creating [Check surface] is the same as the one for [Z-level Rough Cutting]. (P.74-75)
- Precautions for creating [Range surface] is the same as the one for [Calculation area surface] in [Z-level Rough Cutting]. (P.74-77) Besides, be sure not to expand a surface using [Expand Surface] command. (If the expanded surface is used, calculation error occurs.)

How to set cutting model shape

→ Control window

Set [Check surface] in shape setting on the control window.



(1) Click [Check surface] button.

(2) Specify surfaces of cutting shape.

* For cavity shape, specify a parting line surface as well.

(3) Press the center mouse button to execute the command.

To edit the setting, click ▼ button on the right of [Check surface], and select [Clear part] or [Clear All].
When you select [Clear part], the confirmation dialog is displayed. Click [OK] to execute the command.

<<Exercise>>

Set as follows with in model file of ashtray shape that you created in[Text Book for Intensive Course].
If you operate with the model file that you performed Rough cutting, omit the following procedure 1, 2, 4, and set [Layer5] for active layer.

1. Open the model file. CD-ROM /Intensive Course / CAMmodel2.gmd [Layer5]

2. Save the model file. <Destination folder/File name>
D:/cam-tool /cam-toolFile /Exercise2 / CAMmodel2.gmd

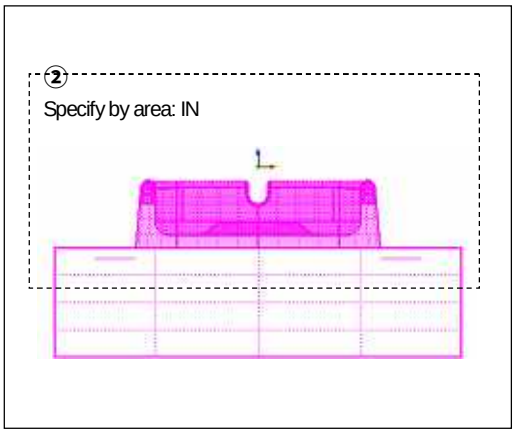
3.Create a new profile (1) Profile name : Scan01
(2) Cutting strategy: Scanning-line Area
(3) Initial setting file : (blank)
(4) Continuous creation : OFF
(5) Add to calculation process: OFF → [Finish]

4. Create a new calculation process sheet. <Destination folder/File name>

D:/cam-tool /cam-toolFile /Exercise2 / CAMcalculation2

5. Add the profile to calculation process sheet.

6. Specify [Check surface] in the profile.

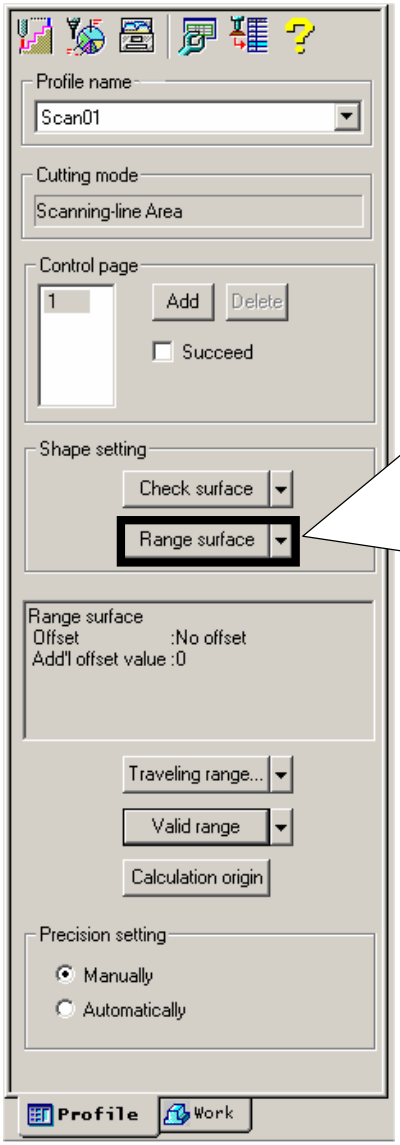


(1) Click [Check surface] button.
(2) Specify a surface for cutting shape. (Specify by area: In)
(3) Press the center mouse button to determine the specifying.

7. After you set, confirm with [CAM mode] function.

How to set cutting range/cutting direction → Control window

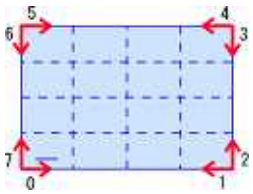
Set [Range surface] in Shape setting on the control window.



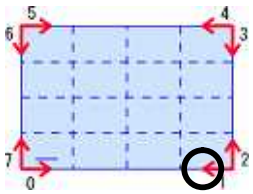
(1)Click [Range surface] button.

(2) Specify a surface that cutter paths are output.
* Only a single surface can be specified as Range surface.

(3) Arrows are displayed at four corners of the specified surface.

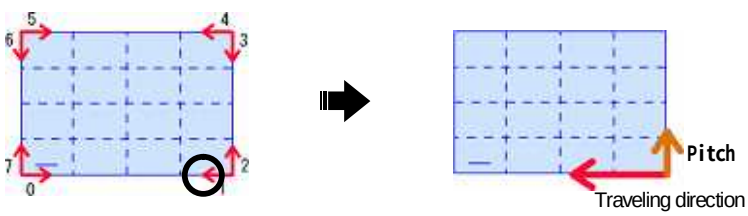


(4) Specify an arrow for traveling direction.



Confirm with CAM mode

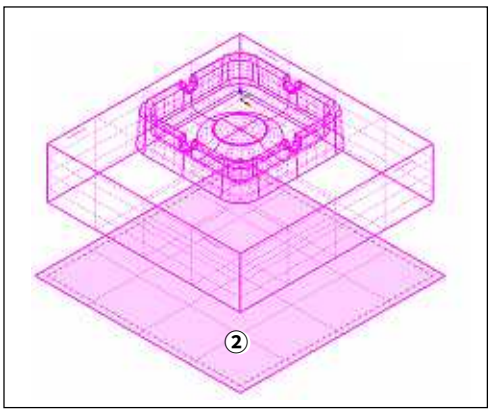
In CAM mode, [Traveling direction(Green) and [Pitch direction(Yellow)] are displayed.



<<Exercise>>

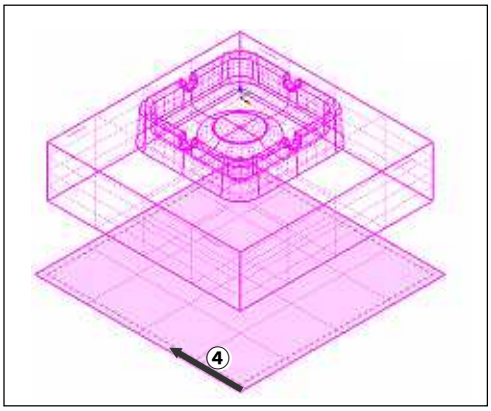
Next, specify [Range surface].

1. Specify [Range surface] on the profile.



(1) Click the [Range surface] button.

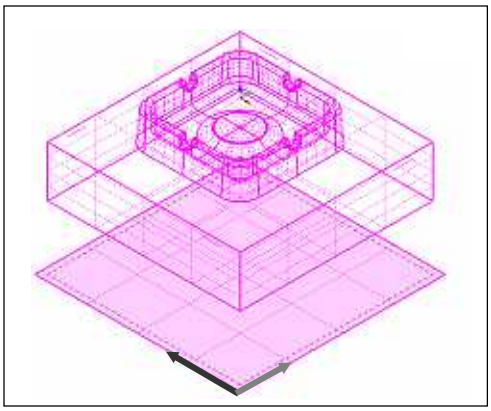
(2) Specify a surface that cutter paths are output.



(3) Four arrows are displayed at four corners of the specified surface.

(4) Specify an arrow for [Traveling direction].

2. Confirm the shape setting with CAM mode.



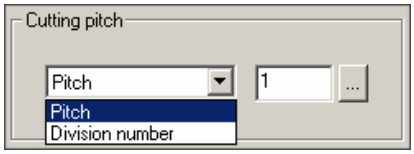
* Long arrow = Traveling direction (Green)

* Short arrow = Pitch direction (Yellow)

How to set cutting pitch

→ Calculation condition sheet:/Machining

Choose either [Pitch] or [Division number] for [Cutting pitch] and input value.



<<Exercise>>

Set [Cutting pitch] and execute calculation.

1. Specify a profile Profile name：Scan01
2. Open the [Calculation condition sheet/Machining] and set the following calculation conditions (1)-(5).

   (1) Cutting pitch：Pitch [1(mm)]
   (2) Traveling type：One way
   (3) Stock：0.5 (mm)
   (4) Division angle：32 degree
   (5) Wall escape allowance：0.1(mm)

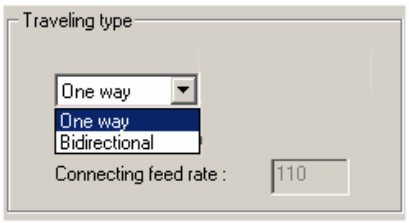
3. Open the [Calculation condition sheet/Tool] and set the following calculation conditions (6)- (7).

   (6) Tool diameter： 6
   (7) Cutting edge R： 3

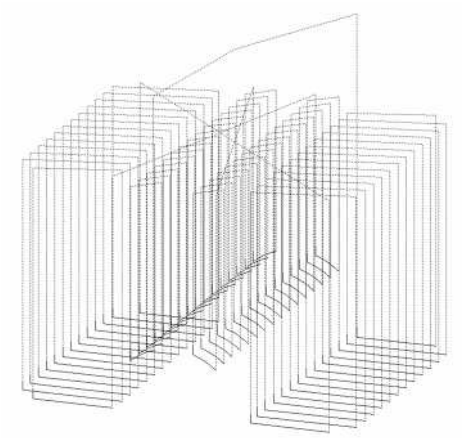
4. After you set the condition, click [OK].
5. Click [Start] button in Control Calculation.
6. When [Normal] is displayed, calculation has ended, and CL data has been created.
7. Right-click on [Normal], and select [Show CL] to confirm [CL data].
   → Cutter paths of one way traveling are output on the near-horizontal area.
8. After you confirm CL, click [Clear CL] button to clear the displayed CL.

How to set One way / Bi-directional traveling → Calculation condition sheet:/Machining

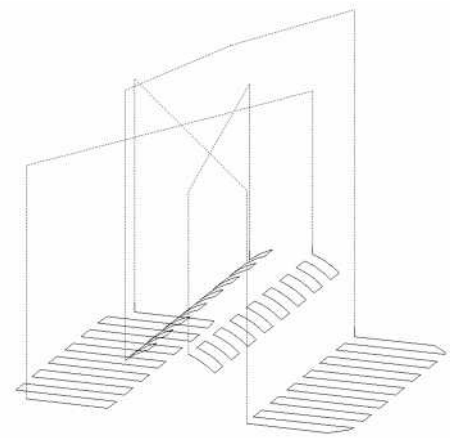
Choose either [One way] or [Bi-directional] for traveling type. For [Bi-directional] traveling, specify connecting feed rate.



• One way



• Bi-directional



<<Exercise>>

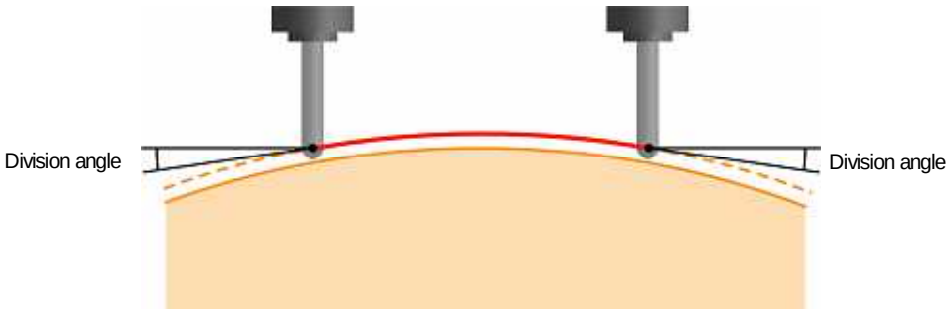
After you confirm the output cuter paths of one way traveling, output paths of Bi-directional traveling.

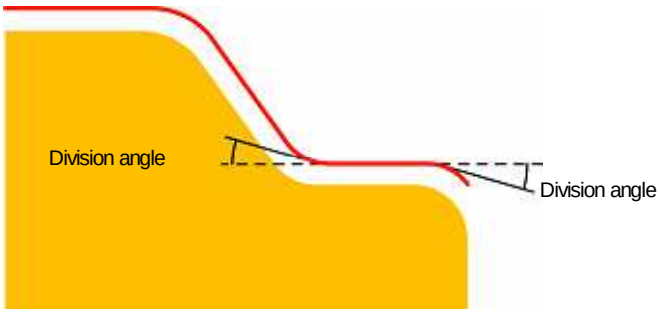
1. Specify a profile Profile name：Scan01
2. Open the [Calculation condition sheet/Machining] and set the following calculation condition (1).
   (1) Traveling type： Bi-directional
3. After you set the condition, click [OK].
4. Click [Start] button in Control Calculation.
5. When [Normal] is displayed, calculation has ended, and CL data has been created.
6. Right-click on [Normal], and select [Show CL] to confirm [CL data].
   → Cutter paths of Bi-directional traveling are output on the near-horizontal area.
7. After you confirm CL, click [Clear CL] button to clear the displayed CL.

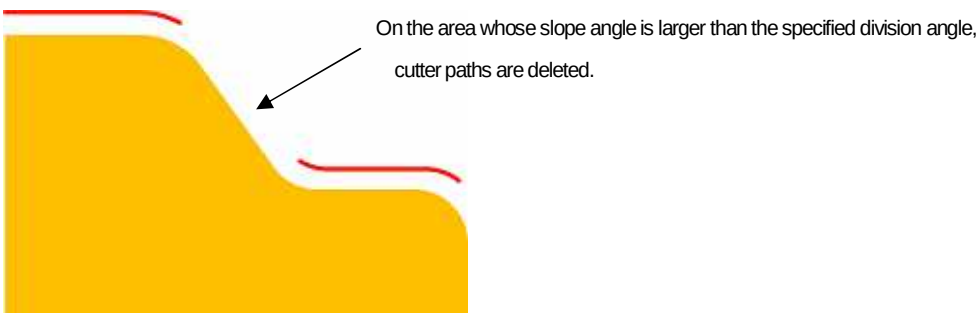
How can I limit area to output paths by specifying angle

→ Calculation condition sheet:/Machining

Specify angle for detecting horizontal area in the specified cutting shape.
Angle between a path and XY plane is described as [Division angle].
When an area whose division angle is smaller than the specified [Division angle] exists in a shape, the area is defined as horizontal area, and paths are output on that area.



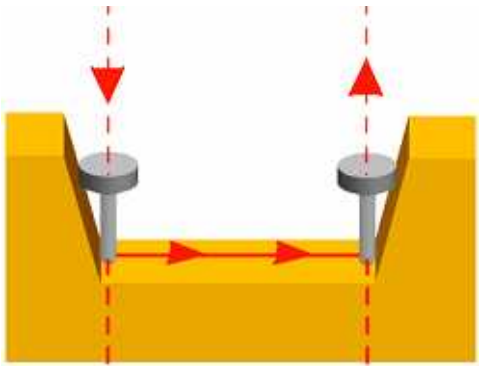




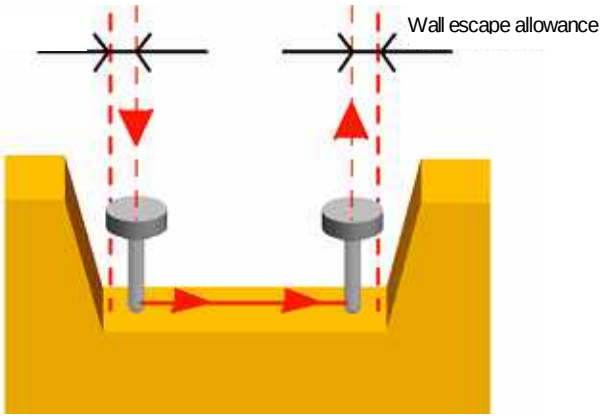
How to set safe retract value toward wall → Calculation condition sheet:/Machining

If a wall exists in a cutting shape, tool may interfere the wall when Approach/Escape is performed. By specifying [Wall escape allowance], safe retract distance toward wall can be set. As a result, it prevents tool's interference with wall from occurring.

• Wall escape allowance : 0



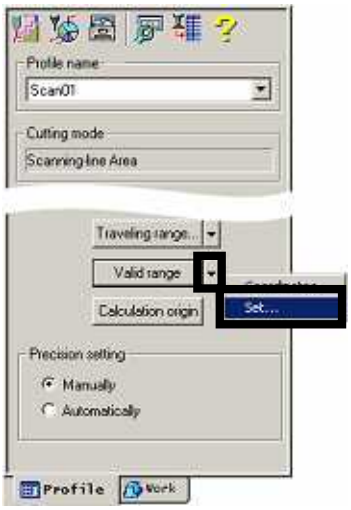
• Wall escape allowance : Specify by numerical value

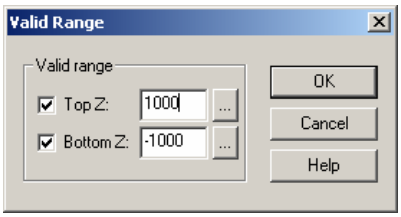


How to set conditions for area cutting (Z-axis direction) → Control window

By specifying valid range, cutter paths can be output only in the specified Z area.
Input value for [Top/Bottom Z] with tool tip value.

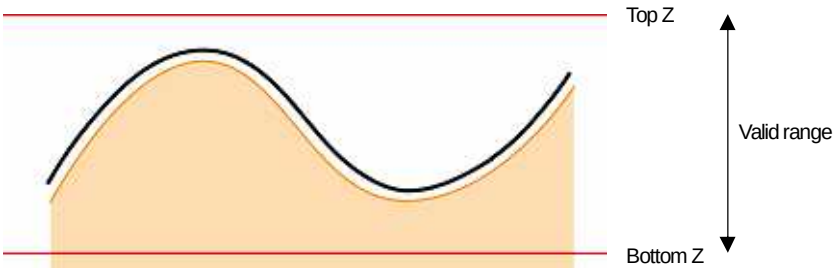
<Control window>



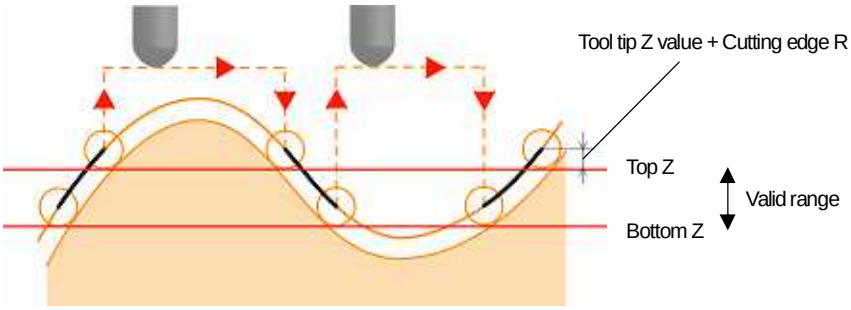


* When the checkbox of each item is OFF, valid range is not specified.

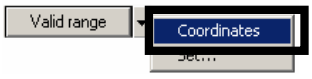
• When not limiting valid range



• When limiting valid range



To set [Top Z]/[Bottom Z] of valid range, you can specify coordinates of the model shape.



* Be sure to specify it in [View direction: (Front)/(Right)].

<<Exercise>>

Set [Valid range], and output cutter paths only for the horizontal area that is on the center of model shape (Z=-20).

1. Specify a profile Profile name：Scan01

2. Click the ▼ button on the right of [Valid range] on the control window, and select [Set].

3. Set [Valid range (1)-(2)].

(1) Top Z: ON [-18]
(2) Bottom Z: ON [-22]

4. After you set the condition, click [OK].
5. Click [Start] button in Control Calculation.
6. When [Normal] is displayed, calculation has ended, and CL data has been created.
7. Right-click on [Normal], and select [Show CL] to confirm [CL data].

→ Cutter paths are output on the area that is lower than [Division angle: 32], and in the range of Z= -18 to Z = -22.

8. After you confirm CL, click [Clear CL] button to clear the displayed CL.

Set [Valid range], and output cutter paths only for the flat area at the bottom of model shape (Z=-30).

2. Click the ▼ button on the right of [Valid range] on the control window, and select [Set].

3. Set [Valid range] (3)-(4).

(3) Top Z: ON [-28]
(4) Bottom Z: ON [-32]

3. After you set the condition, click [OK].
4. Click [Start] button in Control Calculation.
5. When [Normal] is displayed, calculation has ended, and CL data has been created.
6. Right-click on [Normal], and select [Show CL] to confirm [CL data].

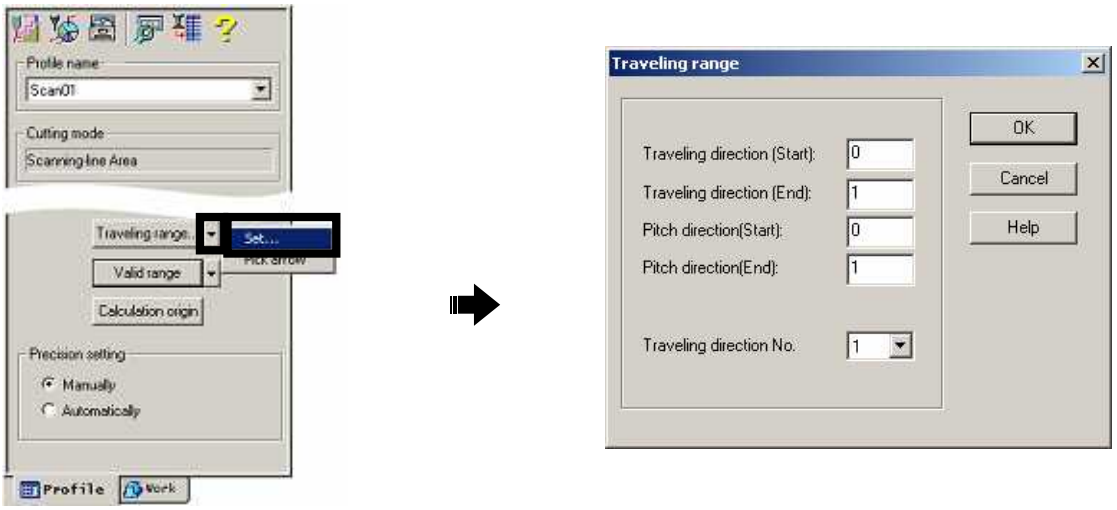
→ Cutter paths are output on the area that is lower than [Division angle: 32], and in the range of Z= -28 to Z = -32.

7. After you confirm CL, click [Clear CL] button to clear the displayed CL.

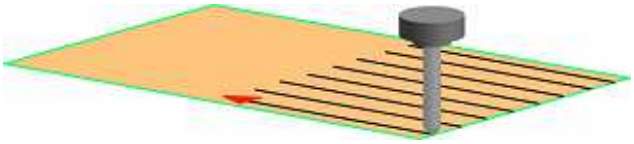
How to set conditions for area cutting (Plane direction) → Control window

For area cutting, create [Range surface] for the limited area. Or, set [Traveling range] by entering value for [Start]/[End] of [Traveling direction] and [Start]/[End] of [Pitch direction]. [Traveling direction number] can be changed.

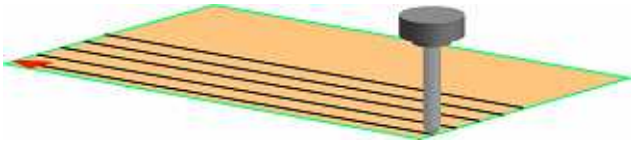
<Control window>



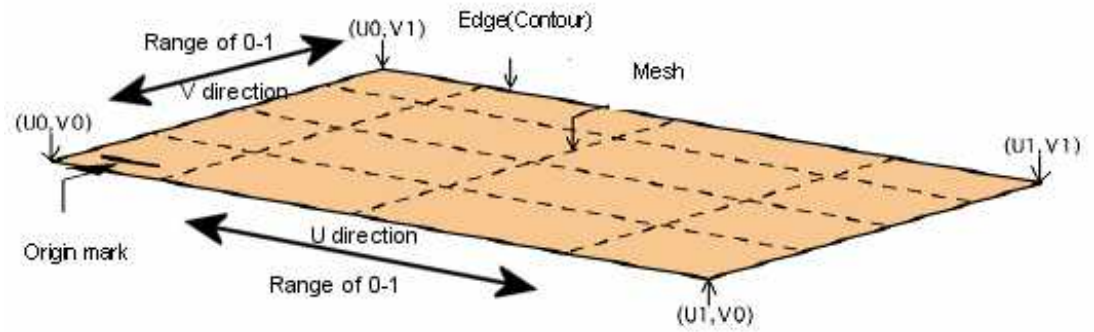
• Traveling direction [Start : 0] [End : 0.5]



• Pitch direction [Start : 0] [End : 0.5]



Surface has parameters of 0-1 in each U/V direction.



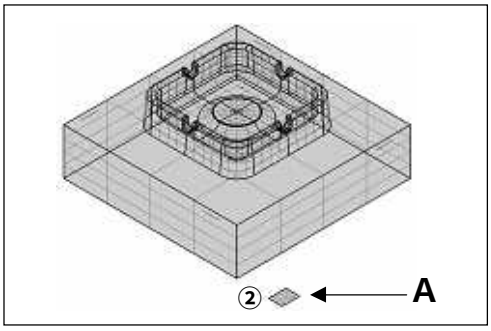
<<Exercise>>

Set [Layer6] for active layer. Output cutter paths only on the area of cigarette to be placed.
Specify [Range surface] for setting cutting range in plane direction, and [Valid range] for setting cutting range in Zdirection.

1. Specify a profile Profile name：Scan01

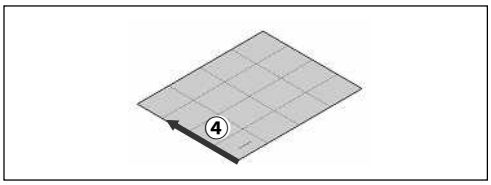
2. Press▼ button on the right of [Range surface] and click [Clear].

3. Click [Range surface] button and specify the [surface A].



(1) Click [Range surface] button.

(2) Specify the [surface A].



(3) Four arrows are displayed at each corner of the specified surface.

(4) Specify an arrow for defining [Traveling direction].

4. Press▼ button on the right of [Valid range] and click [Set].

5. Set [Valid range (1)-(2)], and click [OK].
   (1) Top Z: ON [-12]
   (2) Bottom Z: ON [-15]

6. Open the [Calculation condition sheet/Machining] and set the following calculation conditions (1)-(3).

   (1) Cutting pitch：Pitch [0.2(mm)]
   (2) Traveling type：One way
   (3) Division angle：32 degree

7. Open the [Calculation condition sheet/Tool] and set the following calculation conditions (4)- (5).

   (4) Tool diameter：4
   (5) Cutting edge R：2

8. Click [Start] button in Control Calculation.

9. When [Normal] is displayed, calculation has ended, and CL data has been created.

10. Right-click on [Normal], and select [Show CL] to confirm [CL data].

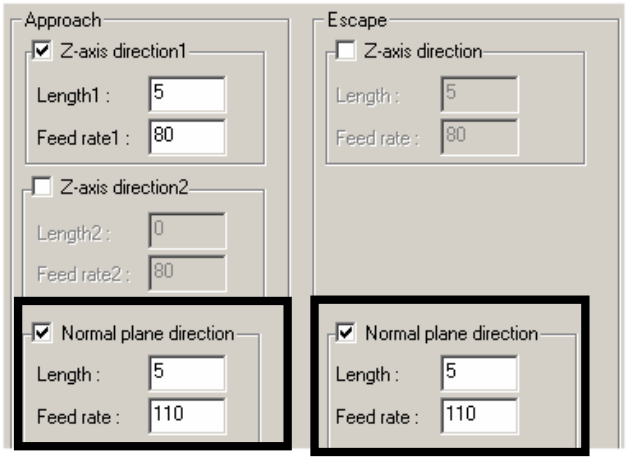
→ Cutter paths are output on the area that is lower than [Division angle: 32], and in the range of Z= -12 to Z = -15.

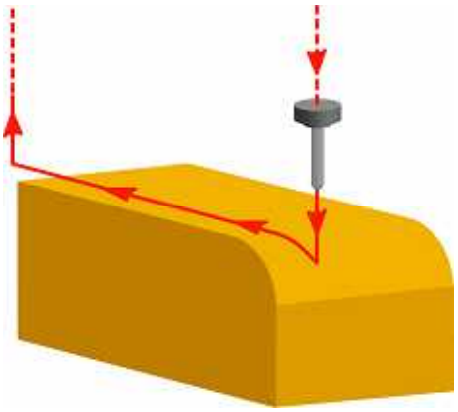
11. After you confirm CL, click [Clear CL] button to clear the displayed CL.

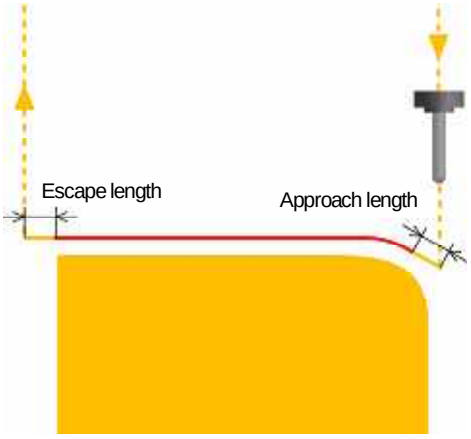
How to set Approach

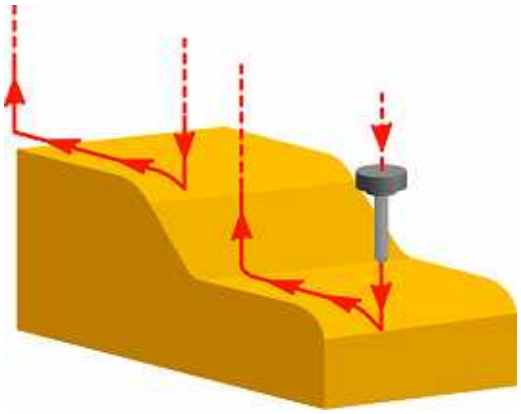
→ Calculation condition sheet：Approach/Escape

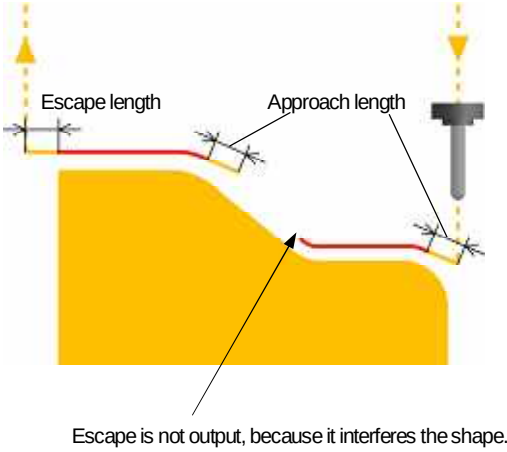
Approach/Escape is performed from normal plane direction (Tangent line direction of cutter paths). If Approach or Escape interferes [Check surface], they are not output.











Escape is not output, because it interferes the shape.

<<Exercise>>

Output Approach/Escape in the [Tangent line direction] of cutter paths.

1. Specify a profile Profile name：Scan01

2. Open the [Calculation condition sheet/Approach/Escape] and set the following calculation conditions (1)-(5).

(1) Approach/Z-axis direction1：ON Length 5(mm)
(2) Approach/Z-axis direction2：ON Length 5(mm)
(3) Approach/Normal plane direction：ON Length 2(mm)
(4) Escape/Z-axis direction：ON Length 5(mm)
(5) Escape/Normal plane direction：ON Length 2(mm)

3. After you confirm the setting, click [OK] button.
4. Click [Start] button in Control Calculation.
5. When [Normal] is displayed, calculation has ended, and CL data has been created.
6. Right-click on [Normal], and select [Show CL] to confirm [CL data].

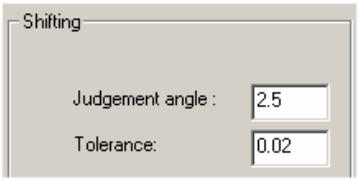
→ Approach(2mm)/Escape(2mm) are output in the [Tangent line direction(direction of expanding path)] of cutter paths.

7. After you confirm CL, click [Clear CL] button to clear the displayed CL.

How to set cutting precision

→ Calculation condition sheet: Precision

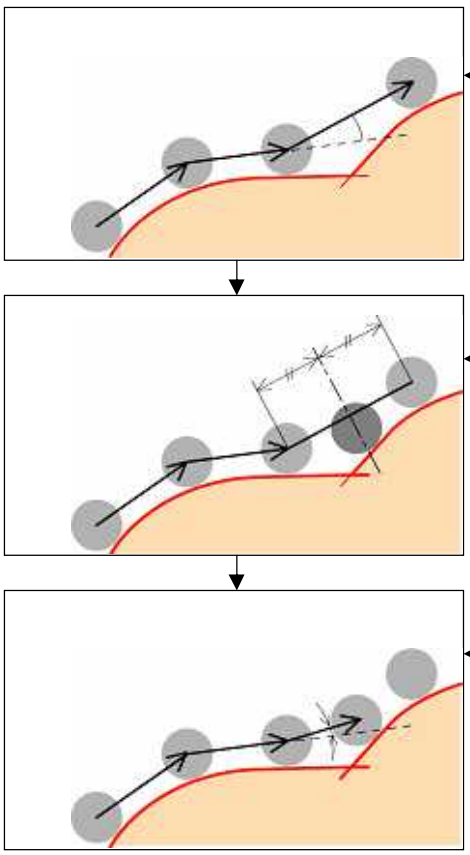
In Scanning-line Area, cutting precision is determined by [Chordal deviation] / [Reduction] / [Tolerance of curvature] / [Shifting].



- Chordal deviation >> Refer to Z-level Rough Cutting (P.105)
- Reduction >> Refer to Z-level Rough Cutting (P.105)
- Tolerance of curvature >> Refer to Z-level Rough Cutting (P.106)

• Shifting - Judgement angle

To output smooth paths for area, which tool moves from a surface to an adjacent surface, partially, calculation needs to be executed with higher precision than the specified value for [Chordal deviation]. When angle that is the change between the two path's vector is larger than the specified [Judgement angle], the path is divided into two paths.



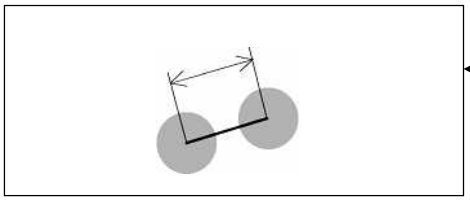
When the angle that is the change between the two path's vector is larger than the specified [Judgement angle]…,

A middle point is positioned, and the path is divided into two paths.

If the angle that is the change between the first path and the divided path is still larger than the specified [Judgement angle], the same process is repeated until the angle becomes smaller.

• Shifting - Tolerance

Specify length of path for [Shifting] processing.



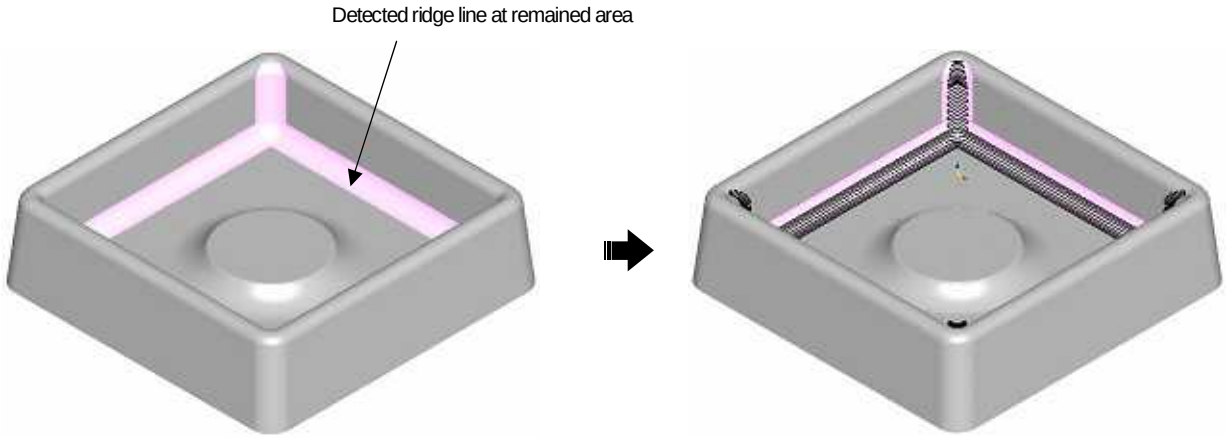
Paths are divided by the specified [Tolerance] value.

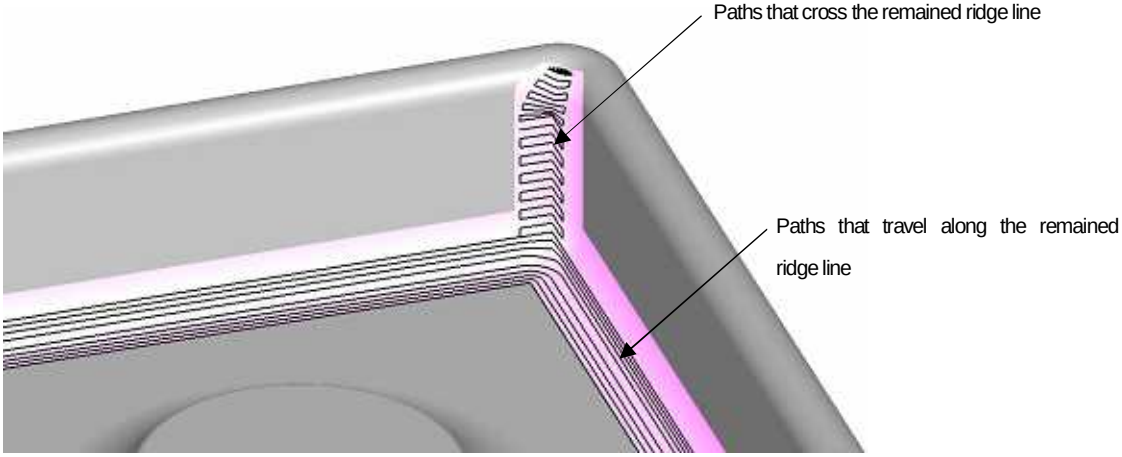
Chapter 6

Cutting Edge - Re-machining -

Overview - Re-machining

This is the cutting strategy that removes the stock left from the previous process. By specifying [Radius of tool that is used in the previous process], area that stock is left is detected by the system. Only Ball end mill tool is available for both [Tool for previous process] and [Tool for current process]. Cutter paths are output either along the detected ridgeline, or across the detected ridge line(travel at right angle).

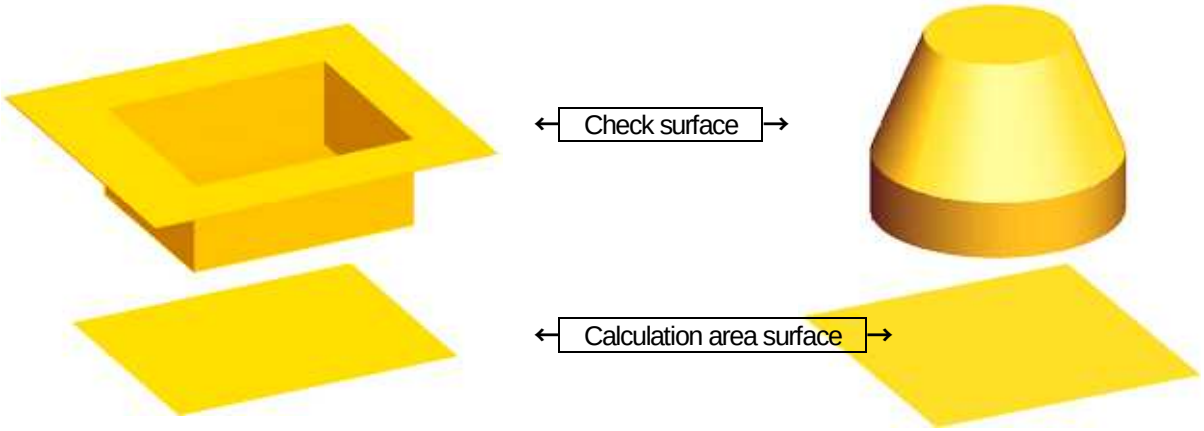




What are needed for model shape? (Precautions/Restrictions)

Prepare the followings as model shape.

(1) Cutting shape + Parting line surface (Check surface)
(2) Surface for determining calculation area in plane direction (Calculation area surface)



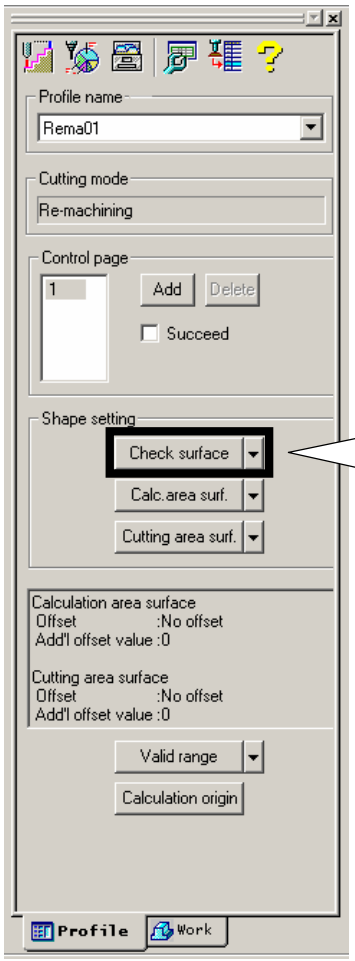
- Precautions for creating [Check surface] is the same as the one for [Z-level Rough Cutting].(P.74-75)
- Precautions for creating [Calculation area surface] is the same as the one for [Z-level Rough Cutting]. (P.74-77)

Besides, be sure not to expand a surface using [Expand Surface] command. (If the expanded surface is used, calculation error may occur.)

How to set cutting model shape

→ Control window

Set [Check surface] in Shape setting on the control window.



(1) Click [Check surface] button.

(2) Specify surfaces of cutting shape.
* For cavity shape, specify a parting line surface as well

(3) Press the center mouse button to determine the specifying.

To edit the setting, click ▼ button on the right of [Check surface], and select [Clear Part] or [Clear All].
When you select [Clear part], the confirmation dialog is displayed. Click [OK] to execute the command.

<<Exercise>>

The model shape of ashtray that you created in [Text Book for Intensive Course - CAD -] has a few corner areas, which stock is left from the previous process. Therefore, practice with the model with many corners.
If you operate with the model file that you performed Rough cutting, omit the following operations 1, 2, 4, and set [Layer7] for active layer.

1. Open the model file. CD-ROM / Intensive Course /CAMmodel2.gmd [Layer7]

2. Save the model file. <Destination folder/File name>
D:/cam-tool /cam-toolFile /Exercise2 / CAMmodel2.gmd

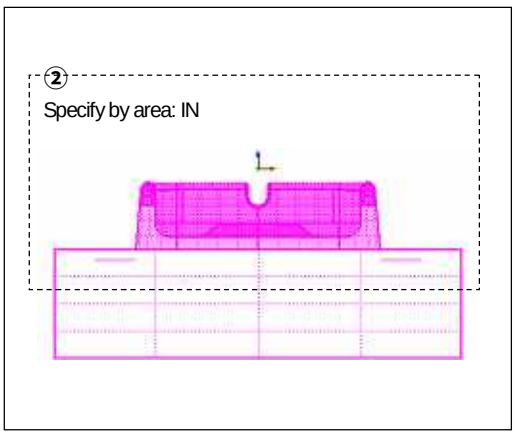
3.Create a new profile (1) Profile name：Rema01
(2) Cutting strategy: Re-machining
(3) Initial setting file：(blank)
(4) Continuous creation：OFF
(5) Add to calculation process: OFF >> [Finish] button

4. Create a new calculation process list. <Destination folder/File name>

D:/cam-tool /cam-toolFile /Exercise2 / CAMcalculation2

5. Add the profile to calculation process list.

6. Specify [Check surface] in the profile.

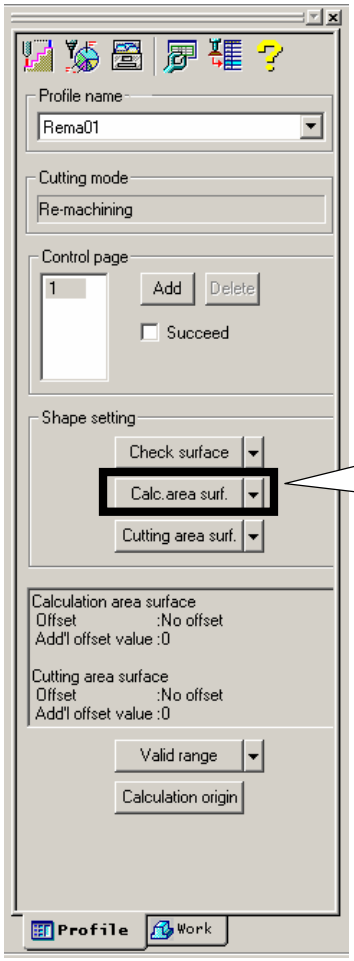


(1) Click [Check surface] button.
(2) Specify a surface for cutting shape. (Specify by area: In)
(3) Press the center mouse button to determine the specifying.

7. After you set, confirm with [CAM mode] function.

How to set cutting range in plane direction → Control window

Set [Calculation area surface] in Shape setting on the control window.



(1)Click [Calculation area surface] button.

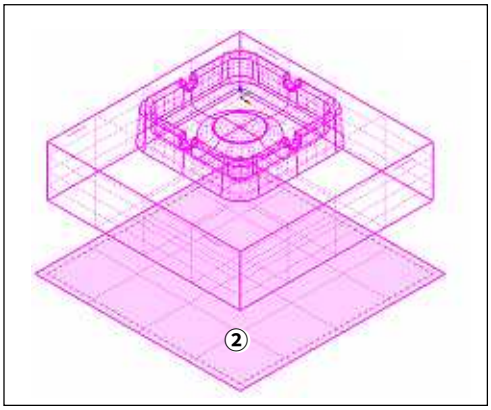
(2) Specify a surface that cutter paths are output.
* Only a single surface can be specified.

To edit the setting, click ▼ button on the right of [Calculation area surface], and click [Clear].
When you click [Clear], the confirmation dialog is displayed. Click [OK] to execute the command.

<<Exercise>>

Next, specify [Calculation area surface].

1. Specify [Calculation area surface] on the profile.



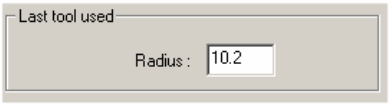
(1) Click the [Calculation area surface] button.

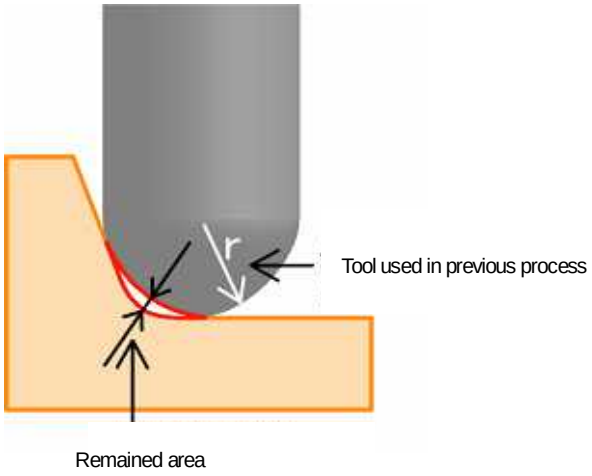
(2) Specify a surface that cutter paths are output.
* Only single surface can be specified as 【Calculation area surface.]

2. Confirm the shape setting with CAM mode.

How to set for detecting remained area → Calculation condition sheet:/Machining

By specifying [Radius of tool that is used in the previous process], the remained stock is detected. Only Ball end mill tool is available for both [Tool for previous process] and [Tool for current process].





For [Last tool used], specify a little larger value than the radius value of tool used in the previous process, so that the path for the remained area and the previous path are connected smoothly.

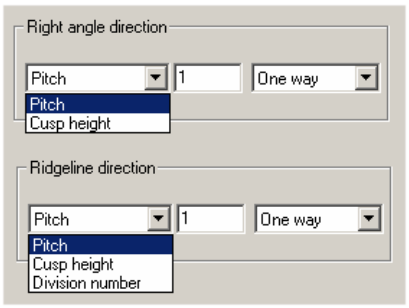
Cutter paths are not output when…

- radius of cavity fillet of a shape and radius of tool used in previous process are the same or approximate.
- radius of tool used in previous process and radius of tool in current process are the same or approximate.
- the detected remained area is smaller than the value for [Chordal deviation].

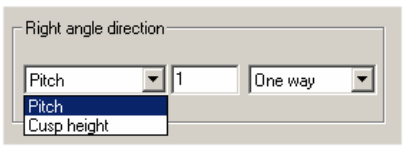
How to set cutting pitch

→ Calculation condition sheet:/Machining

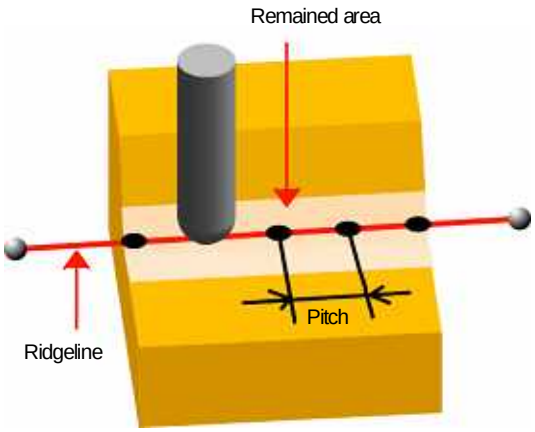
Specify [Cutting pitch] for each [Right angle direction] and [Ridgeline direction].



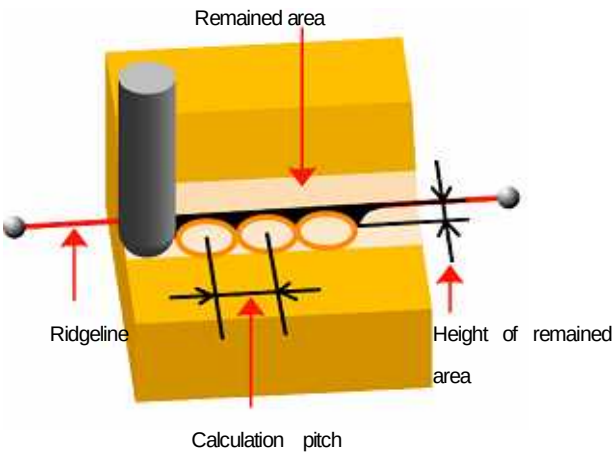
Choose either [Pitch] or [Cusp height] for [Right angle direction pitch].
For [Right angle direction], pitch is calculated based on the detected ridgeline.



<Pitch>

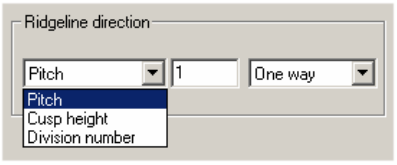


<Cusp height>

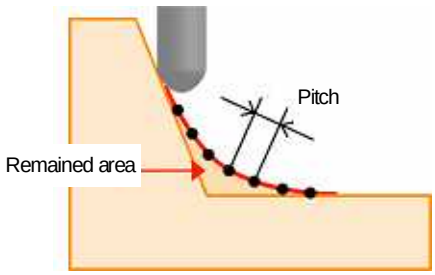


<<Chapter 6 Cutting edge(Re-machining)>>

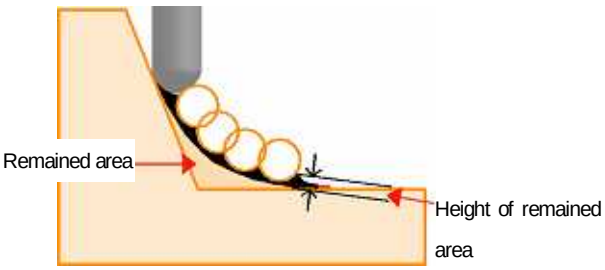
Select [Ridgeline direction pitch] from [Pitch], [Cusp height] or [Division number].
For [Right angle direction], pitch is calculated based on the area that stock is left with the tool in the previous process.



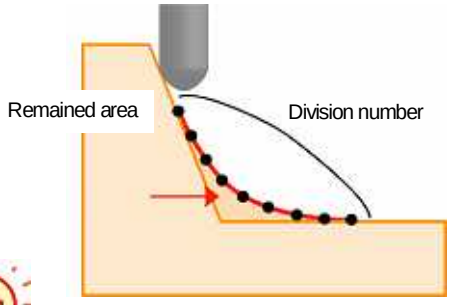
<Pitch>



<Cusp height>

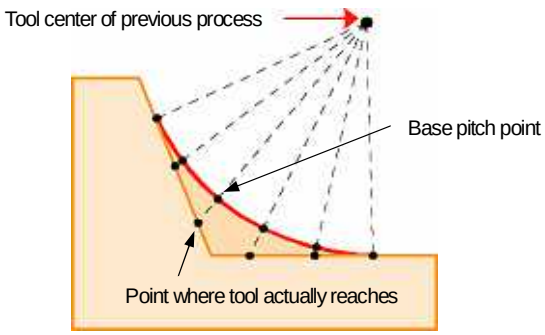


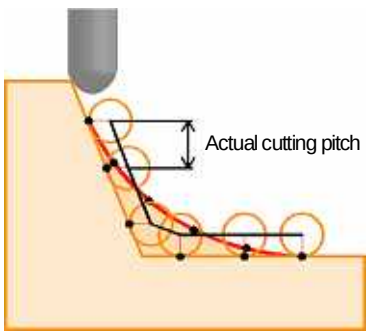
<Division number>



Cutting pitch does not look like equal? ([Calculated cutting pitch]/[Actual cutting pitch])

To calculate cutting pitch, a line is lengthened from the [base pitch point], which is calculated with the tool center of the tool used in previous process, to the [Check surface]. Tool, actually, cuts at each point, where the lengthened line reaches the [Check surface]. Therefore, actual cutting pitch may not be equal.





<<Exercise>>

Set Cutting conditions and execute calculation.

1. Specify a profile Profile name：Rema01

2. Open the [Calculation condition sheet/Machining] and set the following calculation conditions (1)-(9).

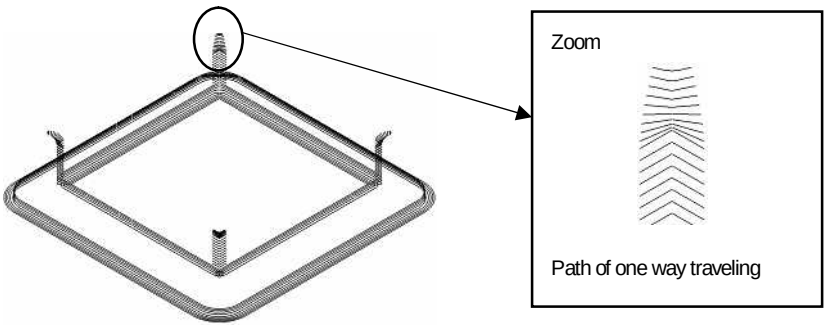
   (1) Last tool used: 5.1
   (2) Right angle direction /Pitch type: Pitch
   (3) Right angle direction/Value：1 (mm)
   (4) Right angle direction/ Cutting type：One way
   (5) Ridgeline direction/ Pitch type: Pitch
   (6) Ridgeline direction/ Value：1 (mm)
   (7) Ridgeline direction/ Cutting type：One way
   (8) Divide ridgeline：OFF
   (9) Stock：0.5 (mm)

3. Open the [Calculation condition sheet/Tool] and set the following calculation conditions (10)- (11).

   (10) Tool diameter： 4
   (11) Cutting edge R： 2

4. After you set the condition, click [OK].
5. Click [Start] button in Control Calculation.
6. When [Normal] is displayed, calculation has ended, and CL data has been created.
7. Right-click on [Normal], and select [Show CL] to confirm [CL data].

   → Cutter paths of [Cutting pitch:1mm] are output in both [right angle direction] and [Ridgeline direction].



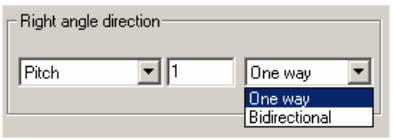
8. After you confirm CL, click [Clear CL] button to clear the displayed CL.

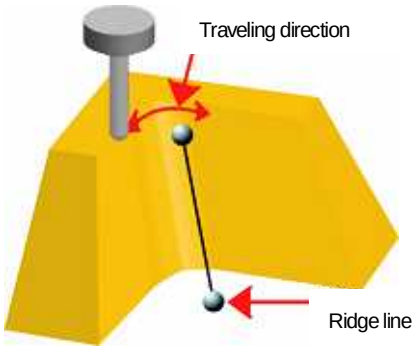
How to set One way / Bi-directional traveling → Calculation condition sheet:/Machining

Specify [One way traveling] or [Bi-directional traveling] for both [Ridgeline direction] and [Right angle direction].

Right angle direction

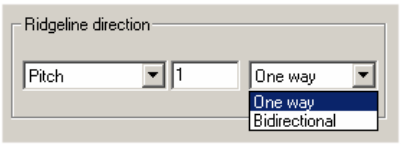
Select [One way] or [Bi-directional] for the type of outputting paths that travel across the remained ridgeline.

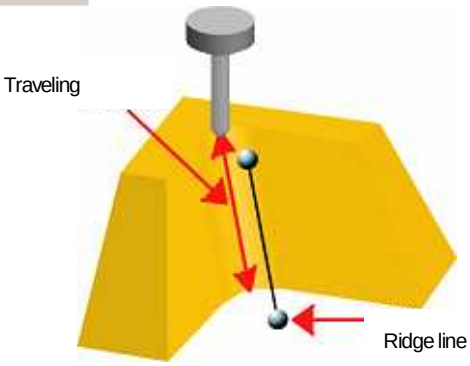




• Ridgeline direction

Select [One way] or [Bi-directional] for the type of outputting paths that travel along the remained ridgeline.

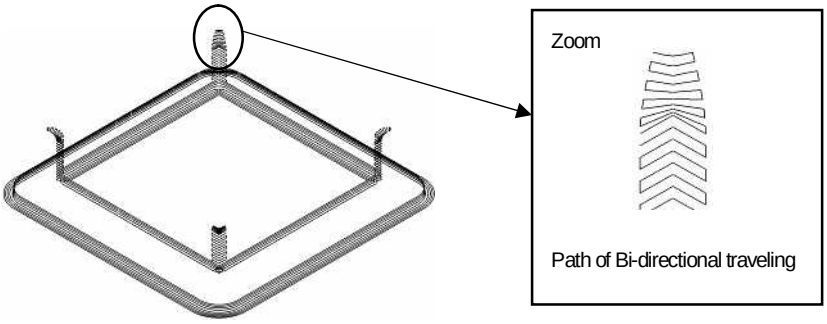




<<Exercise>>

After you confirm the output cuter paths of one way traveling, output paths of Bi-directional traveling.

1. Specify a profile Profile name：Rema01
2. Open the [Calculation condition sheet/Machining] and set the following calculation condition (1).
   (1) Right angle direction/ Cutting type： Bi-directional
3. After you set the condition, click [OK].
4. Click [Start] button in Control Calculation.
5. When [Normal] is displayed, calculation has ended, and CL data has been created.
6. Right-click on [Normal], and select [Show CL] to confirm [CL data].
   → Cutter paths of Bi-directional traveling are output in the [Right angle direction].



7. After you confirm CL, click [Clear CL] button to clear the displayed CL.

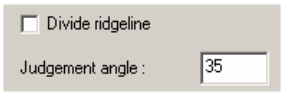
How to set Right angle direction / Ridgeline direction

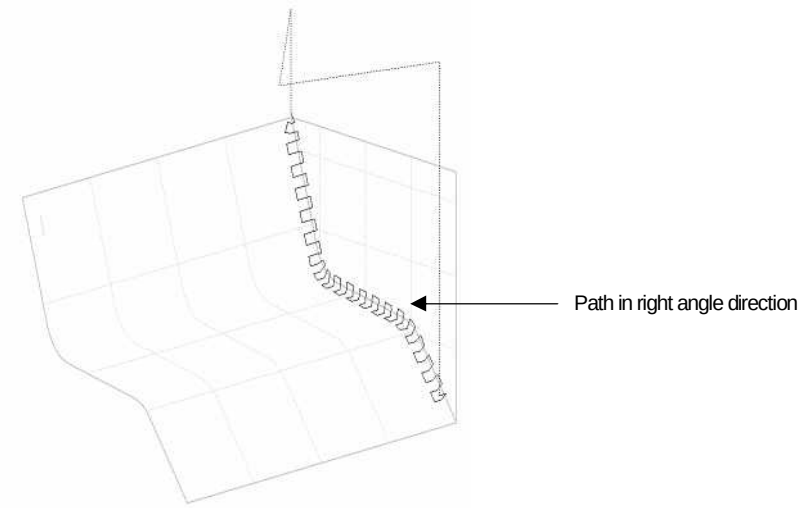
→ Calculation condition sheet:/Machining

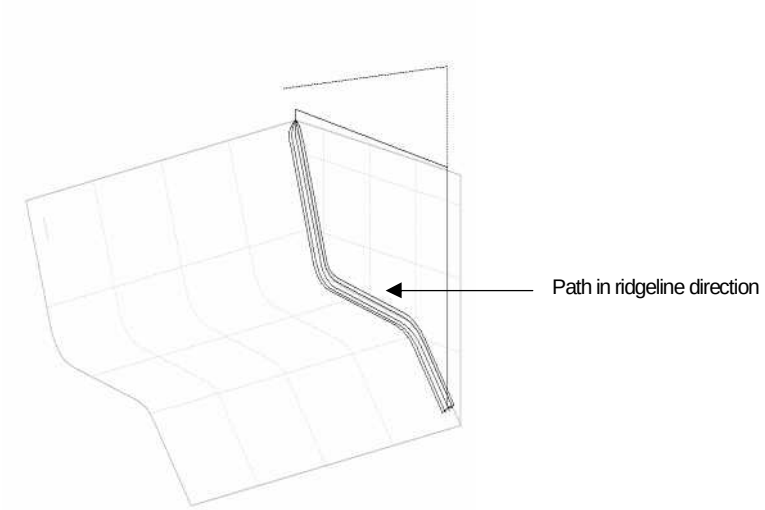
Select whether divide or not divide a remained ridgeline into the area for traveling in right angle direction and the area for traveling in ridgeline direction. Area is divided into the area for [traveling in right angle direction] and the area for traveling in ridgeline direction, by the specified [Judgement angle].

• Divide ridgeline : OFF

Not divide a remained ridgeline into areas. When angle between a line that connects [Maximum Z value] and [Minimum Z value] of ridgeline, and [Work plane] is larger than the specified judgement angle, tool travels in right angle direction. When it is smaller than the specified judgement angle, tool travels in ridgeline direction.

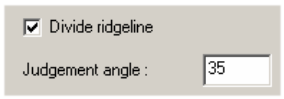


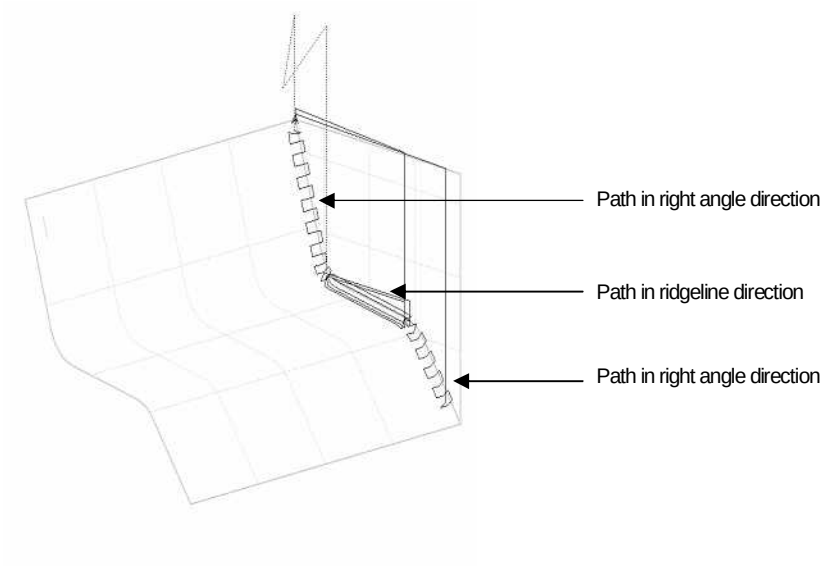




• Divide ridgeline： ON

Divide a remained ridgeline into the area for traveling in right angle direction and the area for traveling in ridgeline direction by the specified [Judgement angle].



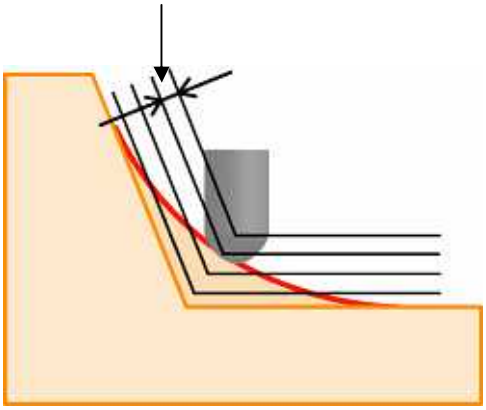


How to set for reducing tool load

→ Calculation condition sheet:/Machining

Specify the volume that tool removes the detected stock for one time. If all of the remained stock can be removed by the specified [Remain step], tool travels the area only on time. If it cannot be removed with one time traveling, tool travels the area multiple times.

Remain step（Cutting volume for one time traveling）



<<Exercise>>

Set [Remain step] and execute calculation.

1. Specify a profile Profile name : Scan01

2. Open the [Calculation condition sheet/Machining] and set the following calculation conditions (1)-(2).

   (1) Right angle direction/ Cutting type : One way

   (2) Remain step : 0.3 (mm)

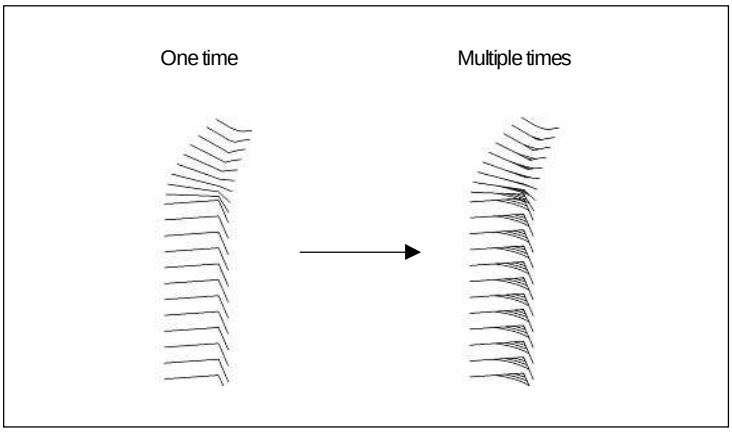
3. After you set the condition, click [OK].

4. Click [Start] button in Control Calculation.

5. When [Normal] is displayed, calculation has ended, and CL data has been created.

6. Right-click on [Normal], and select [Show CL] to confirm [CL data].

   → Tool travels multiple times on the remained area, depending on the specified [Remained step].

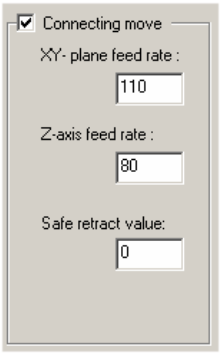


7. After you confirm CL, click [Clear CL] button to clear the displayed CL.

How to set connecting move between remained ridgelines

→ Calculation condition sheet: Approach/Escape

Specify whether to perform or not to perform [Connecting move] when tool travels between the plural detected ridgelines.
Performs connecting move between each path in one way traveling, and connecting move when tool repeats cutting multiple times in Bi-directional traveling.

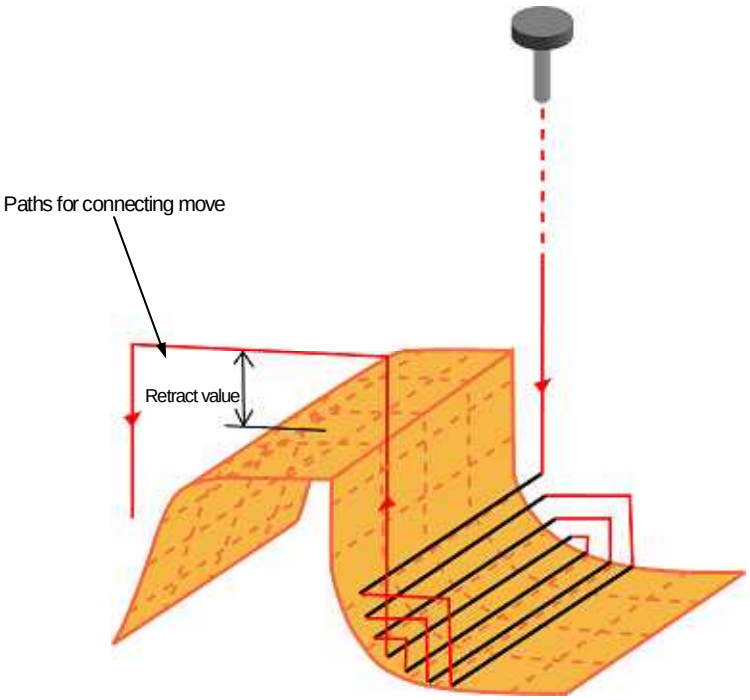


- Safe retract value

  This function is valid when [Ridgeline direction] is set for [Traveling direction].
  Specify [Safe retract value from the check surface] for connecting move between ridgelines.
  It prevents a path of connecting move from interfering check surface. (Connecting move=G01)

- XY plane feed rate/Z-axis feed rate

  Specify feed rate for connecting move in plane direction, and Z-axis direction.



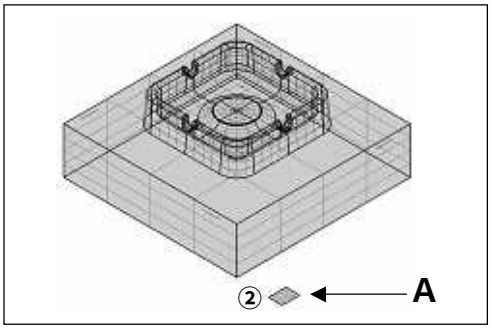
<<Exercise>>

Set [Layer8] for active layer. Set [Calculation area surface] again, and output cutter paths of connecting move.

1. Specify a profile Profile name : Rema01

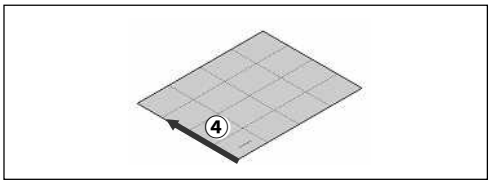
2. Press▼ button on the right of [Calculation area surface] and click [Clear].

3. Click [Calculation area surface] button and specify the [surface A].



(1) Click [Calculation area surface] button.

(2) Specify the [surface A].



4. Open the [Calculation condition sheet/Machining] and set the following calculation conditions (1)-(2).

(1) Ridgeline direction/Cutting type : Bi-directional

(2) Remain step: 0(mm)

5. Open the [Calculation condition sheet/Approach/Escape] and set the following calculation conditions (3)-(4).

(3) Connecting move : ON

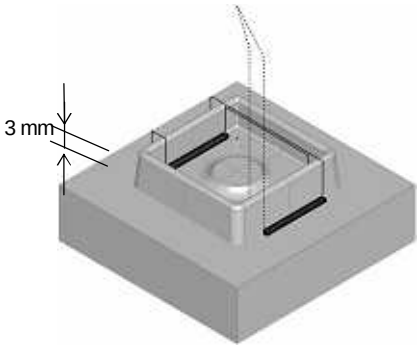
(4) Safe retract value : 3 (mm)

6. Click [Start] button in Control Calculation.

7. When [Normal] is displayed, calculation has ended, and CL data has been created.

8. Right-click on [Normal], and select [Show CL] to confirm [CL data].

→ Tool moves between the remained ridgelines with the [3mm] safe retract value from the [Check surface].



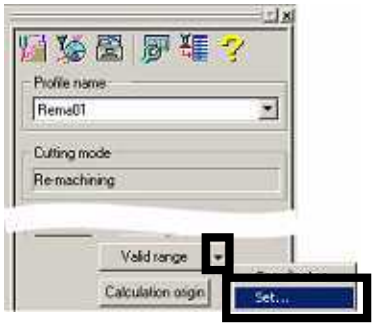
9. After you confirm CL, click [Clear CL] button to clear the displayed CL.

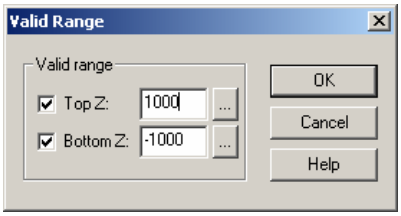
How to set conditions for area cutting

→ Control window

By specifying valid range, cutter paths can be output only in the specified Z area.
Input value for [Top Z/Bottom Z] with tool tip value.

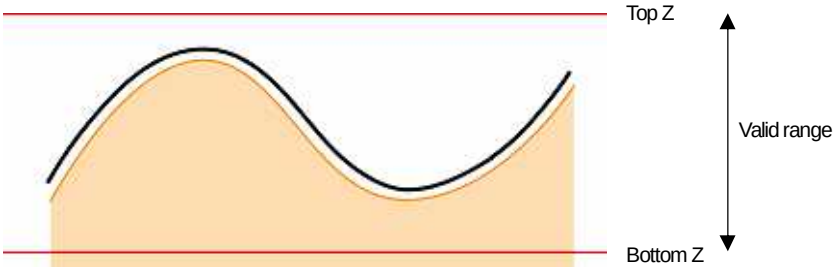
<Control window>



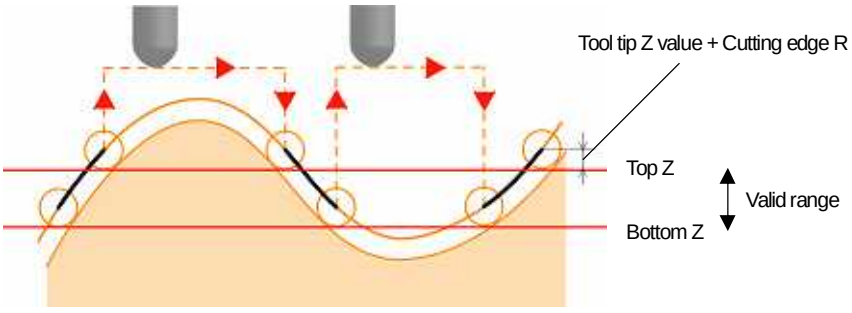


* When the check box of each item is OFF, valid range is not limited.

■ When not limiting valid range



■ When limiting valid range



To set [Top Z]/[Bottom Z] of valid range, you can specify coordinates of the model shape.

* Be sure to specify it in [View direction: (Front)/(Right)].

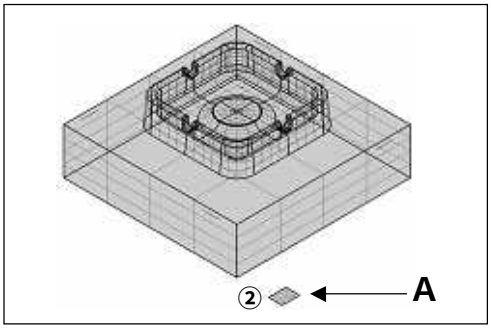
<<Exercise>>

Set [Layer9] for active layer. Set [Valid range], [Calculation area surface] and [Cutting area surface], and output cutter paths only for side surfaces that are outside of the shape, and corner areas of the bottom surface.

1. Specify a profile Profile name : Rema01

2. Press▼ button on the right of [Calculation area surface] and click [Clear].

3. Click [Calculation area surface] and [Cutting area surface] button and specify the [surface A] and [surface B].

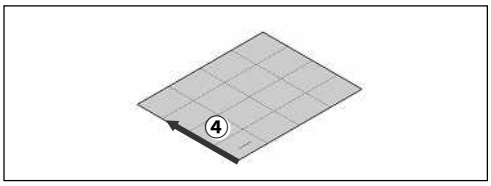


(1) Click [Calculation area surface] button.

(2) Specify the surface A.

(3) Click [Cutting area surface] button.

(4) Specify the surface B.



4. Press▼ button on the right of [Valid range] and click [Set].

5. Set [Valid range (1)-(2)], and click [OK].
   (1) Top Z: ON [-28]
   (2) Bottom Z: ON [-32]

6. Open the [Calculation condition sheet/Machining] and set the following calculation condition (1).
   (1) Ridgeline direction/Cutting type : One way

7. Open the [Calculation condition sheet/Approach/Escape] and set the following calculation condition (2).
   (2) Connecting move : OFF

8. After you set the conditions, click [OK].

9. Click [Start] button in Control Calculation.

10. When [Normal] is displayed, calculation has ended, and CL data has been created.

11. Right-click on [Normal], and select [Show CL] to confirm [CL data].

   → Cutter paths are output in the range of [Z = -28] - [Z = -32].

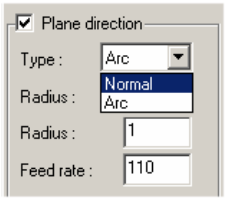
12. After you confirm CL, click [Clear CL] button to clear the displayed CL.

<<Chapter 6 Cutting edge(Re-machining)>>

How to set Approach

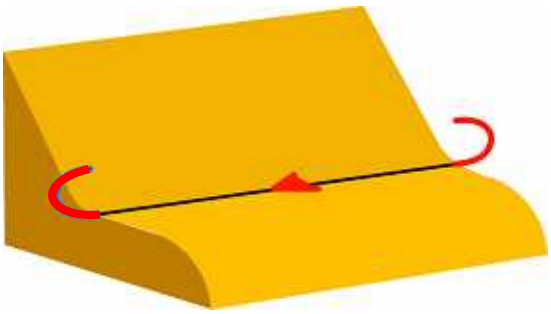
→ Calculation condition sheet: Approach/Escape

Choose either [Arc] or [Normal] for [Plane direction Approach/Escape].
(For Plane direction, the setting of Approach and Escape are the same.)



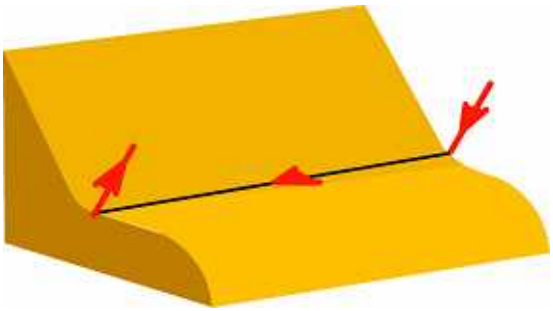
• Arc

Perform Arc Approach/Escape of the specified radius from the normal direction of the surface.



• Normal

Perform Arc Approach/Escape from the normal direction of the surface.



<<Exercise>>

Output Approach/Escape and execute calculation.

1. Specify a profile Profile name：Rema01

2. Open the [Calculation condition sheet/Approach/Escape] and set the following calculation condition (1)-(4).

(1) Approach/Z-axis direction1：ON Length 5(mm)
(2) Approach/Z-axis direction2：OFF
(3) Approach/Plane direction：ON
Type：Arc
Radius：2(mm)

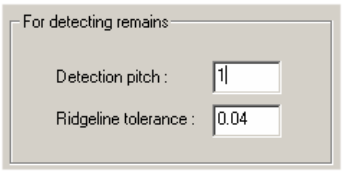
(4) Escape/Z-axis direction：ON Length 5(mm)

3. After you confirm the setting, click [OK] button.
4. Click [Start] button in Control Calculation.
5. When [Normal] is displayed, calculation has ended, and CL data has been created.
6. Right-click on [Normal], and select [Show CL] to confirm [CL data].
   → Arc Approach(R2)/Arc Escape(R2) are output from the Normal line direction of cutter paths.
7. After you confirm CL, click [Clear CL] button to clear the displayed CL.

<<Chapter 6 Cutting edge(Re-machining)>>

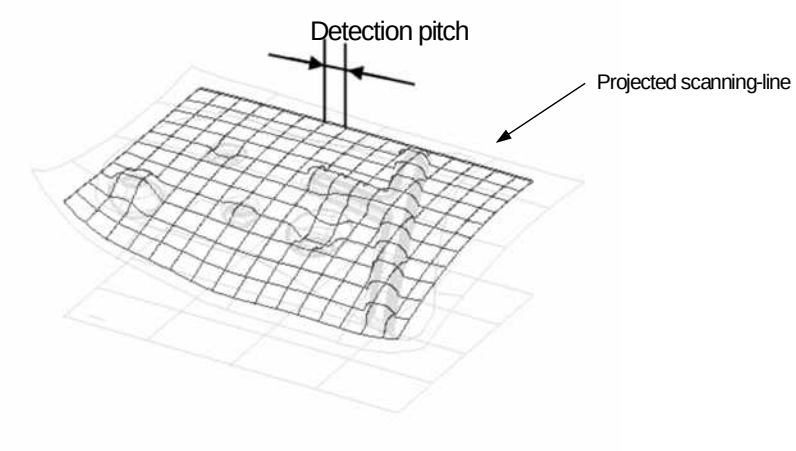
How to set precision for detecting remained area → Calculation condition sheet: Precision

Set precision for detecting remained area by [Detection pitch] and [Ridgeline tolerance].



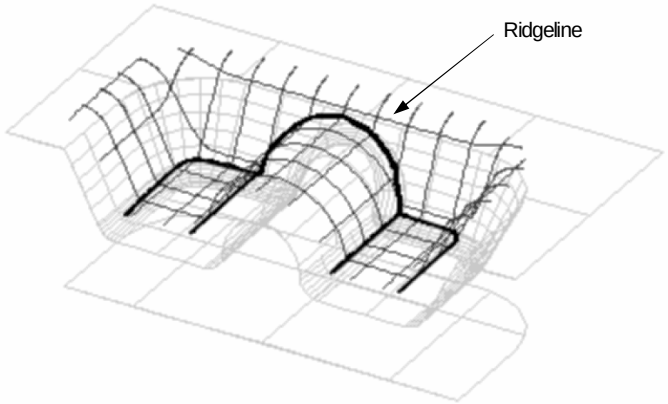
• Detection pitch

Specify pitch for scanning-line, which is used to detect ridgeline on the remained area.
As a guide, input about [tool radius]. For the shape that has many slopes, specify smaller value depending on the shape.



• Ridgeline tolerance

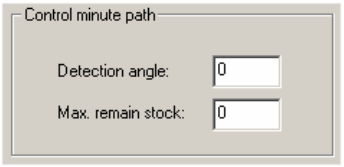
Specify precision for detecting ridgeline of the shape. When it is not specified, an error occurs.
Basically, input larger value than the specified [Chordal deviation].



How to set not to output micro paths

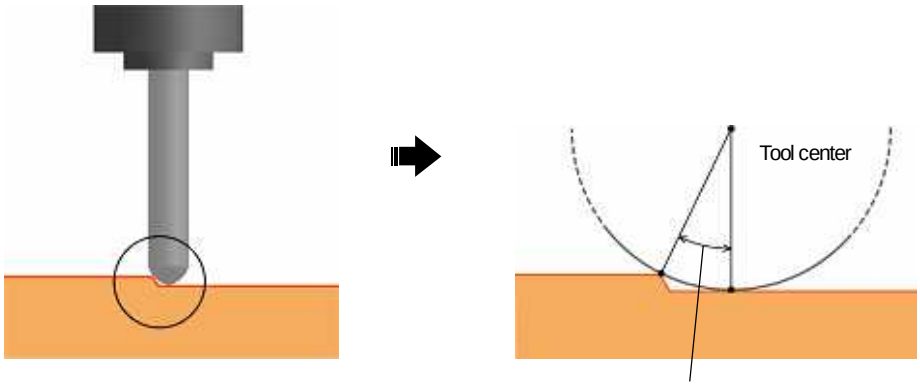
→ Calculation condition sheet: Precision

Delete micro(unnecessary) paths in the paths, which are output for the remained area, by specifying [Detection angle] and [Max. remain stock].



- Detection angle

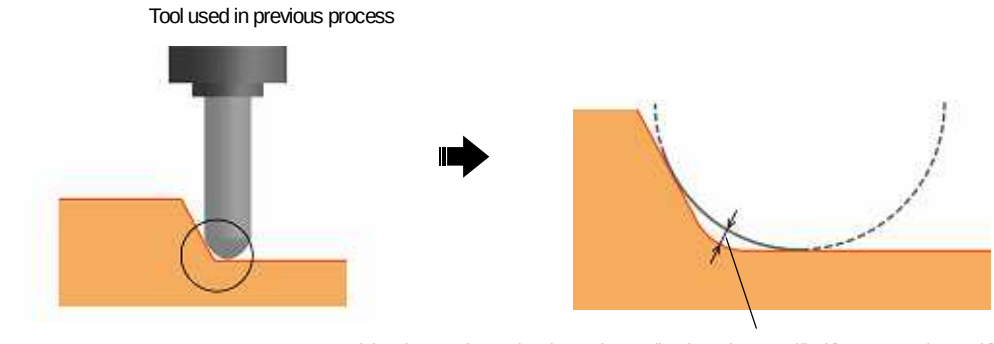
When angle between the two lines, which connects the [two points that a tool is on the shape] and [Tool center], is smaller than the specified [Detection angle], no cutter path is output. It enables deleting unnecessary paths such as a path for a minute step.



If this angle is smaller than the specified [Detection angle], no cutter path is output.

- Max. remain stock

When remained volume is smaller than the specified [Max. remain stock], no path is output.
Unnecessary paths can be omit in case such as when stock is left on a minute area.



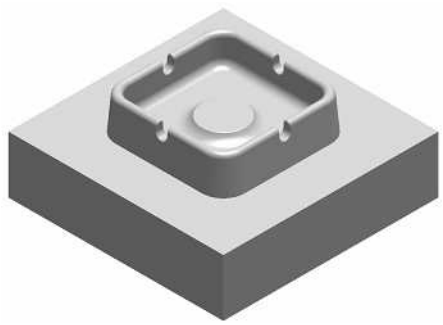
If the detected remained area is smaller than the specified [Max. remain stock], no cutter path is output.

Chapter 7

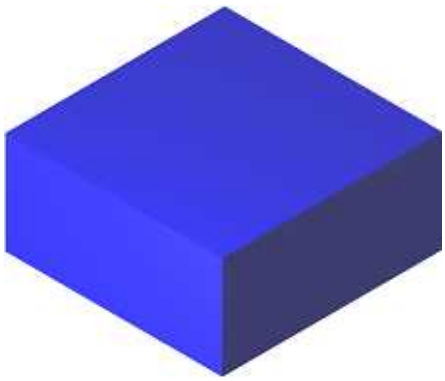
Example of creating process

Example of creating processes for cutting core shape

<Model shape>

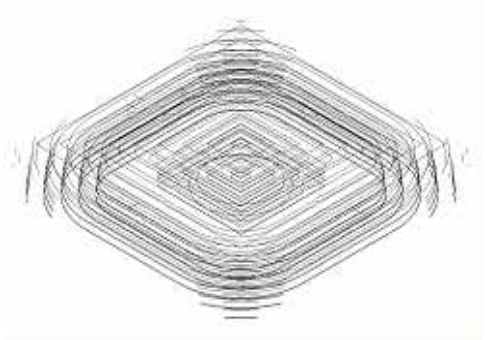


<Initial work solid>

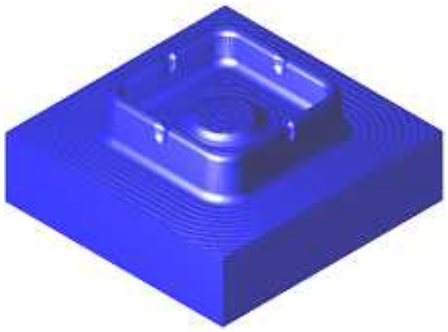


■ First Rough cutting process (Z-level Rough Cutting)

<CL data>

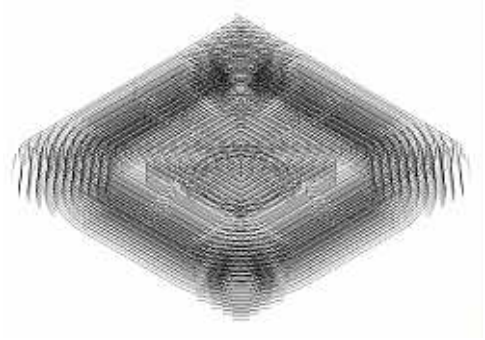


<Solid shape after cutting has been performed>

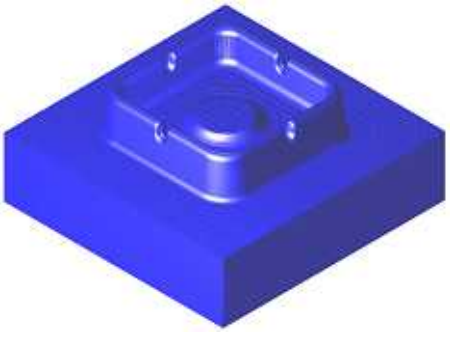


■ Semi Rough cutting process (Z-level Rough Cutting)

<CL data>

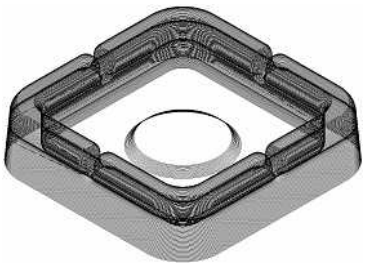


<Solid shape after cutting has been performed>

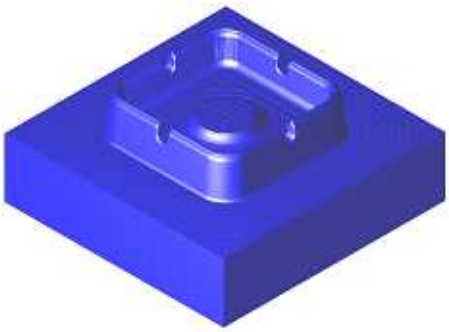


■ Semi Finishing process (Z-level Finishing)

<CL data>

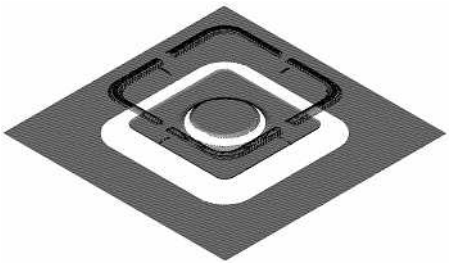


<Solid shape after cutting has been performed>



■ Semi Finishing process (Scanning-line Area)

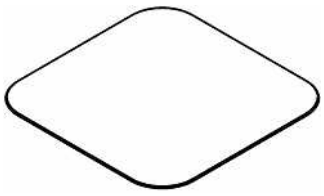
<CL data>



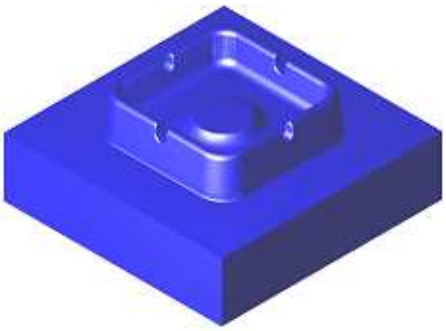
<Solid shape after cutting has been performed>

■ Semi Finishing process (Re-machining)

<CL data>

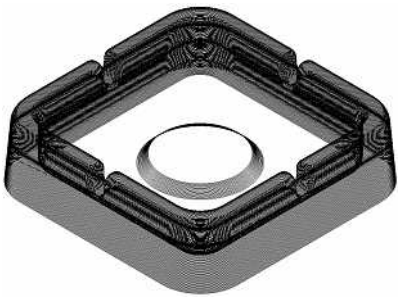


<Solid shape after cutting has been performed>

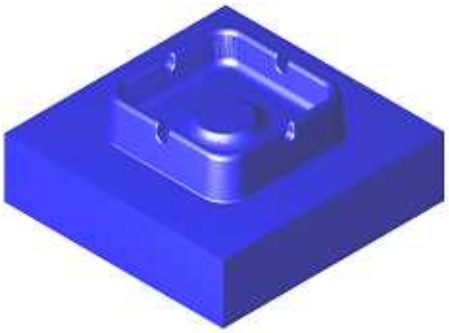


■ Finishing process (Z-level Finishing)

<CL data>

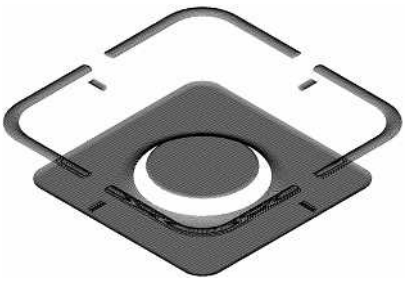


<Solid shape after cutting has been performed>

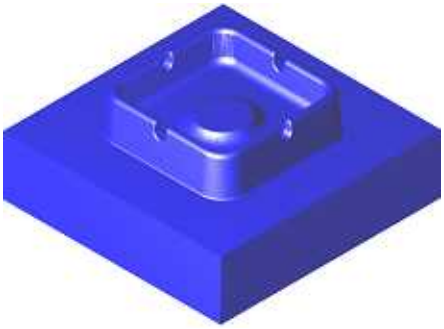


■ Finishing process (Scanning-line Area)

<CL data>

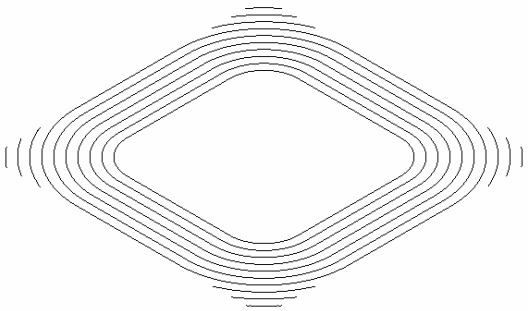


<Solid shape after cutting has been performed>

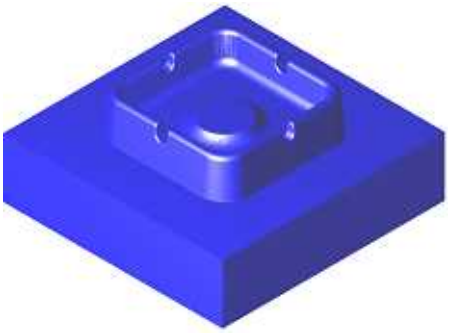


■ Finishing process (Z-level Finishing)

<CL data>

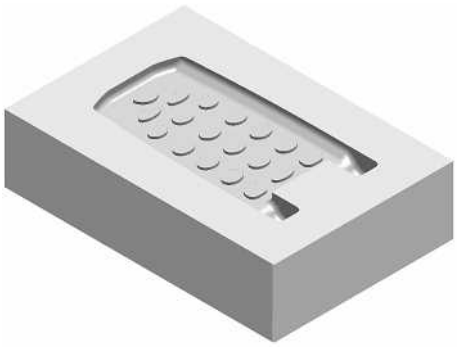


<Solid shape after cutting has been performed>

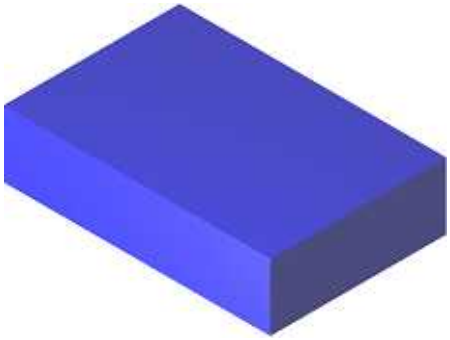


Example of creating processes for cutting cavity shape

<Model shape>

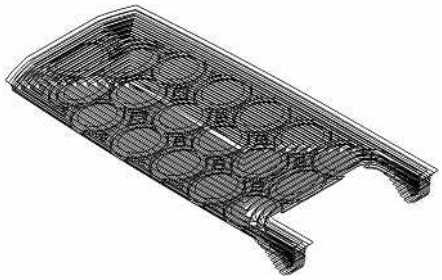


<Initial work solid>

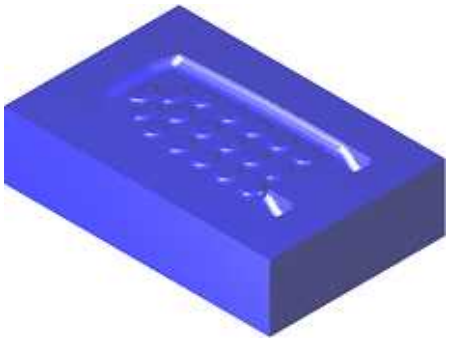


■ Rough cutting process (Z-level Rough Cutting)

<CL data>



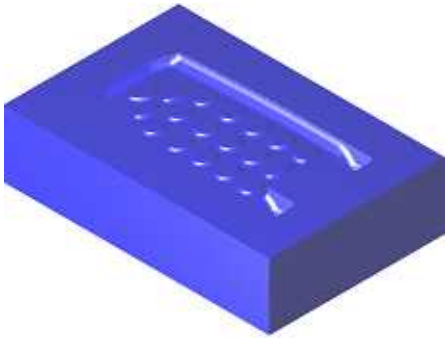
<Solid shape after cutting has been performed>



■ Semi Finishing process (Z-level Finishing)

<CL data>

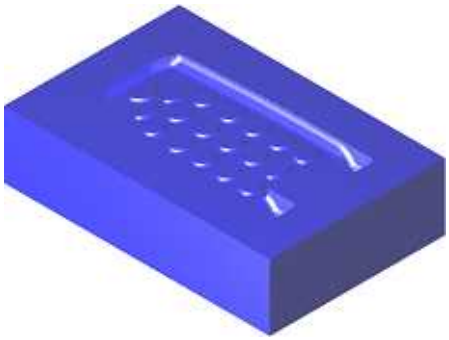
<Solid shape after cutting has been performed>



■ Semi Finishing process (Re-machining)

<CL data>

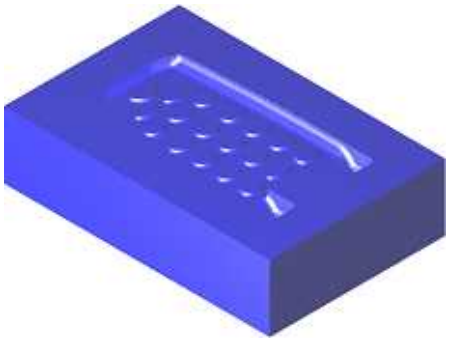
<Solid shape after cutting has been performed>



■ Semi Finishing process (Scanning-line Area)

<CL data>

<Solid shape after cutting has been performed>



Chapter 8

Exercise

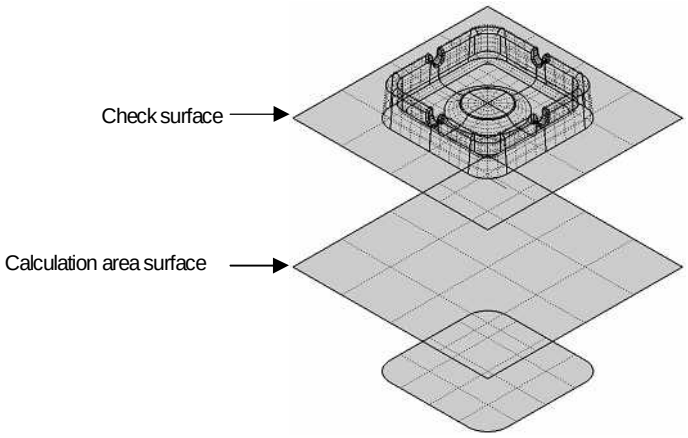
- Creating process/CAM Calculation -

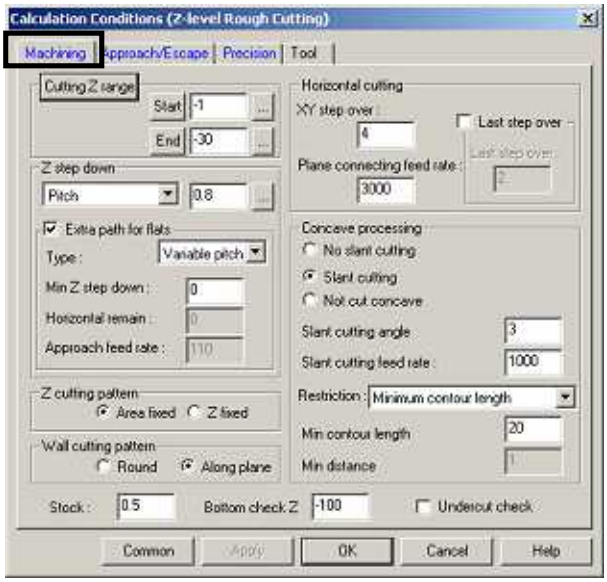
Exercise1 (Core shape)

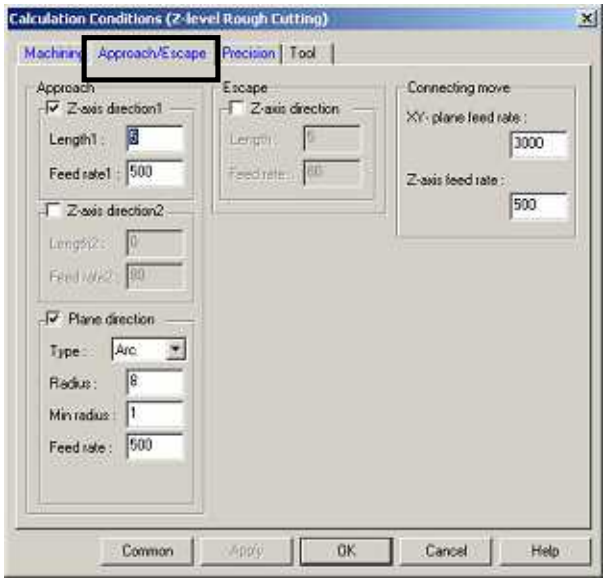
Create profiles of eight processes with the following conditions, and execute calculation.
CD-ROM /Intensive Course/CAMmodel3.gmd

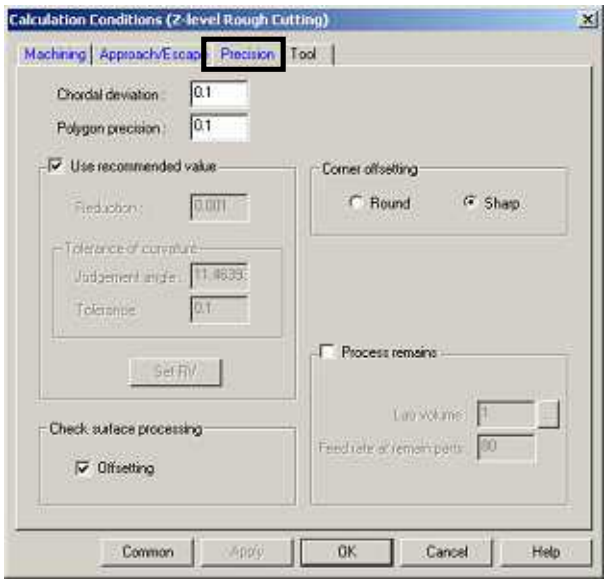
■ Process 1 (Z-level Rough Cutting)

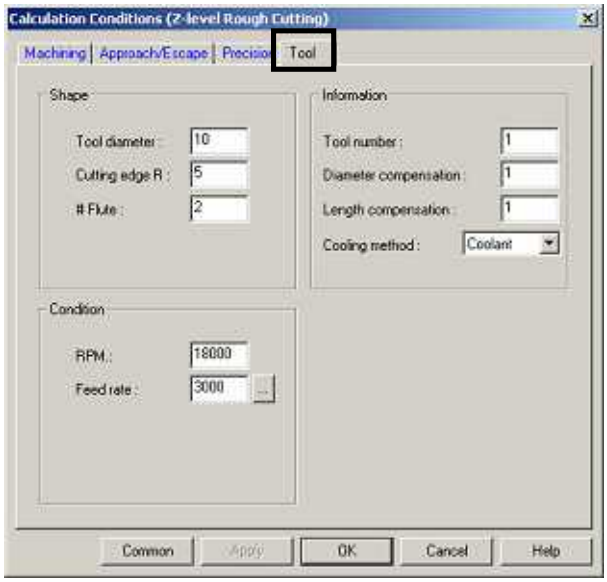
- Check surface [Layer1]
- Calculation area surface [Layer1]
- Work [Layer10]

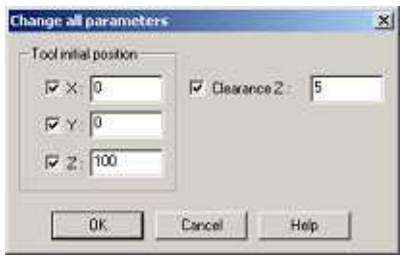






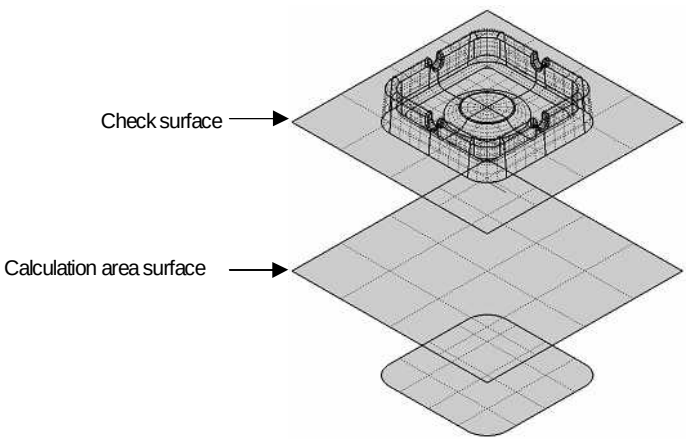

Calculation Conditions (Z-level Rough Cutting)
Machining
Approach/Escape
Precision
Tool
Chordal deviation :
0.1
Polygon precision :
0.1
Use recommended value
Reduction :
0.001
Tolerance of corner
Judgement angle :
11.4635
Tolerance :
0.1
Set RV
Corner offsetting
Round
Sharp
Process remains
Lap volume :
1
Feed rate of remain parts :
30
Check surface processing
Offsetting
Common
Apply
OK
Cancel
Help

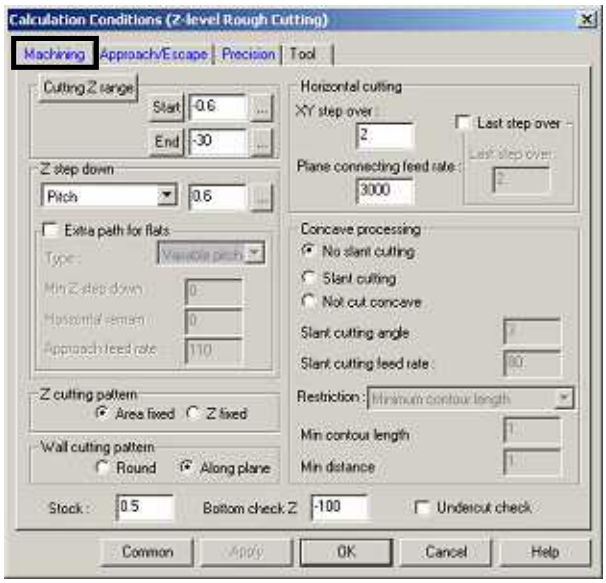

Calculation Conditions (Z-level Rough Cutting)
Machining
Approach/Escape
Precision
Tool
Shape
Tool diameter :
10
Cutting edge R :
5
Flute :
2
Information
Tool number :
1
Diameter compensation :
1
Length compensation :
1
Cooling method :
Coolant
Condition
RPM :
18000
Feed rate :
3000
Common
Apply
OK
Cancel
Help

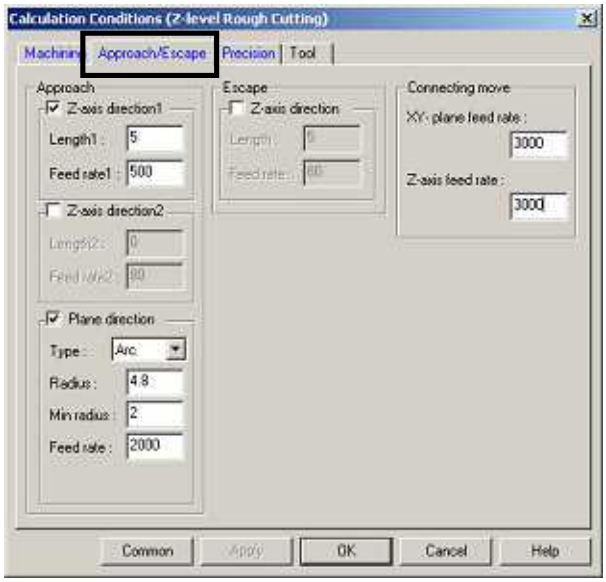

Change all parameters
Tool initial position
X :
0
Y :
0
Z :
100
Clearance Z :
5
OK
Cancel
Help

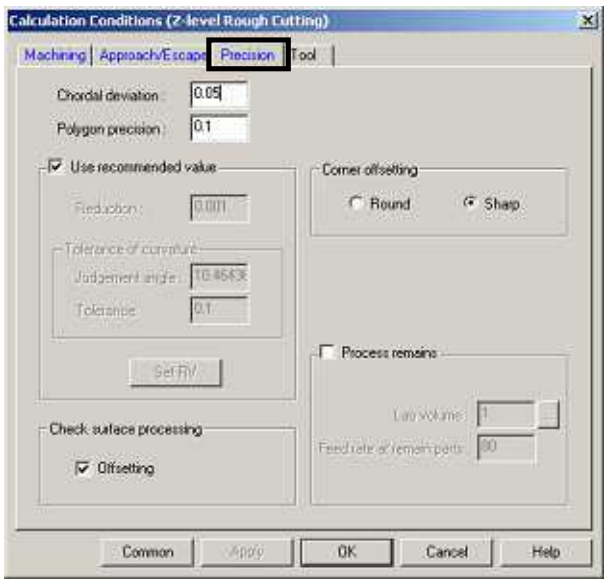
■ Process 2 (Z-level Rough Cutting)

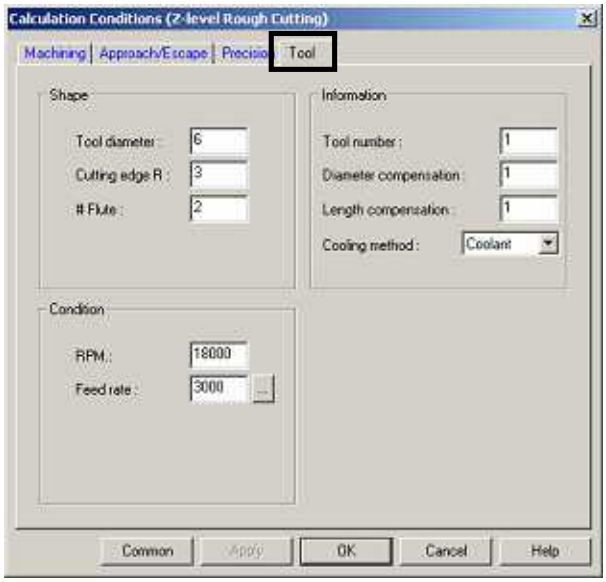
- Check surface [Layer1]
- Calculation area surface [Layer1]

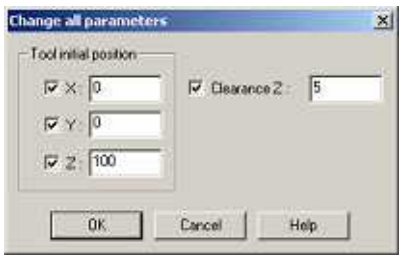






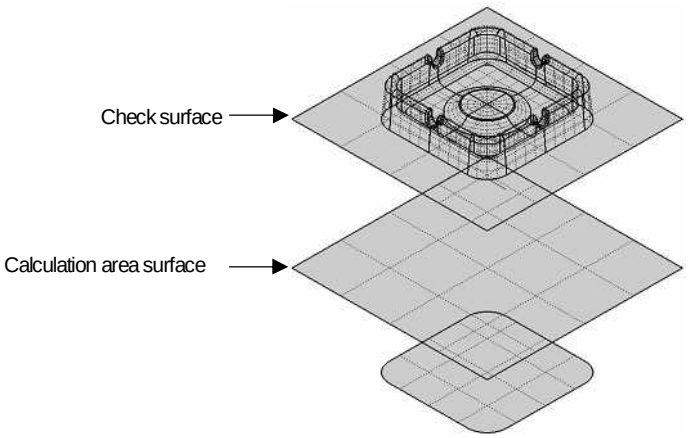

Calculation Conditions (Z-level Rough Cutting)
Machining
Approach/Escape
Precision
Tool
Chordal deviation :
0.05
Polygon precision :
0.1
Use recommended value
Reduction :
0.001
Tolerance of corner
Judgement angle :
10.46433
Tolerance :
0.1
Set RV
Corner offsetting
Round
Sharp
Process remains
Lap volume :
1
Feed rate of remain parts :
30
Check surface processing
Offsetting
Common
Apply
OK
Cancel
Help

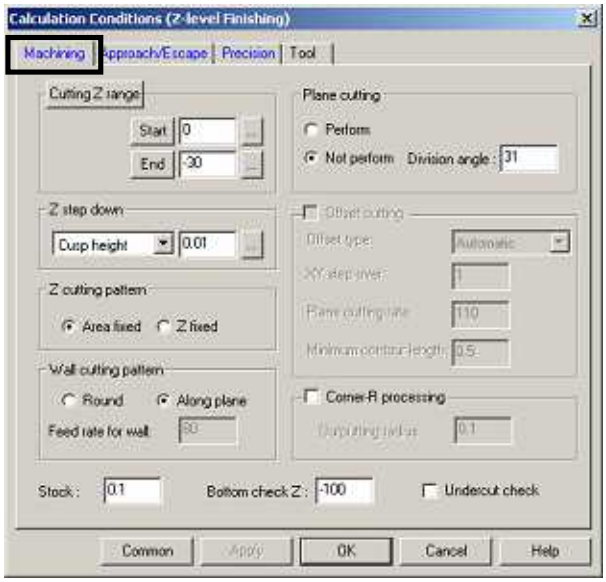

Calculation Conditions (Z-level Rough Cutting)
Machining
Approach/Escape
Precision
Tool
Shape
Tool diameter :
6
Cutting edge R :
3
Flute :
2
Information
Tool number :
1
Diameter compensation :
1
Length compensation :
1
Cooling method :
Coolant
Condition
RPM :
18000
Feed rate :
3000
Common
Apply
OK
Cancel
Help

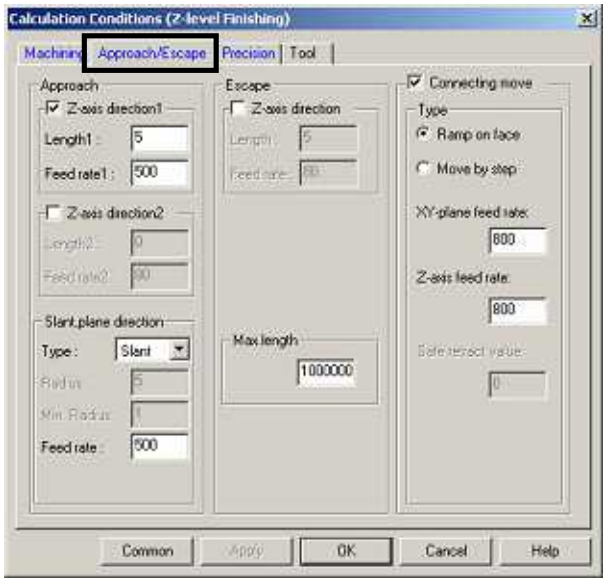

Change all parameters
Tool initial position
X :
0
Y :
0
Z :
100
Clearance Z :
5
OK
Cancel
Help

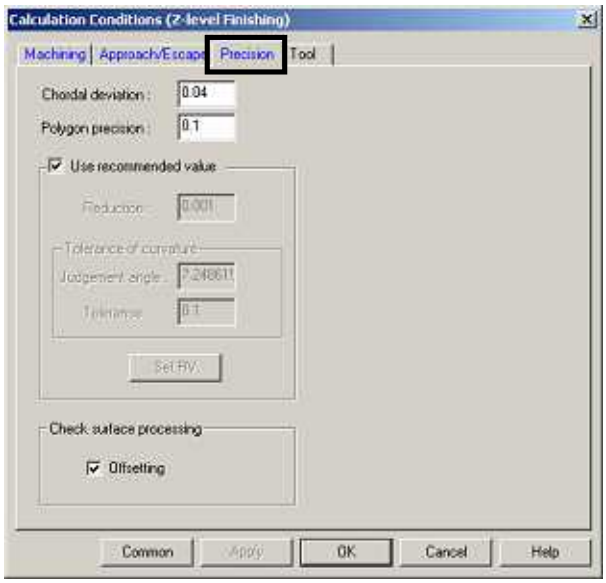
■ Process 3 (Z-level Finishing)

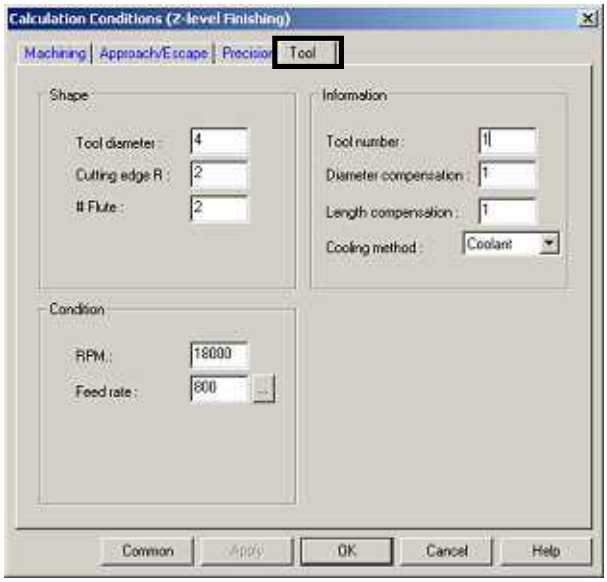
- Check surface [Layer1]
- Calculation area surface [Layer1]

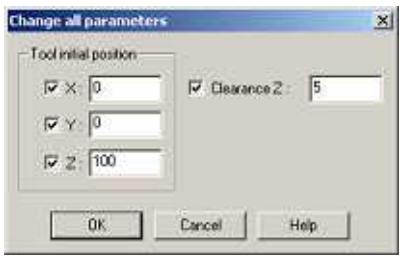






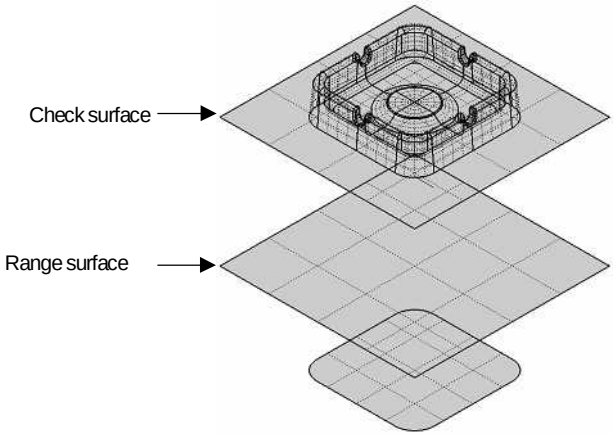

Calculation Conditions (Z-level Finishing)
Machining
Approach/Escape
Precision
Tool
Chordal deviation :
0.04
Polygon precision :
0.1
Use recommended value
Reduction :
0.001
Tolerance of curvature
Judgement angle :
7.248611
Tolerance :
0.1
Set RV
Check surface processing
Offsetting
Common
Apply
OK
Cancel
Help

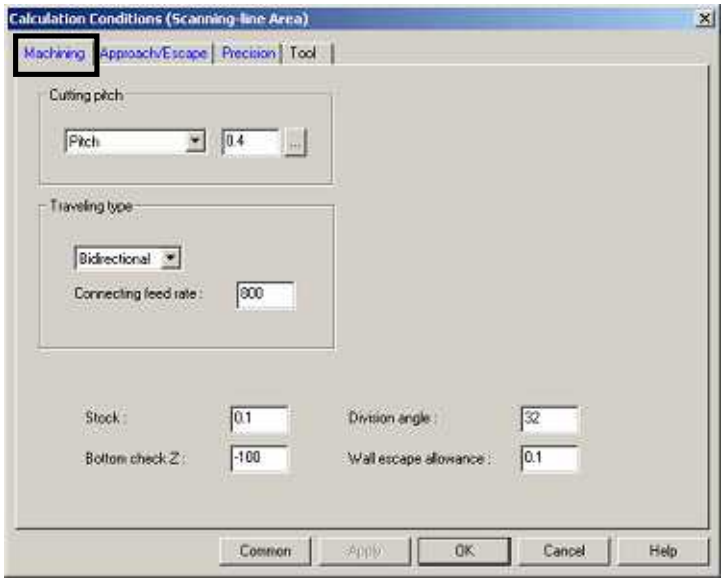

Calculation Conditions (Z-level Finishing)
Machining
Approach/Escape
Precision
Tool
Shape
Tool diameter :
4
Cutting edge R :
2
Flute :
2
Information
Tool number :
1
Diameter compensation :
1
Length compensation :
1
Cooling method :
Coolant
Condition
RPM :
18000
Feed rate :
800
Common
Apply
OK
Cancel
Help

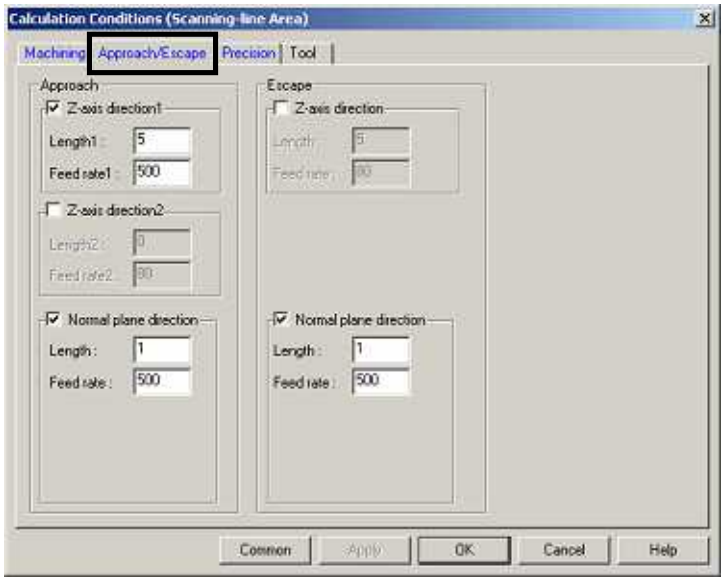

Change all parameters
Tool initial position
X :
0
Y :
0
Z :
100
Clearance Z :
5
OK
Cancel
Help

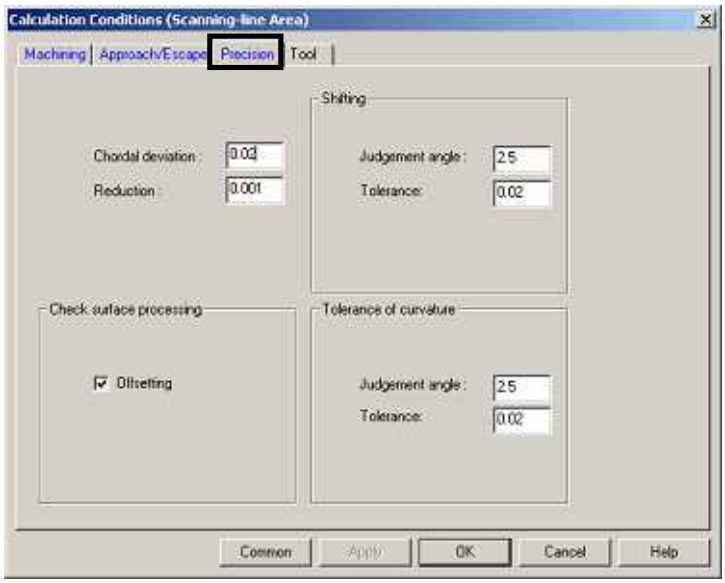
■ Process 4 (Scanning-line Area)

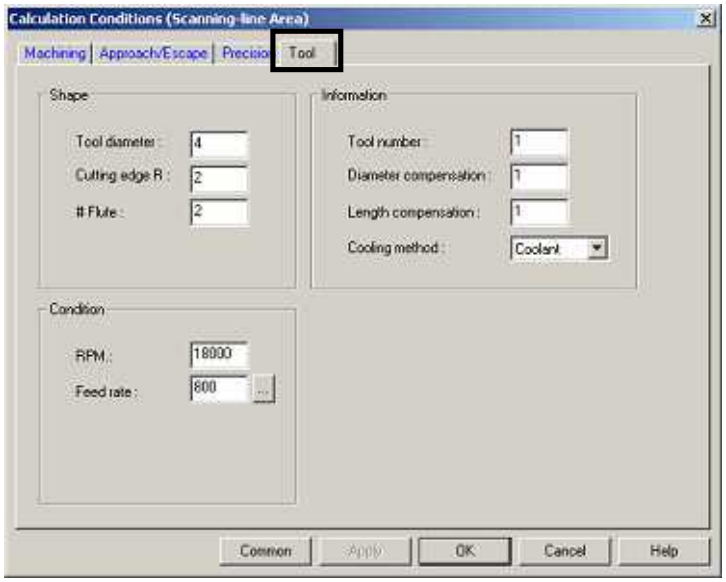
- Check surface [Layer1]
- Range surface [Layer1]

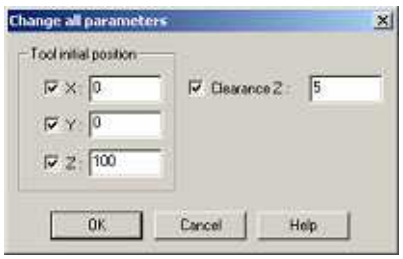






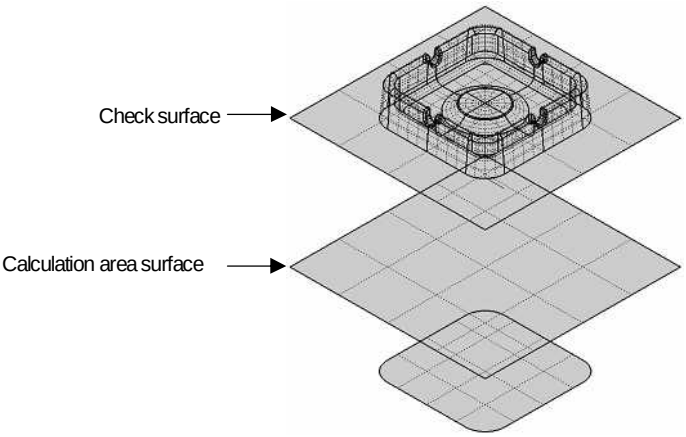

Calculation Conditions (Scanning-line Area)
Machining
Approach/Escape
Precision
Tool
Shifting
Chordal deviation :
0.02
Reduction :
0.001
Judgement angle :
2.5
Tolerance:
0.02
Check surface processing
Offsetting
Tolerance of curvature
Judgement angle :
2.5
Tolerance:
0.02
Common
Apply
OK
Cancel
Help

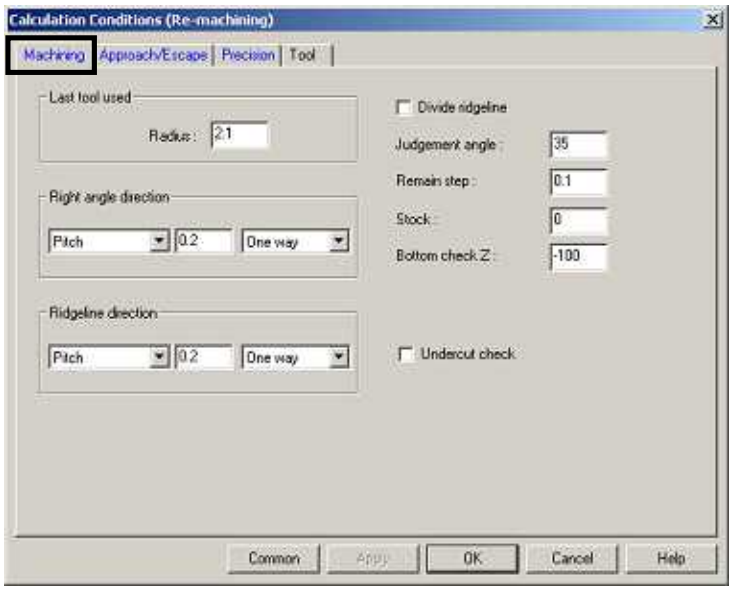

Calculation Conditions (Scanning-line Area)
Machining
Approach/Escape
Precision
Tool
Shape
Tool diameter :
4
Cutting edge R :
2
Flute :
2
Information
Tool number :
1
Diameter compensation :
1
Length compensation :
1
Cooling method :
Coolant
Condition
RPM :
18000
Feed rate :
800
Common
Apply
OK
Cancel
Help

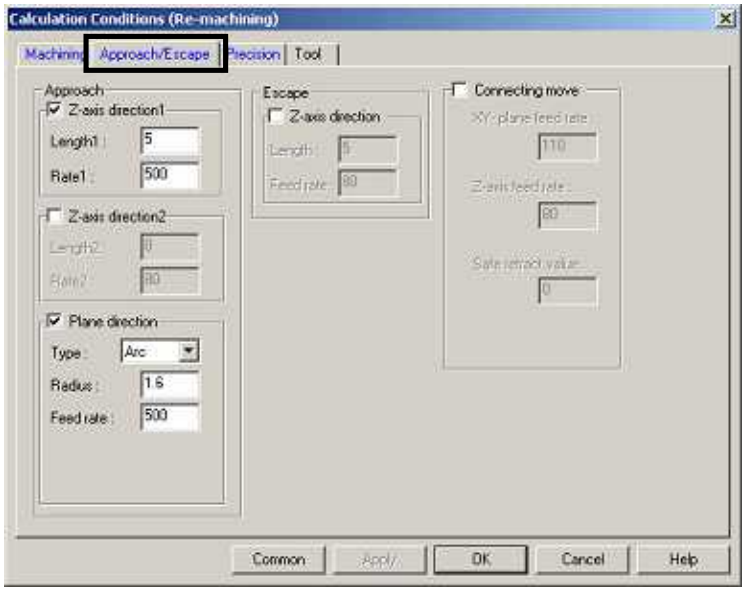

Change all parameters
Tool initial position
X :
0
Y :
0
Z :
100
Clearance Z :
5
OK
Cancel
Help

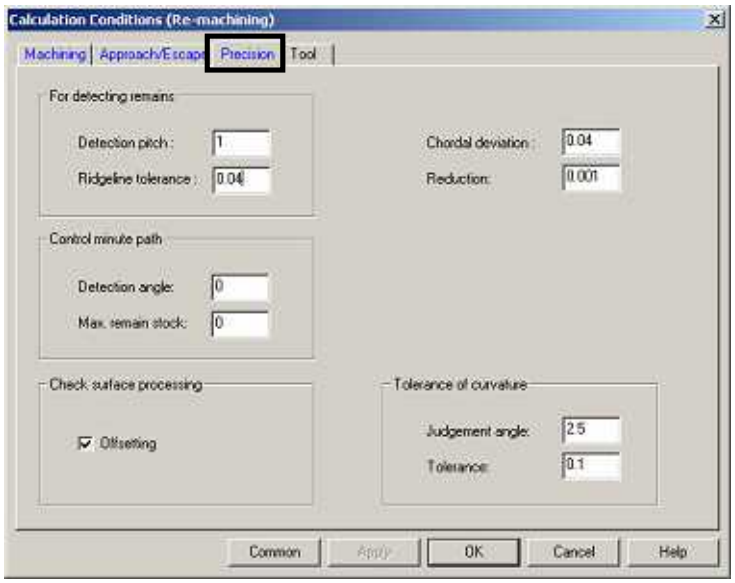
■ Process 5 (Re-machining)

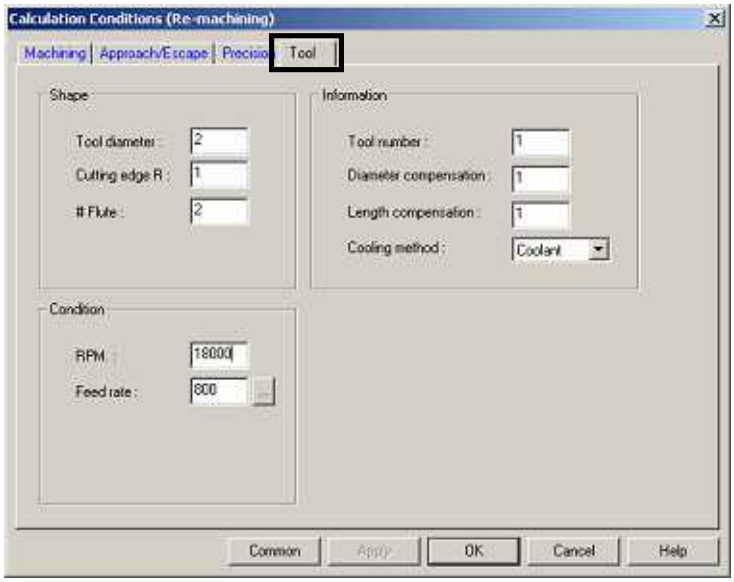
- Check surface [Layer1]
- Calculation area surface [Layer1]

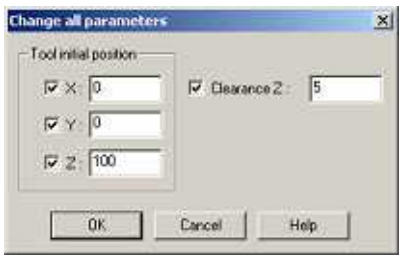






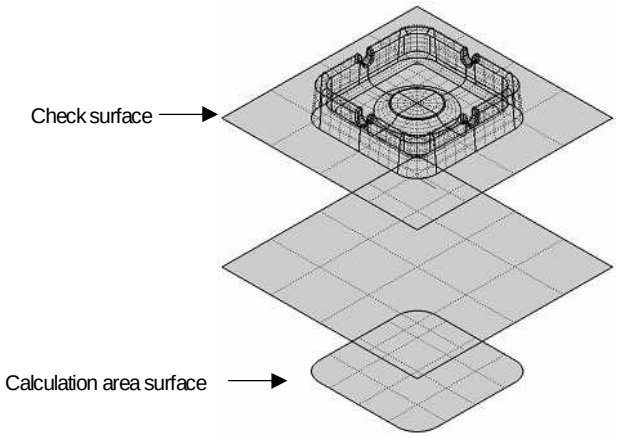

Calculation Conditions (Re-machining)
Machining
Approach/Escape
Precision
Tool
For detecting remains
Detection pitch : 1
Ridgeline tolerance : 0.04
Chordal deviation : 0.04
Reduction: 0.001
Control minute path
Detection angle: 0
Max. remain stock: 0
Check surface processing
Offsetting
Tolerance of curvature
Judgement angle: 2.5
Tolerance: 0.1
Common
Apply
OK
Cancel
Help

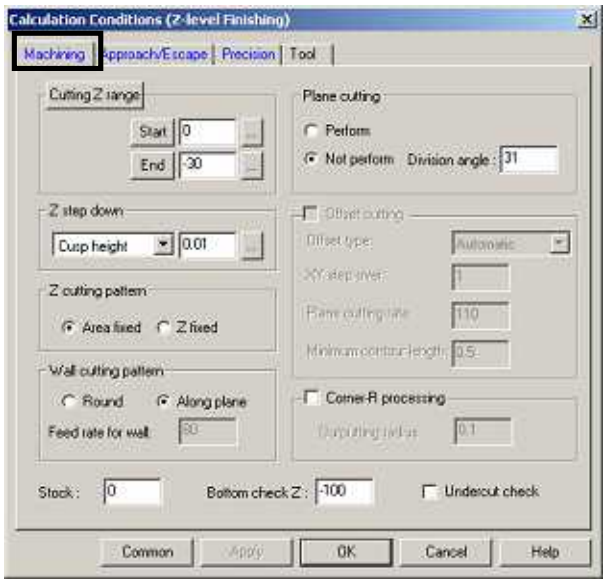

Calculation Conditions (Re-machining)
Machining
Approach/Escape
Precision
Tool
Shape
Tool diameter : 2
Cutting edge R : 1
Flute : 2
Information
Tool number : 1
Diameter compensation : 1
Length compensation : 1
Cooling method : Coolant
Condition
RPM : 18000
Feed rate : 800
Common
Apply
OK
Cancel
Help

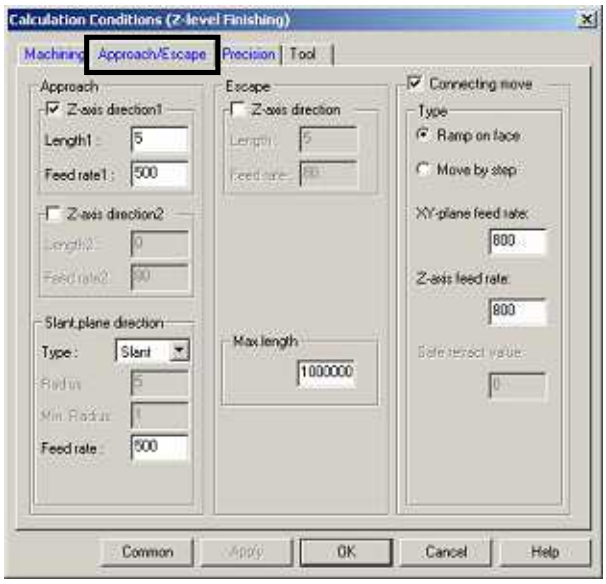

Change all parameters
Tool initial position
X: 0
Y: 0
Z: 100
Clearance Z : 5
OK
Cancel
Help

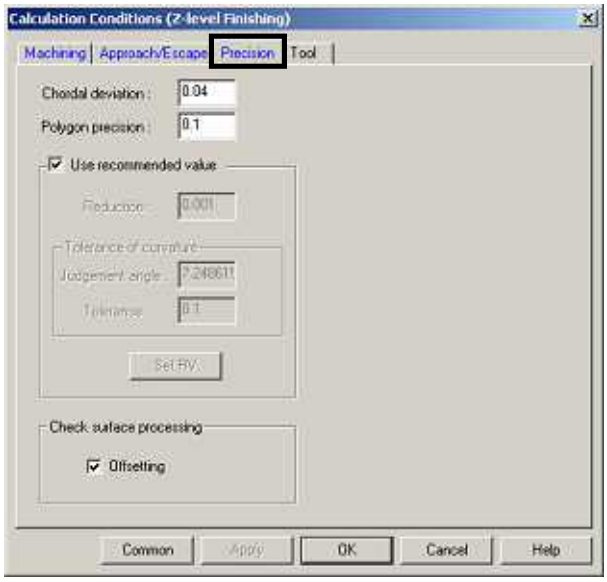
■ Process 6 (Z-level Finishing)

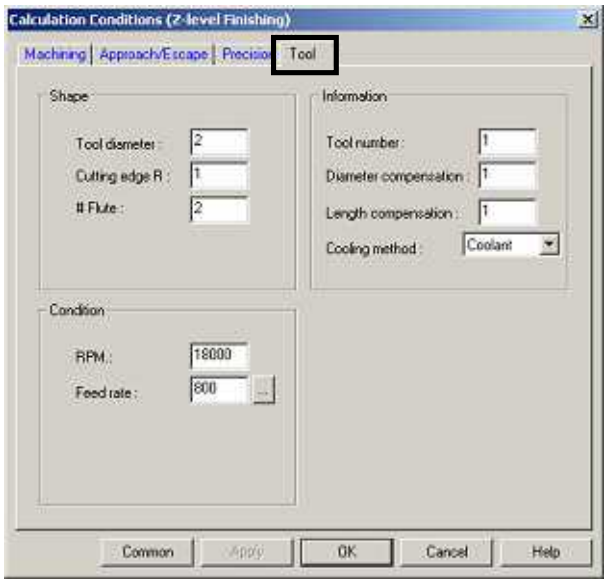
- Check surface [Layer1]
- Calculation area surface [Layer1]

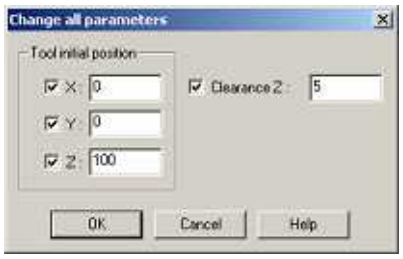






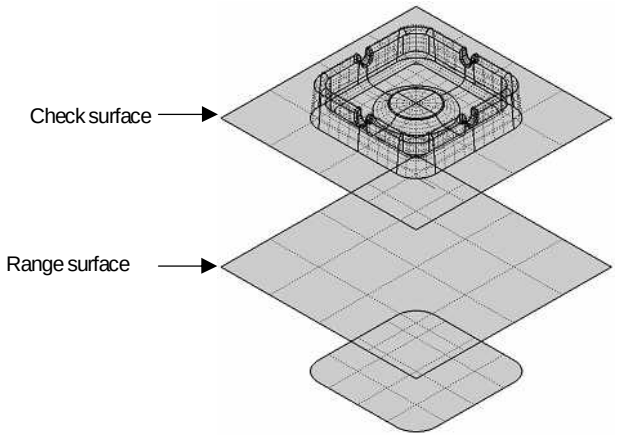

Calculation Conditions (Z-level Finishing)
Machining
Approach/Escape
Precision
Tool
Chordal deviation :
0.04
Polygon precision :
0.1
Use recommended value
Reduction :
0.001
Tolerance of curvature
Judgement angle :
7.248611
Tolerance :
0.1
Set RV
Check surface processing
Offsetting
Common
Apply
OK
Cancel
Help

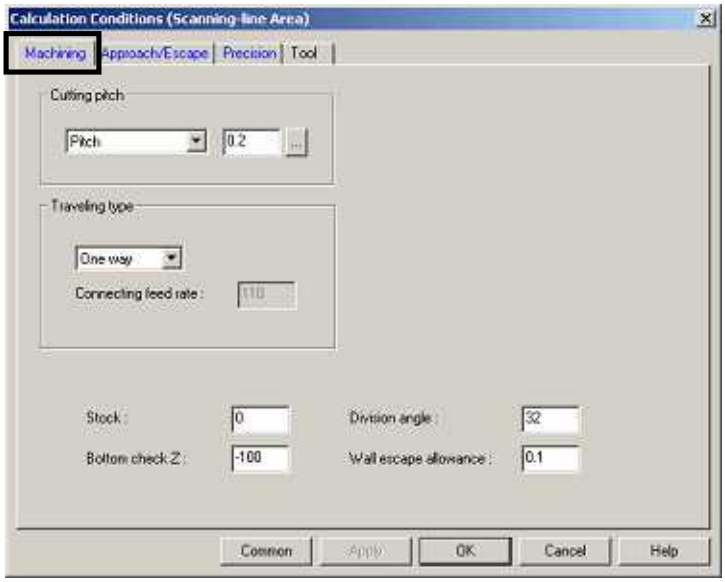

Calculation Conditions (Z-level Finishing)
Machining
Approach/Escape
Precision
Tool
Shape
Tool diameter :
2
Cutting edge R :
1
Flute :
2
Information
Tool number :
1
Diameter compensation :
1
Length compensation :
1
Cooling method :
Coolant
Condition
RPM :
18000
Feed rate :
800
Common
Apply
OK
Cancel
Help

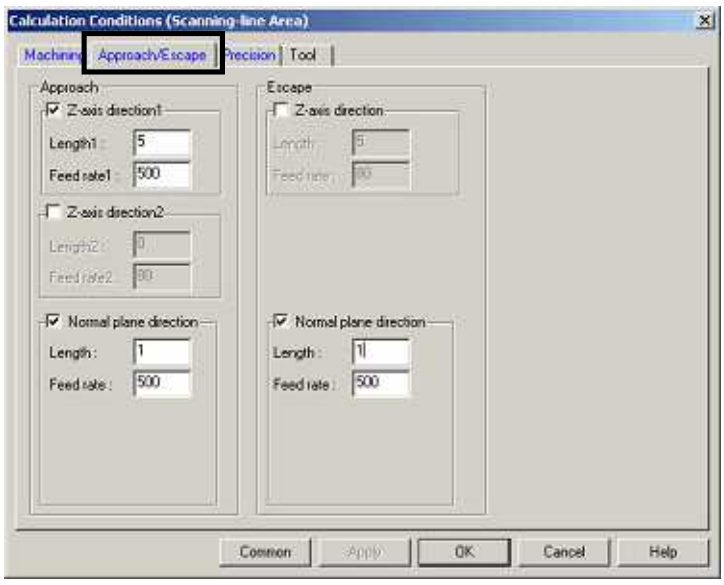

Change all parameters
Tool initial position
X :
0
Y :
0
Z :
100
Clearance Z :
5
OK
Cancel
Help

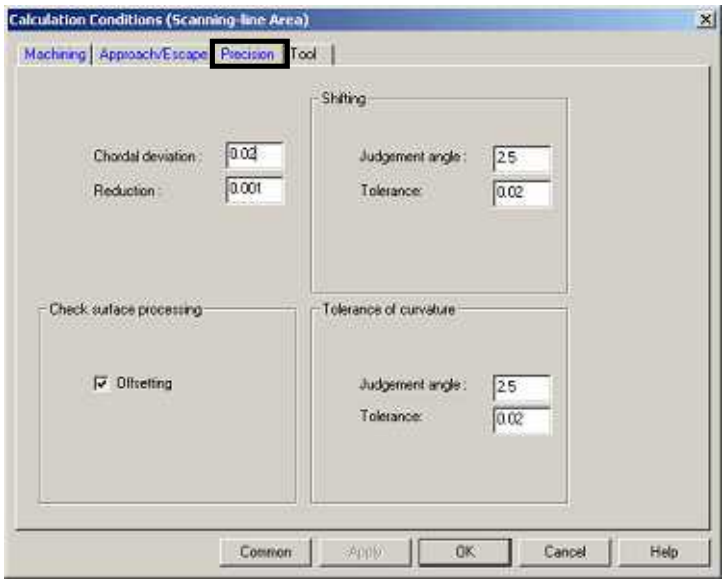
■ Process 7 (Scanning-line Area)

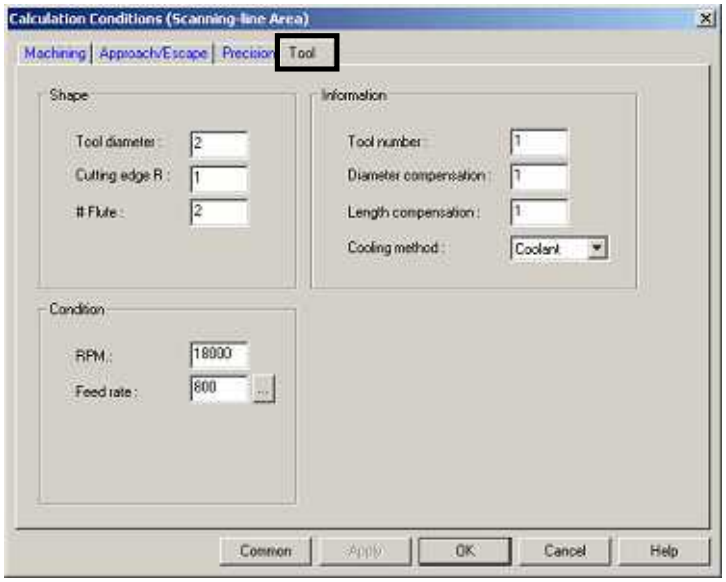
- Check surface [Layer1]
- Range surface [Layer1]

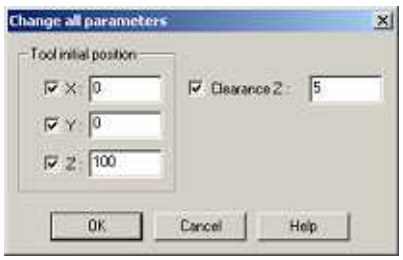






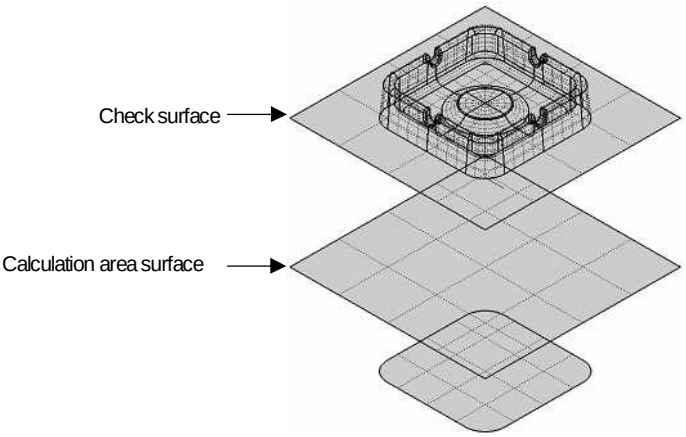

Calculation Conditions (Scanning-line Area)
Machining
Approach/Escape
Precision
Tool
Shifting
Chordal deviation :
0.02
Reduction :
0.001
Judgement angle :
2.5
Tolerance:
0.02
Check surface processing
Offsetting
Tolerance of curvature
Judgement angle :
2.5
Tolerance:
0.02
Common
Apply
OK
Cancel
Help

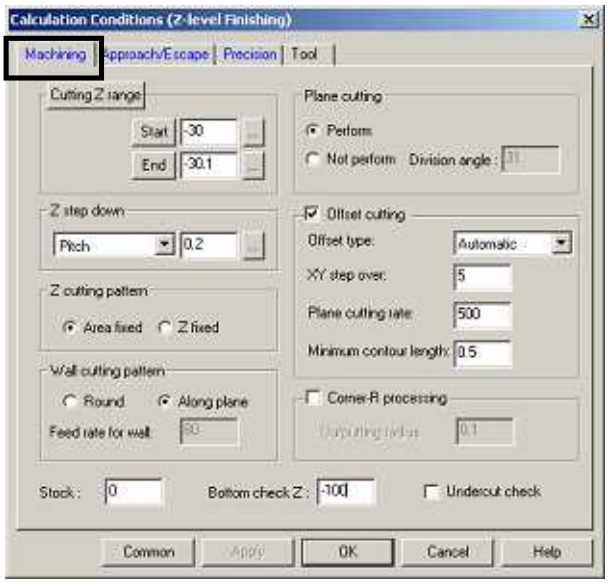

Calculation Conditions (Scanning-line Area)
Machining
Approach/Escape
Precision
Tool
Shape
Tool diameter :
2
Cutting edge R :
1
Flute :
2
Information
Tool number :
1
Diameter compensation :
1
Length compensation :
1
Cooling method :
Coolant
Condition
RPM :
18000
Feed rate :
800
Common
Apply
OK
Cancel
Help

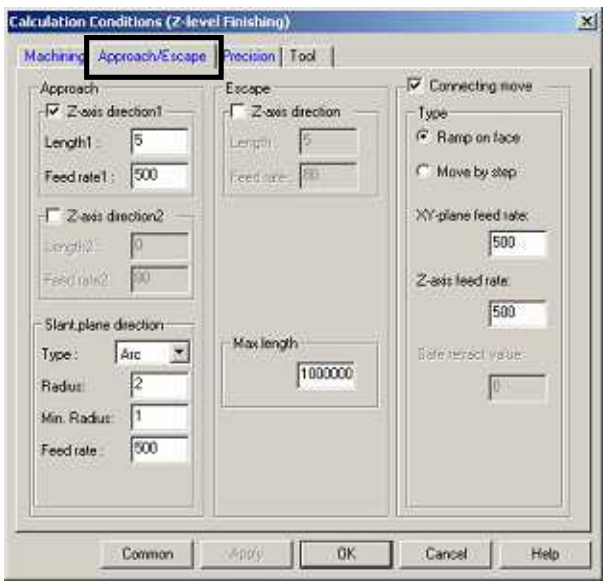

Change all parameters
Tool initial position
X :
0
Y :
0
Z :
100
Clearance Z :
5
OK
Cancel
Help

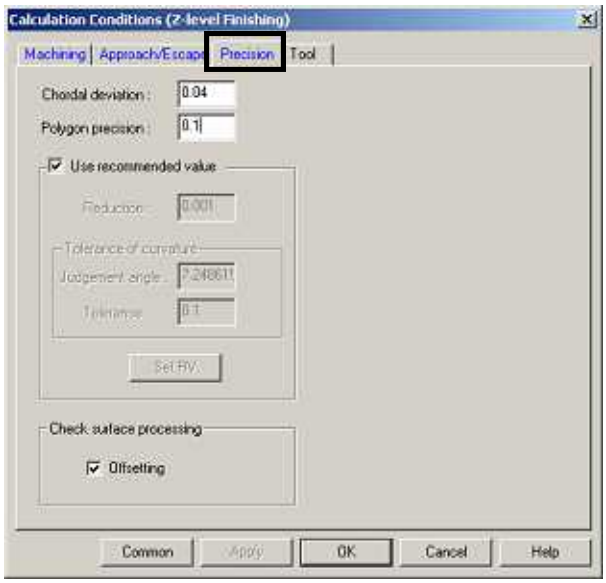
■ Process 8 (Z-level Finishing)

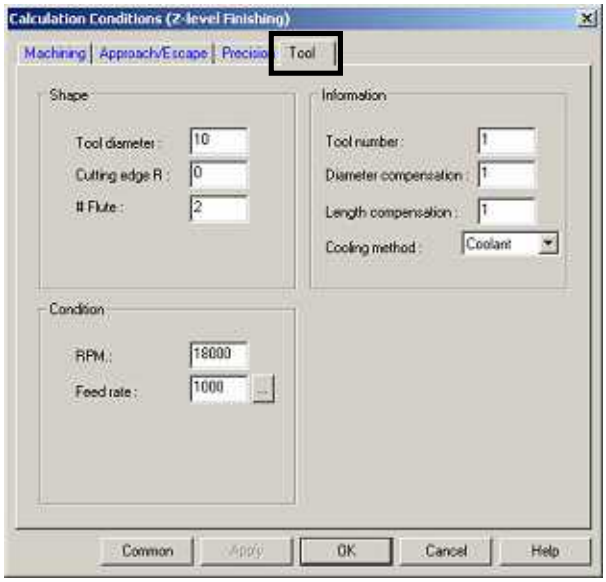
- Check surface [Layer1]
- Calculation area surface [Layer1]

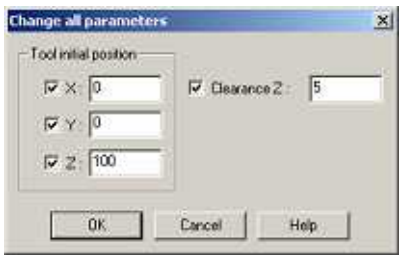







Calculation Conditions (Z-level Finishing)
Machining
Approach/Escape
Precision
Tool
Chordal deviation :
0.04
Polygon precision :
0.1
Use recommended value
Reduction :
0.001
Tolerance of curvature
Judgement angle :
7.248611
Tolerance :
0.1
Set RV
Check surface processing
Offsetting
Common
Apply
OK
Cancel
Help


Calculation Conditions (Z-level Finishing)
Machining
Approach/Escape
Precision
Tool
Shape
Tool diameter :
10
Cutting edge R :
0
Flute :
2
Information
Tool number :
1
Diameter compensation :
1
Length compensation :
1
Cooling method :
Coolant
Condition
RPM :
18000
Feed rate :
1000
Common
Apply
OK
Cancel
Help

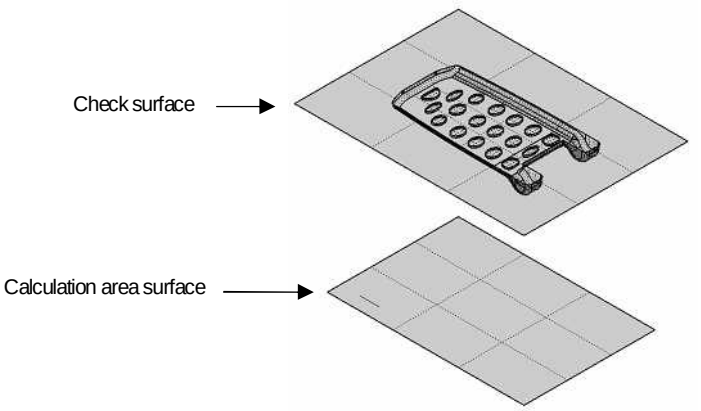

Change all parameters
Tool initial position
X :
0
Y :
0
Z :
100
Clearance Z :
5
OK
Cancel
Help

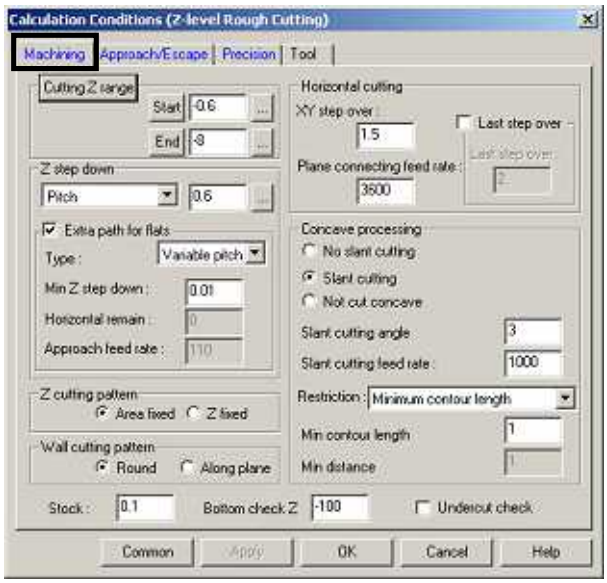
Exercise2 (Cavity shape)

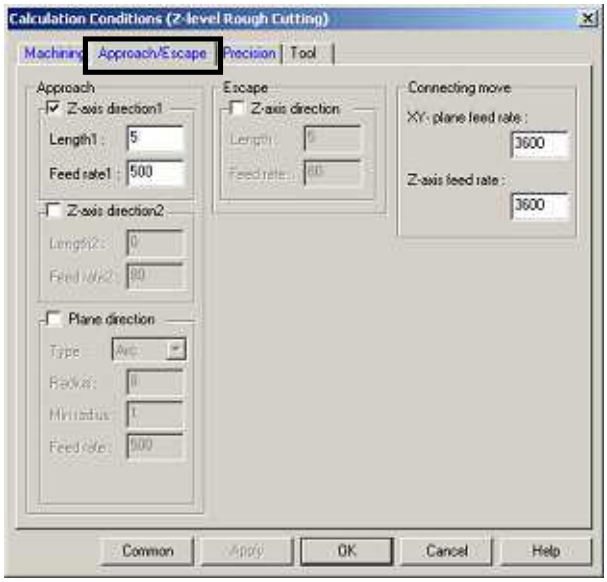
Create profiles of four processes with the following conditions, and execute CAM calculation.
CD-ROM /Intensive Course/CAMmodel4.gmd

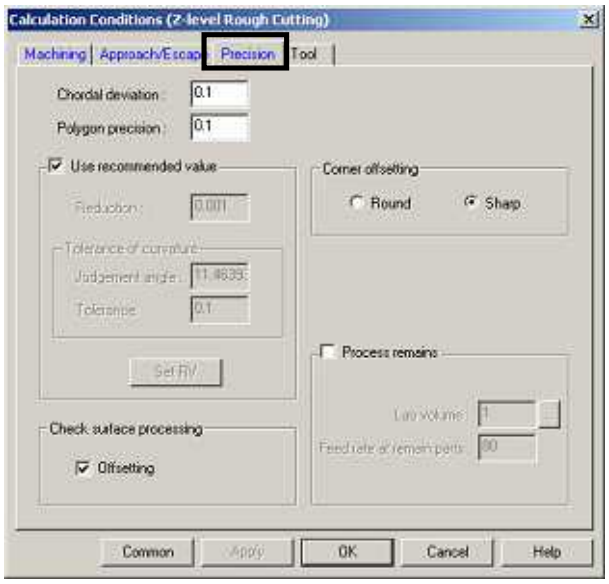
■ Process 1 (Z-level Rough Cutting)

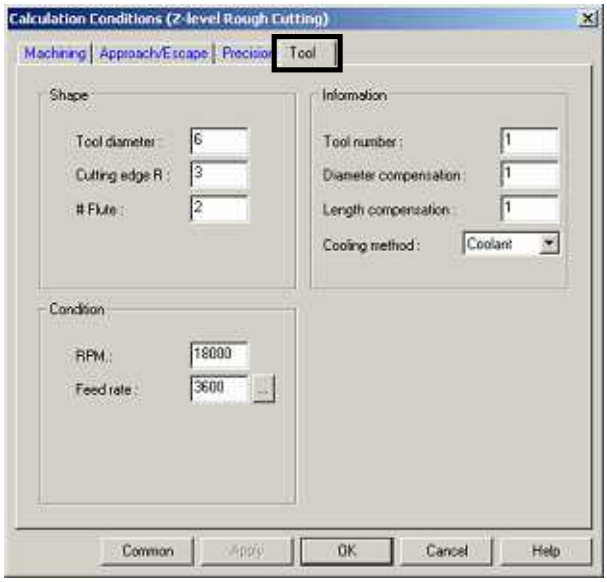
- Check surface [Layer1]
- Calculation area surface [Layer1]
- Work [Layer10]

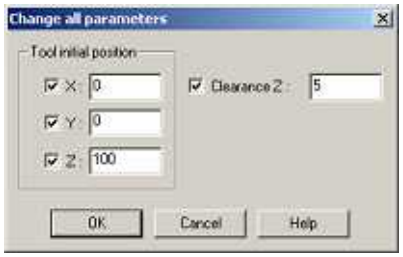






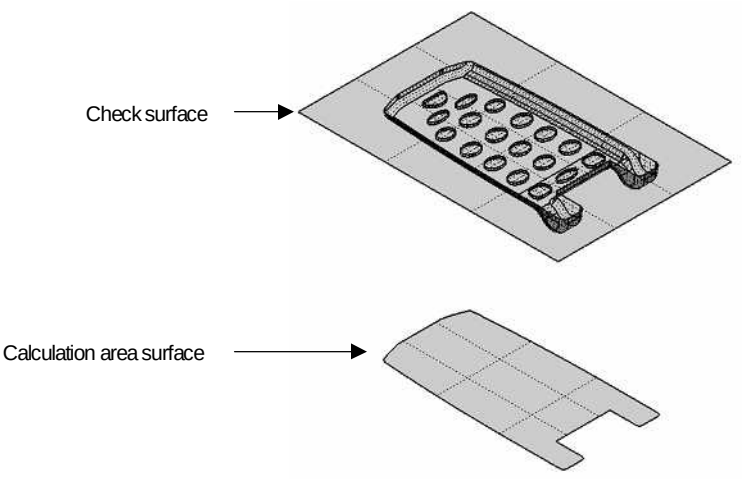

Calculation Conditions (Z-level Rough Cutting)
Machining
Approach/Escape
Precision
Tool
Chordal deviation :
0.1
Polygon precision :
0.1
Use recommended value
Reduction :
0.001
Tolerance of curvature
Judgement angle :
11.4635
Tolerance :
0.1
Set RV
Corner offsetting
Round
Sharp
Process remains
Lap volume :
1
Feed rate at remain parts :
50
Check surface processing
Offsetting
Common
Apply
OK
Cancel
Help

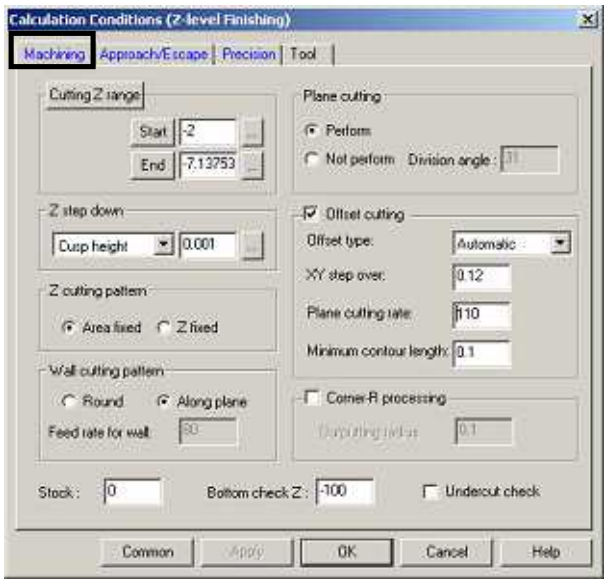

Calculation Conditions (Z-level Rough Cutting)
Machining
Approach/Escape
Precision
Tool
Shape
Tool diameter :
6
Cutting edge R :
3
Flute :
2
Information
Tool number :
1
Diameter compensation :
1
Length compensation :
1
Cooling method :
Coolant
Condition
RPM :
18000
Feed rate :
3600
Common
Apply
OK
Cancel
Help

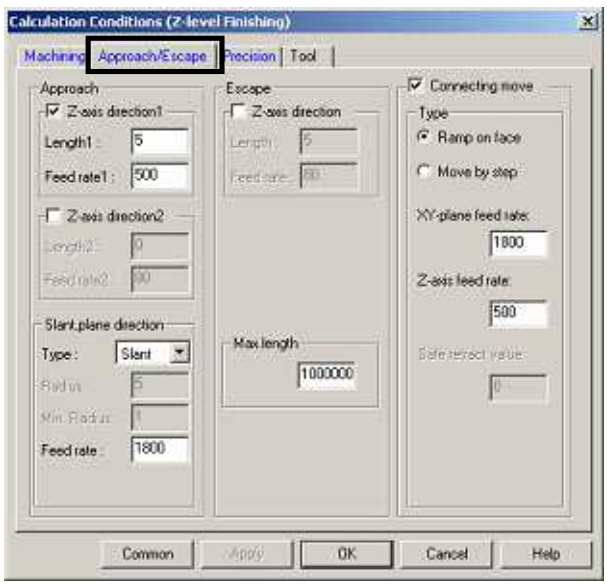

Change all parameters
Tool initial position
X :
0
Y :
0
Z :
100
Clearance Z :
5
OK
Cancel
Help

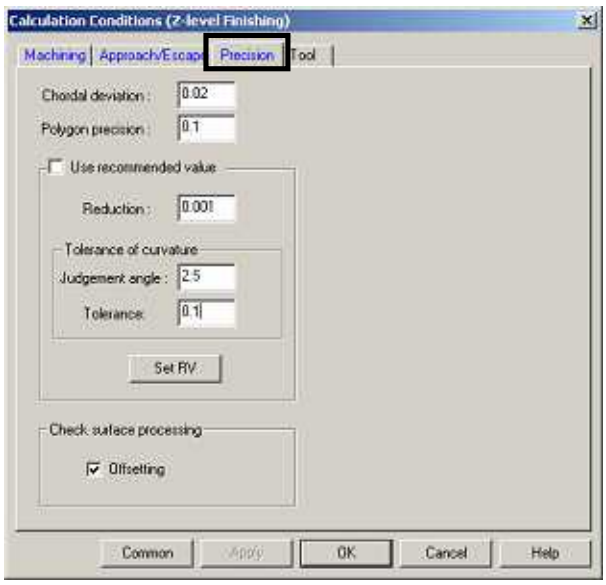
■ Process 2 (Z-level Finishing)

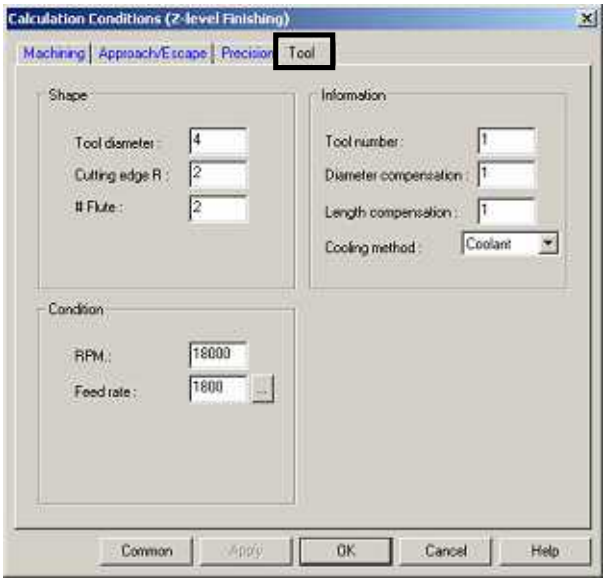
- Check surface [Layer2]
- Calculation area surface [Layer2]

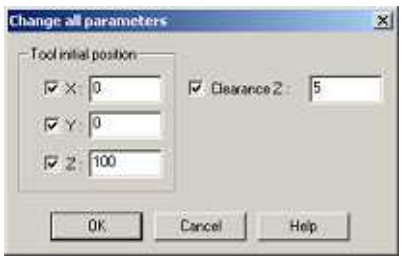






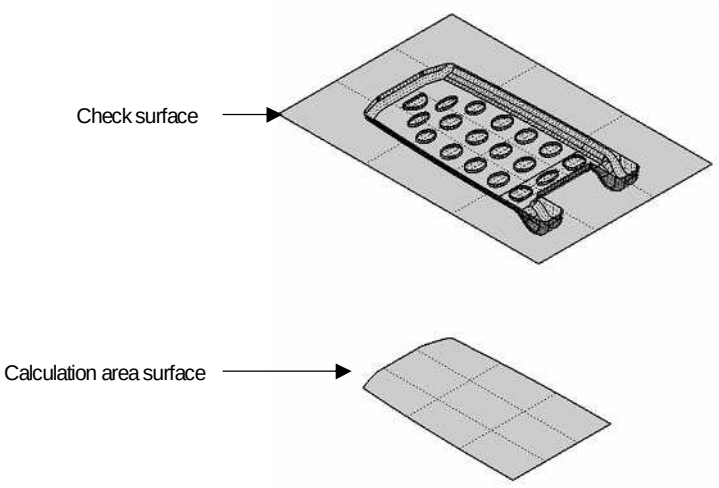

Calculation Conditions (Z-level Finishing)
Machining
Approach/Escape
Precision
Tool
Chordal deviation :
0.02
Polygon precision :
0.1
Use recommended value
Reduction :
0.001
Tolerance of curvature
Judgement angle :
2.5
Tolerance :
0.1
Set RV
Check surface processing
Offsetting
Common
Apply
OK
Cancel
Help

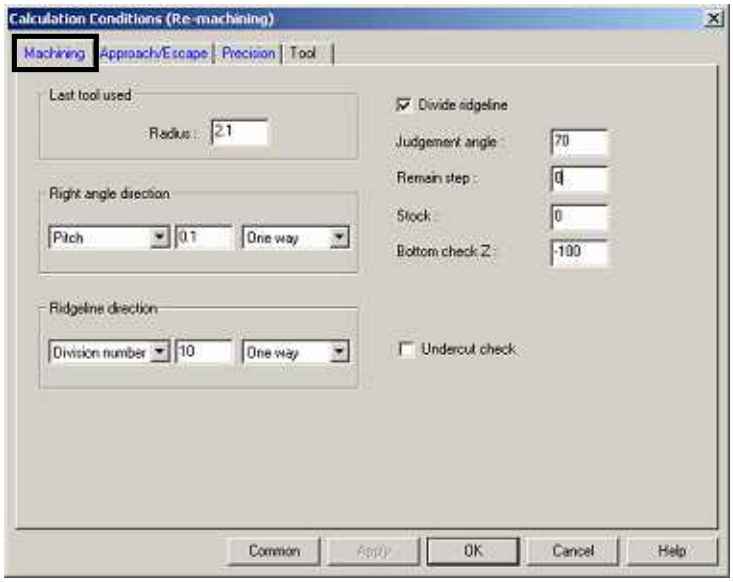

Calculation Conditions (Z-level Finishing)
Machining
Approach/Escape
Precision
Tool
Shape
Tool diameter :
4
Cutting edge R :
2
Flute :
2
Information
Tool number :
1
Diameter compensation :
1
Length compensation :
1
Cooling method :
Coolant
Condition
RPM :
18000
Feed rate :
1800
Common
Apply
OK
Cancel
Help

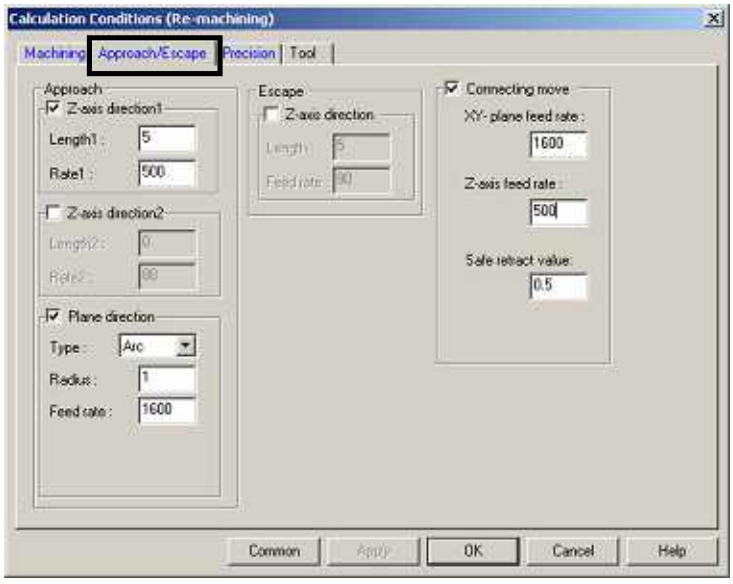

Change all parameters
Tool initial position
X :
0
Y :
0
Z :
100
Clearance Z :
5
OK
Cancel
Help

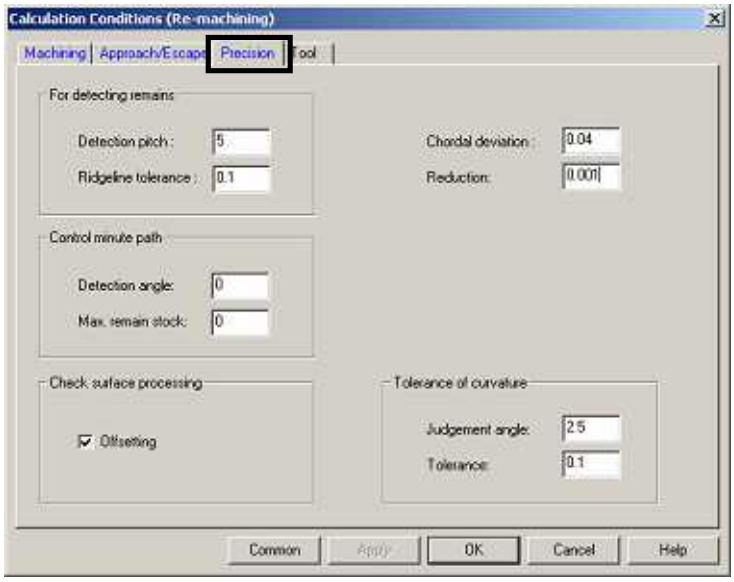
■ Process 3 (Re-machining)

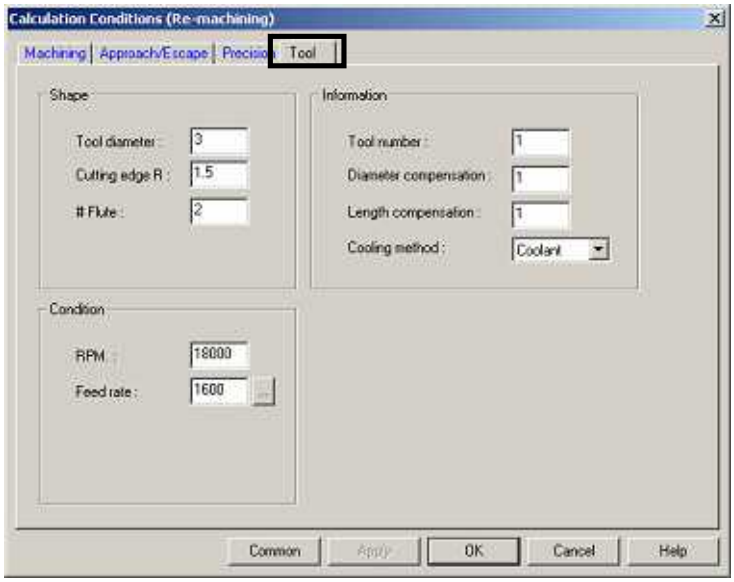
- Check surface [Layer3]
- Calculation area surface [Layer3]

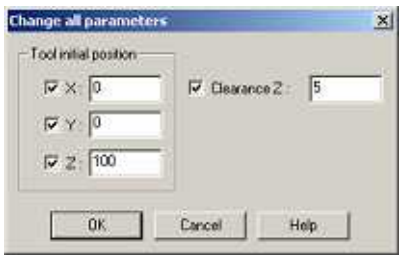







Calculation Conditions (Re-machining)
Machining
Approach/Escape
Precision
Tool
For detecting remains
Detection pitch :
5
Ridgeline tolerance :
0.1
Chordal deviation :
0.04
Reduction:
0.001
Control minute path
Detection angle:
0
Max. remain stock:
0
Check surface processing
Offsetting
Tolerance of curvature
Judgement angle:
2.5
Tolerance:
0.1
Common
Apply
OK
Cancel
Help


Calculation Conditions (Re-machining)
Machining
Approach/Escape
Precision
Tool
Shape
Tool diameter :
3
Cutting edge R :
1.5
Flute :
2
Information
Tool number :
1
Diameter compensation :
1
Length compensation :
1
Cooling method :
Coolant
Condition
RPM :
18000
Feed rate :
1600
Common
Apply
OK
Cancel
Help

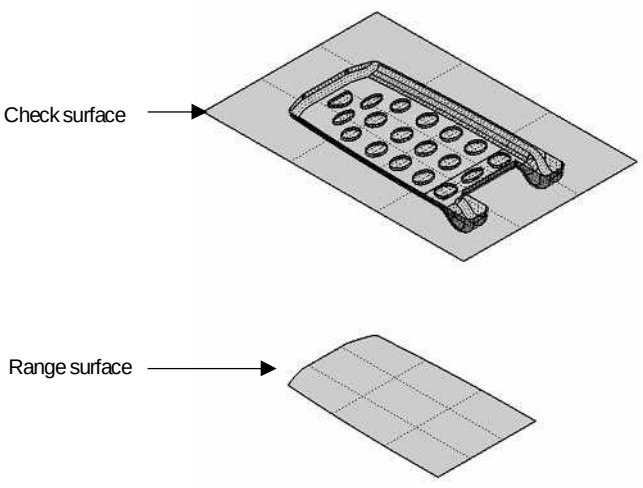

Change all parameters
Tool initial position
X :
0
Y :
0
Z :
100
Clearance Z :
5
OK
Cancel
Help

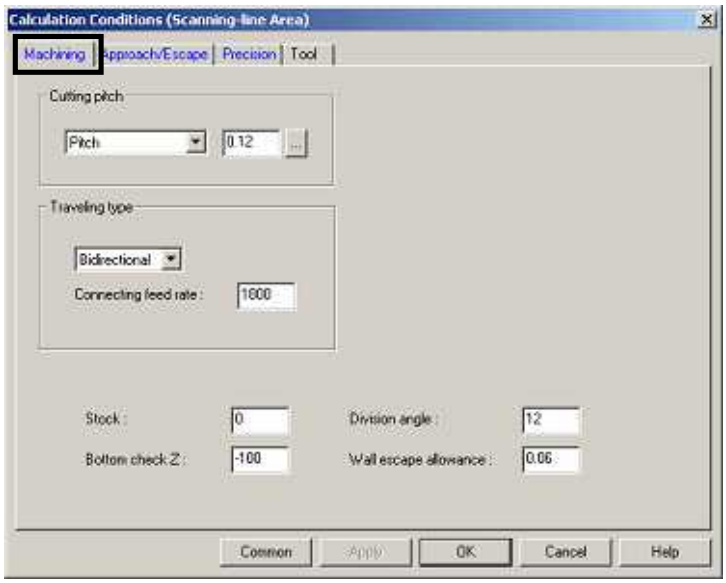
■ Process 4 (Scanning-line Area)

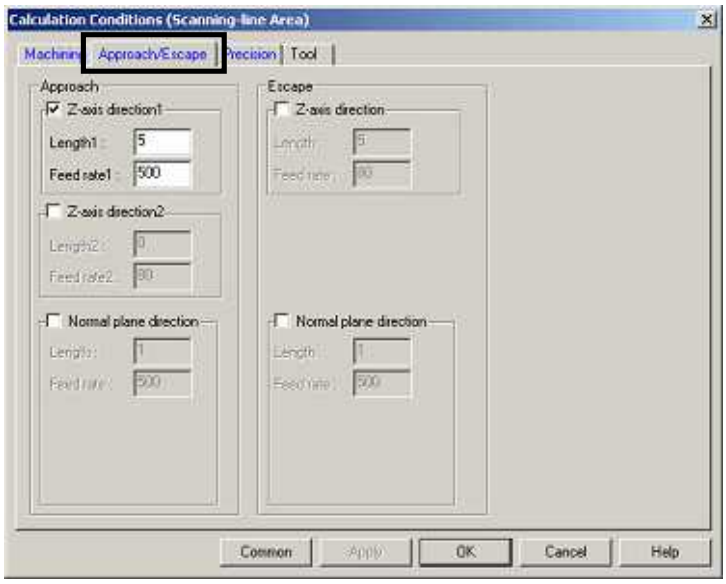
- Check surface [Layer4]
- Range surface [Layer4]

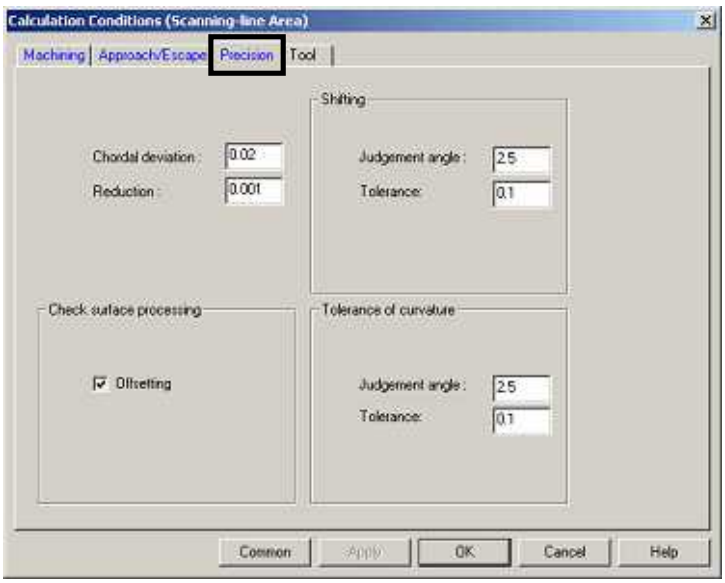
Top Z = -3.6

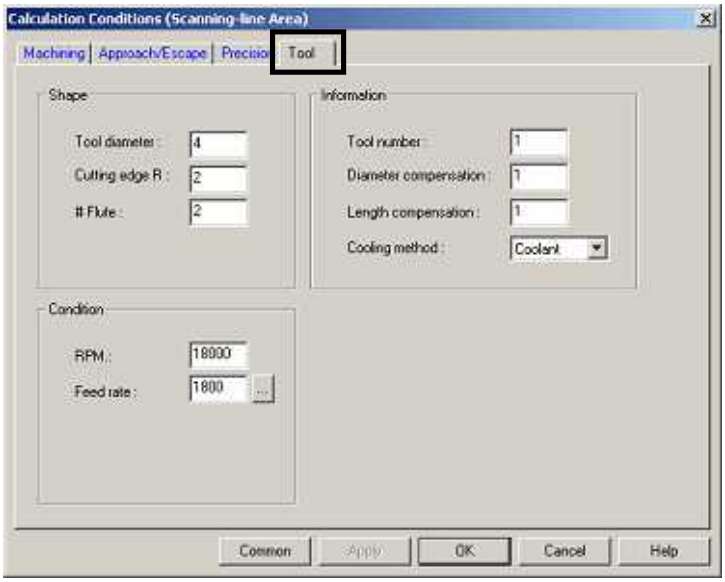
Bottom Z = -7.62

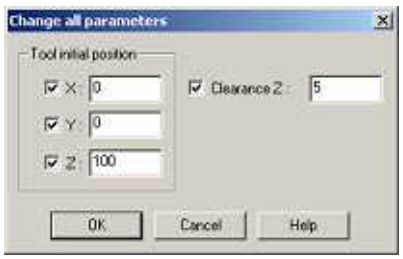







Calculation Conditions (Scanning-line Area)
Machining
Approach/Escape
Precision
Tool
Shifting
Chordal deviation :
0.02
Reduction :
0.001
Judgement angle :
25
Tolerance:
0.1
Check surface processing
Offsetting
Tolerance of curvature
Judgement angle :
25
Tolerance:
0.1
Common
Apply
OK
Cancel
Help


Calculation Conditions (Scanning-line Area)
Machining
Approach/Escape
Precision
Tool
Shape
Tool diameter :
4
Cutting edge R :
2
Flute :
2
Information
Tool number :
1
Diameter compensation :
1
Length compensation :
1
Cooling method :
Coolant
Condition
RPM :
18000
Feed rate :
1800
Common
Apply
OK
Cancel
Help


Change all parameters
Tool initial position
X :
0
Y :
0
Z :
100
Clearance Z :
5
OK
Cancel
Help