ZW3D CAM

3X Milling

Contents

3X Milling

Key Points:

- ✧ Roughing tool path generation
- ✧ Path pattern & boundary setting
- ✧ Finishing operation type & tool path generation
- ✧ Text engraving creation
- ✧ Tool path edit & transfer

This document is used for advanced 3x milling training. The most important and frequently used operations and its parameter settings will be introduced in this document. In addition, the document will share some skills/tips to help users get familiar with ZW3D 3X milling. For new users, we strongly recommend them starting with another fundamental CAM tutorial (ZW3D_FromEntryToMaster_2X Machining.pdf) before learning this tutorial.

Notice:

1) This tutorial is based on ZW3D 2019 version, some functions or icons may not match the current version.
2) All the tutorial models can be found in installation folder: ...training\3X machining model

1.1 3X Milling Introduction

Generally, when we start a new 3X milling programing, the first step is to evaluate the manufacturing model so that we will have a general idea on how to simplify a complicated model, and then apply proper roughing, rest roughing, semi finishing, finishing operations to generate tool path. The tool path should be verified to avoid work pieces damage and to ensure high-quality machining. Finally we will specify a suitable post processor to transfer the tool path to GM code for manufacturing. The work flow in ZW3D 3X milling is as below:

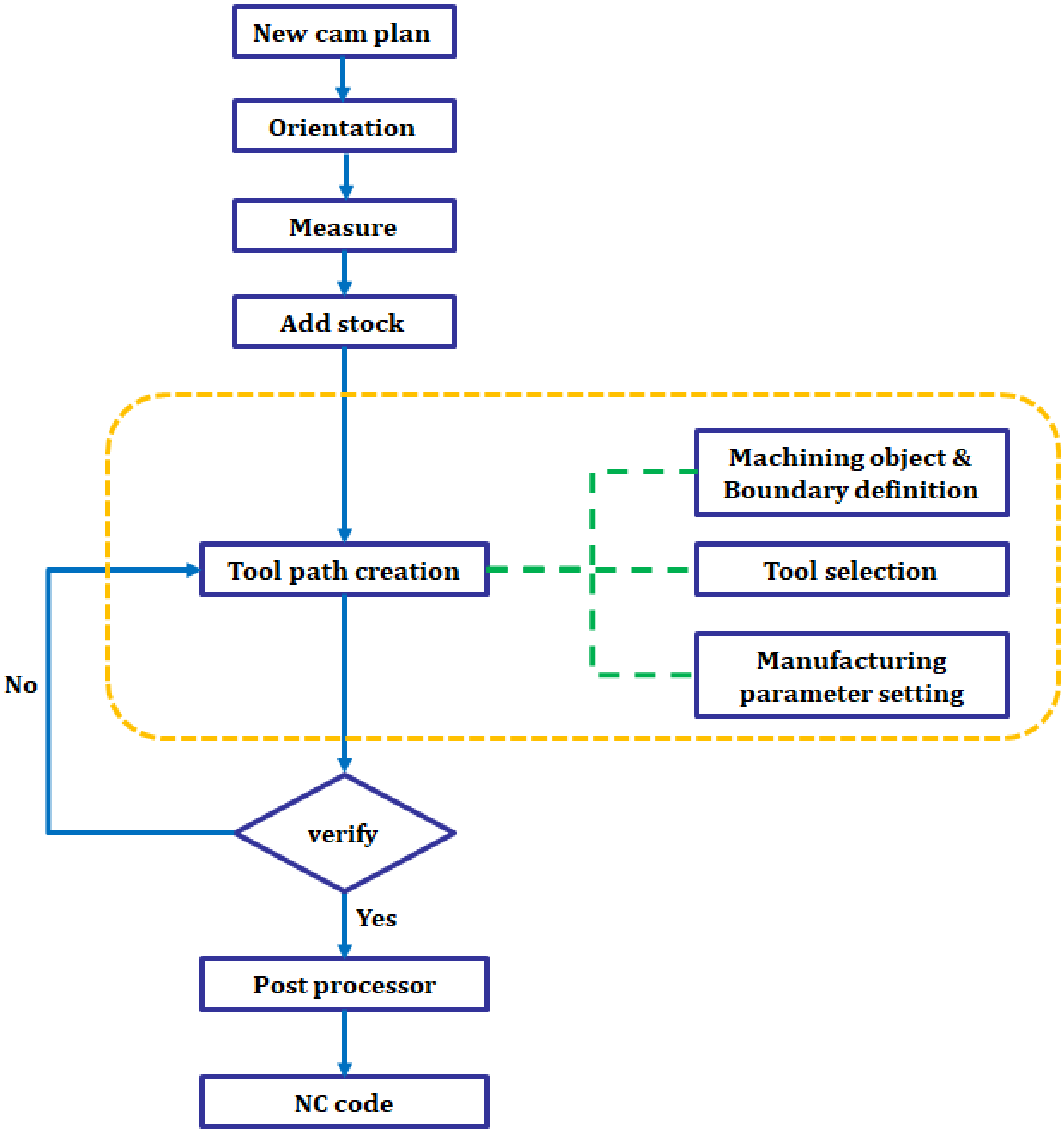


Figure1 3X Workflow

1.2 Roughing

Roughing operation is used to quickly remove the material from stock, which will use a big size tool and big stepdown to ensure manufacturing efficiency. And of course, we need to consider leaving a uniform rest material after roughing operation for next process. ZW3D provides 4 types of roughing operations to match different machine situations.

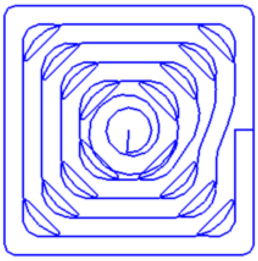
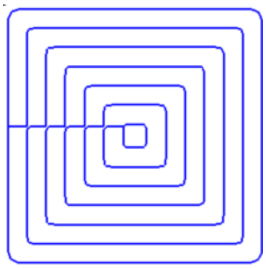
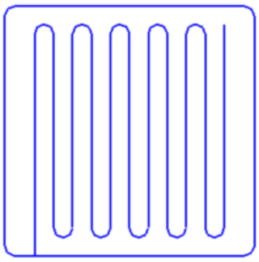
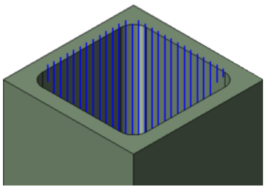
Operation	Pattern	Application scenarios
Smooth Flow		Smooth flow roughing operation used for hard material which requested a uniform chip-load.
Offset 2D		Offset 2D roughing operation can be used for almost all the material. This operation is recommended as the first choice for roughing operation.
Lace		Lace roughing operation is used for soft material.
Plunge		Plunge roughing operation is used for generating tool path in some model with big size, request special machine to work with it.

Figure2 Roughing operation type

1.2.1 Requirement for generate roughing tool path

To generate roughing tool path successfully there are three necessary requirements:

- Specify roughing operation type
- Specify tool (tool need to smaller than machine area)
- Specify Feature (part, and stock or profile boundary)

Generally, if we specified these three necessary requirements, we can get a tool path. No matter whether the tool path is proper or not.

Task1: Create a roughing tool path

STEP 01 Open "Roughing.Z3" file and double click "3X_CAM" to go into CAM space.

STEP 02 Click **Setup** > **Add Stock** to add a stock for the part. (All the setting are by default)

STEP 03 Click "Yes" to hide the stock

STEP 04 Select **3x Quick** > **Offset 2D** roughing operation

STEP 05 Select part and stock, then click OK button to define machining object and area for this operation.

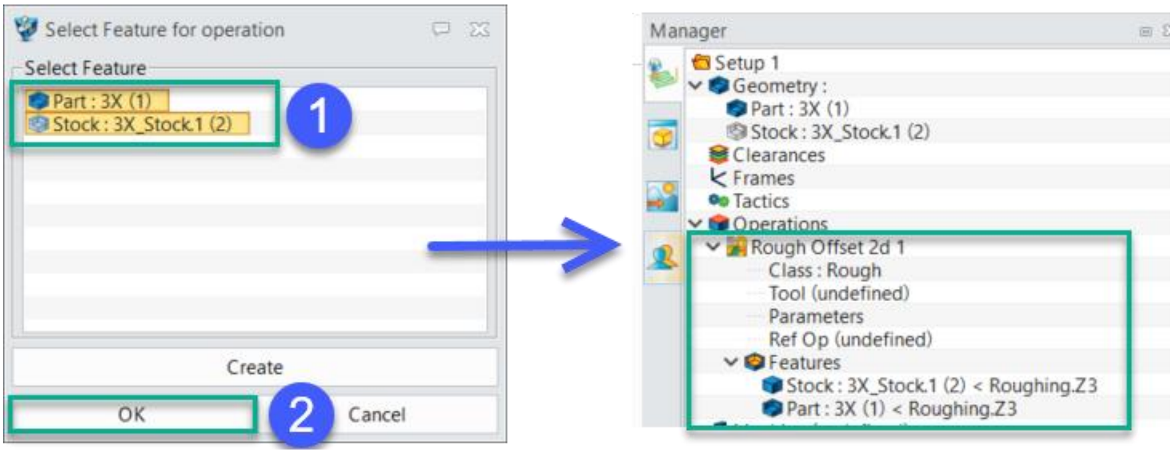


Figure3 Define machining object and area

STEP 06 Input "D10R0" and change radius to "0" in pop-up tool define form.

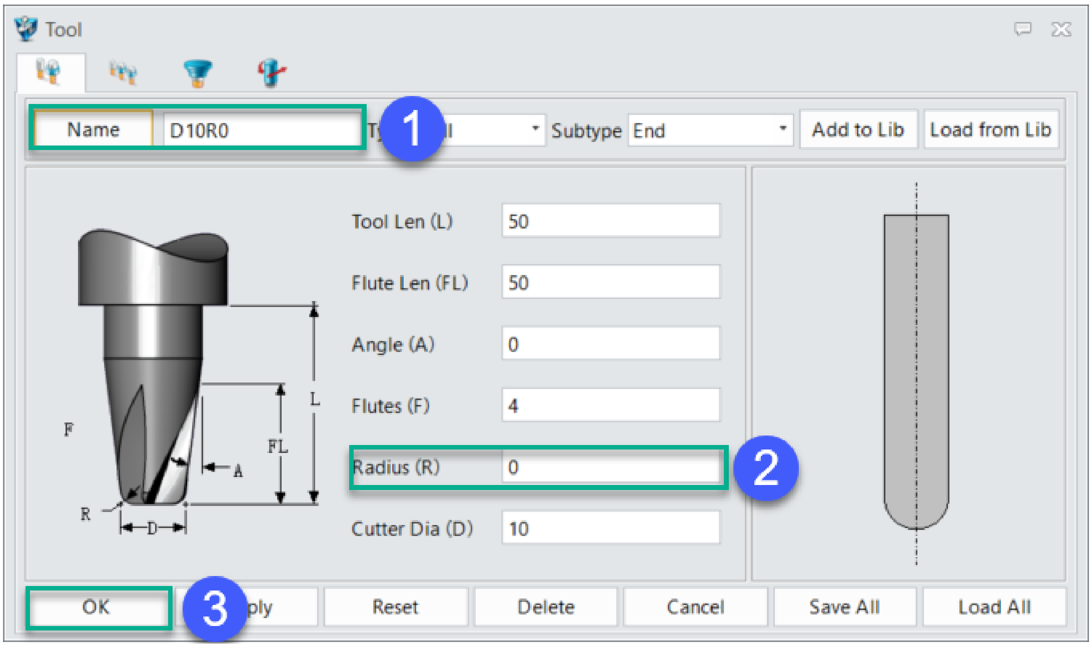


Figure4 Cutter definition

Tips: if the tool define form didn't pop up. Double click "Tool (undefined) "in operation list to call out it.

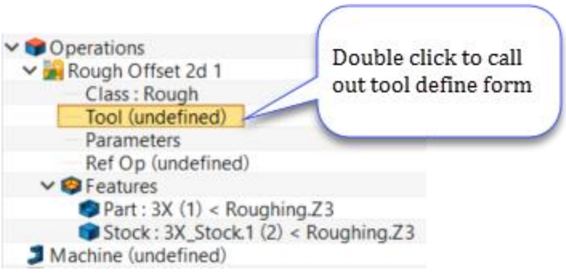


Figure5 Callout cutter definition

STEP 06 Click "Yes" to calculate the roughing tool path.

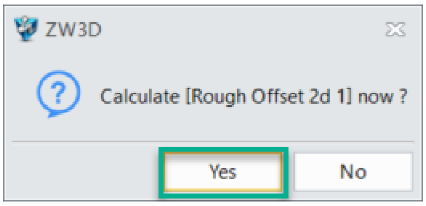


Figure6 Tool path calculation

Tips: if the calculation dialog didn't pop up, right click the operation named "Rough Offset 2d 1", then click "Calculate" button to generate the tool path.

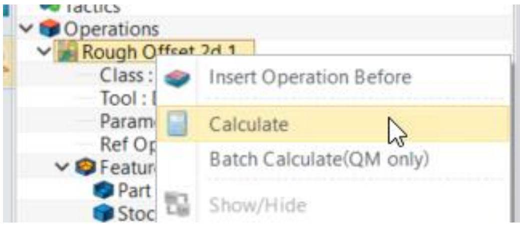


Figure7 Callout tool path calculation

STEP 07 After calculation we get a result as below:

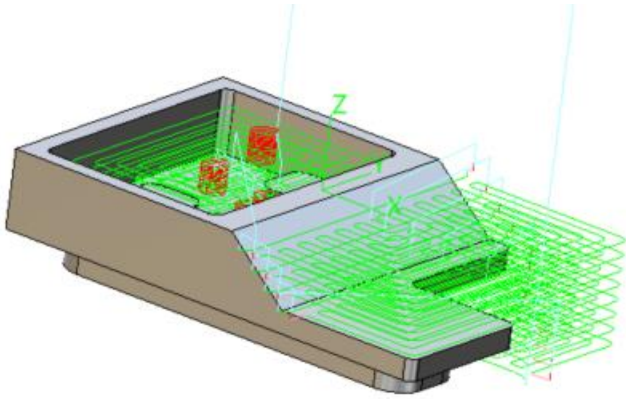


Figure8 Roughing tool path

1.2.2 Detail setting in 3X roughing operation

In section 1.2.1 we discussed the necessary condition to generate the tool path, but to generate a proper tool path to match real machine situation, we need more parameters to control the tool path generation.

Double click "parameters" on operation list, it will pop up an operation detail setting form.

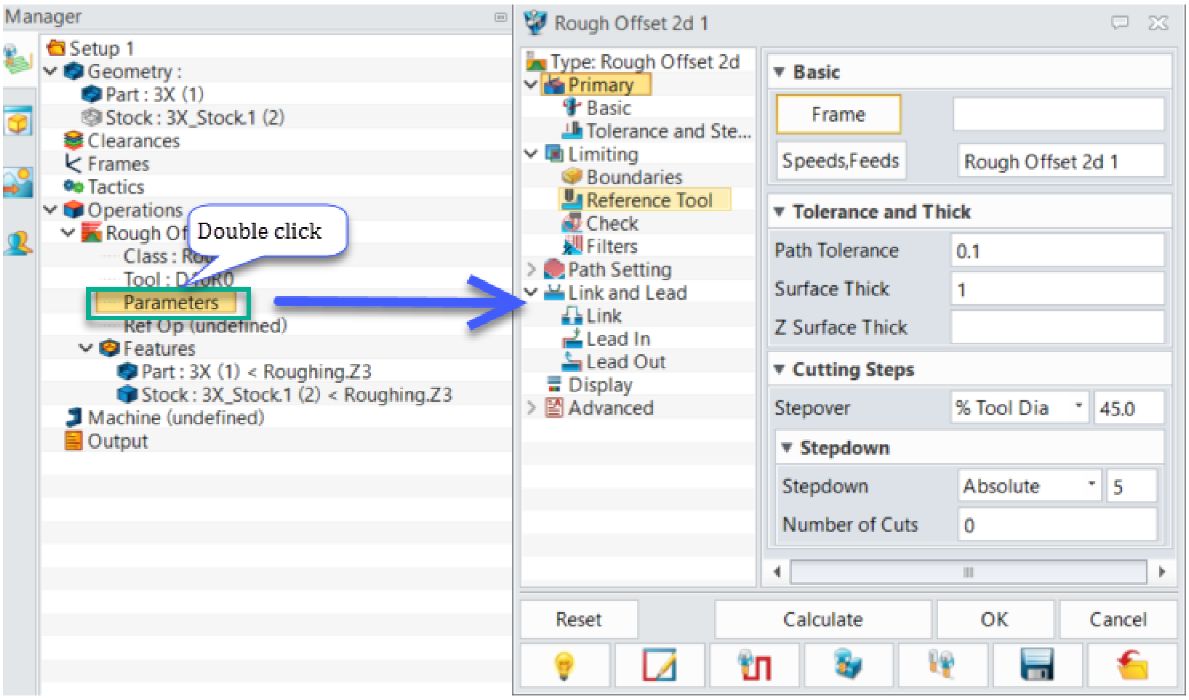


Figure9 Parameters setting form

1) **Primary** parameters

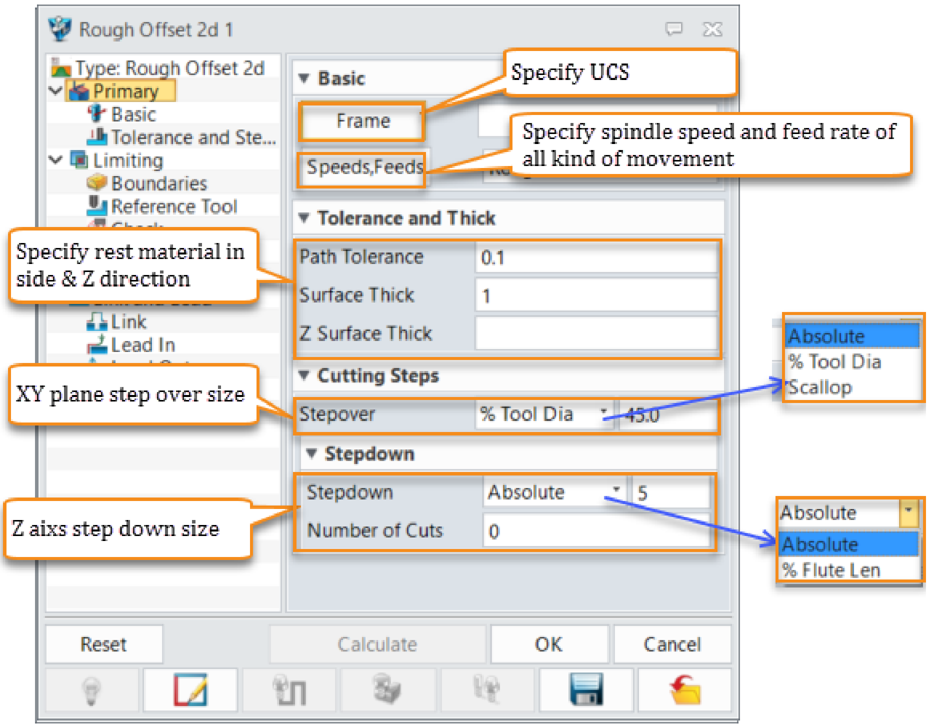


Figure10 Primary parameter tab

Frame: Specify a UCS (user coordinate system) to calculate the tool path. The frame should be created before we specify it in the setting form.

Task: Create a new frame and use it to calculate the tool path in the bottom side.

STEP 01 Double click the icon of the stock to show the stock

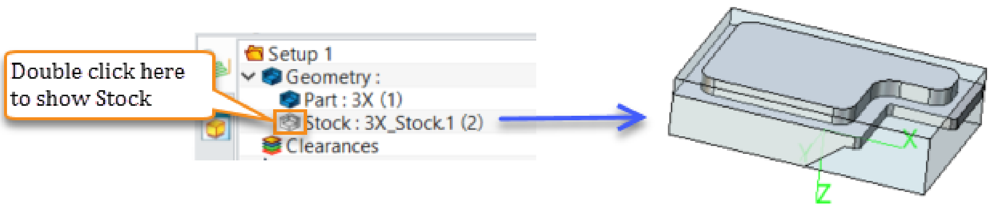


Figure11 Show stock

STEP 02 Create a new frame at the lower left conner of the stock.

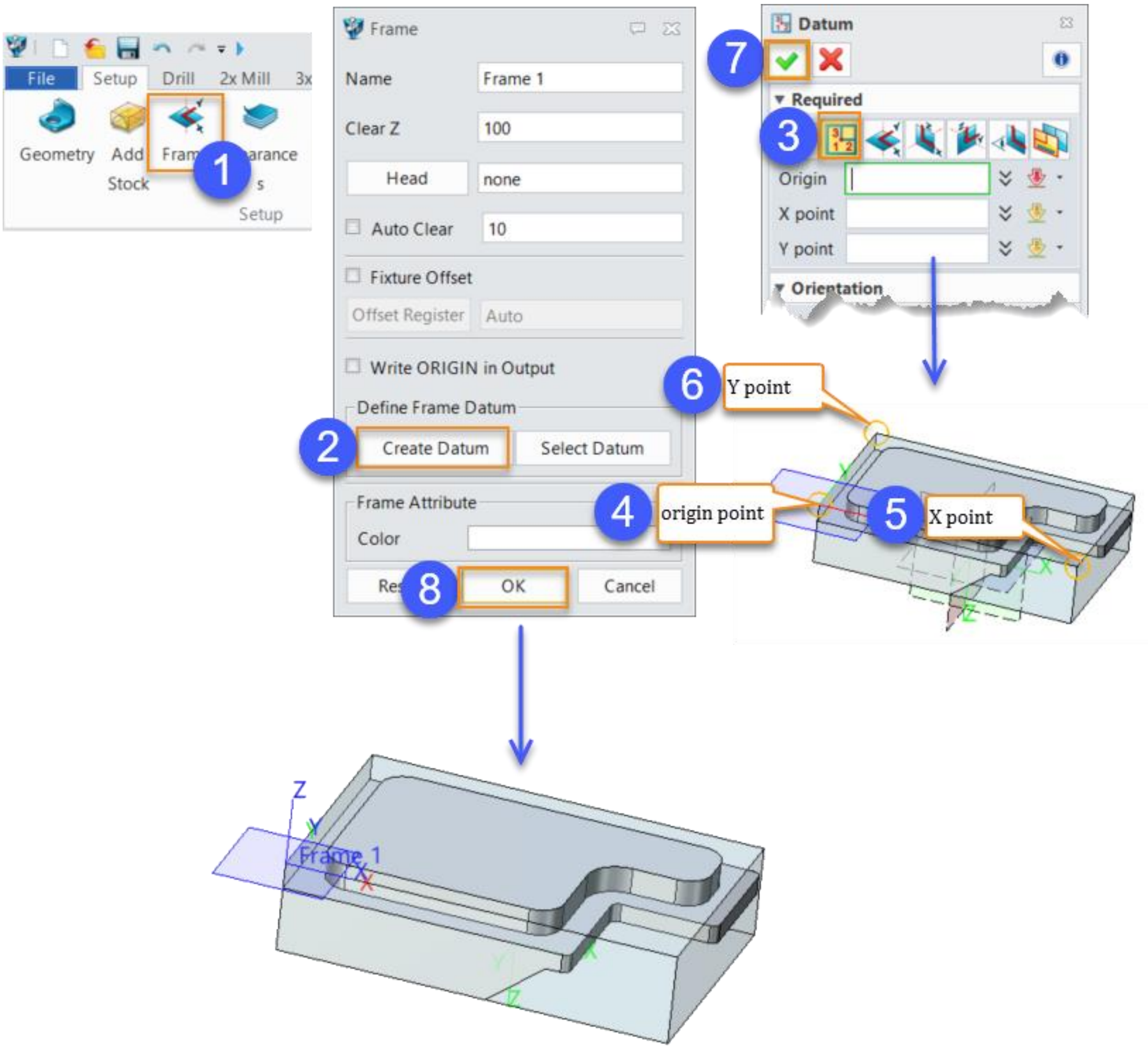


Figure12 Create a frame (UCS)

STEP 03 Right click on Rough offset 2d 1 operation and copy it.

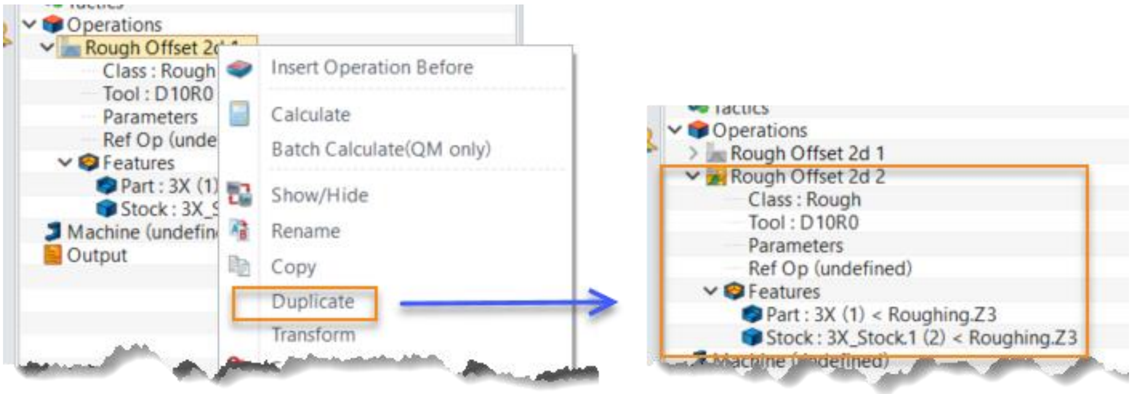


Figure13 Duplicate operation

Tips: "Duplicate" option can help you quickly create a new operation by referring an existing operation.

STEP 04 Double click Parameter on new copied operation, then select the Frame 1

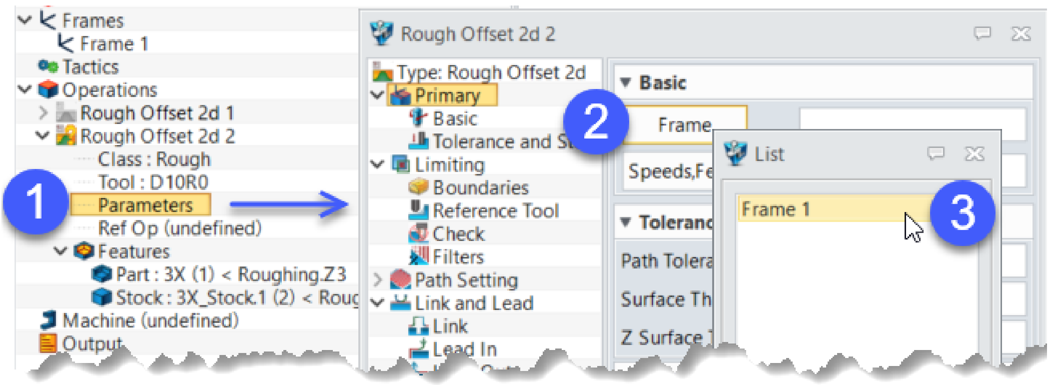


Figure14 Specify a frame (UCS)

STEP 05 Calculate the tool path of Rough Offset 2d 2 operation.

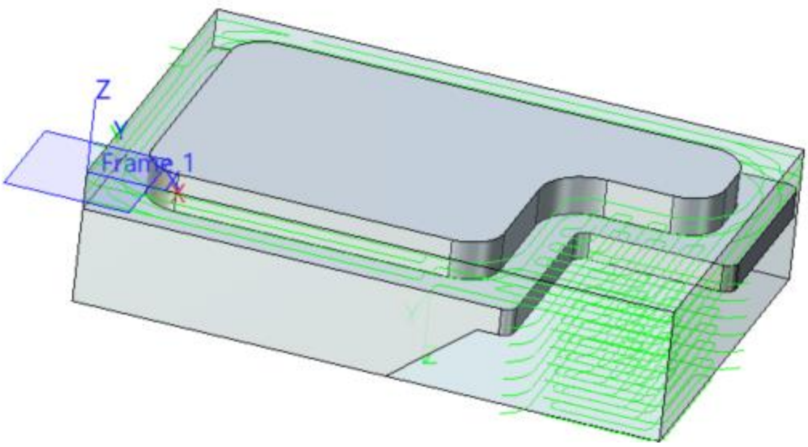


Figure15 Specify a frame (UCS)

The Frame have two application scenarios:

1, To create the tool path which the Z Axis direction of the machining area is different from the system default Z Axis direction. See the example above.

2, To output the NC code, the XYZ coordinate value will be set according to the specified frame to output.

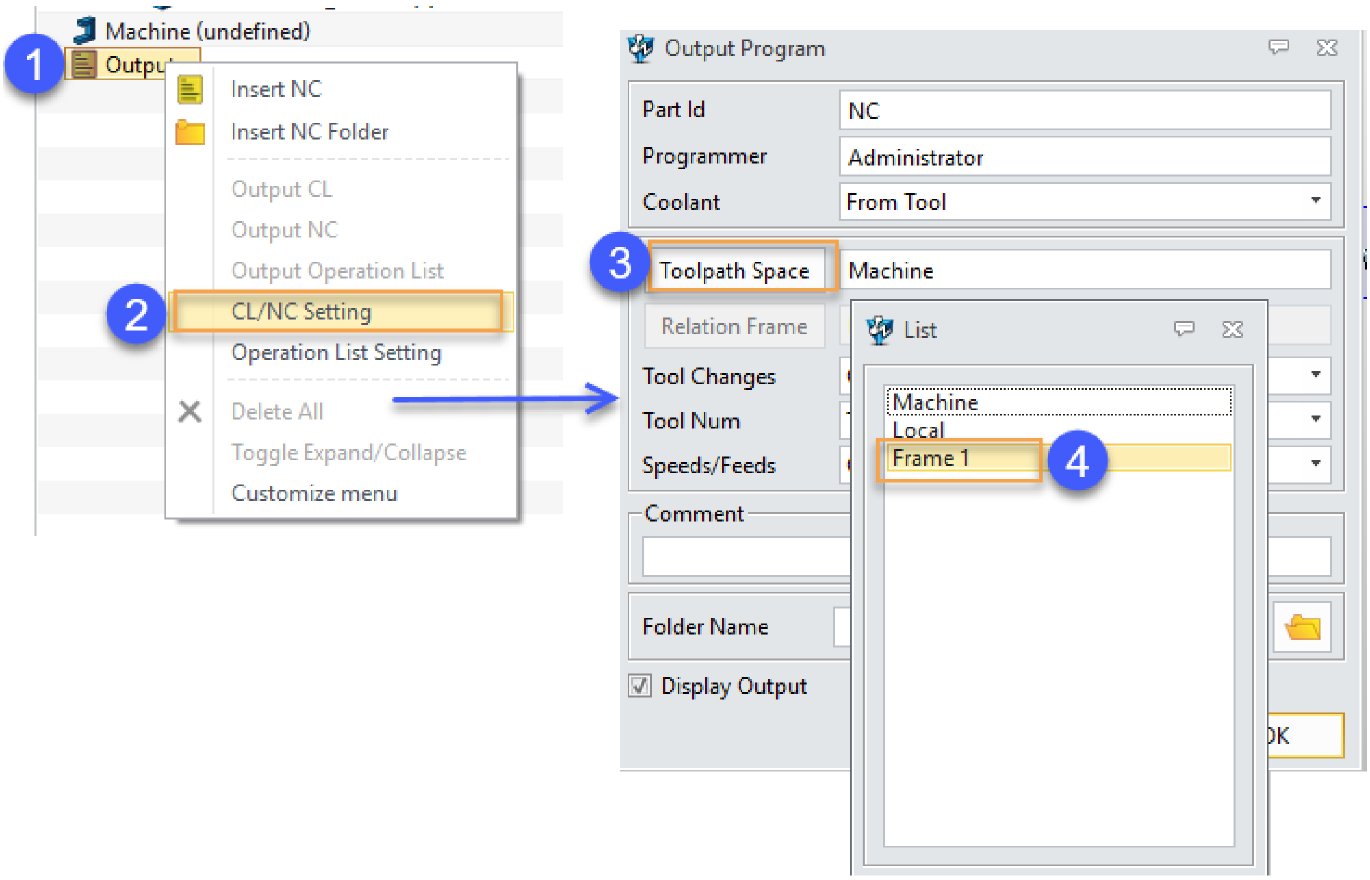


Figure16 Output NC with a specified frame (UCS)

Tips: Holding the mouse pointer above the input box of the correspoinding option, it will pop up an illusalution of funciton description.

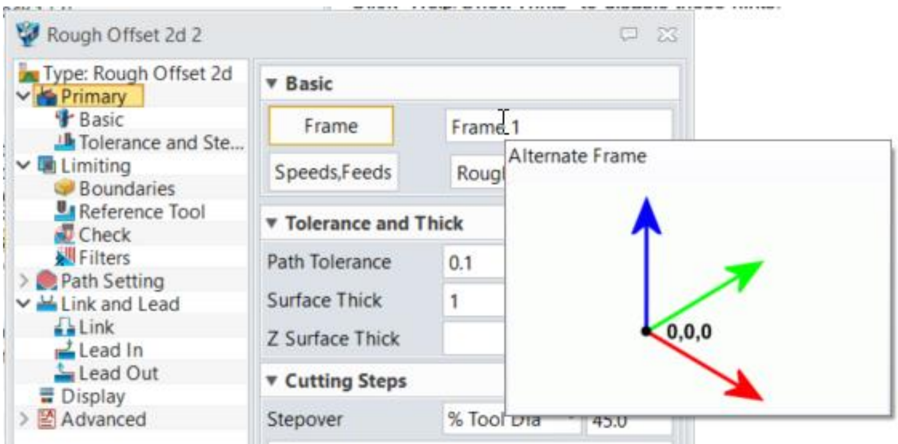


Figure17 Show the illustration

Path Tolerance: This sets the tolerance for internally converting the part geometry to tessellation (triangulated) geometry

Surface Thick & Z Surface Thick: Specify the rest material for the operation.

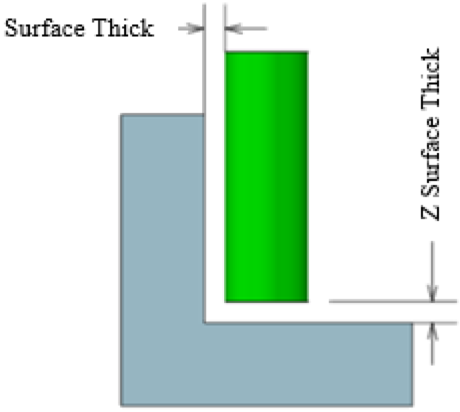


Figure18 Surface & Z Surface Thick

Tips: The Surface Thick or Z Surface Thick can be left blank without any value. Different settings will result in different effects. For more details please refer to the image as below:

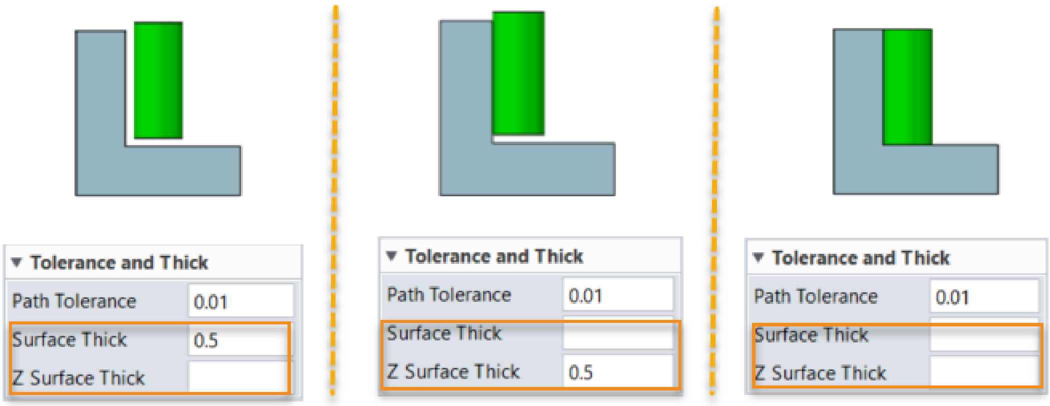


Figure19 Surface & Z Surface Thick setting

For the beginner we suggest setting both the “Surface Thick” and “Z Surface Thick” to avoid mistake.

Number of Cuts: To uniform the rest material at incline surface, ZW3D allow adding specified number of cut at each stepdown layer.

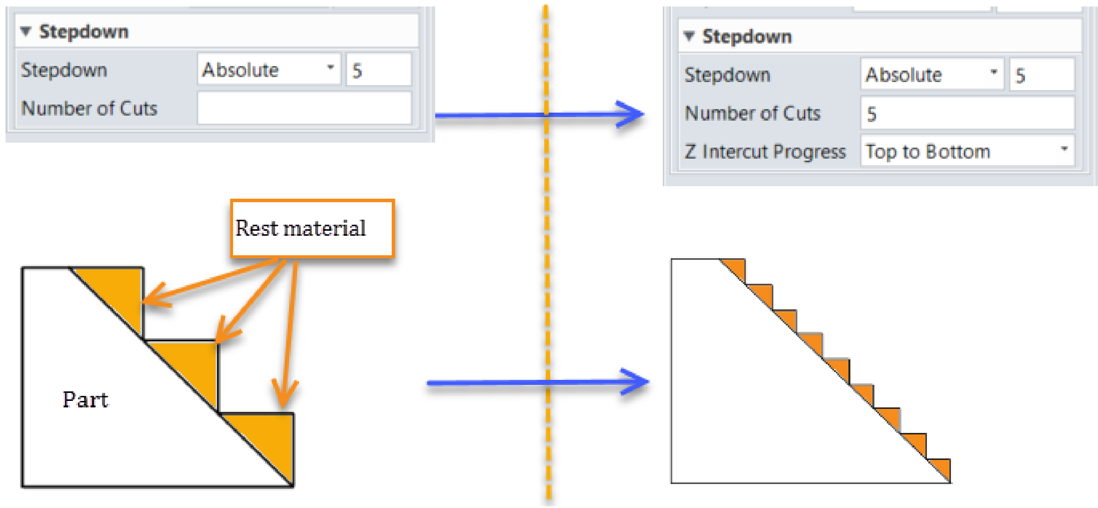


Figure20 Number of cuts

2) **Limiting** parameters

Limiting tab is used for defining the machining area. In other words, we can set different conditions to control the tool path generating area, such as boundaries, rest material area by calculating a reference tool, checking table, clamps, holder collision, and use filter to remove some tiny tool paths.

- Boundaries

In ZW3D, we allow users to calculate the roughing tool path without defining a stock. Thus, when talk about the boundaries, we can divide it into two parts to discuss – with or without stock.

a) Without stock

When we didn't define a stock for calculating the tool path, the boundary will be defined by containment type – Silhouettes & Simple box. The containment type is only used for calculating the boundary without specifying a stock.

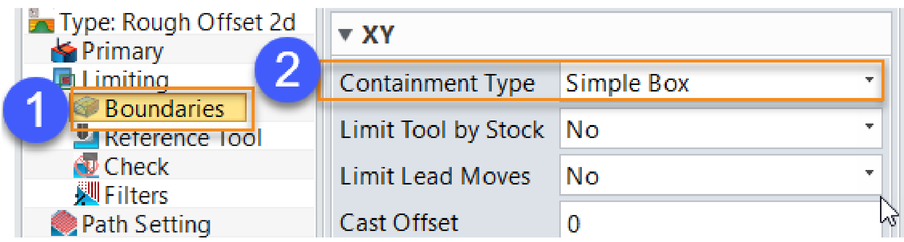


Figure21 Containment type

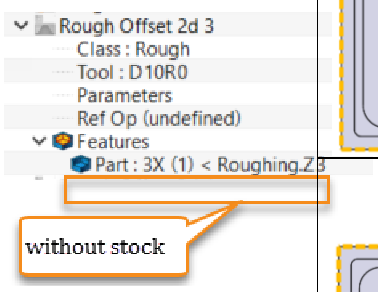
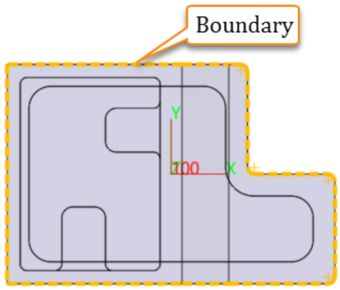
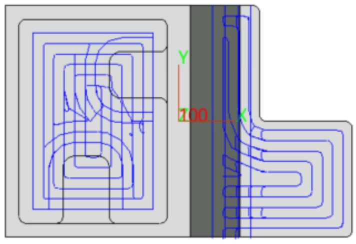
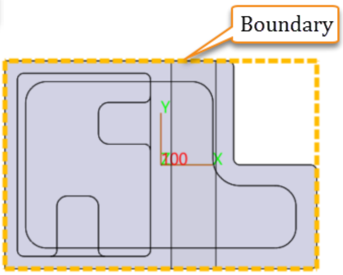
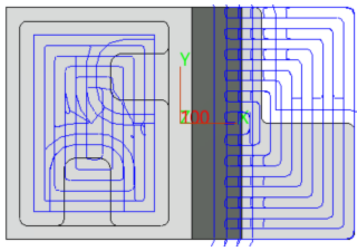
Type	Stock setting	Calculate boundary	Tool path
Silhoueete			
Simple box			

Figure22 Boundary setting (without stock)

b) With stock

When we only defined a stock for roughing operation, the machining area will be defined by stock. But if we also define a profile containment in it, the machining area will be defined by the intersection area of stock and profile containment.

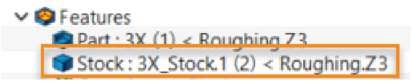
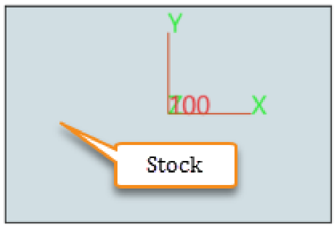
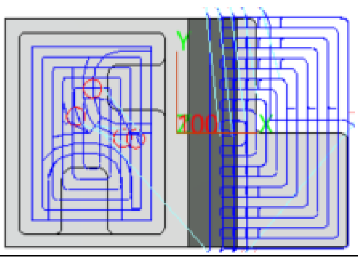
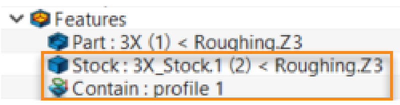
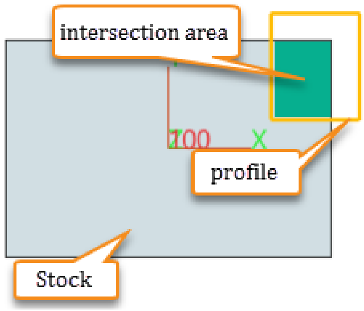
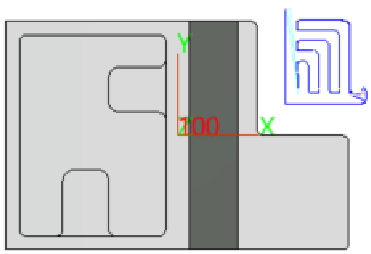
Boundary type		Calculate boundary	Tool path
Stock only			
Stock with profile			

Figure23 Boundary setting (with stock)

- Other boundaries setting

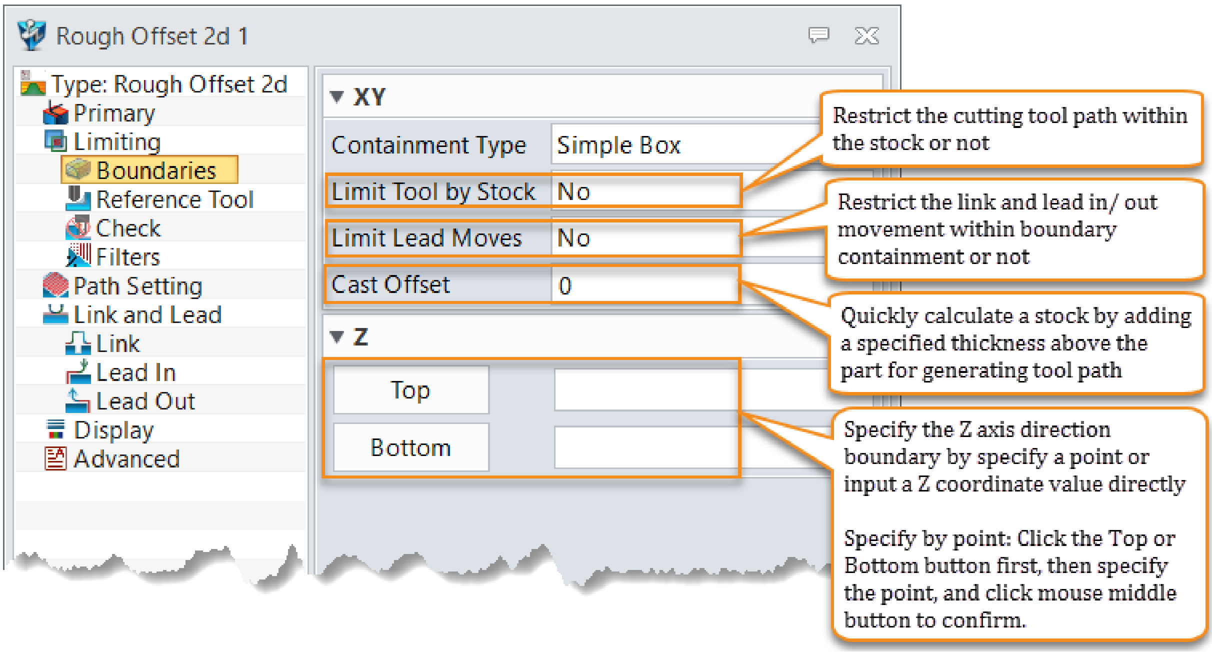


Figure24 Other boundary settings

- Reference Tool

A specified reference tool used to determine which regions could not be cut by the reference tool, then generate tool path to remove the rest material.

- Check

To check the collision of additional parts (table, clamp, tool holder and so on) and correct the tool path.

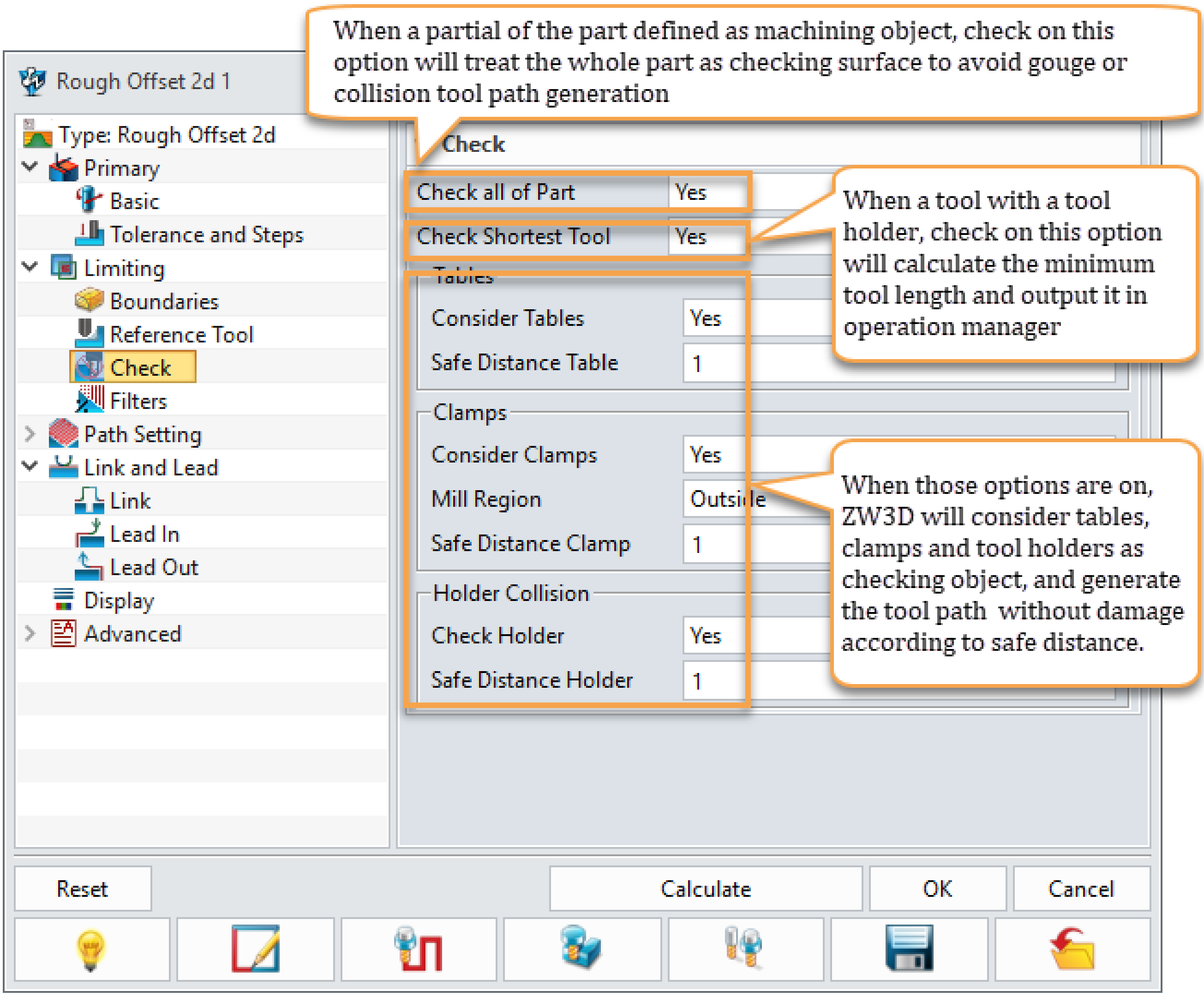


Figure25 Check options

- Filters

To filter out some tiny or unnecessary tool path segments.

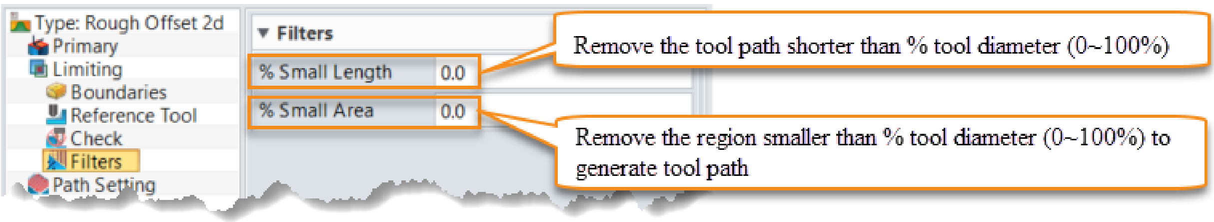


Figure26 Tool path filter

3) **Path Setting**

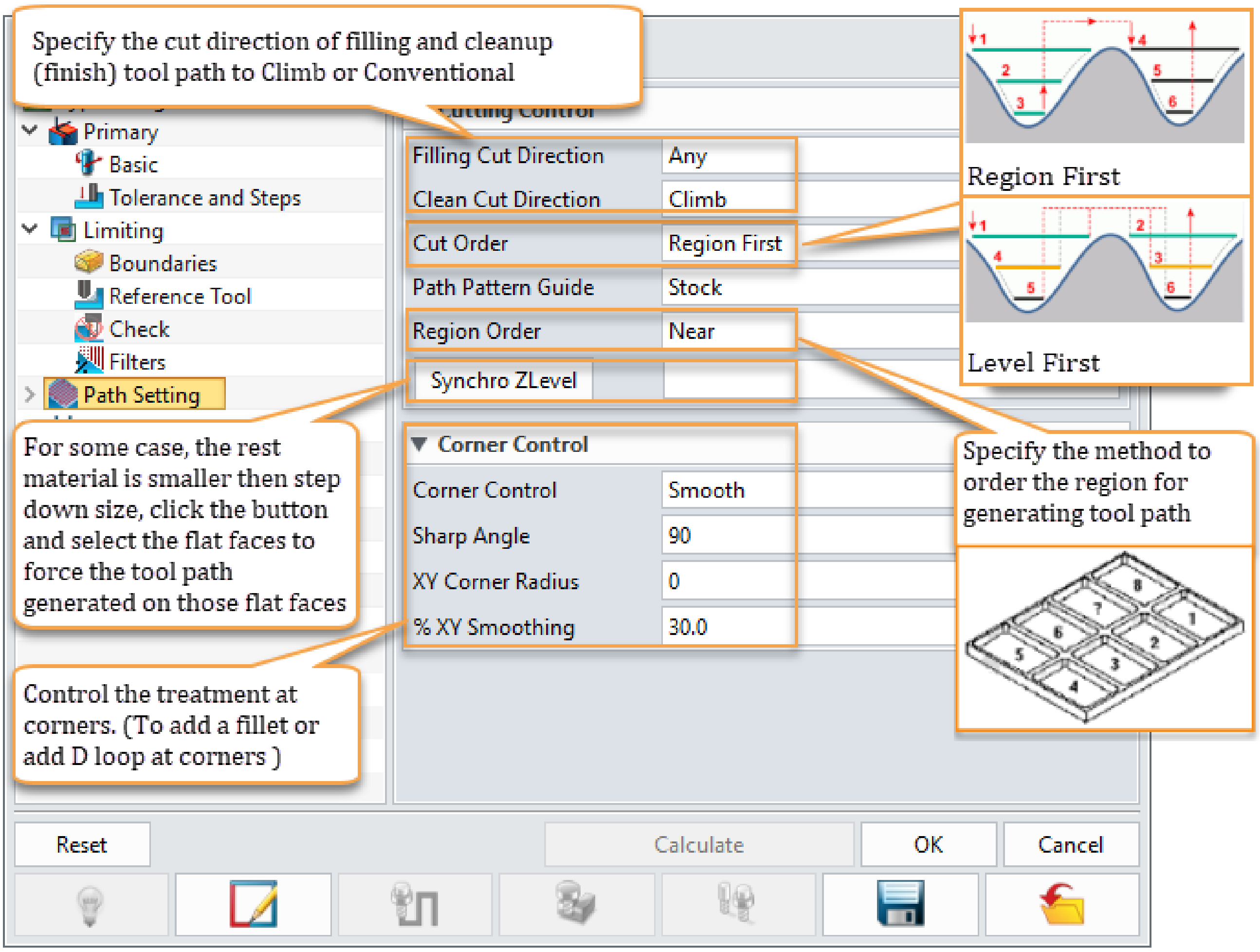


Figure27 Path setting

- Filling & Cleanup tool path

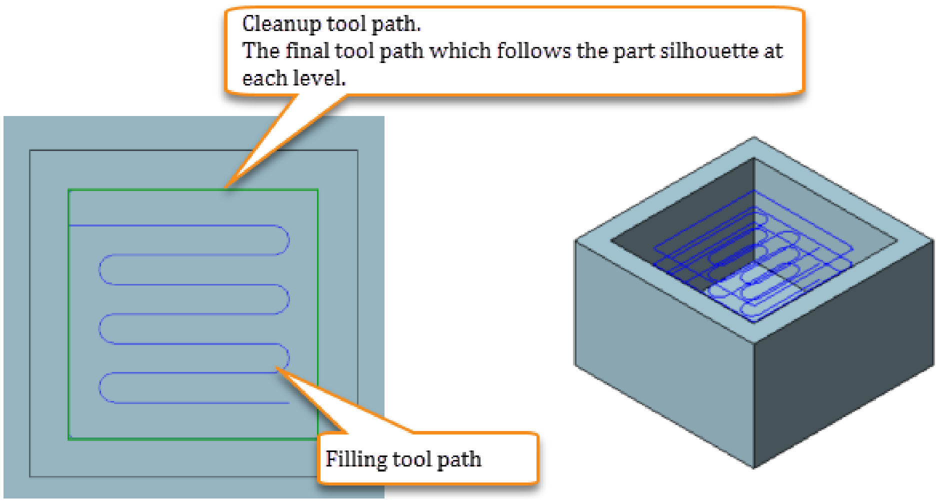


Figure28 Filling & Cleanup tool path

- Cut Direction

User can set the cut direction to Climb, Conventional, Automatic and Any.

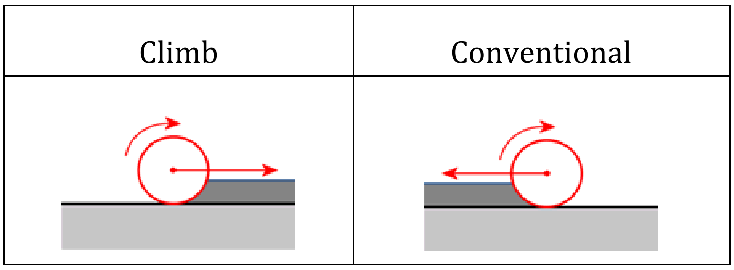


Figure29 Cut direction

Automatic: Selecting automatic will set the direction of cleanup tool path to be the same as filling tool path.

Any: System will assign the tool path to Climb or Conventional in order to get a Zigzag tool path.

- Path Pattern Guide

Control the tool path generated according to part silhouette, stock silhouette, or control by both part and stock.

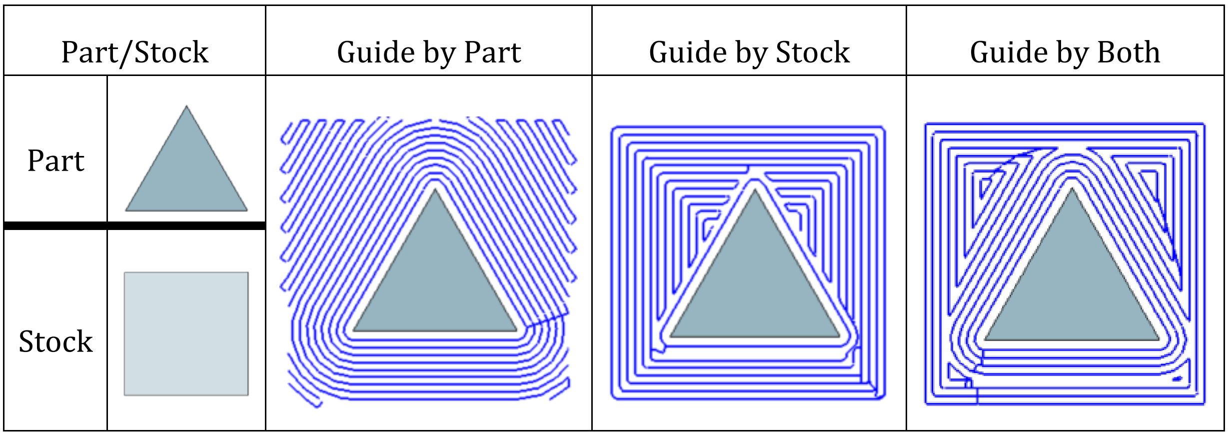


Figure30 Path pattern guide

- Synchro Z Level

Specify the location that are forced to generate tool path to remove material, so as to ensure a uniform rest material for next process.

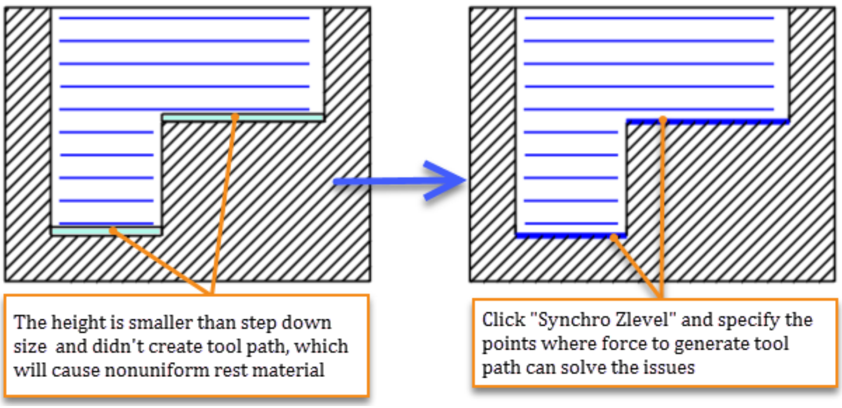


Figure31 Synchro Z level

- Corner Control

Specify the method to treat the corner by Smooth or D loop.

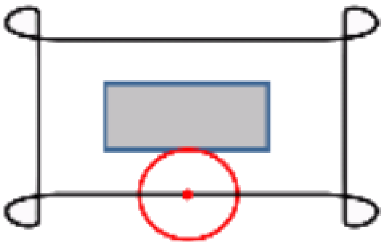


Figure32 Corner control

- Sharp Angle (only available for D loop)

If the corner angle of tool path is smaller than specified sharp angle value, the corner will add a D loop.

- XY Corner Radius

Specify the fillet value of the corner.

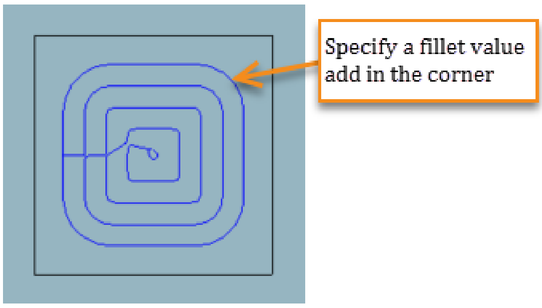


Figure33 XY corner radius

- %XY Smoothing

Smooth all tool paths based on a percentage of the Step Size.

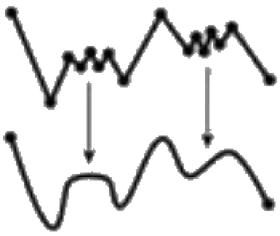


Figure34 XY Smoothing

4) **Link and lead**

In ZW3D, the tool movement will be defined as lead in (engage), lead out (retract), cut, plunge, rapid up/down and link segments. In some case, to optimize tool path, some short links will be treated as cutting movement.

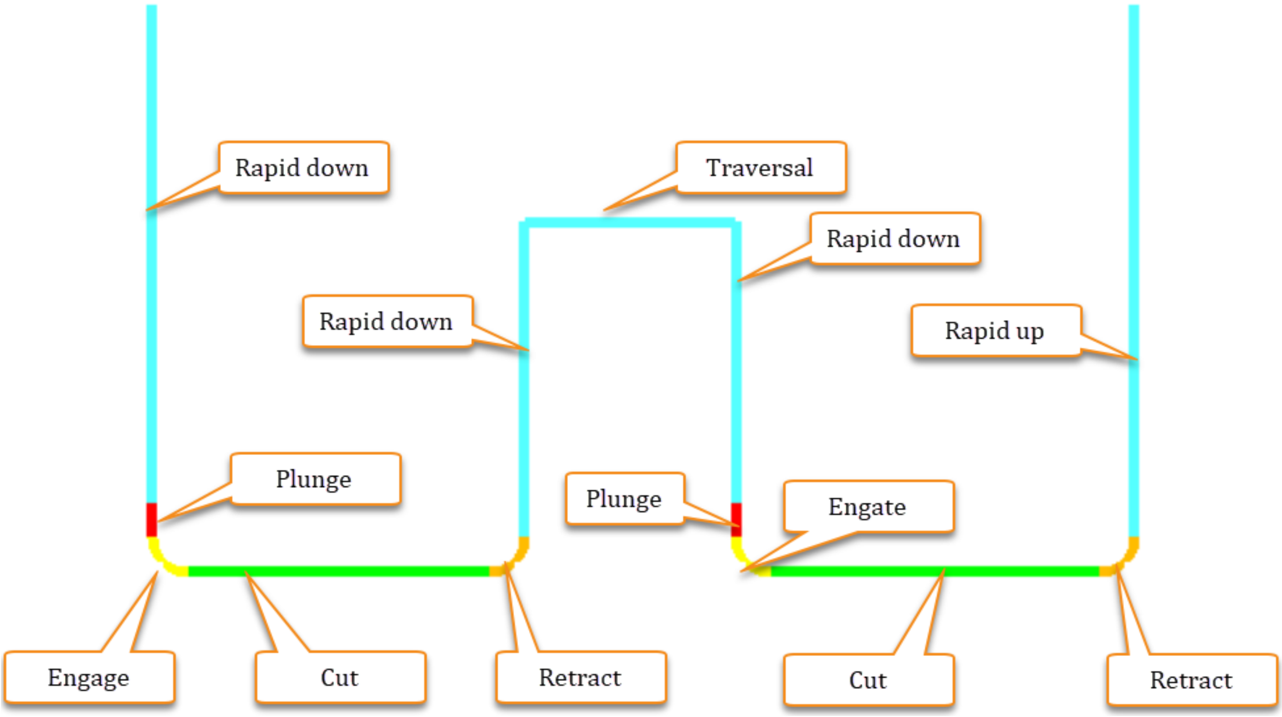


Figure35 Tool movement types

In some cases, to optimize tool path, some short link will be treated as cutting movement. Thus the speed setting in Speeds and Feeds table will be as below:

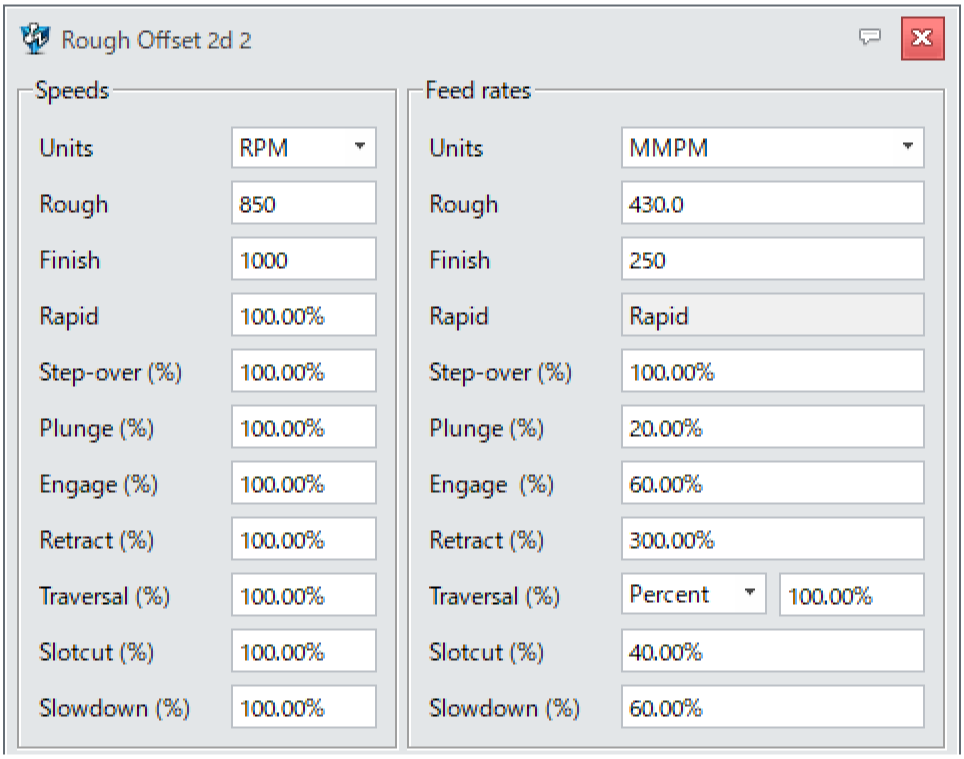


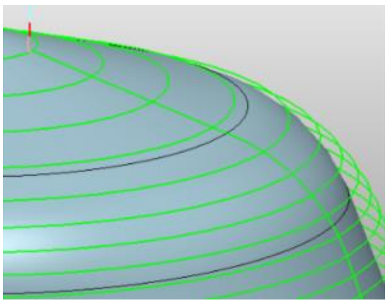
Figure36 Speed and Feed

- Link

The link is used to join the cutting tool path segment. In ZW3D, there are 2 link types: short link and long link. Short link will be mostly used to join the cutting tool path segments in the same cutting region. And long links will be used to join the cutting tool path segments in different cutting regions.

a) Short link

ZW3D provides 5 types of short link to help users meet different machining needs.

Link type	Description	Tool path	Tool lift
On surface	The link will connect the next paths with a straight line and project the line on the surface to get a 3D link		No

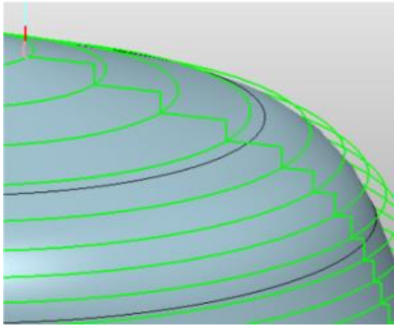
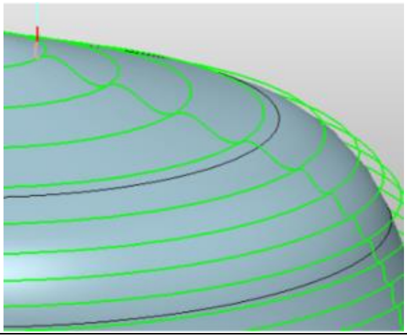
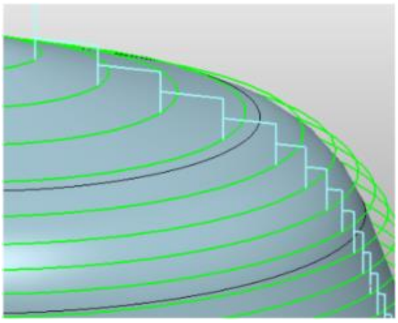
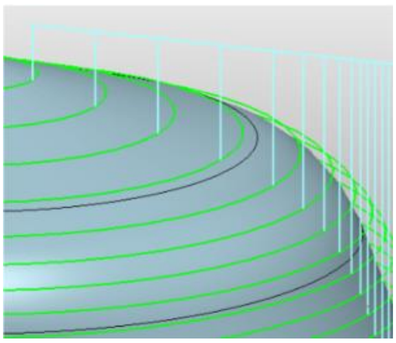
Step	The link will move straight to the next path by separated XY movement and Z movement.		No
Spline	System will create a smooth spline between two paths		No
Optimized	The link will lift the tool to safe distance and then move straight to next path by separated XY movement and Z movement.		Yes
Clearance	The link will lift the tool to clearance plane and then move straight to next path by separated XY movement and Z movement.		Yes

Figure37 Short link types

b) Long link

Long link only has optimized and clearance link types which are the same as short link.

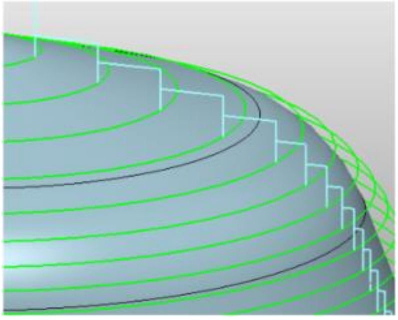
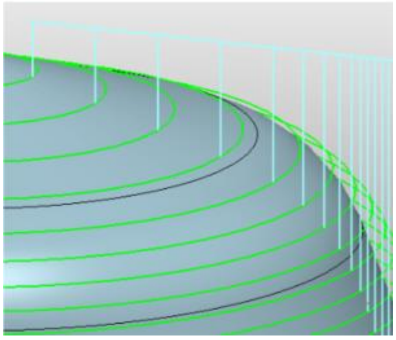
Link type	Description	Tool path	Tool lift
Optimized	The link will lift the tool to safe distance and then move straight to next path by separated XY movement and Z movement.		Yes
Clearance	The link will lift the tool to clearance plane and then move straight to the next path by separated XY movement and Z movement.		Yes

Figure38 Long link types

c) Link setting

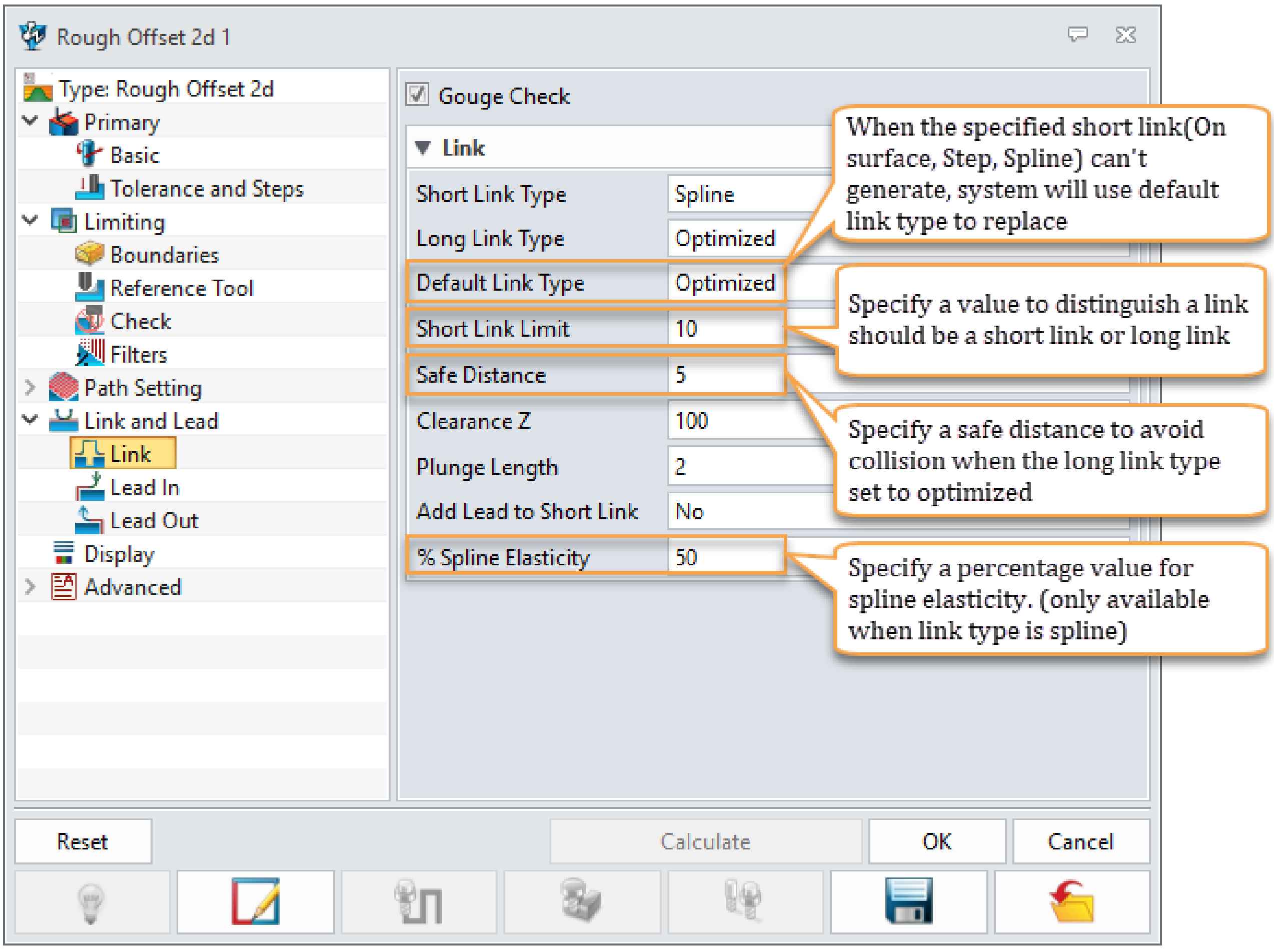


Figure39 Link setting

- Safe Distance

Specify a safe distance to avoid collision when long link type is set to “Optimized”.

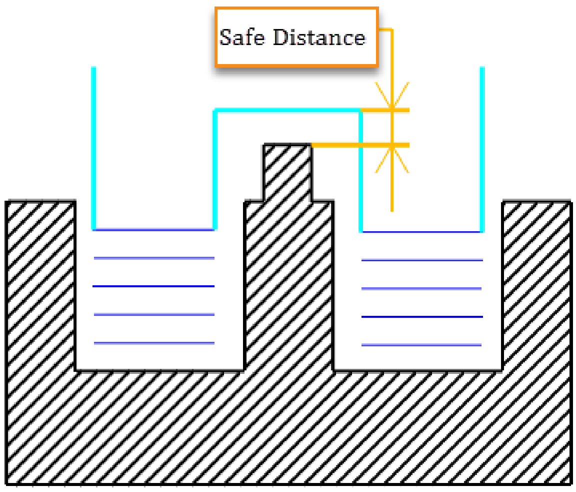


Figure40 Safe distance

- Lead in/out

ZW3D 3X milling provides 4 types of lead in/out for user, the details are as below.

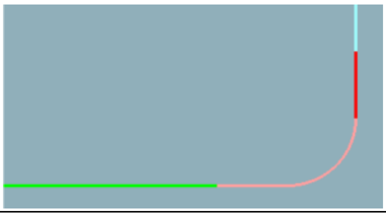
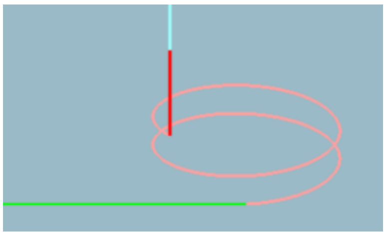
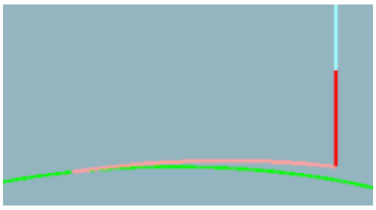
Lead type	Description	Tool path
None	The lead in/out is created and empty when selecting this type.	
Arc_Line	Extend a tangent line and insert a tangential arc move at the end of the line. So the lead in goes the arc first then the line, the lead out goes the line first then the arc.	
Ramp_Arc	The tool will do a ramp into the material at a specified angle, following an arc with specified radius.	
Along tool path	The tool will do a ramp into the material at a specified angle, following the direction of the cut. If the cut is not close to a span, the lead will follow the reverse direction of the cut.	

Figure41 Lead in/out types

Tips: In some narrow areas, if the space is not enough to create a "Ramp_Arc" type lead in, it will firstly degrade to "Along tool path" type. And if also fail to generate, it will degree to "None" type. The degrade rule will follow the order: "Ramp_Arc" >> "Along tool path" >>"None"

- Lead in/out setting

The setting parameter for lead in and lead out is the same.

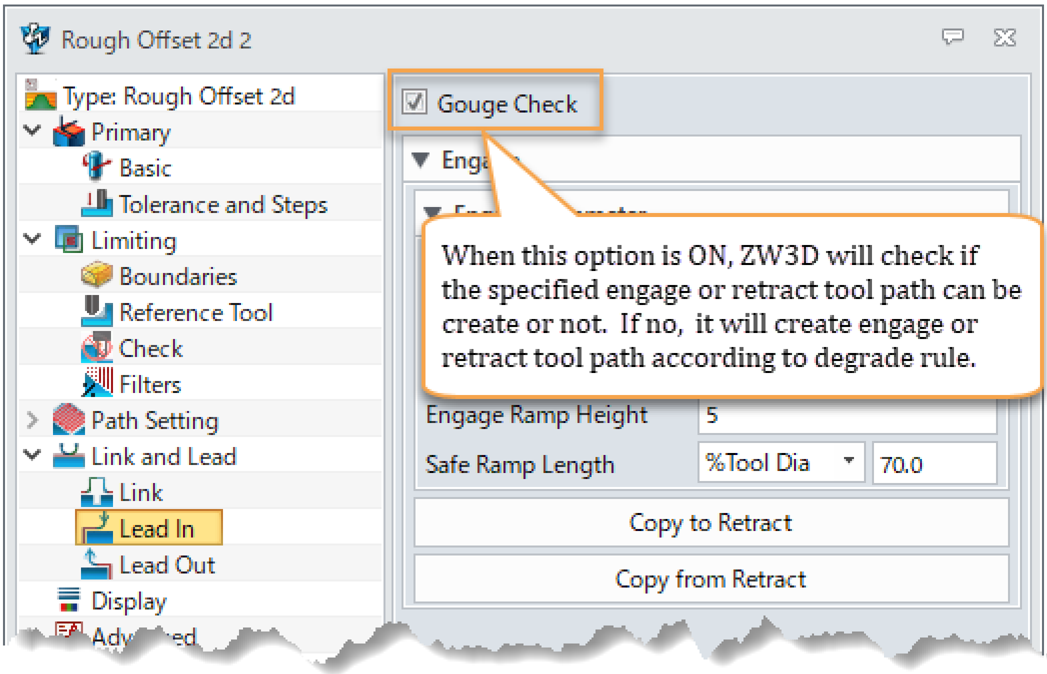


Figure42 Engage

○ Safe Ramp Length

When users use a facing cutter, if the machining region is too small, it may cause tool damage. ZW3D will check the engage tool path according to safe ramp length, if engage tool path is shorter than specified "Safe Ramp Length", it will remove the unsafe tool path to avoid tool damage.

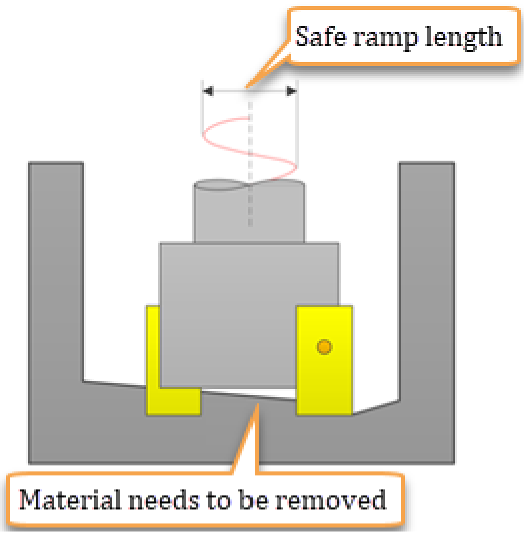


Figure43 Safe ramp length

5) **Display**

The parameters in this tab are used to control tool path display status, such as color, line type, and display mode and each movement type on or off.

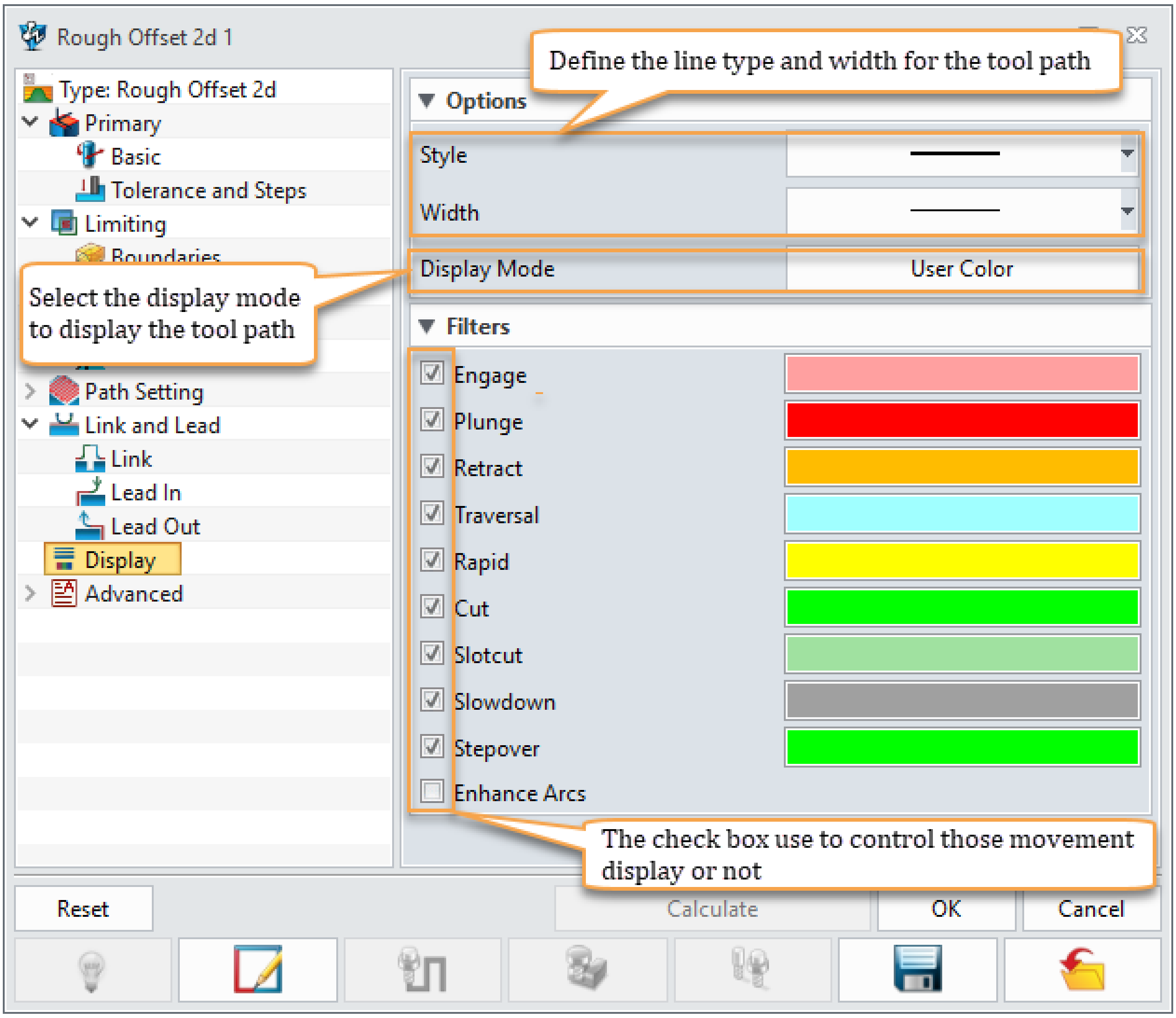


Figure44 Tool path display

6) **Advanced** parameter

For some special cases or advanced users, ZW3D provides some advanced parameters to help users get the result they expected.

- Path Pattern

a) Spiral

Convert the tool path patter to a spiral one, more details setting as below:

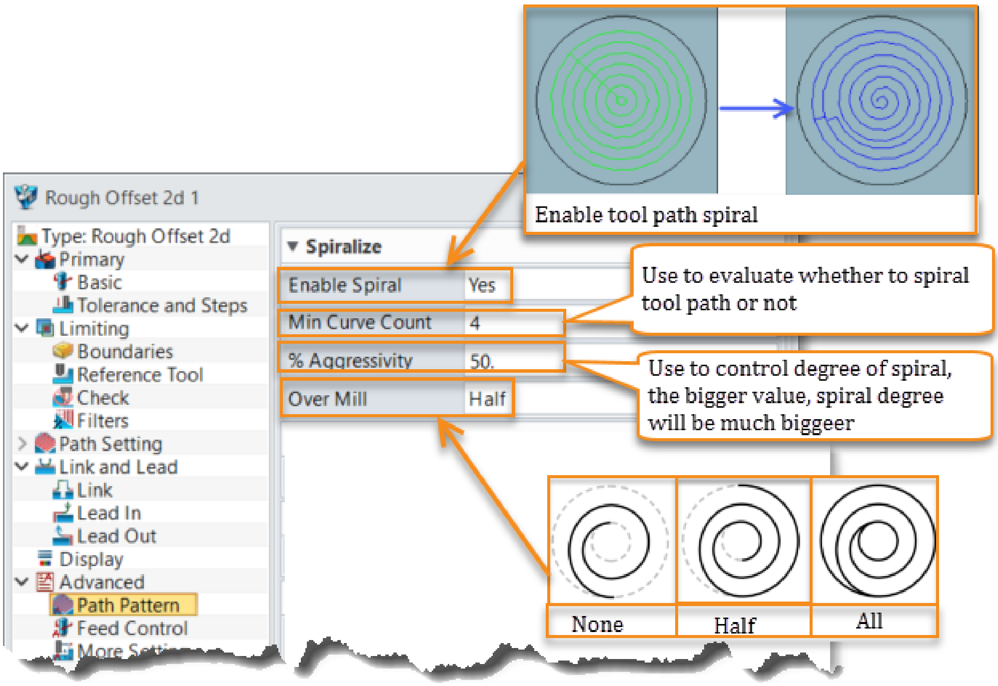


Figure45 Advanced setting -- Spiralize

○ Min curve count

If turns of the filling tool path is smaller than min curve count. It won't spiral the tool path.

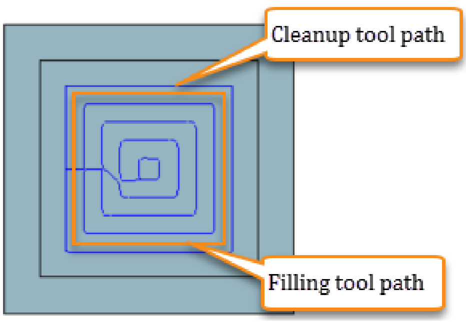
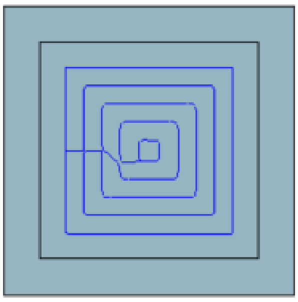
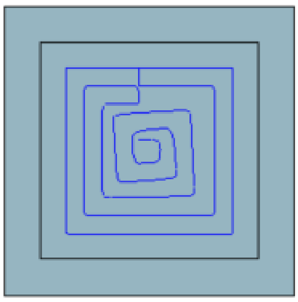
Spiral off	Min Curve Count = 4	Min Curve Count = 3
		

Figure46 Min curve count

b) Waves

Transfer a normal tool path pattern to a wave one.

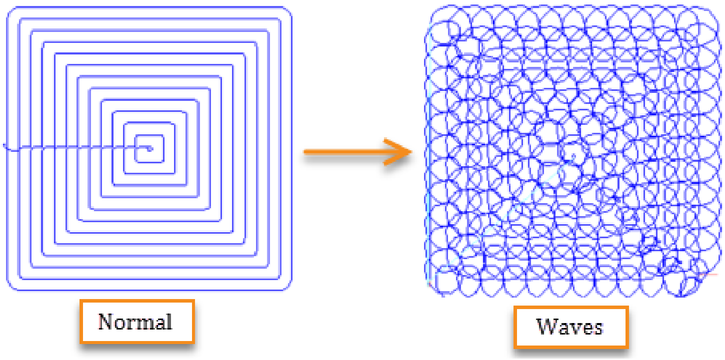


Figure47 Advanced setting -- Waves

c) Plunge

Transfer a normal tool path pattern to a plunge one. The result will be the same as using “Plunge” roughing operation

d) Pause/Break

When machining some hard material, to avoid the tool damage we need to lift the tool and break for a while, the machining operation is repeated in each specified time or distance.

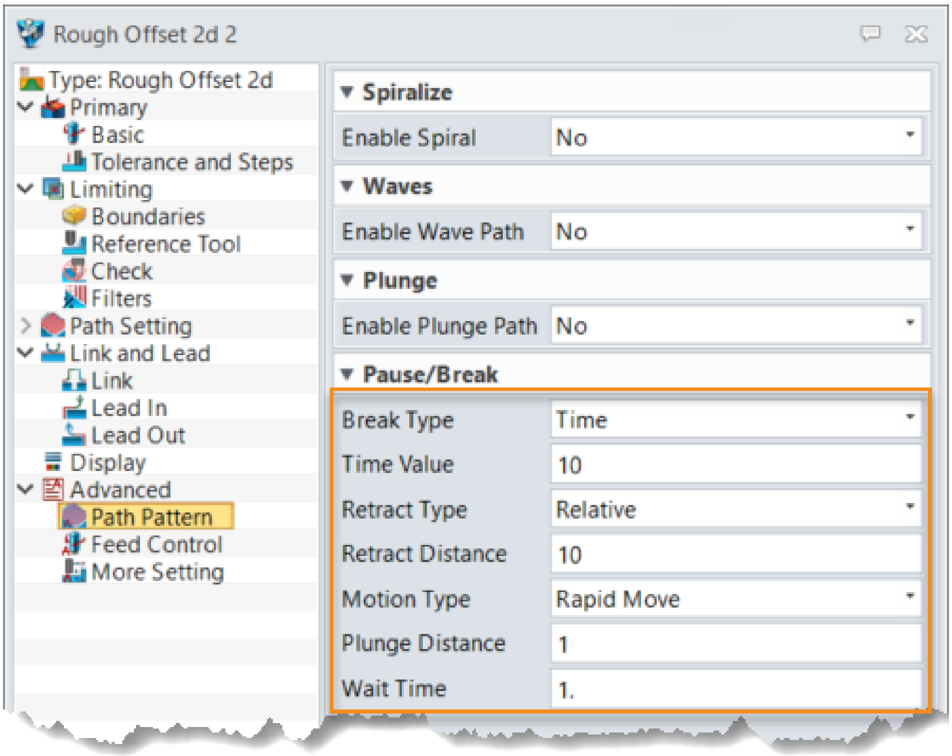


Figure48 Advanced setting – Pause/Break

- Feed Control

AFC (Advanced Feed Control) is an alternative method of calculating QM feeds. It will automatically change the feed rate according to the chip-load (remove volume of material), smooth the machining operation to get a high-quality and safer milling result, and increase the tool life.

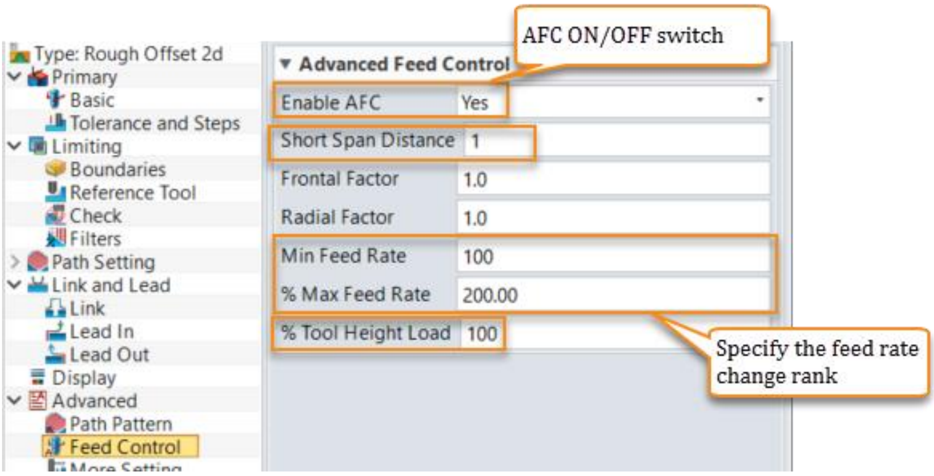


Figure49 Advanced setting -- AFC

- More setting

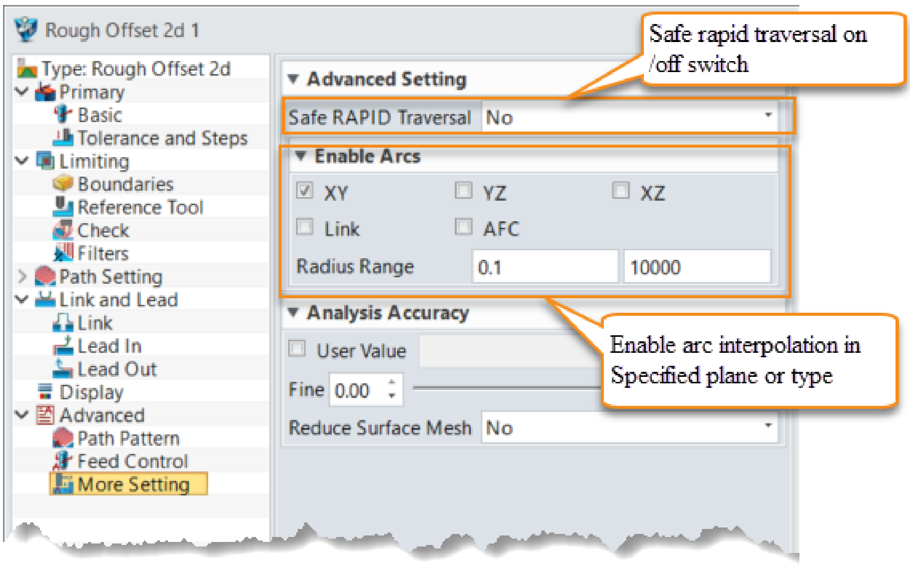


Figure1 Advanced setting – More setting

○ Safe RAPID

For some old type machine, they will divide a line traversal to two pieces. Which will move △X and △Y with the same feed rate, if X or Y didn't finish the movement, it will only move the X or Y. Thus, a traversal movement will divide as below:

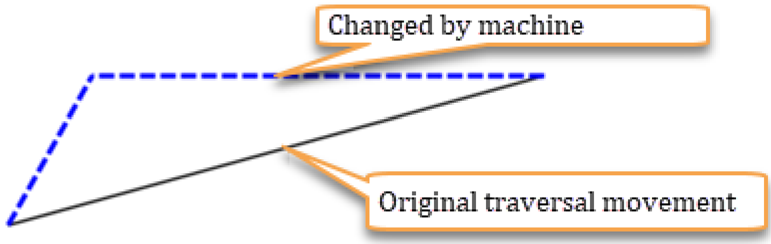


Figure50 Advanced setting – Safe rapid

If the calculated traversal movement is different from the real machine movement, it is very dangerous. So ZW3D provides a method to create the traversal which matches special machine type and automatically adjust the improper traversal movement.

1.2.3 Rest roughing operation

Rest roughing operation is used to uniform the rest material for the process of finishing operation after the roughing operation. ZW3D 3X milling provides 3 method to calculate the rest roughing tool path -- reference operation, reference tool, rest stock. Normally, we suggest users using reference operation and rest stock to create the rest roughing operation.

How to create a rest roughing operation?

Open the file “Rest roughing operation.Z3”.

- Create by reference operation

  STEP 01 Select an operation which you use to calculate the rest roughing tool path

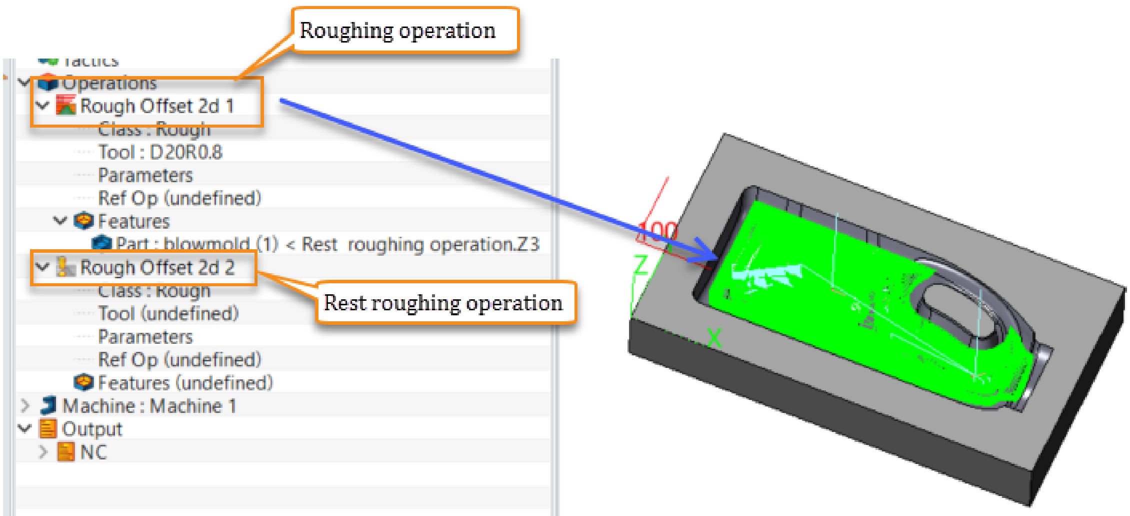


Figure51 Reference operation step1

STEP 02 Select a reference operation to detect the area of rest material.

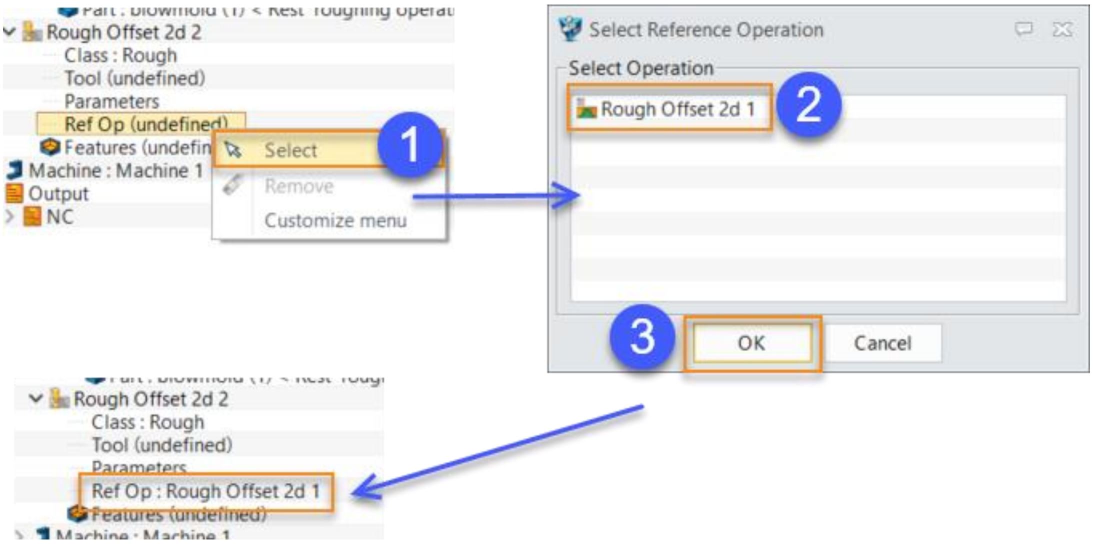


Figure52 Reference operation step2

STEP 03 Setting the machining parameters, such as tool, features, step down, step over, lead in/out and so on.

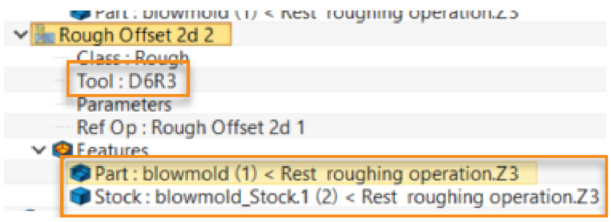


Figure53 Reference operation step3

STEP 04 Calculate the tool path

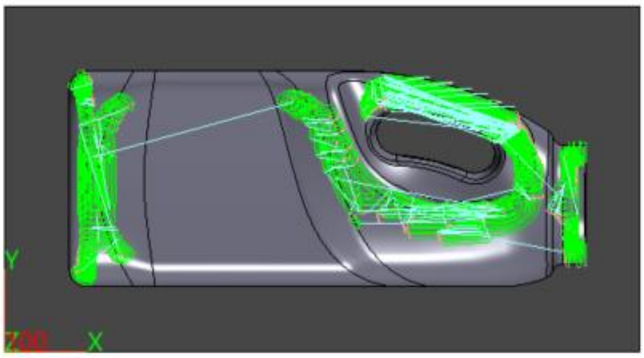


Figure54 Reference operation step4

- Create by rest stock

STEP 01 Solid verify the roughing operation

STEP 02 Create the rest stock

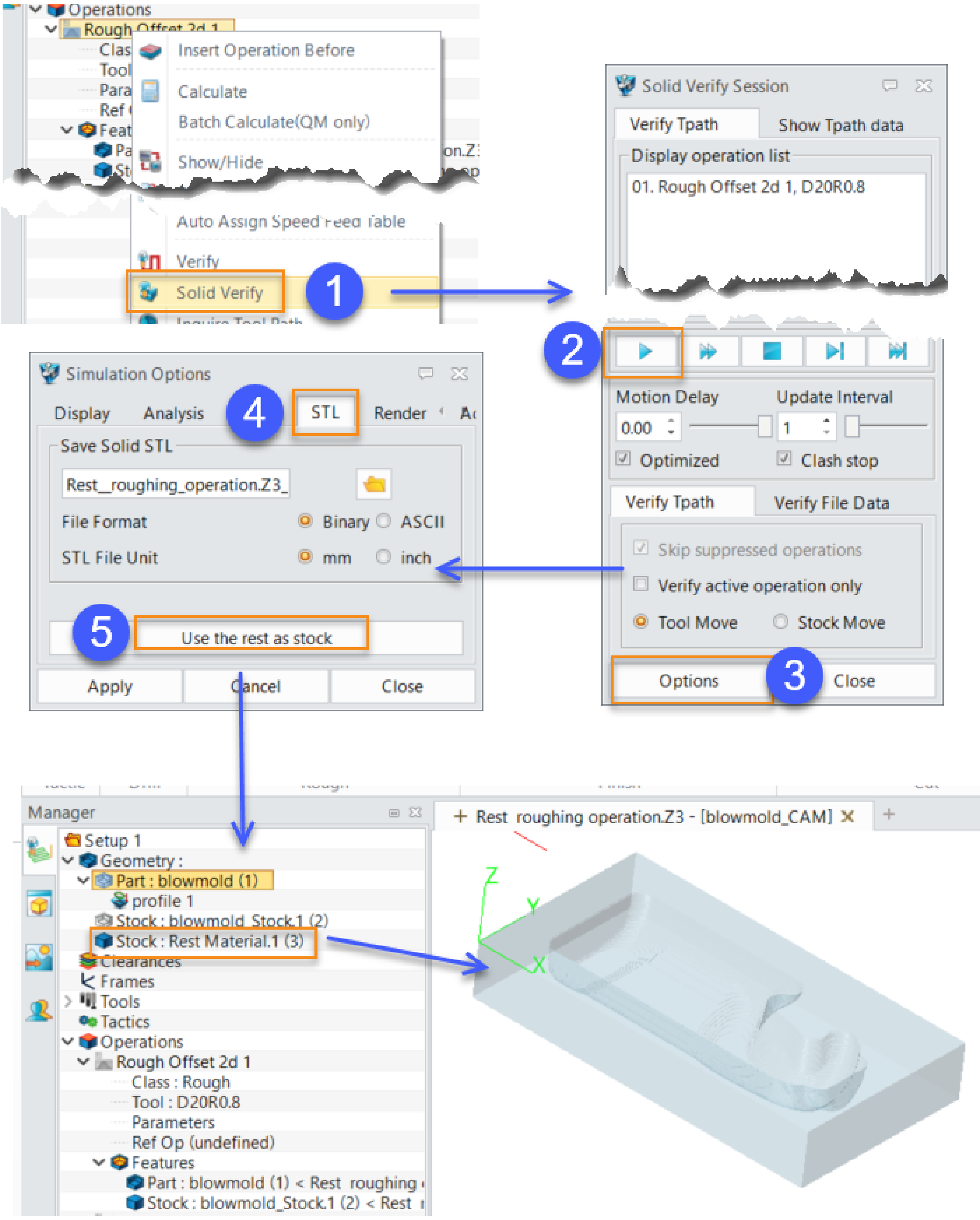


Figure55 Rest stock step2

STEP 03 Select an operation which you use to calculate the rest roughing tool path.

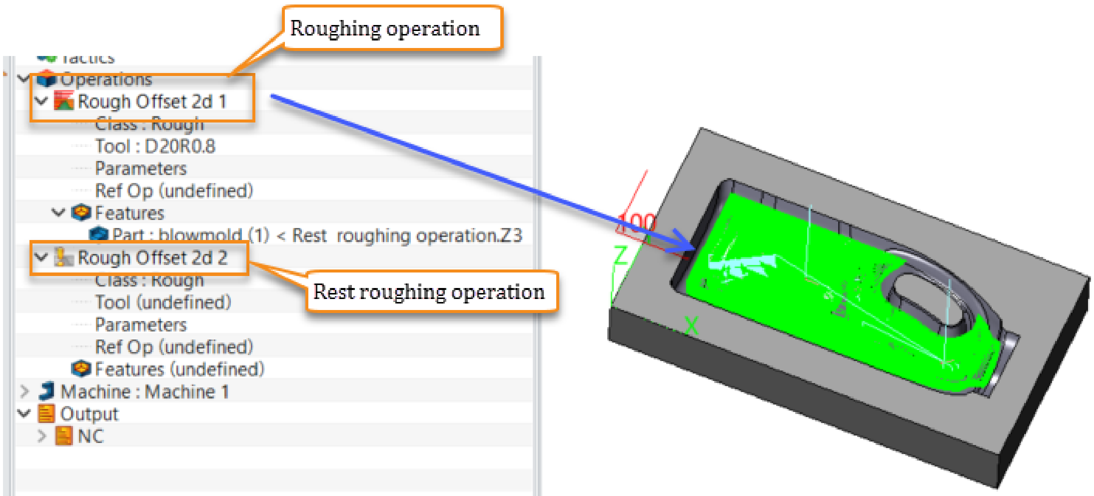


Figure56 Rest stock step3

STEP 04 Add the rest stock to the feature and setting associate parameters

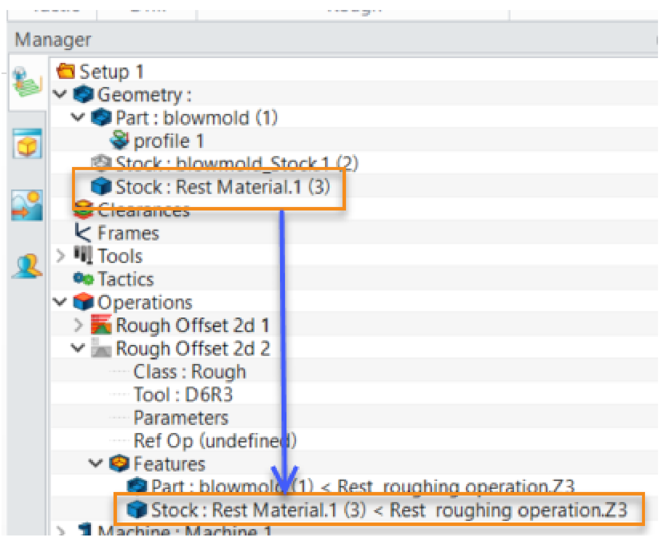


Figure57 Rest stock step4

STEP 05 Calculate the tool path

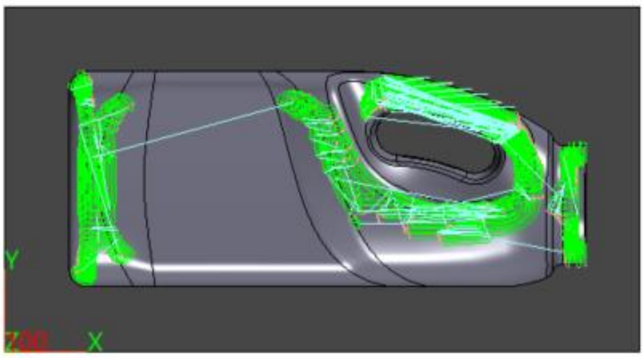


Figure58 Rest stock step5

1.3 Finishing

The machining purpose of finishing operation is very different from roughing operation. The roughing operation focuses on removing the unnecessary material as much as possible in a short time, thus the roughing operation will use a big step and tool. But finishing operation will focus on the dimension accuracy and surface quality, thus the finishing operation will use a high speed, small step and suitable tool to ensure a high-quality result.

Most of the parameters in finishing operation is the same as roughing operation, we won't explain it one more time, and will introduce the different one or the one that only exists in finishing operation. In addition, different types of finishing operations will have different tool path generation conditions, we will introduce the operations one by one.

1.3.1 Lace operation

Lace operation will generate a group of uniform parallel tool path with equal distance. The lace operation tool path generation theory is that ZW3D will create a group of parallel tool path in XY plane first, and then project it to 3D model. Because of the tool path generation theory, the tool path in steep area will become nonuniform. Thus, this kind of tool path pattern will be suitable for a smooth and gently change machine area but not steep area. Refer to the case as below

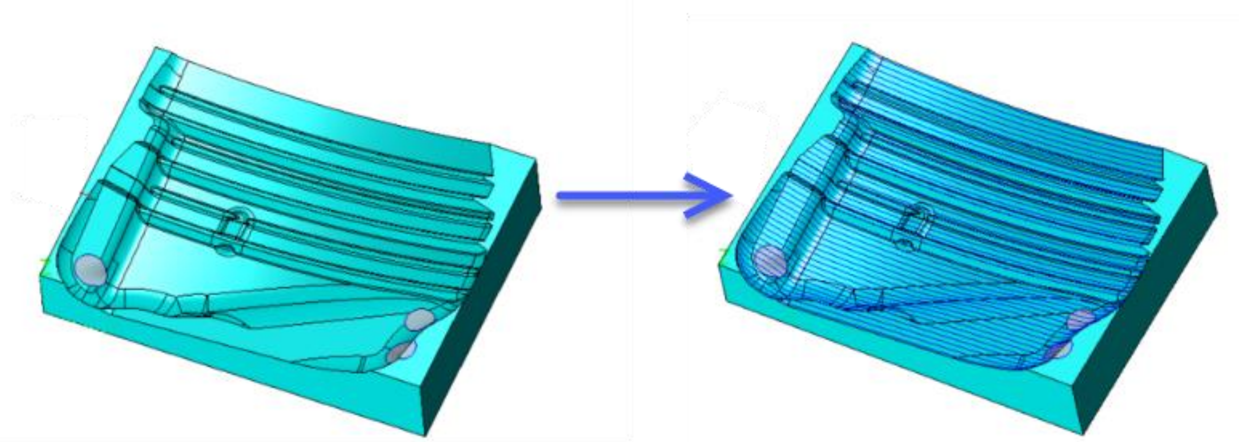


Figure59 Lace operation

- Path pattern

- Tool path generation requirement

- Specify tool
- Specify feature (part or part with profile boundary)

- Associate parameters setting introduce
  - Primary

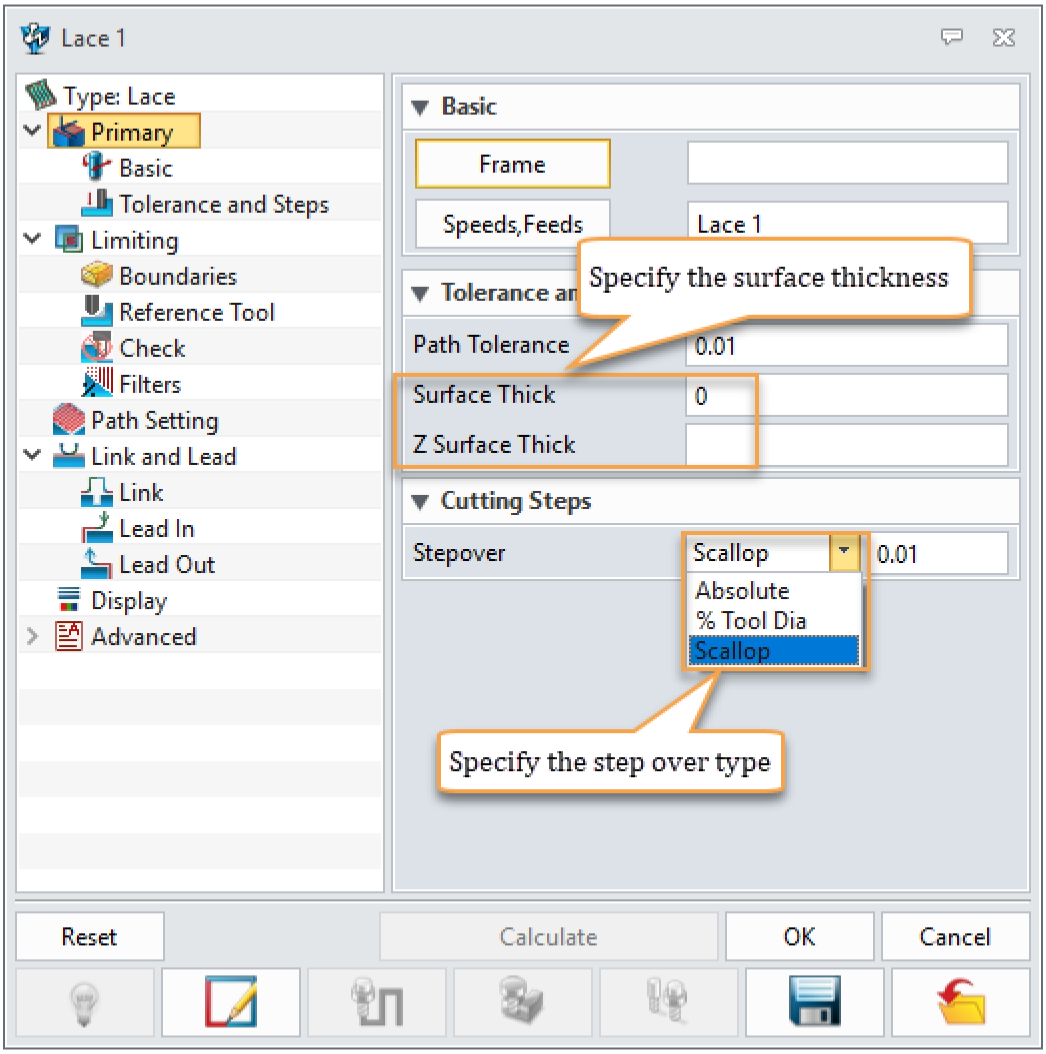


Figure60 Lace operation -- primary

  - Limiting

The containment type definition in finishing operation is a little bit different from roughing operation, and there is a new containment type named "cut contact" in finishing operation

a) Simple box: all the faces inside the boundary box will join into the tool path calculation

b) Silhouette: the silhouette will be the same as the boundary to calculate the tool path.

c) Cut contact: system will not use the boundary to trim the tool path directly but will calculate the point on the boundary where the tool contact to the part. Ensure get a completed tool path to machine the area.

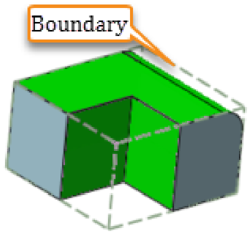
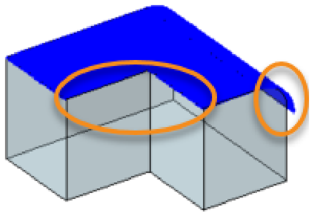
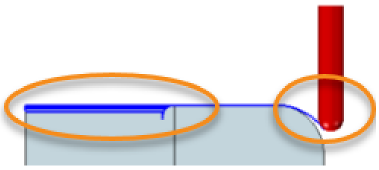
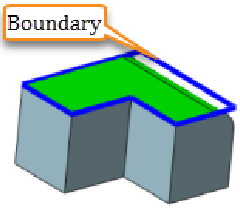
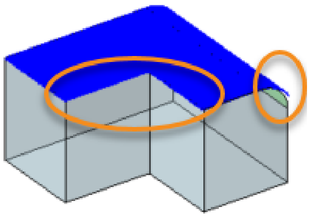
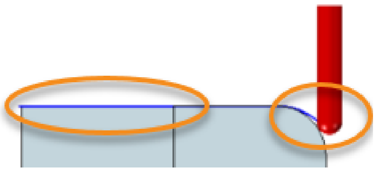
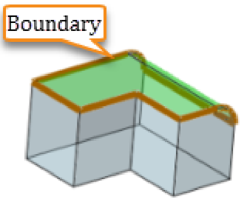
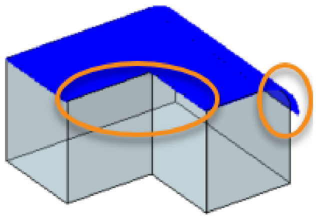
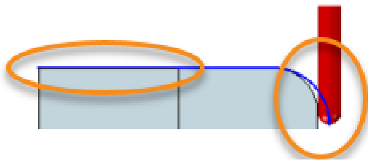
type	Boundary 1*	Tool path	
Simple box			
Silhouette			
Cut contact			

Figure61 Boundaries

Note: 1* The green faces will be joined into the tool path calculation.

○ Filter

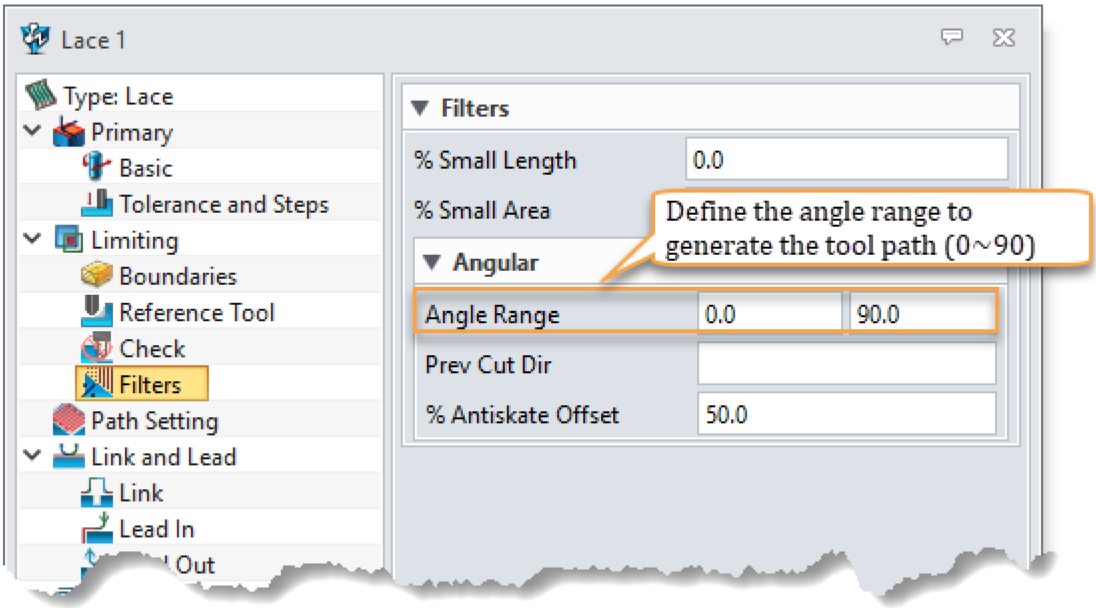


Figure62 Filter

○ Path setting

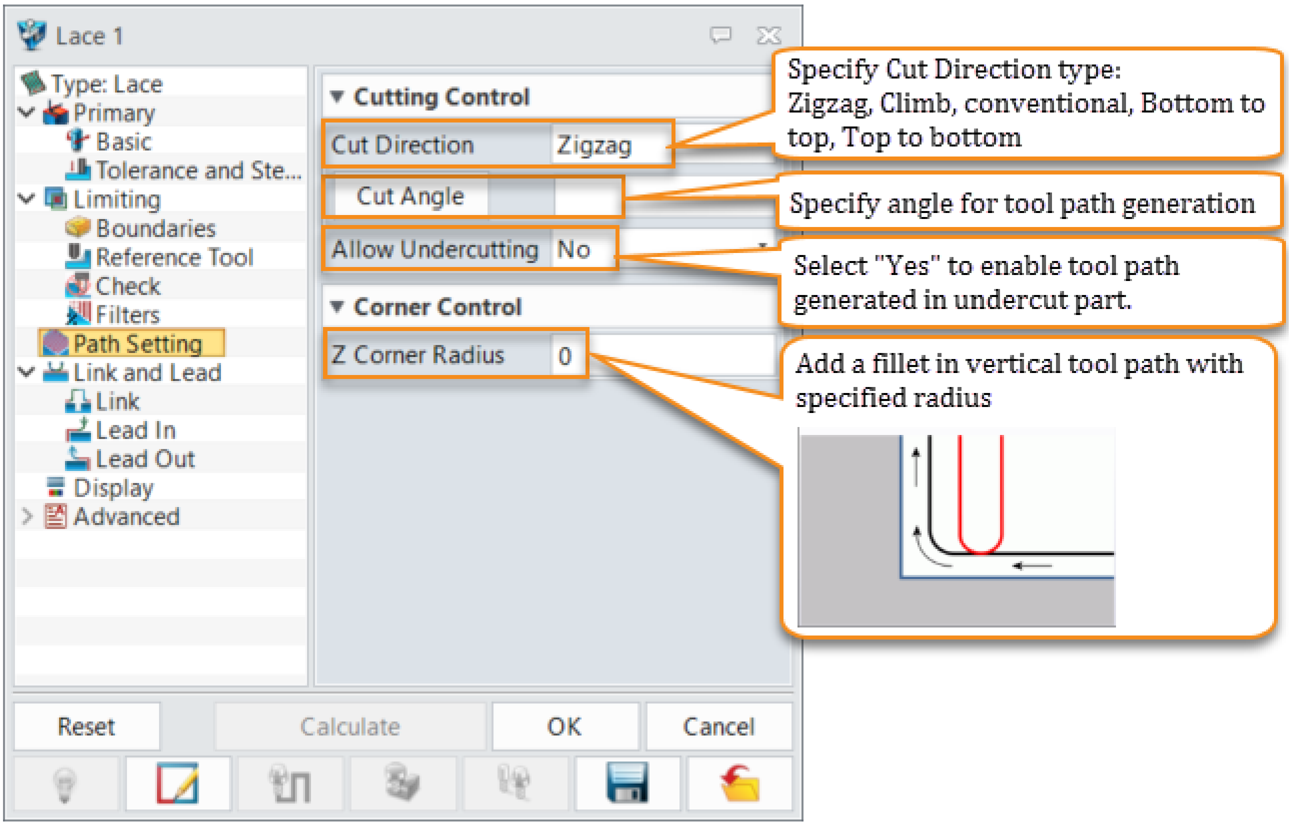


Figure63 Path setting

a) Cut direction

ZW3D provides several cut direction types for users to select.

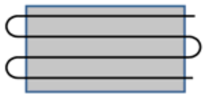
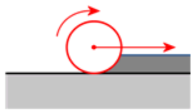
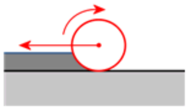
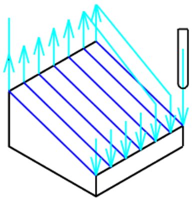
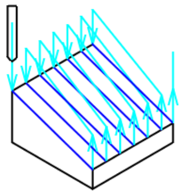
Type	Zigzag	Climb	Conventional	Bottom to top	Top to Bottom
Pattern					

Figure64 Cut direction

b) Cut angle

The cut angle defined by the angle between the horizontal directions and the counter clock wise. After setting an angle with horizontal, system will automatically change the start point in order to generate a continue tool path.

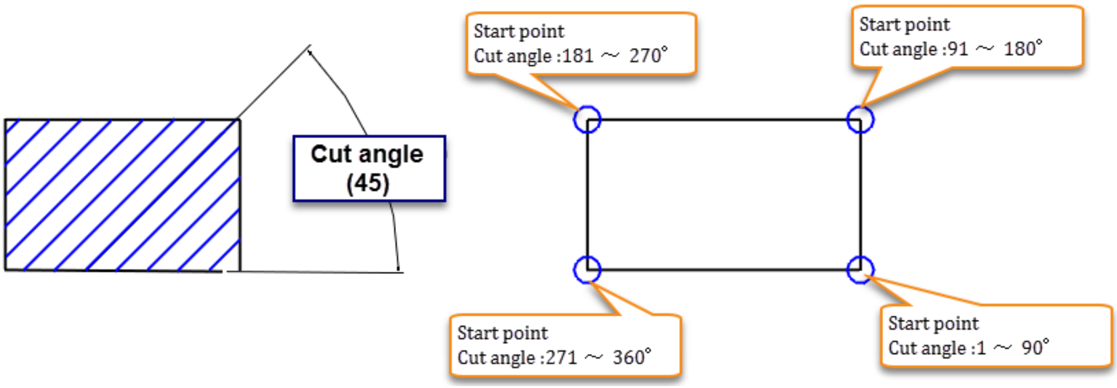


Figure65 Cut angle

c) Allow undercutting

Normally, ZW3D will not calculate the tool path where is an undercut. Only choose “allow undercutting” option, and select a proper tool which can cut the material in undercut places, ZW3D will generate the tool paths.

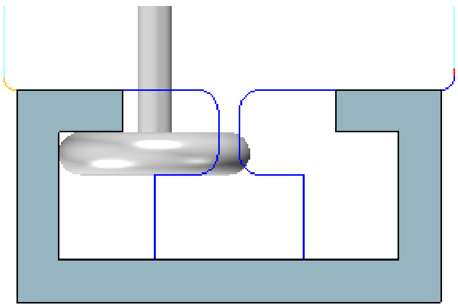


Figure66 Under cut

- Advanced > Uniform cut

Enable the uniform cut, ZW3D will detect the distance between tool paths, if the distance is bigger than the specified step over, it will create a 3D offset pattern tool path to uniform the step over.

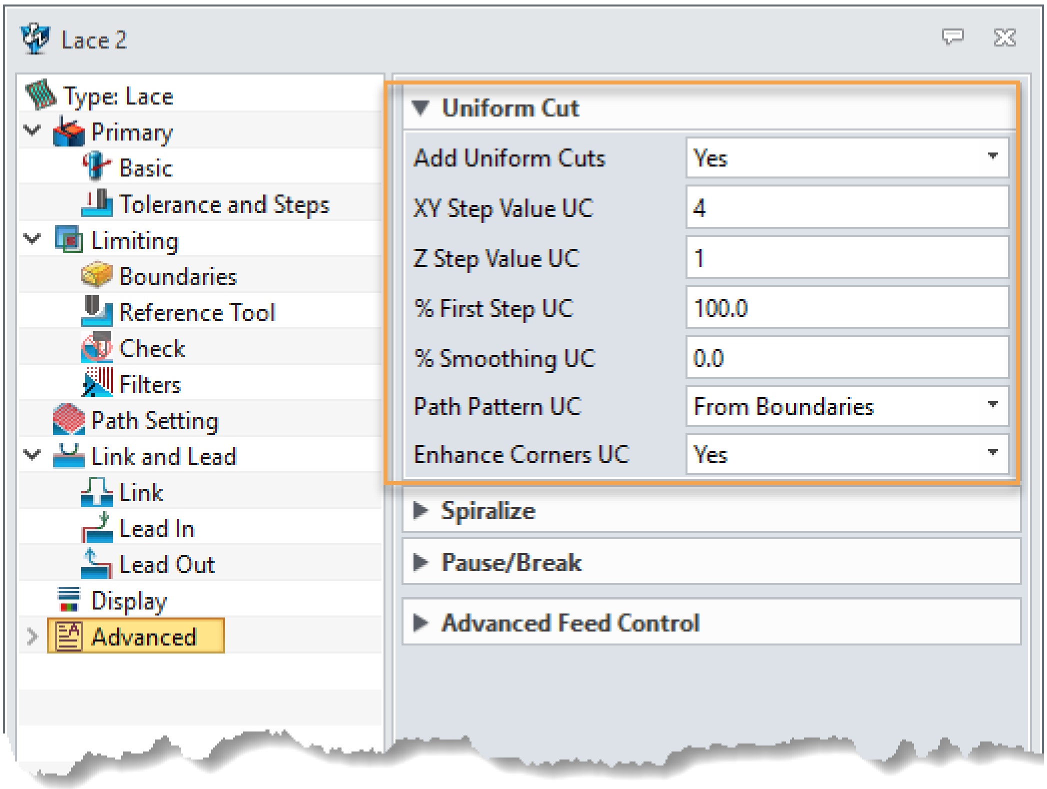


Figure67 Uniform cut 1

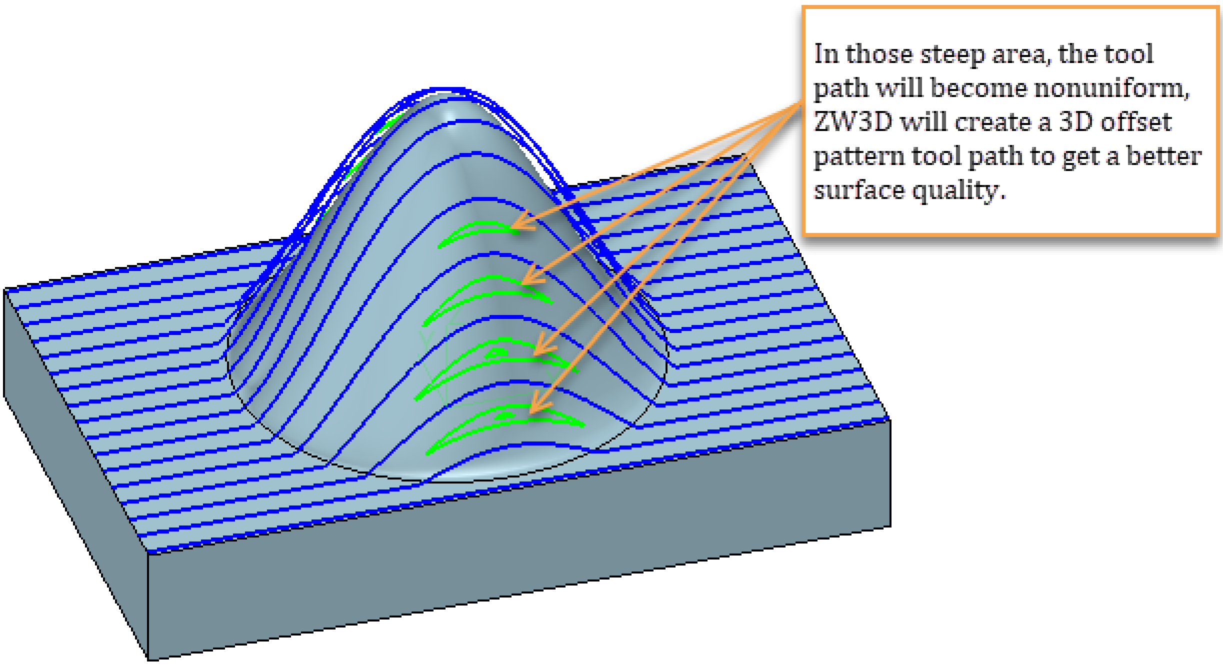


Figure68 Uniform cut 2

- Example

STEP 01 Open "Lace.Z3" file and go into CAM space

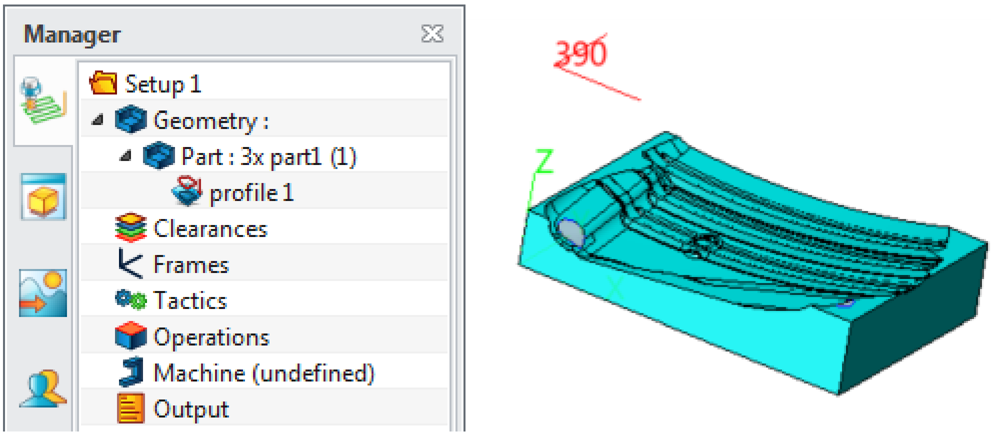


Figure69 Lace operation example Step1

STEP 02 Select Lace finishing operation

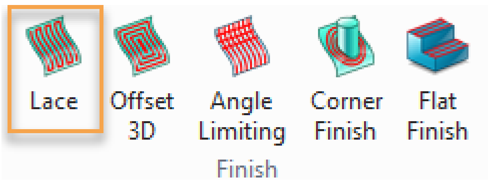


Figure70 Lace operation example Step2

STEP 03 Select Part and profile1 to the feature

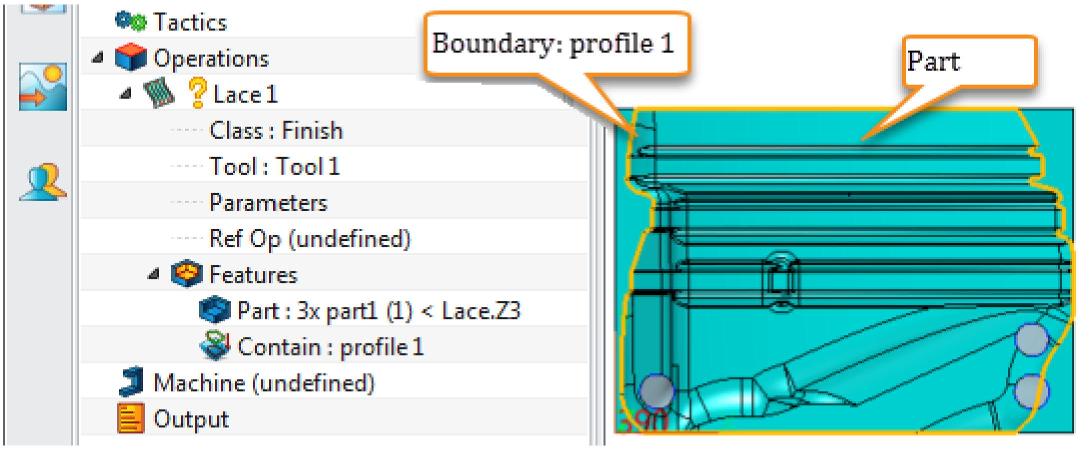


Figure71 Lace operation example Step3

STEP 04 Specify a D12R6 tool and use default setting, then calculate the tool path

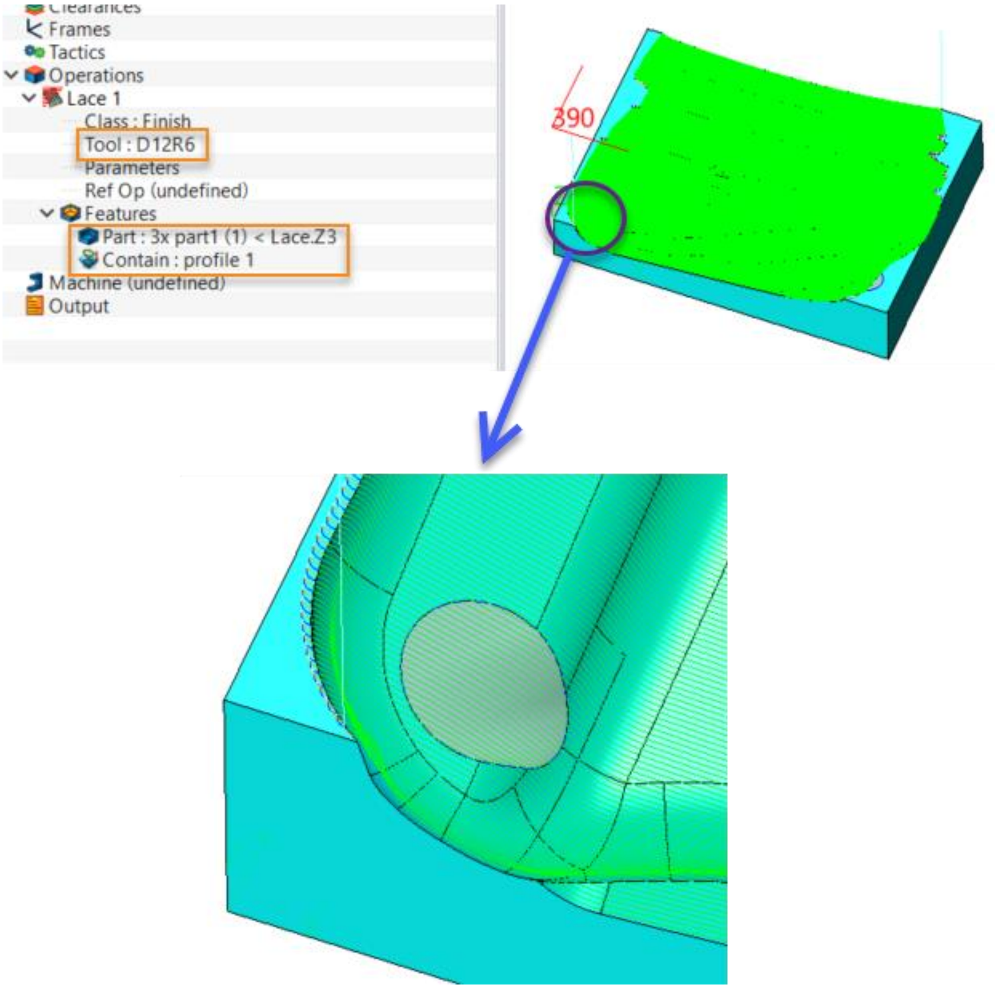


Figure72 Lace operation example Step4

1.3.2 Offset 3D operation

Offset 3D will generate a 3D equal step tool path all over the part following the silhouette or 3D boundary profile. If the boundary profile is not specified, system will use the part silhouette as base to offset and generate the whole tool path. Otherwise it will follow the 3D boundary profile to offset the tool path.

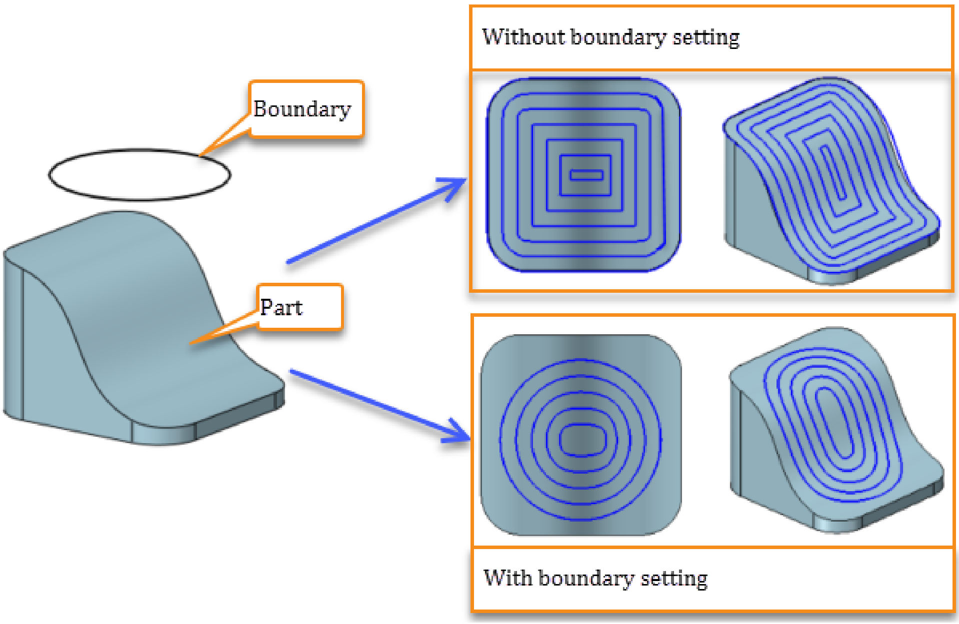
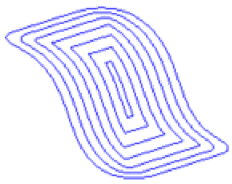


Figure73 Offset 3D operation

- Path pattern



- Tool path generation requirement
  - Specify tool
  - Specify feature (part or part with profile boundary)
- Associate parameters setting
  - Primary

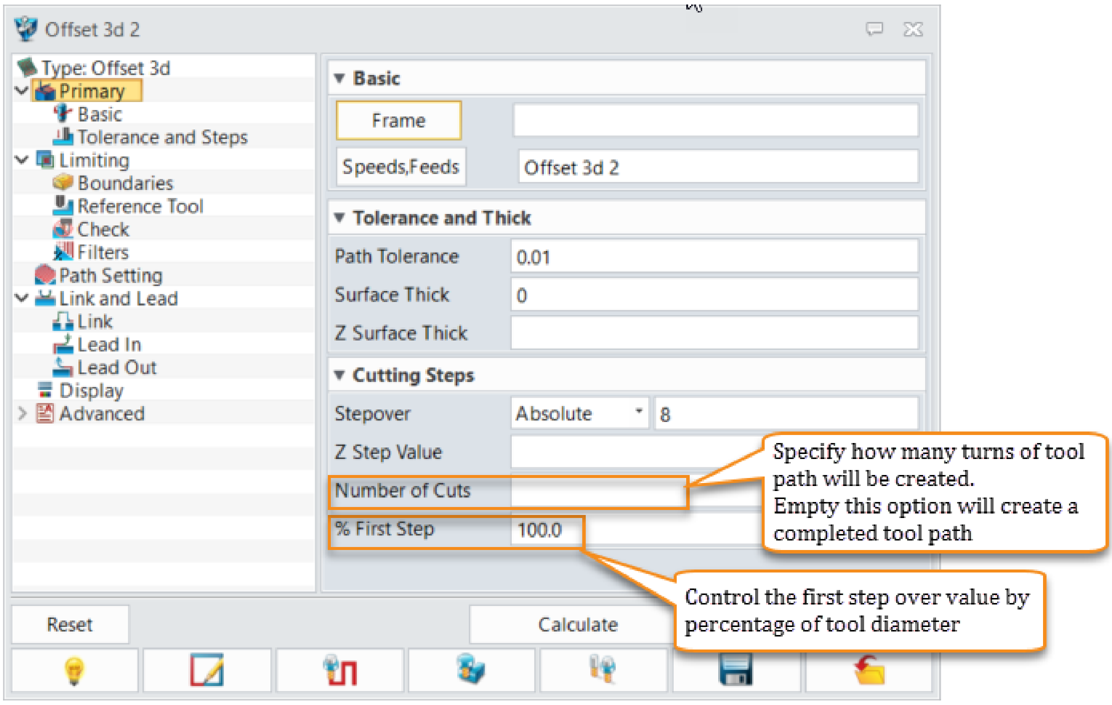


Figure74 Offset 3D operation -- Primary

○ Path pattern

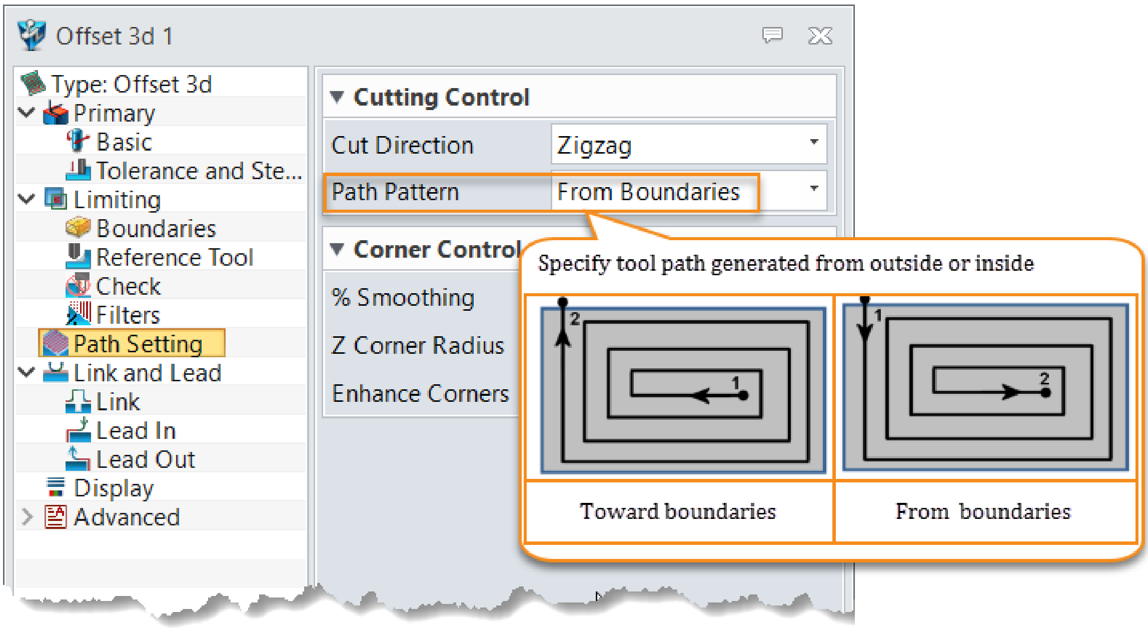


Figure75 Offset 3D operation – Path setting

- Example

STEP 01 Open “CAM_TM_Model.Z3” file and go into CAM space

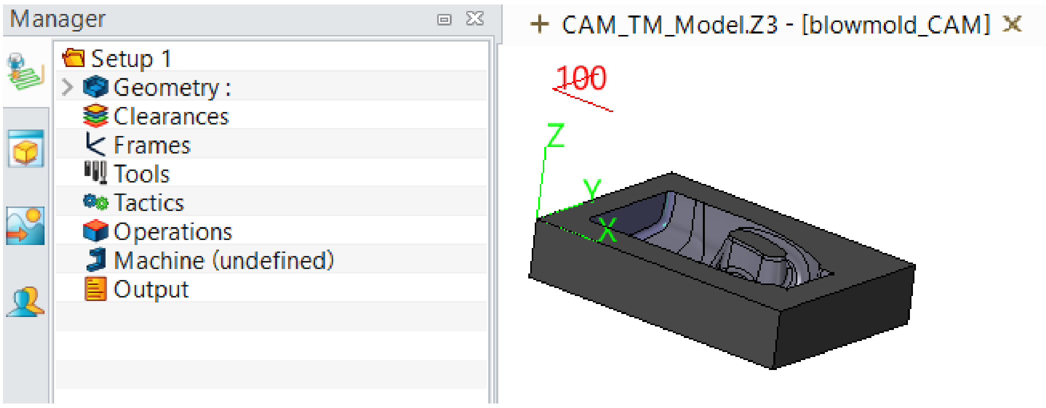


Figure76 Offset 3D operation example Step1

STEP 02 Select offset 3D operation

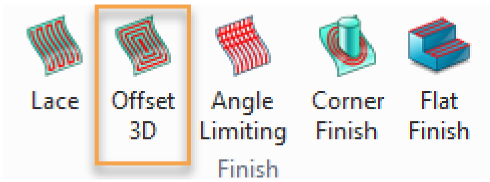


Figure77 Offset 3D operation example Step2

STEP 03 Add part to the feature and specify a profile boundary feature.

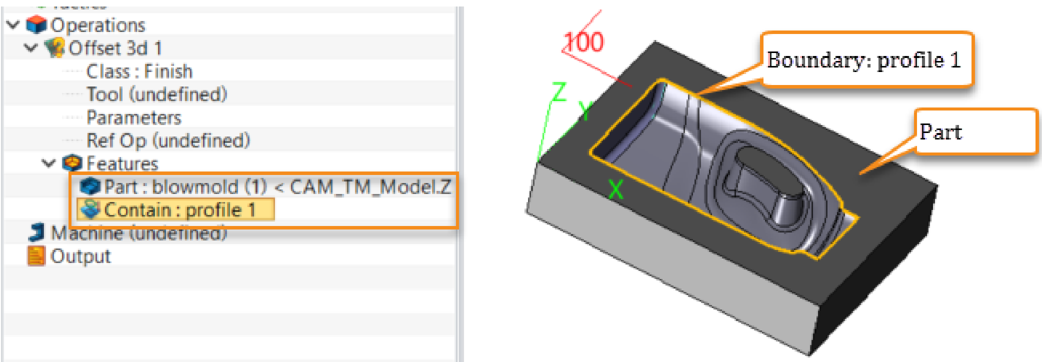


Figure78 Offset 3D operation example Step3

STEP 04 Specify a D8R4 tool for this operation and set the step over to 5(mm)

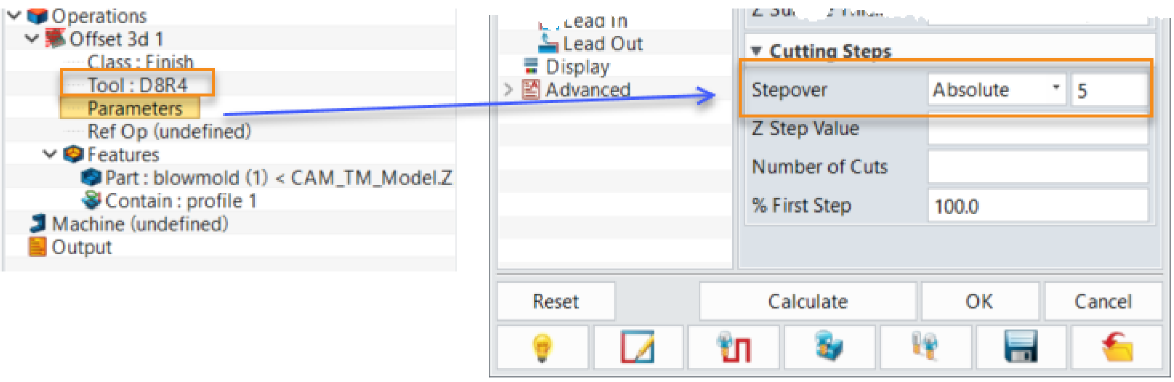


Figure79 Offset 3D operation example Step4

STEP 05 Calculate the tool path

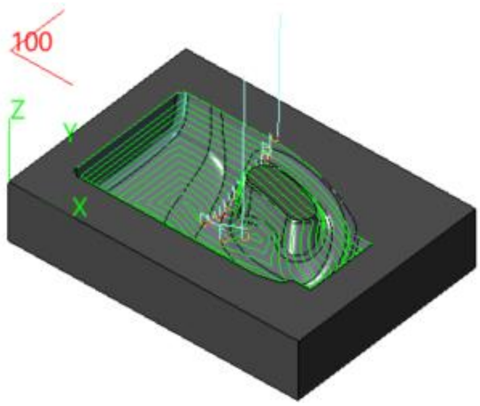
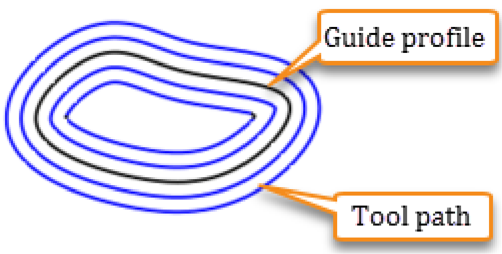


Figure80 Offset 3D operation example Step5

1.3.3 Drive curve operation

Drive curve operation will generate the tool path by offsetting a specified guide profile in both sides, and the number of turns can be defined by users. The profile can be opened or closed. Users can specify more than one guide profile to generate the tool path.

- Path pattern



- Tool path generation requirement
  - Specify tool
  - Specify feature (part or part with profile boundary)
  - Specify drive curve (☆ the profile type should be part)

  Tips: The drive curves can be closed or opened
- Parameter setting
  - Primary

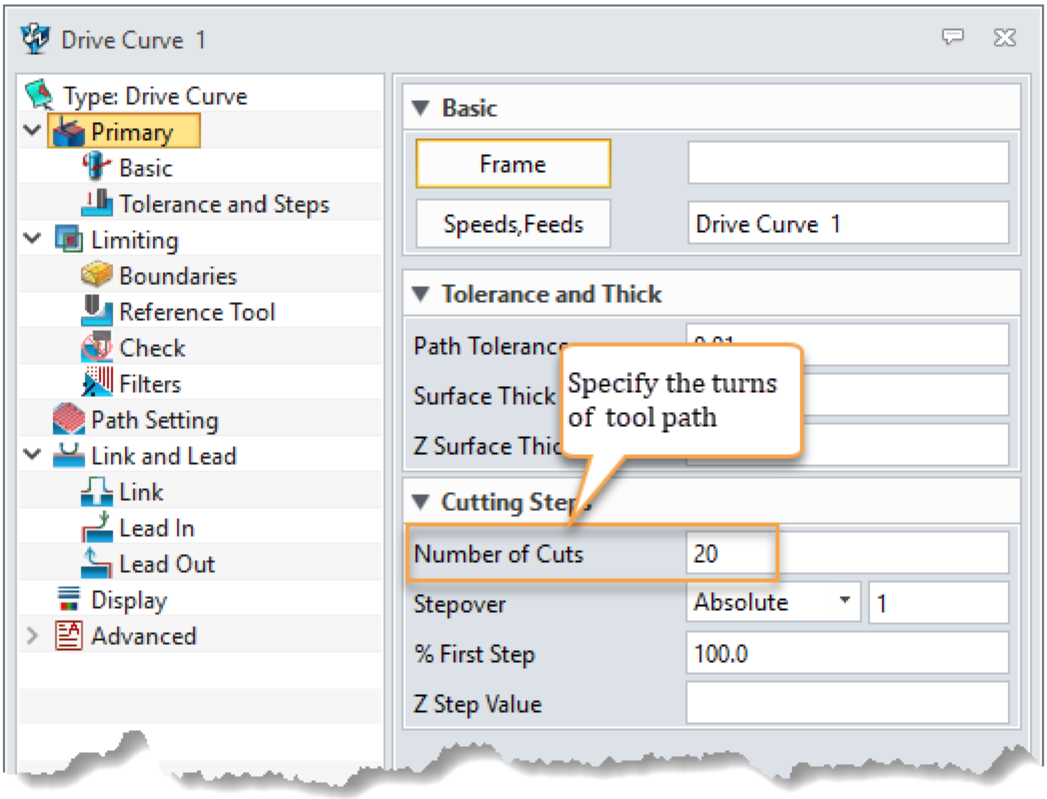


Figure81 Drive curve operation -- Primary

Number of cut option can specify how many turns of tool path will be generated by offsetting from the guide profile. When option is empty, ZW3D will generate a tool path along the guide profile. Otherwise it will offset guide profile in both sides to generate the tool path.

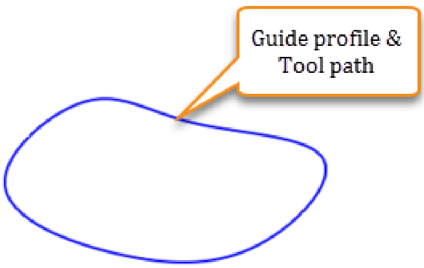
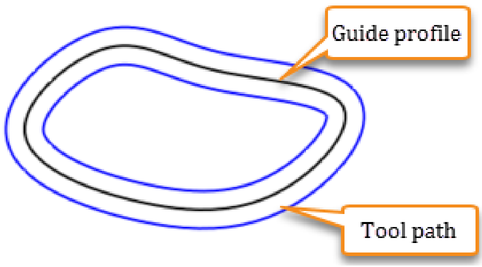
Number of cuts	0	1
Path pattern		

Figure82 Drive curve operation – Number of cuts

○ Path setting

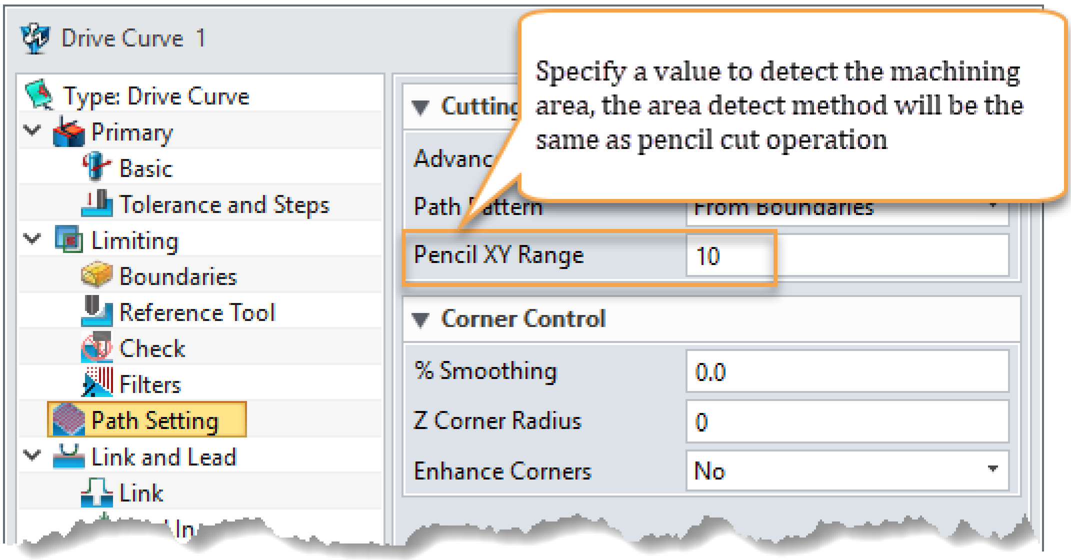


Figure83 Drive curve operation – Pencil XY Range 1

In some cases, the drive curve will be very close to the part which is impossible to create the tool path along the drive curve. Set a proper value in "Pencil XY Range", then it will detect a machining area and create a tool path guide by the drive curve.

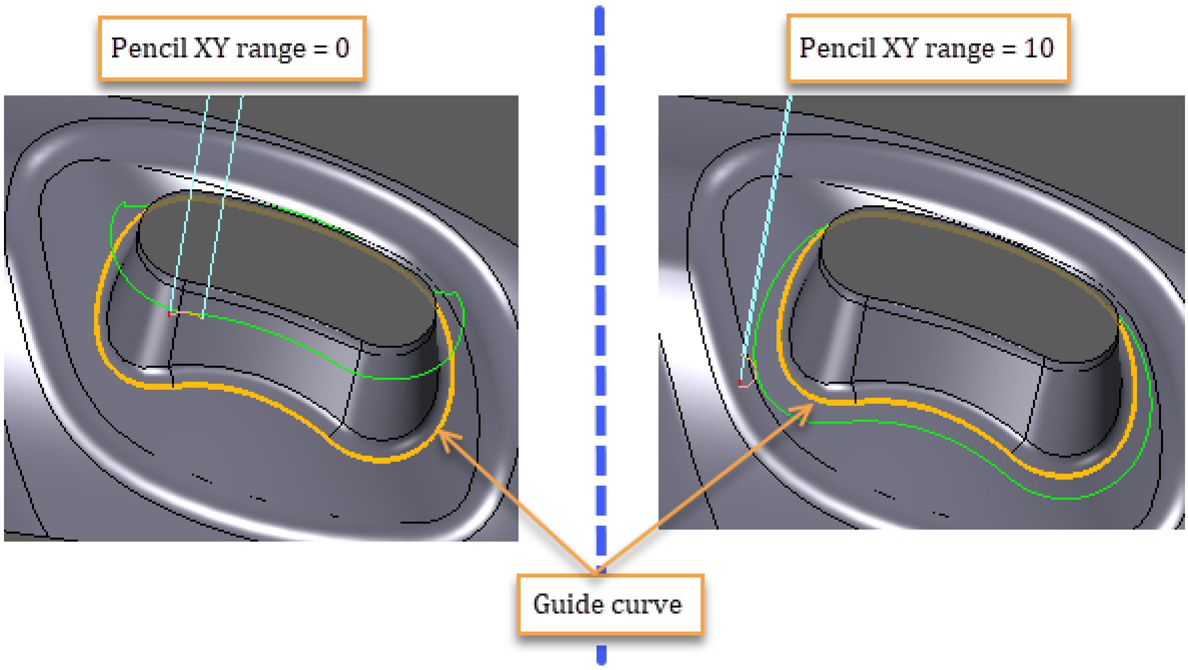


Figure84 Drive curve operation – Pencil XY Range 2

- Example

STEP 01 Open "Drive curve .Z3" file and go into CAM space

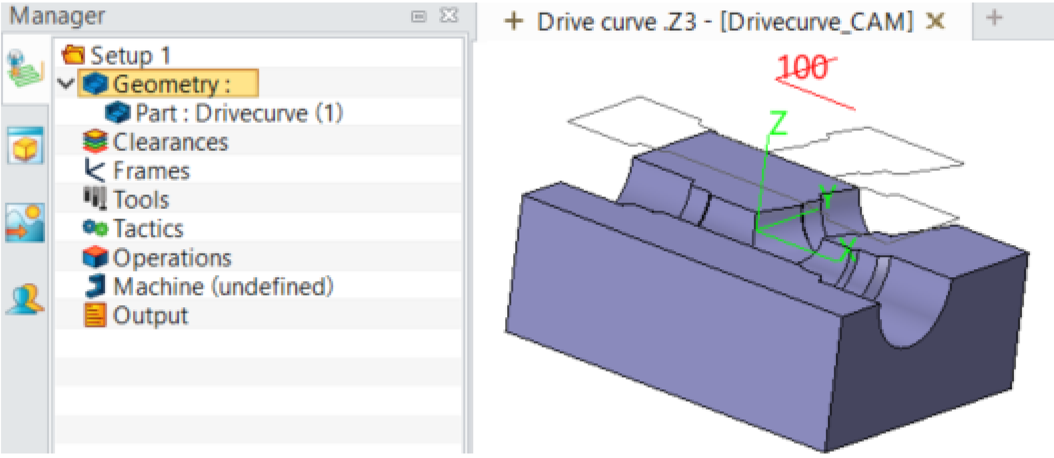


Figure85 Drive curve operation example Step1

STEP 02 Select Drive curve operation

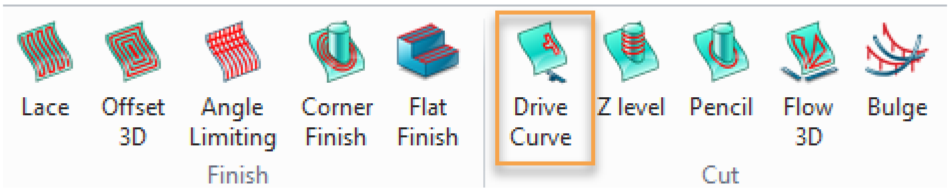


Figure86 Drive curve operation example Step2

STEP 03 Add the part to the feature

STEP 04 Define drive curve

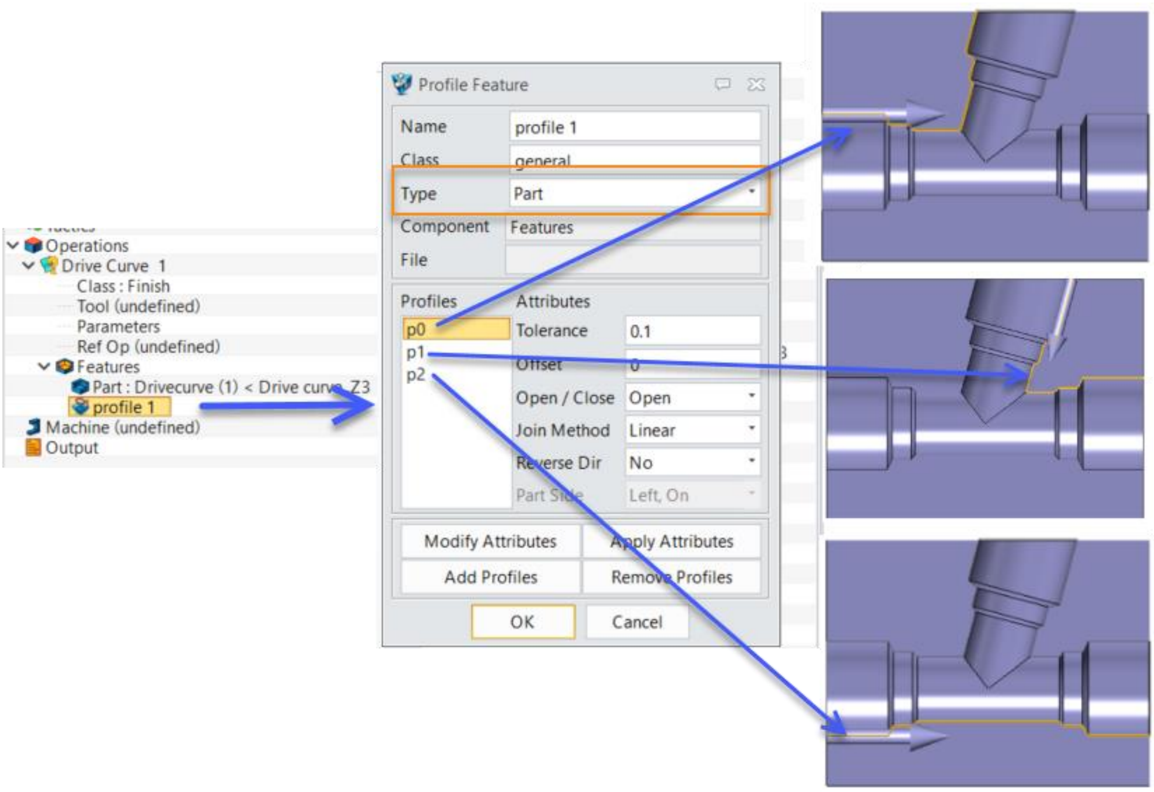


Figure87 Drive curve operation example Step4

STEP 05 Define boundary feature

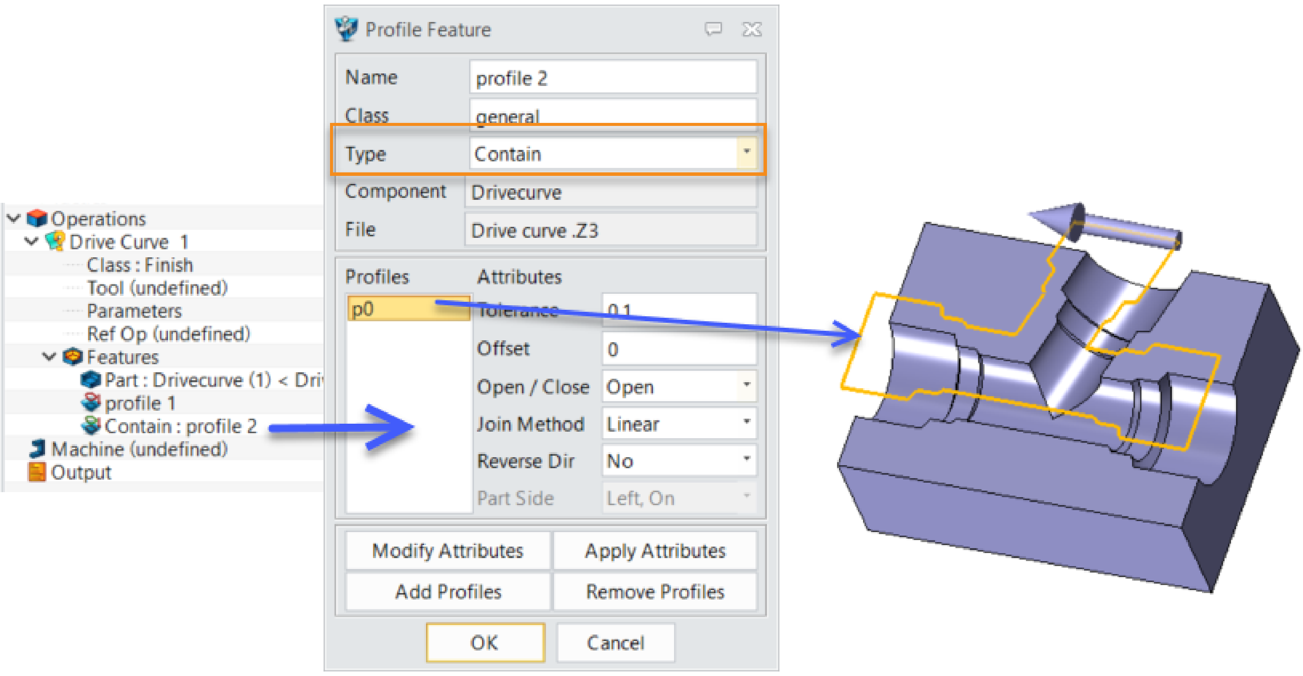


Figure88 Drive curve operation example Step5

STEP 06 Specify a D18R9 tool for this operation and set the associate parameters as below:

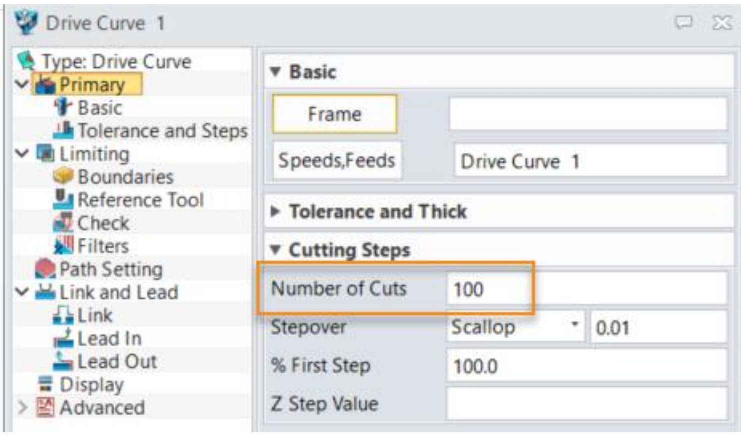


Figure89 Drive curve operation example Step6-1

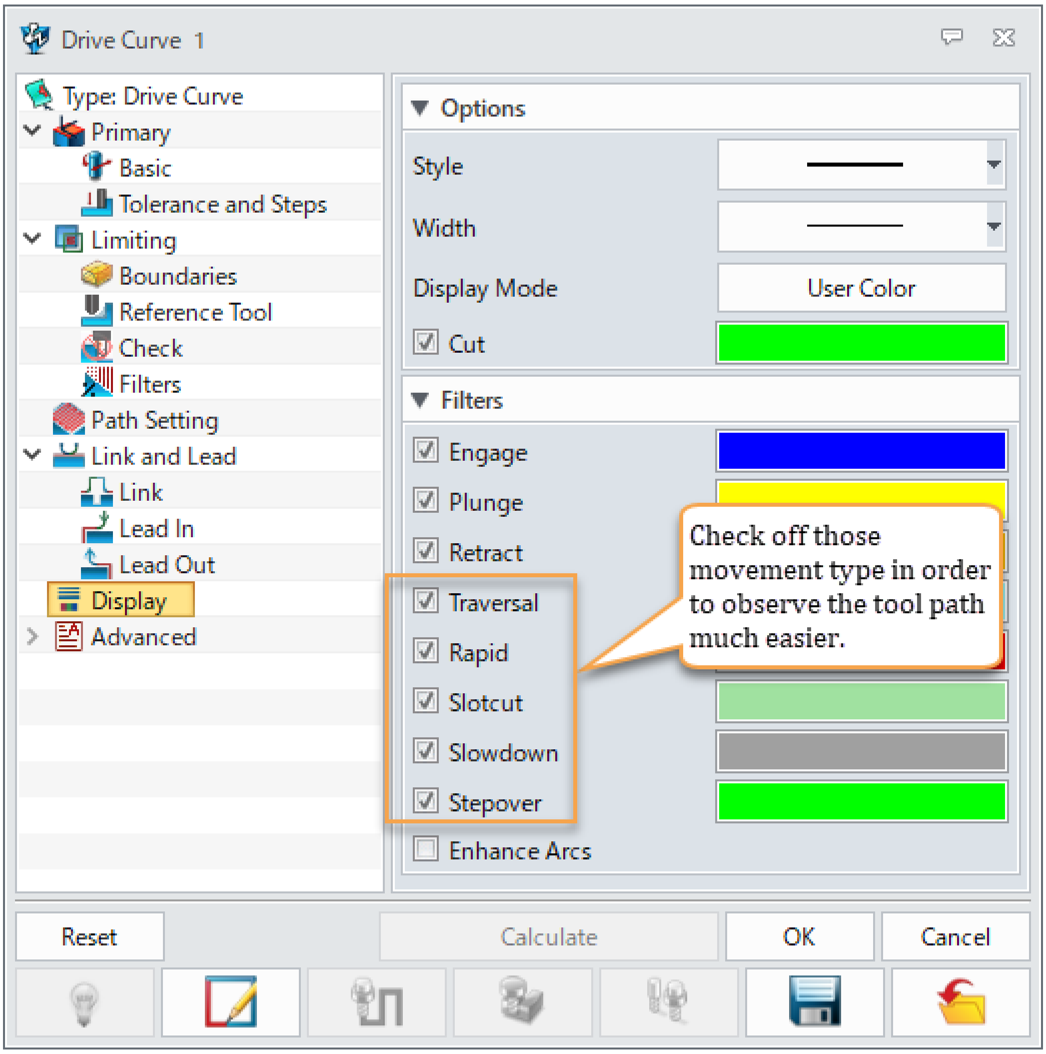


Figure90 Drive curve operation example Step6-2

STEP 07 Calculate the tool path

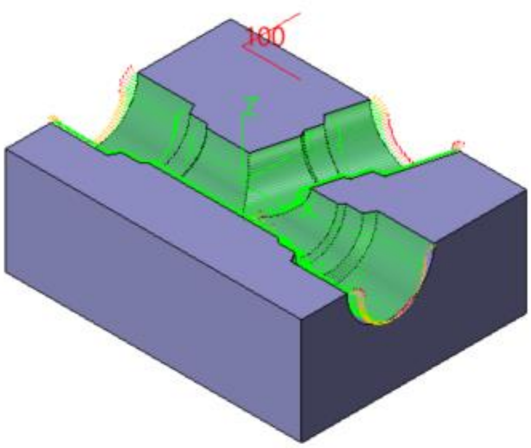


Figure91 Drive curve operation example Step7

1.3.4 Flow 3D operation

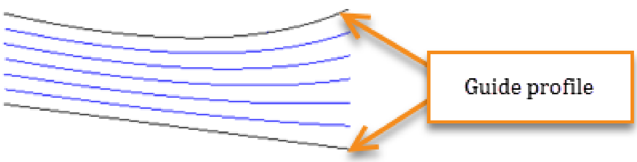
Flow 3D operation will morph a pair of guide profile to generate a set of tool path with 3D equal distance to fill with the area between the guide profiles. The guide profile can be opened or closed.

Note:

1. Please pay attention to the direction of the guide profile should be the same, otherwise it will create a twist tool path.

2. Two guide profiles have to be separated in two profile features.

- Path pattern



- Tool path generation requirement
  - Specify tool
  - Specify feature (part or part with profile boundary)
  - Specify a pair of guide curves (☆ the profile type should be part)
- Example

STEP 01 Open "Flow_3D.Z3" file and go into CAM space.

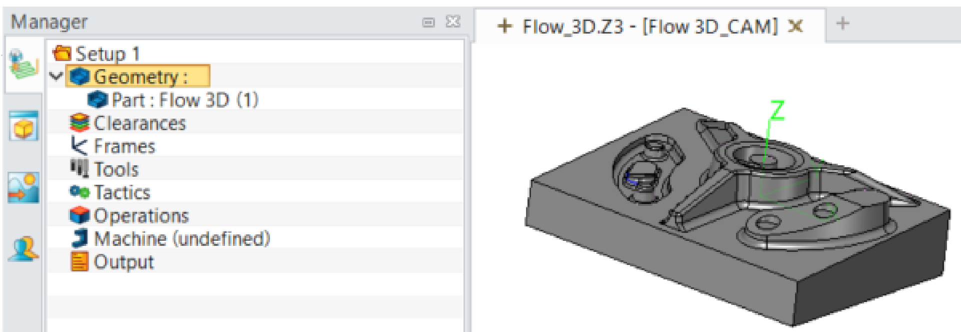


Figure92 Flow 3D operation example Step1

STEP 02 Select flow 3D operation

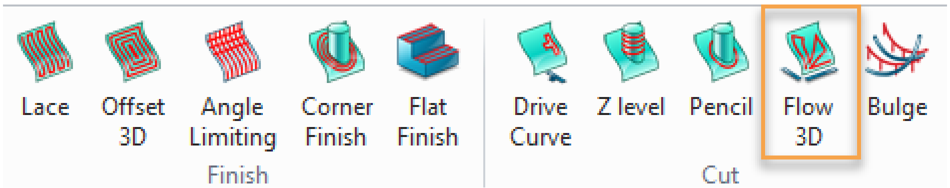


Figure93 Flow 3D operation example Step2

STEP 03 Add the part to feature

STEP 04 Define guide curves for flow 3D

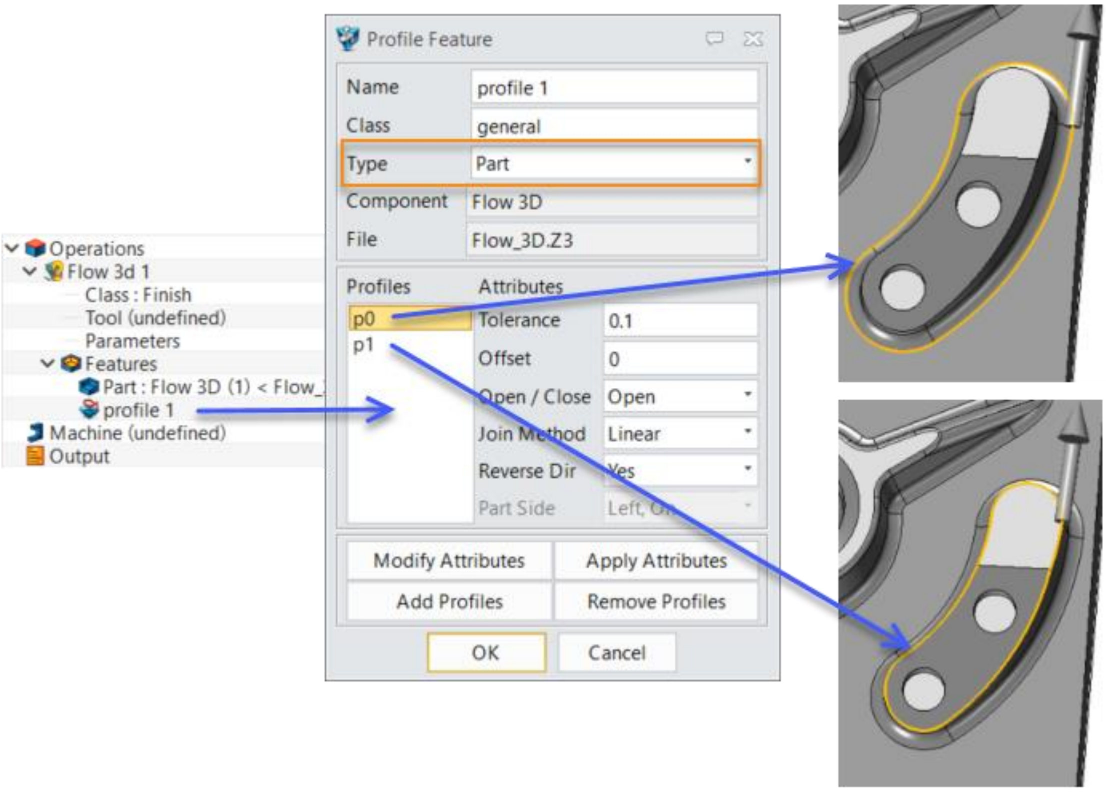


Figure94 Flow 3D operation example Step4

STEP 05 Specify a 2mm ball end tool

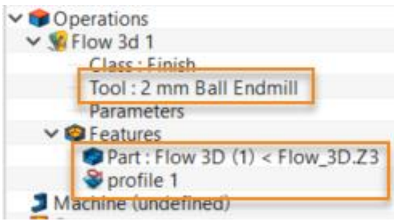


Figure95 Flow 3D operation example Step5

STEP 06 Calculate the tool path.

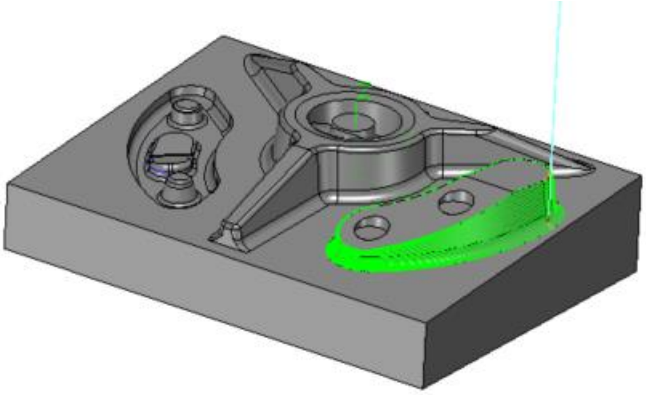
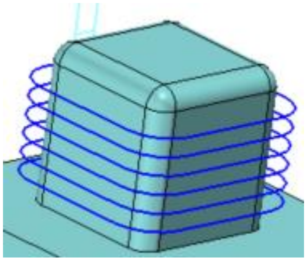


Figure96 Flow 3D operation example Step6

1.3.5 Z level operation

Z level operation will generate a set of contour tool path in Z axis direction which are used to machine steep wall area.

- Path pattern



- Tool path generation requirement
  - Specify tool
  - Specify feature (part or part with profile boundary)
- Associate parameters setting introduce
  - Non-uniform cuts

    a) This option is by default set "NO" which means that ZW3D will use uniform step down to create the tool path.

    b) For some parts with different angle steep wall, we need to use "non-uniform cuts" to get a better surface quality as shown below:

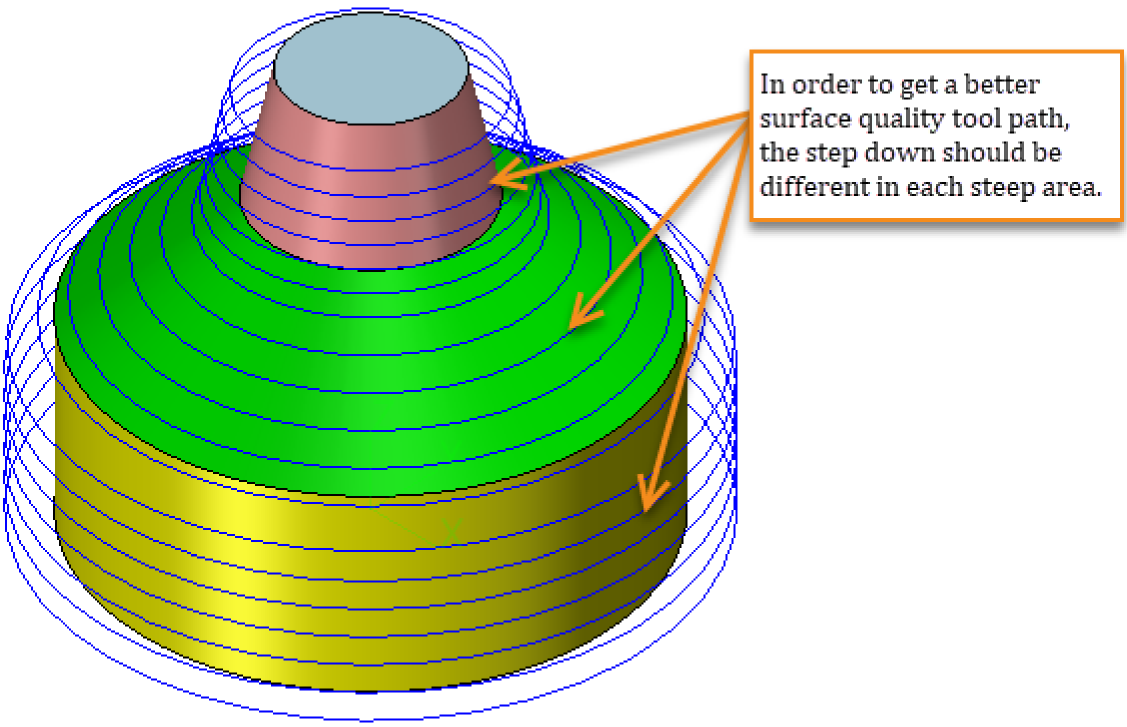


Figure97 Z level operation – Non-uniform cut 1

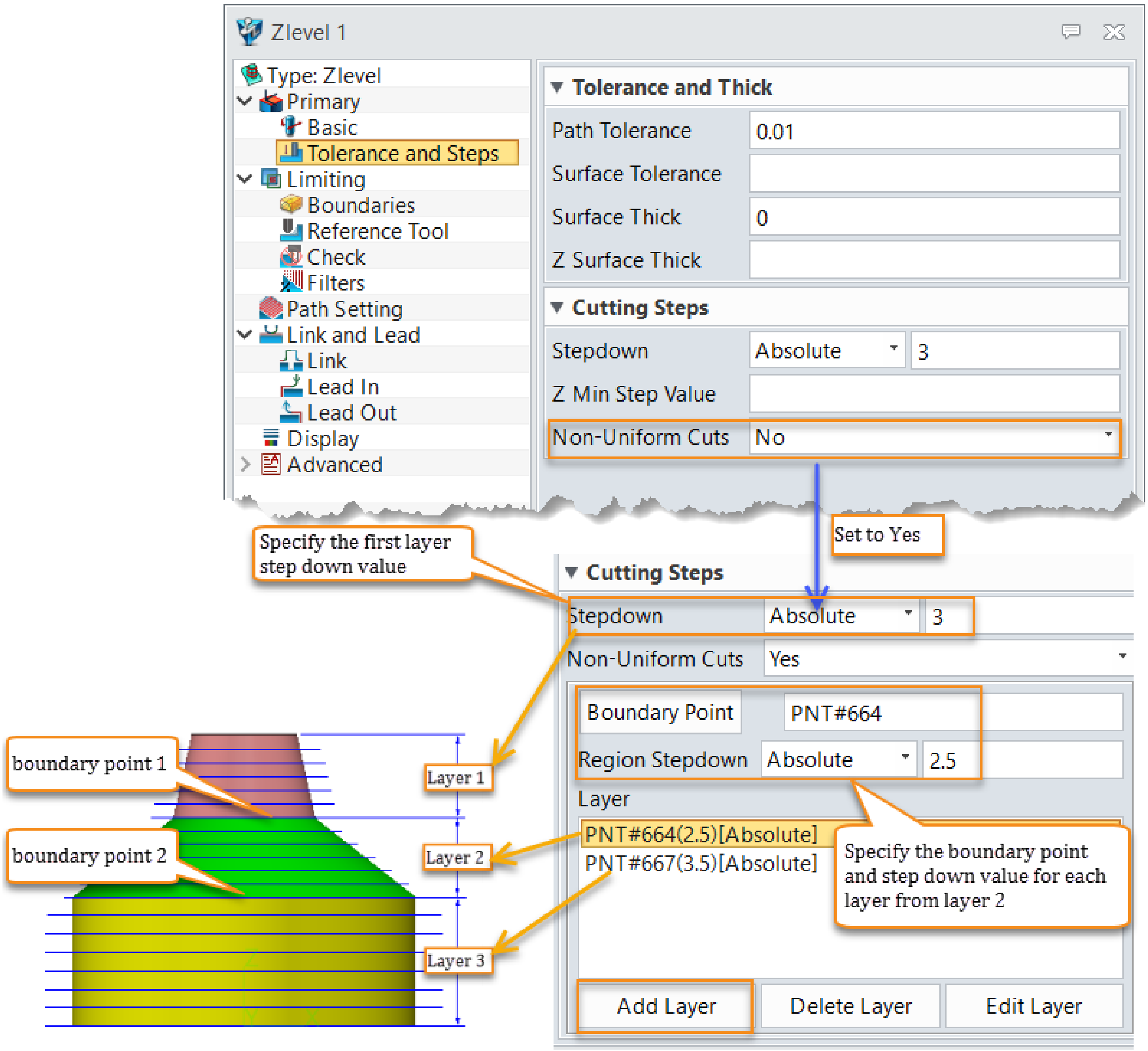


Figure1 Z level operation – Non-uniform cut 2

c) How to add a new layer

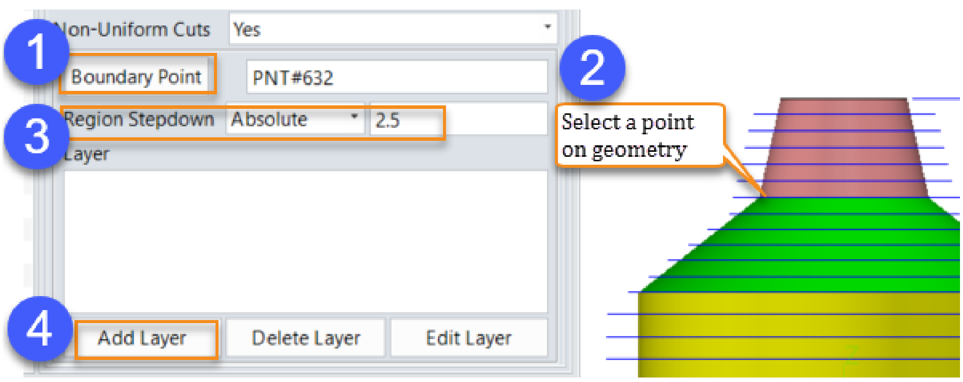


Figure98 Z level operation – Non-uniform cut 3

d) How to delete a layer

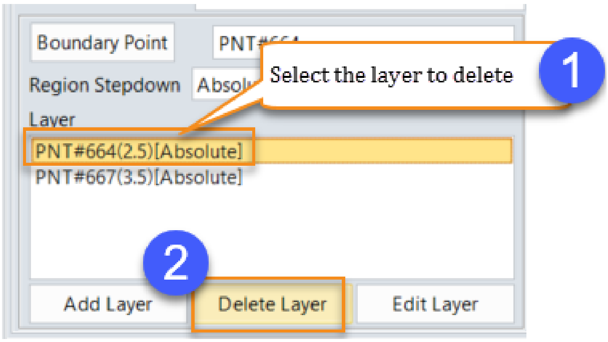


Figure99 Z level operation – Non-uniform cut 4

e) How to edit a layer

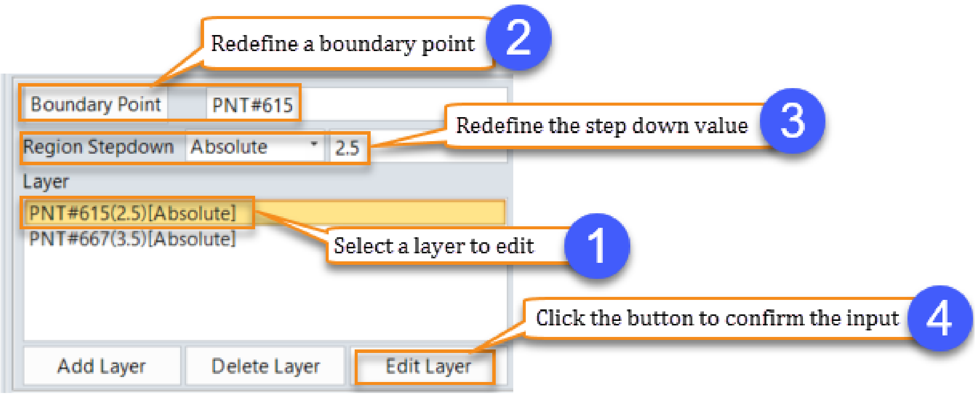


Figure100 Z level operation – Non uniform cut 5

- Path Setting

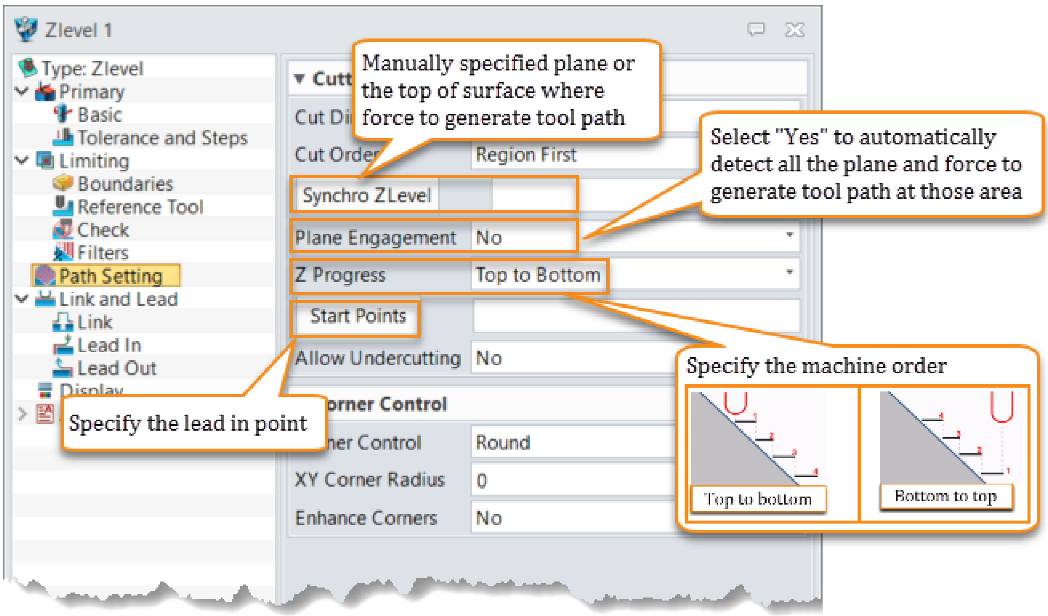


Figure1 Z level operation – Path setting

- Example

STEP 01 Open “CAM_TM_Model.Z3” and go in to CAM space

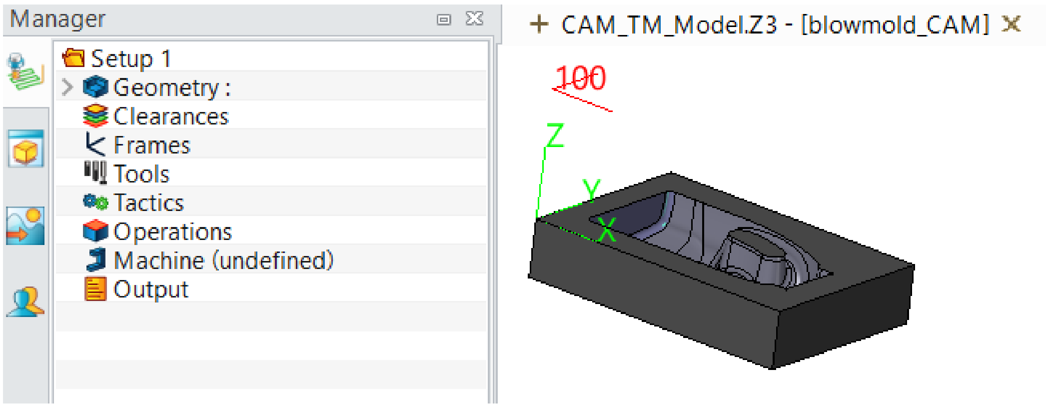


Figure101 Z level operation example step1

STEP 02 Select Z level operation

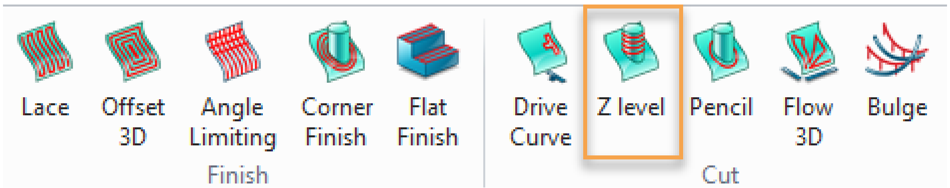


Figure102 Z level operation example step2

STEP 03 Add part to the feature and specify a profile boundary feature.

STEP 04 Specify a D8R4 tool for this operation

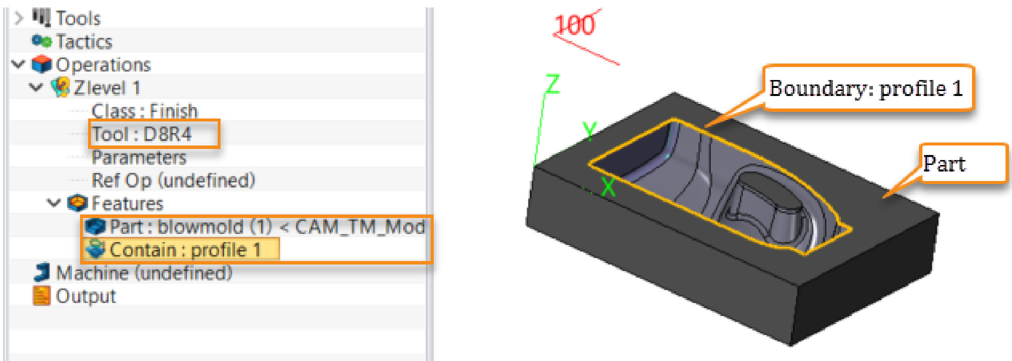


Figure103 Z level operation example step5

STEP 05 Specify steep area angle range for generating Z level tool path

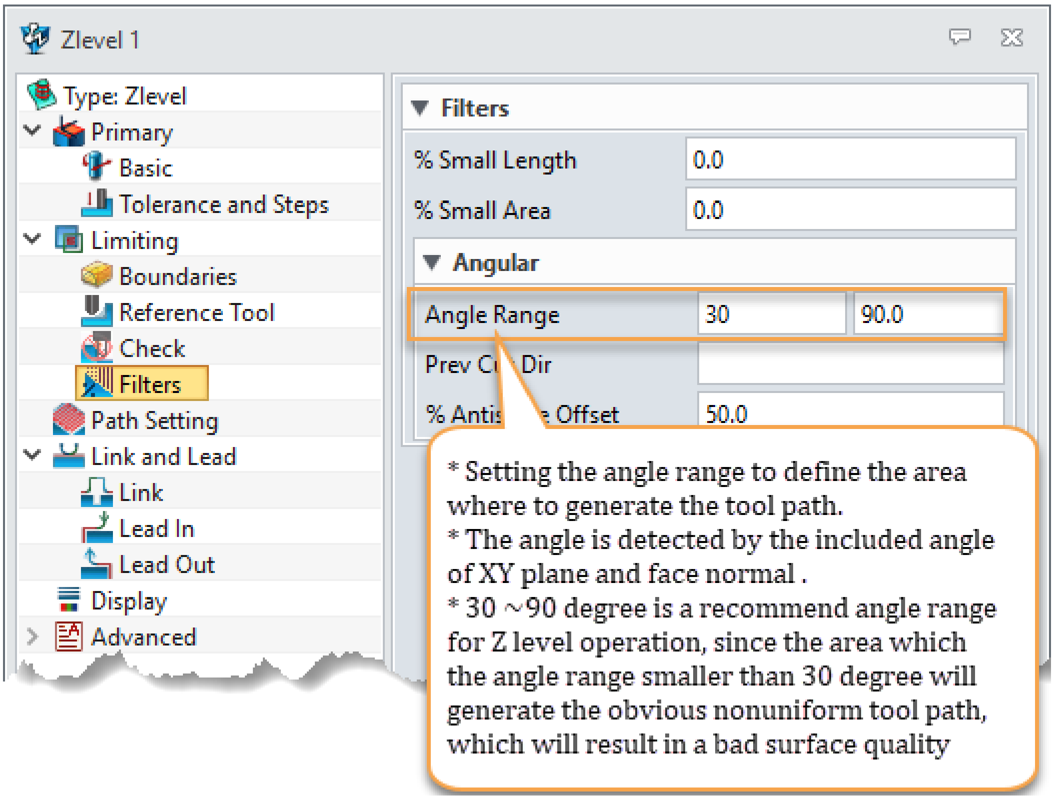


Figure104　　Z level operation example step5

STEP 06 Calculate the tool path

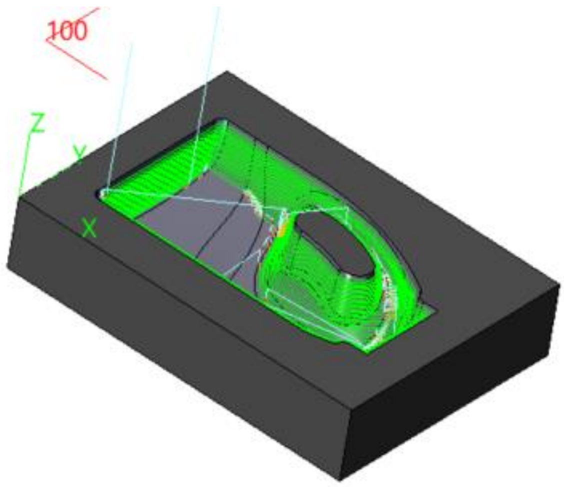
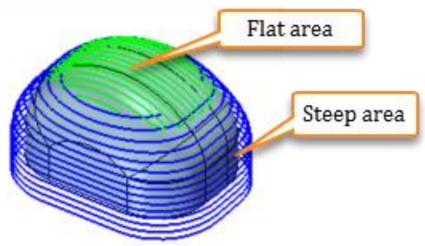


Figure105　　Z level operation example step6

1.3.6 Angle limiting operation

The angle limiting combined several finishing operations in it, which will automatically detect the machine area is steep or flat, then assign proper operation to those area. Lace or offset 3D part pattern is recommended for shallow area, Z level path pattern is recommended for steep area. The definition of steep area can be adjusted by user. This operation will highly improve our machine efficiency and surface finish.

- Path pattern



- Tool path generation requirement
  - Specify tool
  - Specify feature (part or part with profile boundary)
- Associate parameters setting introduce
  - Primary

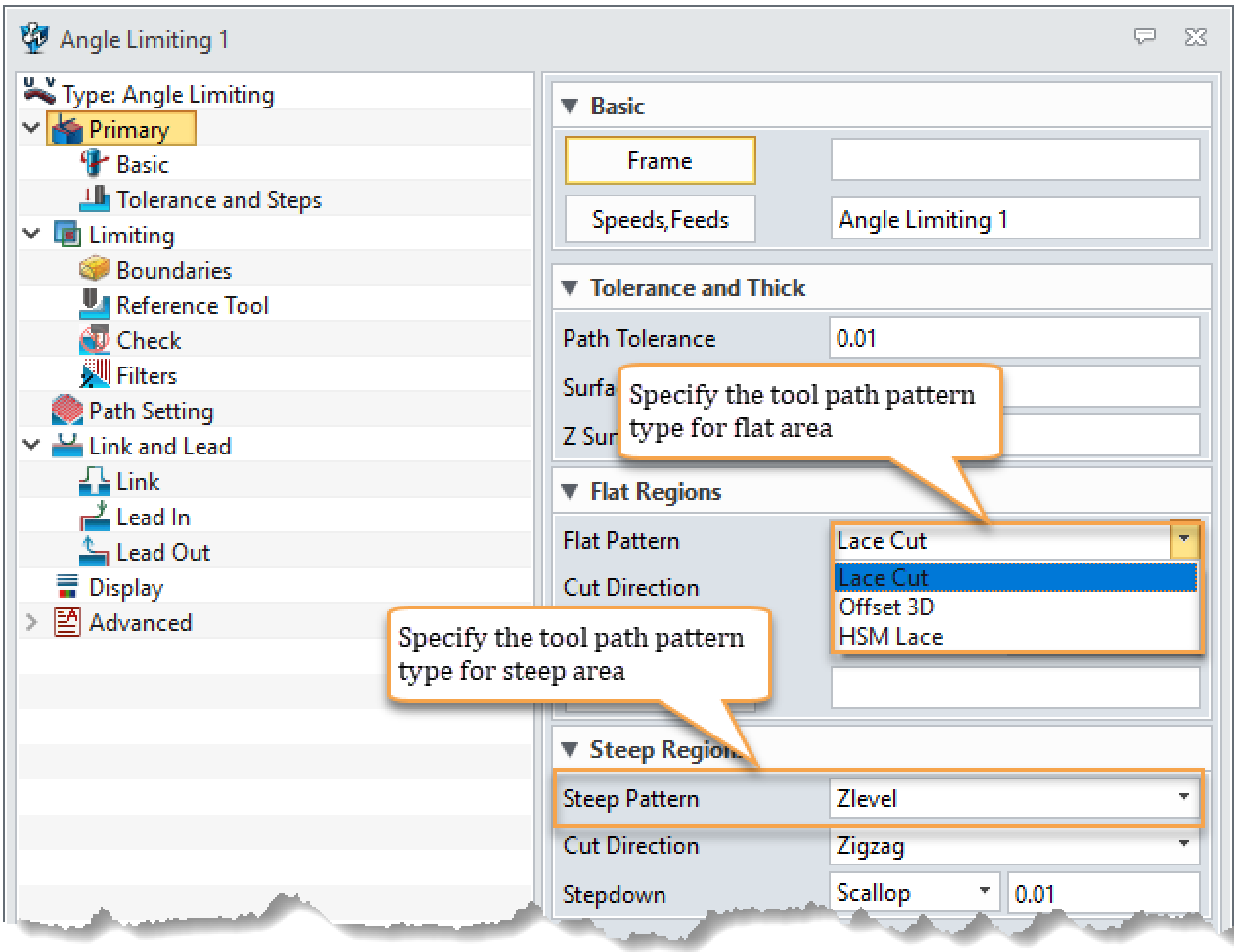


Figure106 Angle limiting operation – Primary

○ Path Setting

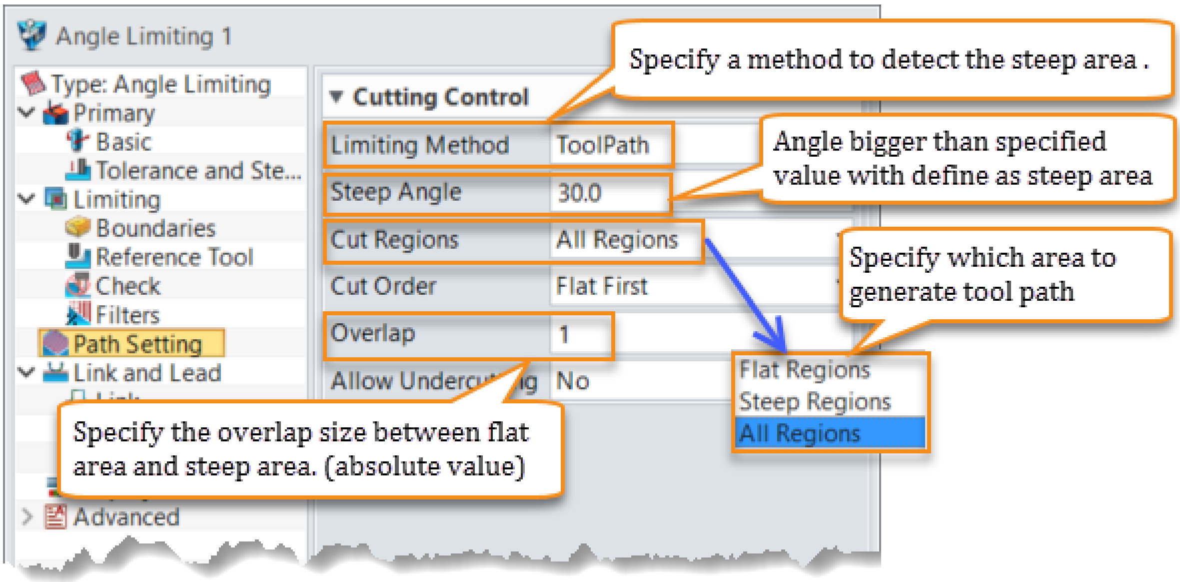


Figure107 Angle limiting operation – Path setting

- Example

STEP 01 Open "CAM_TM_Model.Z3" and go to CAM space

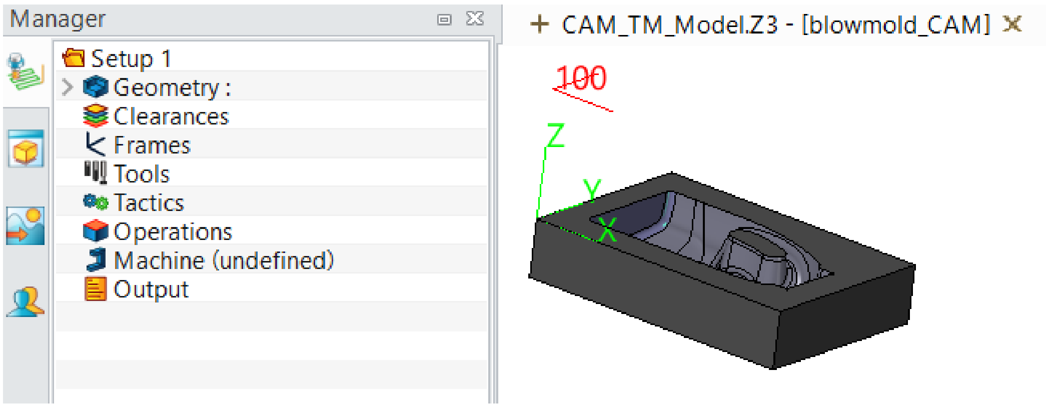


Figure108 Angle limiting operation example step1

STEP 02 Select Angle limiting operation

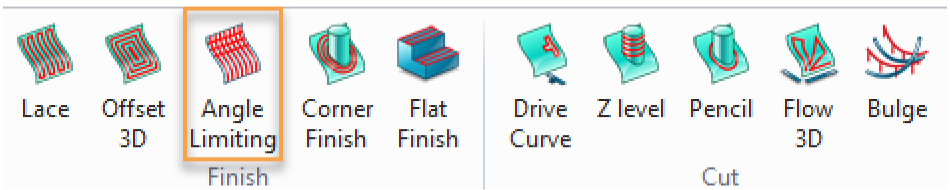


Figure109 Angle limiting operation example step2

STEP 03 Add part to the feature and specify a profile boundary feature.

STEP 04 Specify a D8R4 tool for this operation

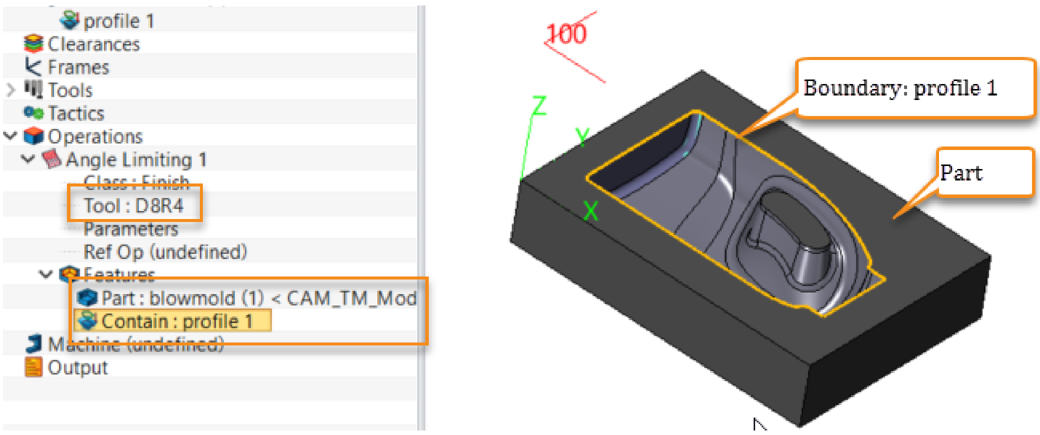


Figure110 Angle limiting operation example step4

STEP 05 Specify path pattern for flat area and steep area

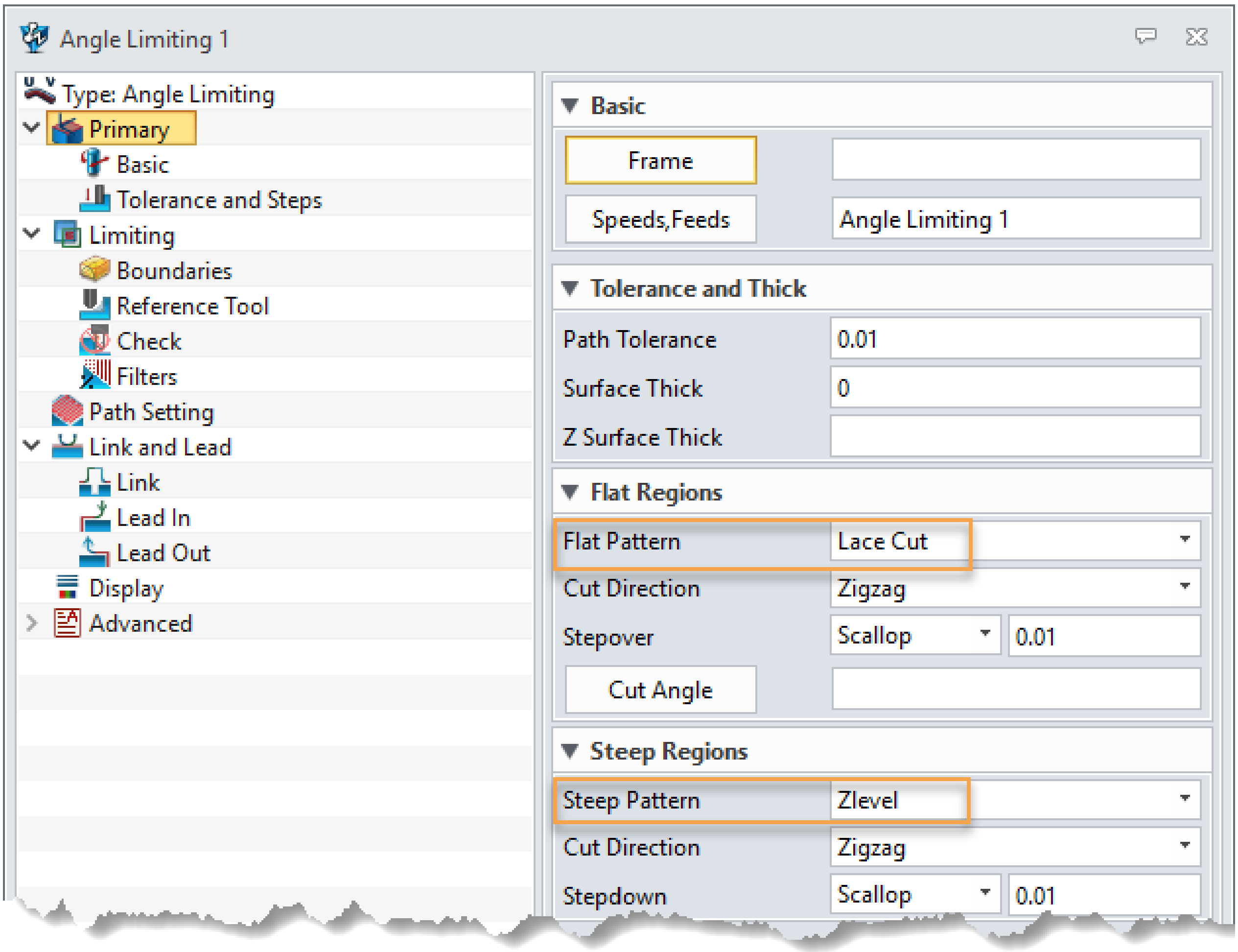


Figure111 Angle limiting operation example step5

STEP 06 Define angle range of steep area

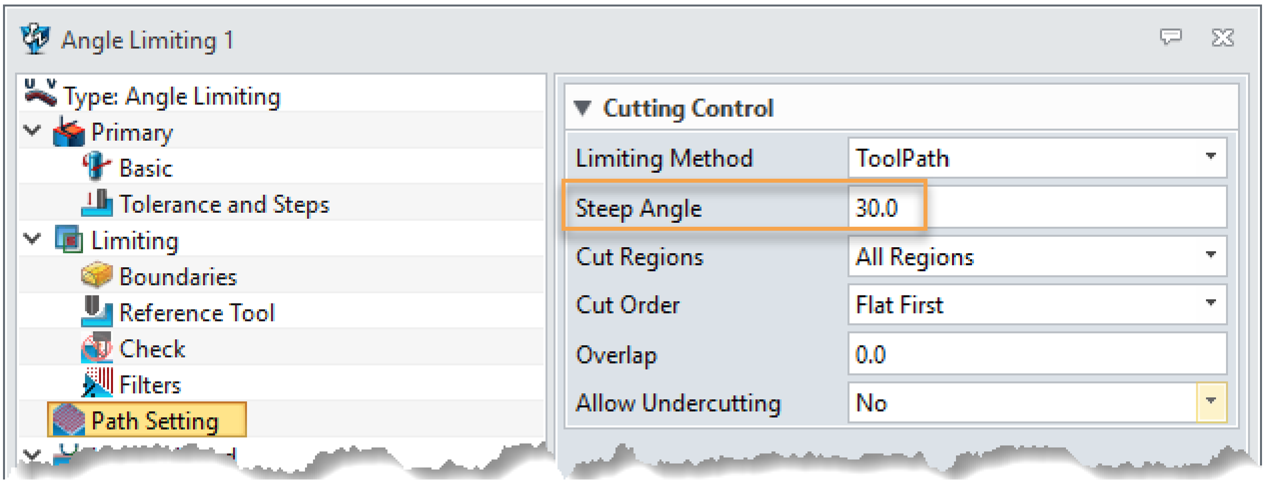


Figure112 Angle limiting operation example step6

STEP 07 Calculate the tool path

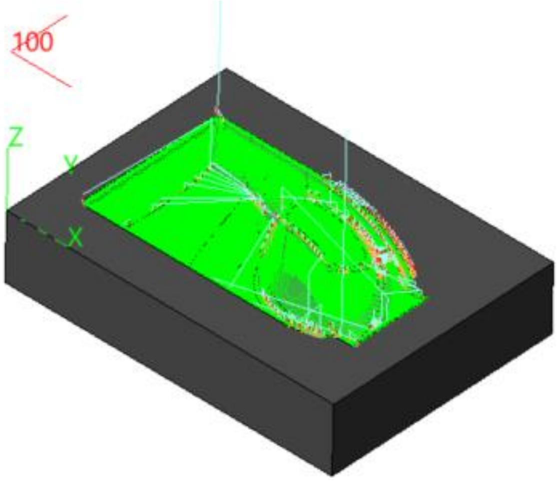
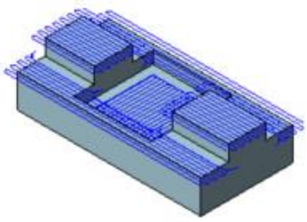


Figure113 Angle limiting operation example step7

1.3.7 Flat finishing operation

Flat finishing operation will detect all of the planar surface within given flat tolerance automatically, and then use lace or offset 2D path pattern to create tool path for all planar surfaces in one operation.

- Path pattern



- Tool path generation requirement
    - Specify tool
    - Specify feature (part or flat region feature)
- Associate parameters setting
    - Path Setting

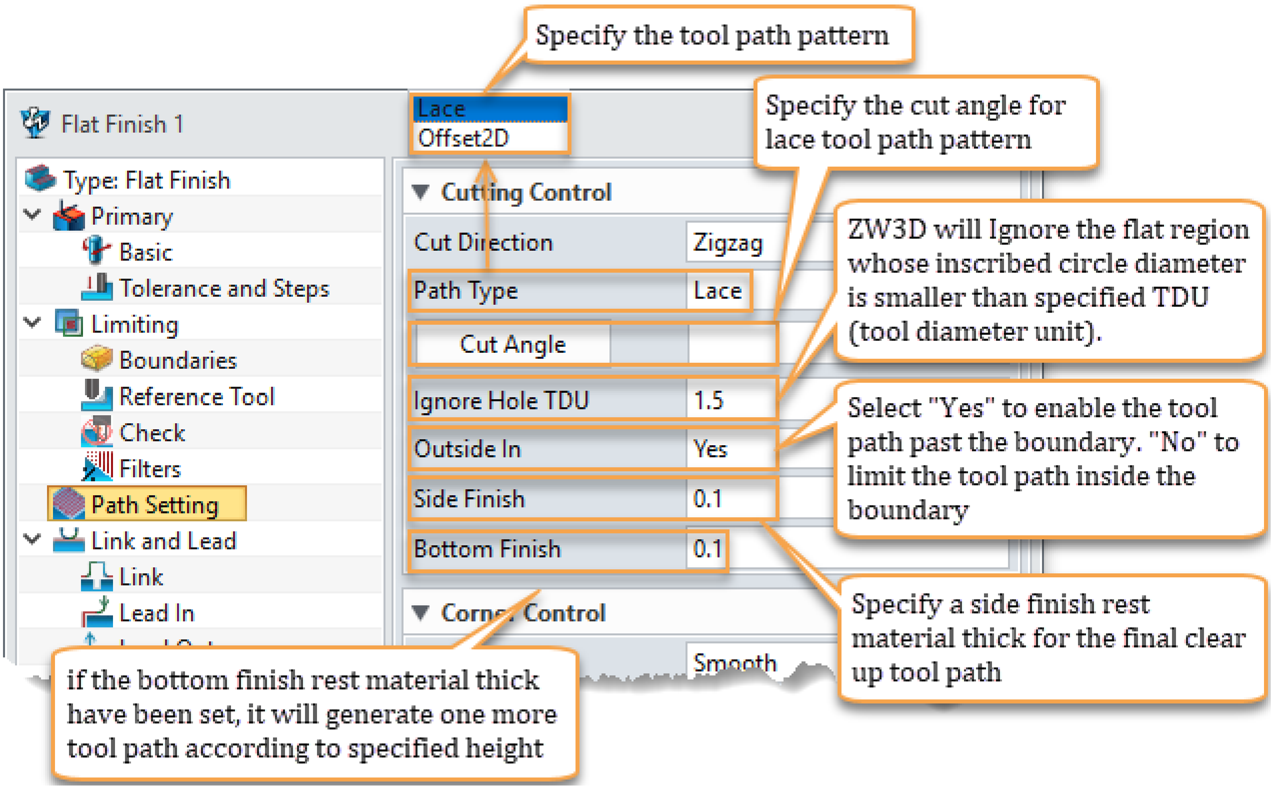


Figure114 Flat finishing operation – Path setting

- Example

STEP 01 Open "Flat Finishing.Z3" file and go to CAM space

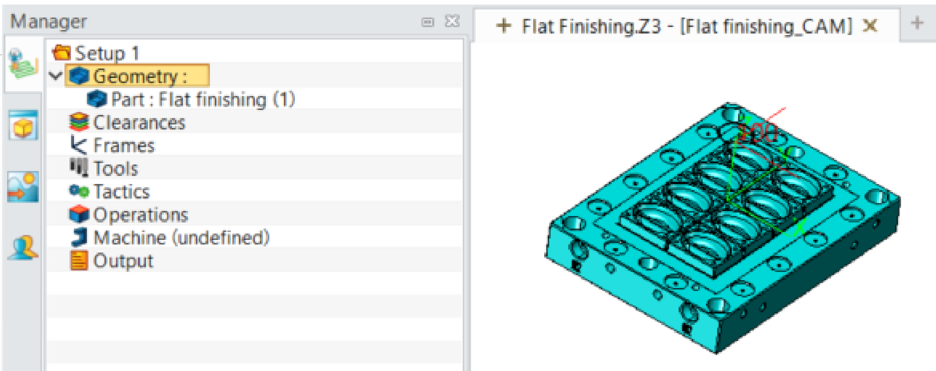


Figure115 Flat finishing operation example step3

STEP 02 Select flat finishing operation

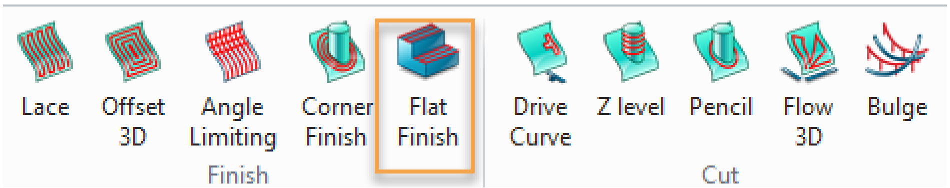


Figure116 Flat finishing operation example step2

STEP 03 Add part to the feature and specify a 10mm flat end tool as the tool

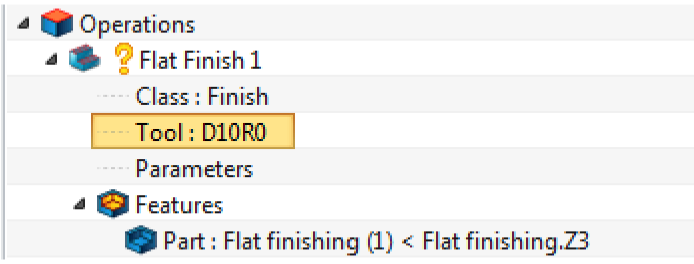


Figure117 Flat finishing operation example step3

STEP 04 Specify the path pattern for flat operation.

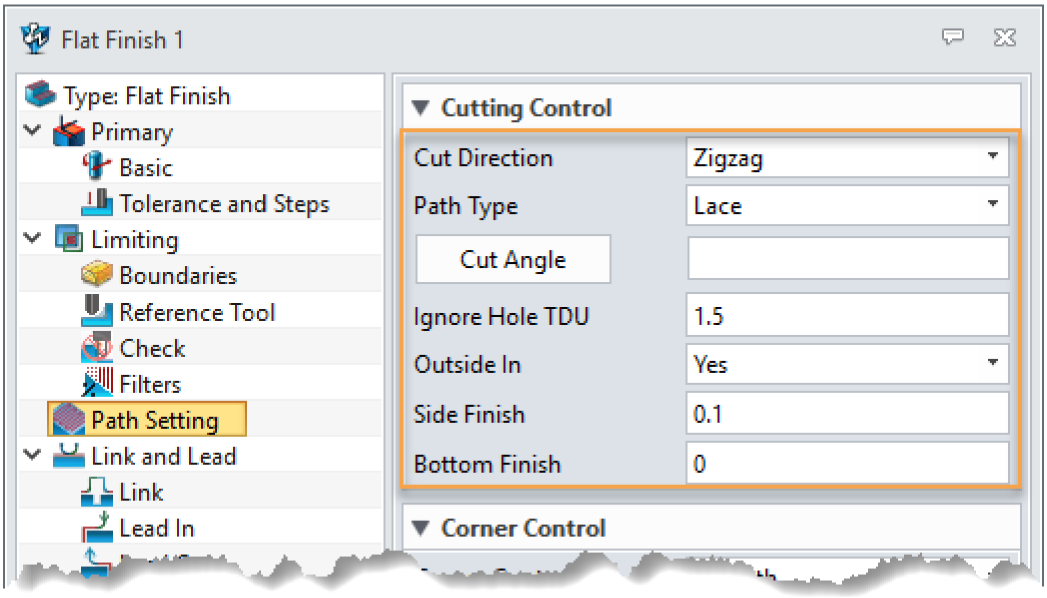


Figure118 Flat finishing operation example step4

STEP 05 Calculate the tool path

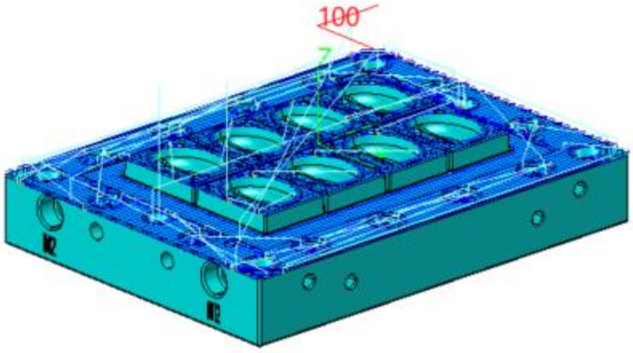
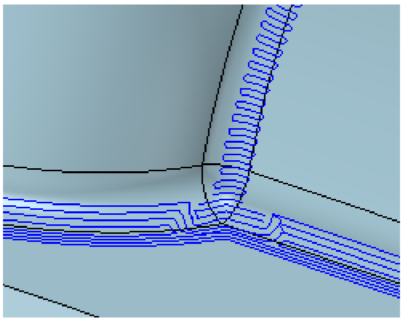


Figure119 Flat finishing operation example step5

1.3.8 Corner finishing operation

Corner finishing operation is a corner cleanup operation, it will detect the rest material by specifying a reference tool and generate tool path to remove those rest materials.

- Path pattern



- Tool path generation requirement
  - Specify tool
  - Specify a reference tool
  - Specify feature (part or part with profile boundary)
- Associate parameters setting introduce
  - Primary

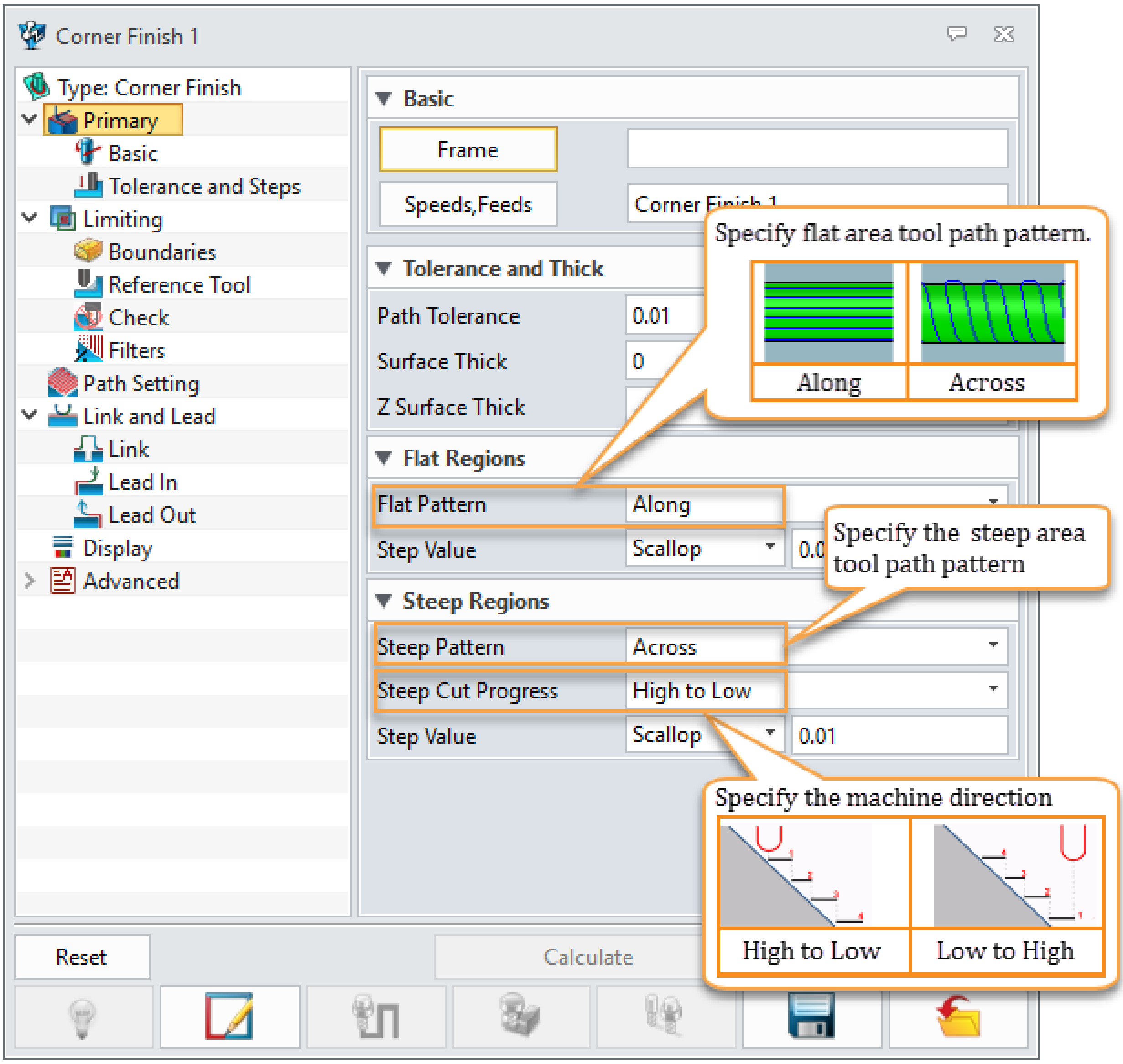


Figure120 Corner finishing operation -- Primary

- Example

STEP 01 Open “Mouse_end.Z3” file and go to CAM space

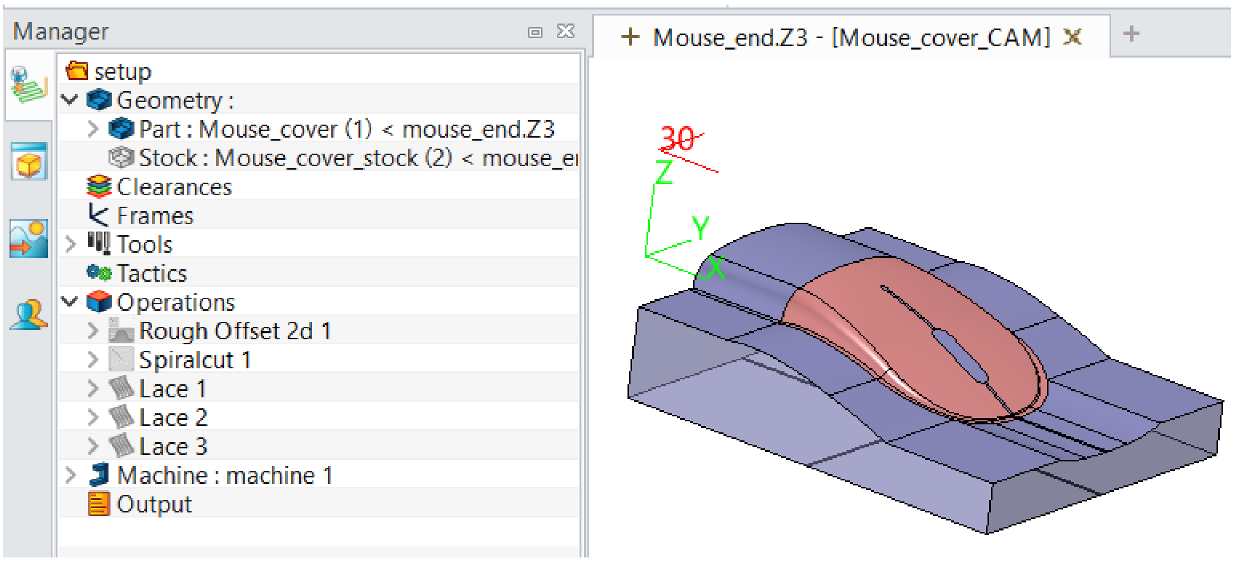


Figure121 Corner finishing operation example step1

STEP 02 Select corner finishing operation.

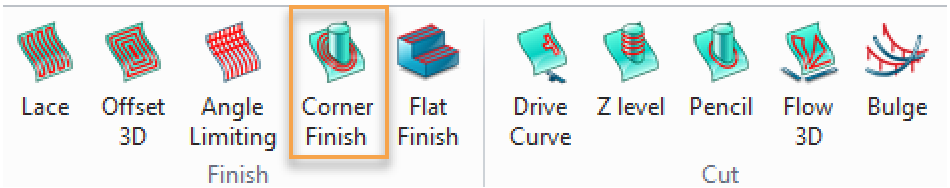


Figure122 Corner finishing operation example step2

STEP 03 Add the part to the feature and specify a D4R2 tool for this operation

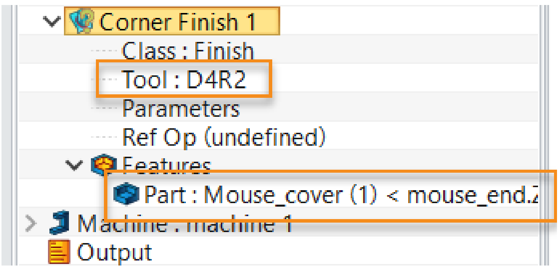


Figure123 Corner finishing operation example step3

STEP 04 Specify path pattern for flat and steep area.

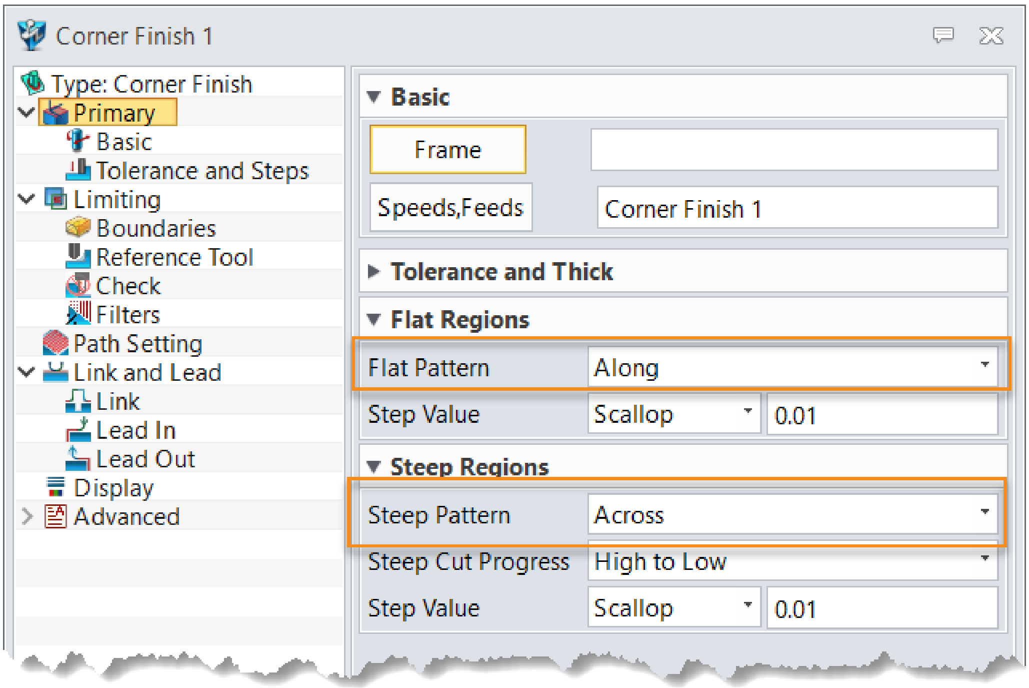


Figure124 Corner finishing operation example step4

STEP 05 Specify a reference tool to detect the rest material area and set the min rest height

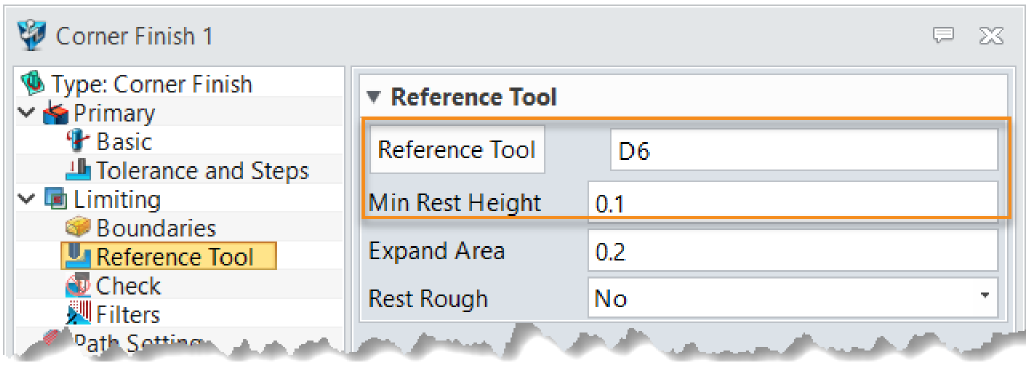


Figure125 Corner finishing operation example step5

Min Rest Height: This parameter sets the minimum height to detect the rest material area according to the specified reference tool.

STEP 06 Calculate the tool path

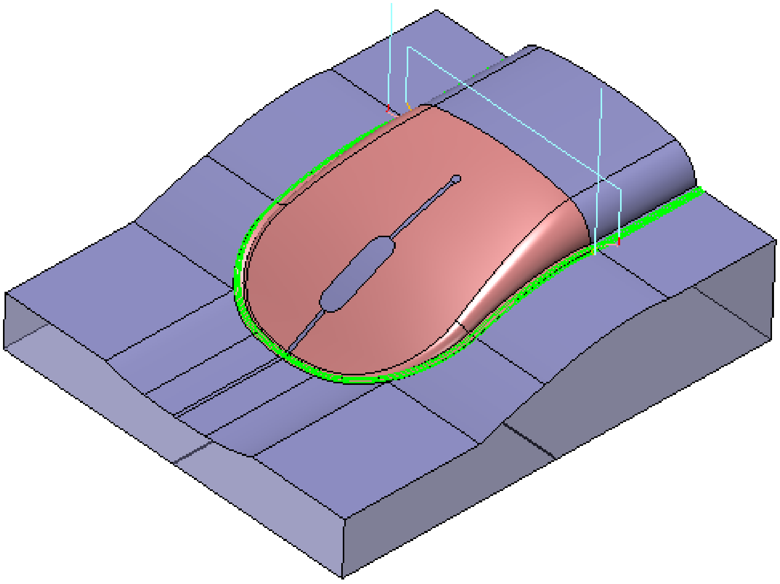


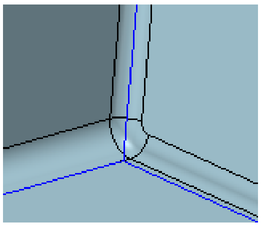
Figure126 Corner finishing operation example step6

1.3.9 Pencil operation

Pencil operation is another corner cleanup operation, it will detect all the corner where the corner radius is equal to or smaller than the specified tool, then create tool path and follow the corner shape to rest material.

Note: The tool radius should be bigger than or equal to the cleanup corner radius, otherwise it can't generate tool path.

- Path pattern



- Tool path generation requirement
  - Specify tool
  - Specify feature (part or part with profile boundary)
- Example

STEP 01 Open "Mouse_end.Z3" file and go to CAM space

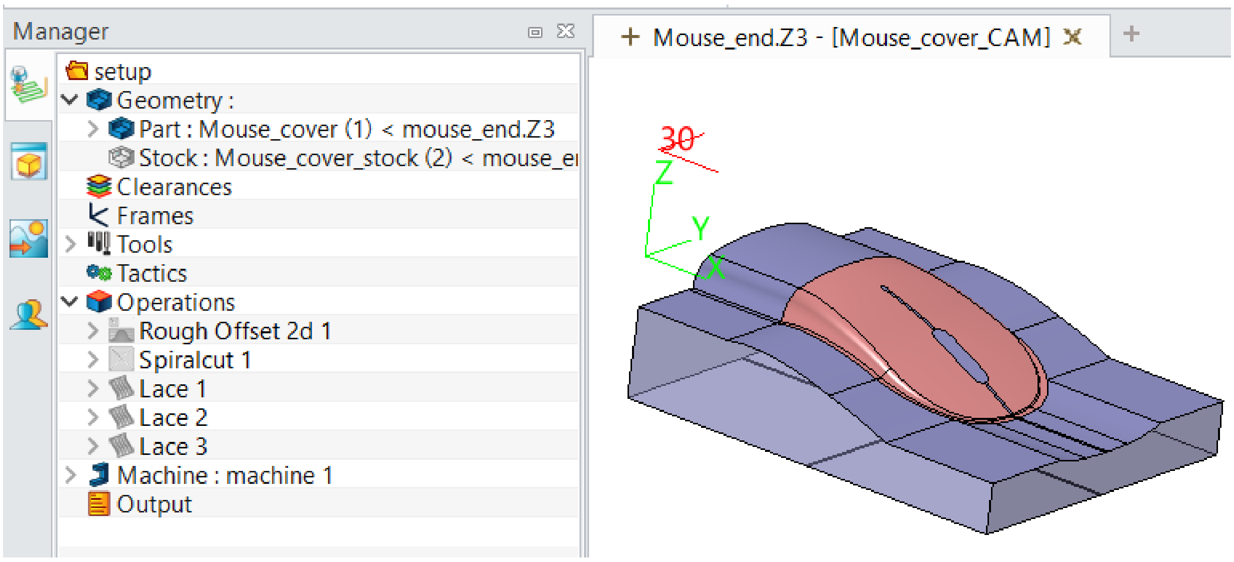


Figure127 Pencil operation example step1

STEP 02 Select pencil operation.

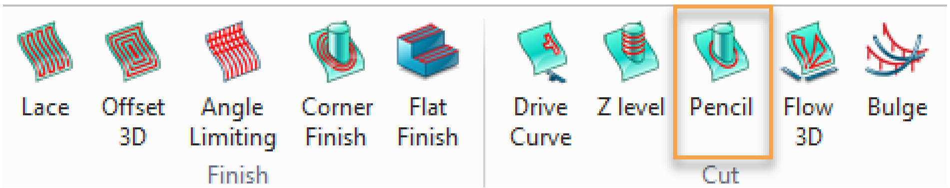


Figure128 Pencil operation example step2

STEP 03 Add the part to the feature and specify a D4R2 tool for this operation

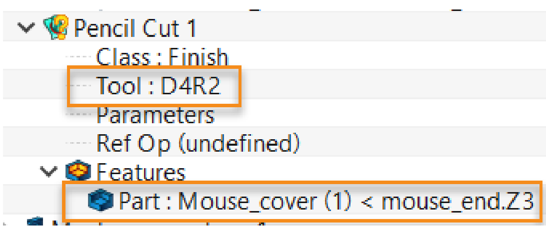


Figure129 Pencil operation example step3

STEP 04 Specify the number of cuts

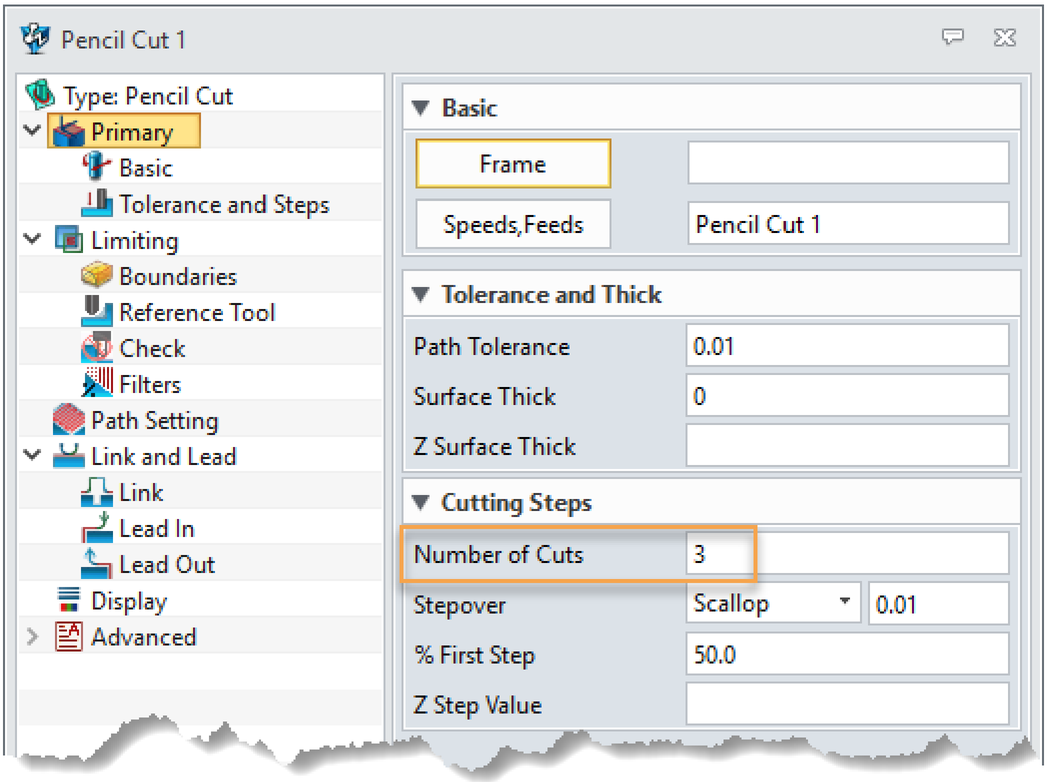


Figure130 Pencil operation example step4

STEP 05 Calculate the tool path

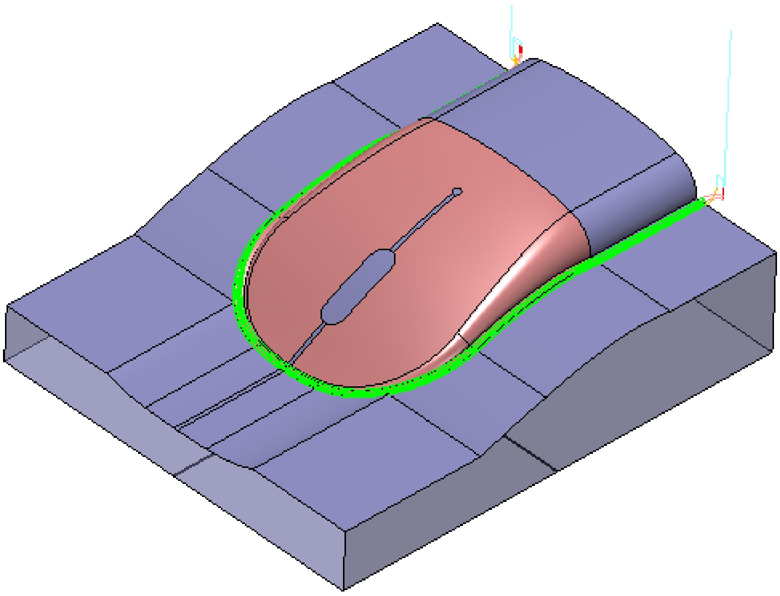


Figure131 Pencil operation example step5

1.3.10 Bulge operation

Bulge operation creates a network of bulges on a surface using two intersecting curves referred to as a drive curve (Directrix) and a generator curve (Generatrix). This operation is useful in dispersion networks for automotive lighting parts or as an artistic hammer-paint effect. The Generatrix is used as a pattern generator that is guided by the Directrix.

Note:

1) The first profile in feature group will be defined as Generatrix

2) The second profile in feature group will be defined ad Directrix

3) The interaction point of Generatrix & Directrix will be defined as the origin to array the tool path

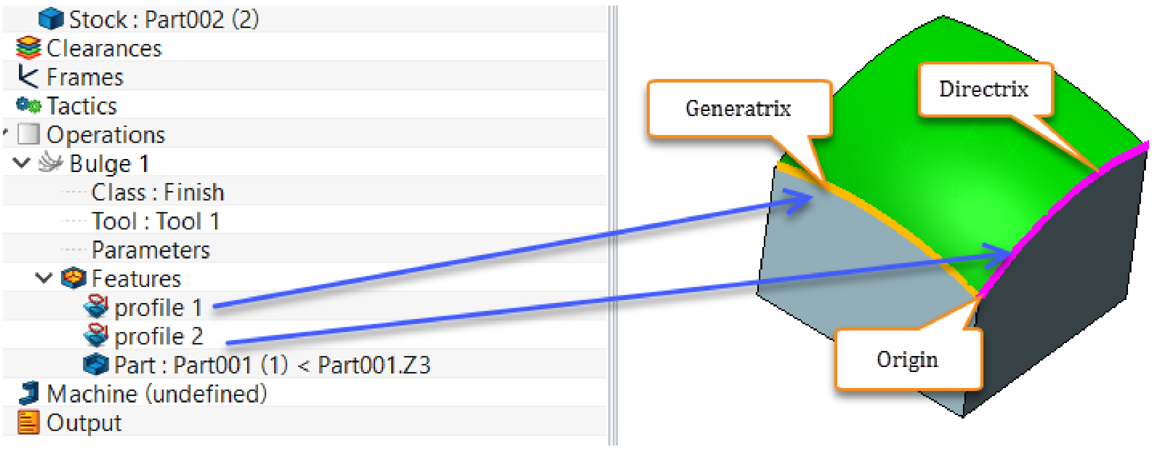
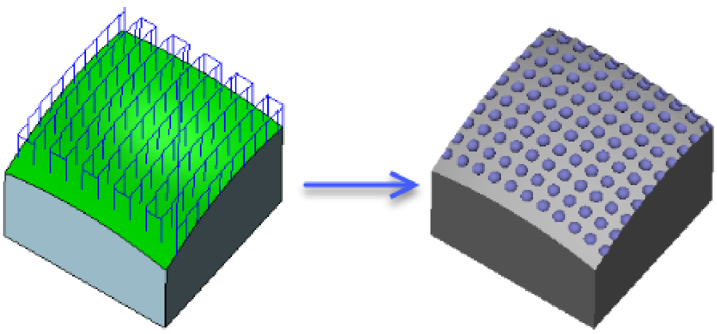


Figure132 Bulge operation

- Path pattern



- Tool path generation requirement
  - ○ Specify tool
  - ○ Specify array guide curve (Generatrix & Directrix)
  - ○ Specify feature (part or part with profile boundary)

- Associate parameters setting
  - Primary

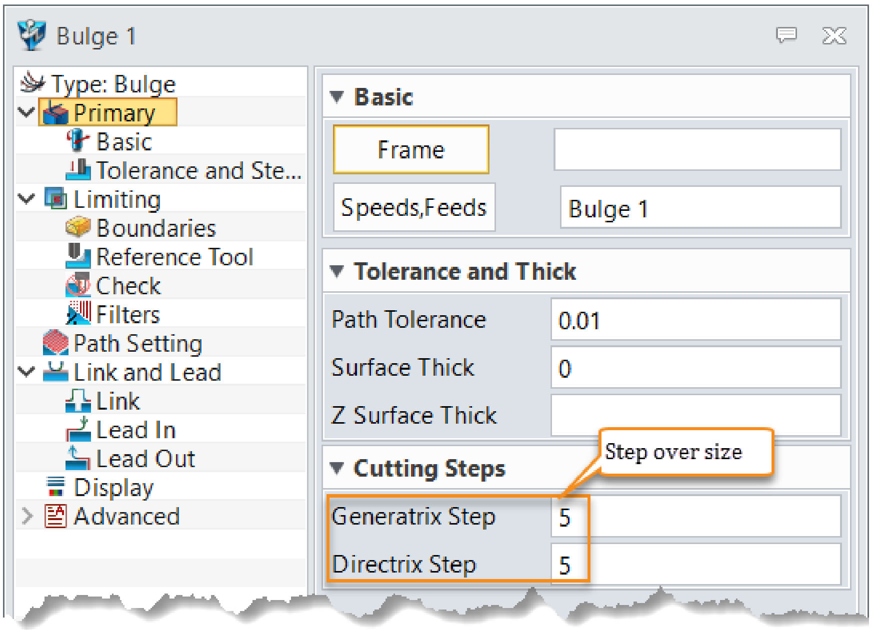


Figure133 Bulge operation – Primary

  - Path setting

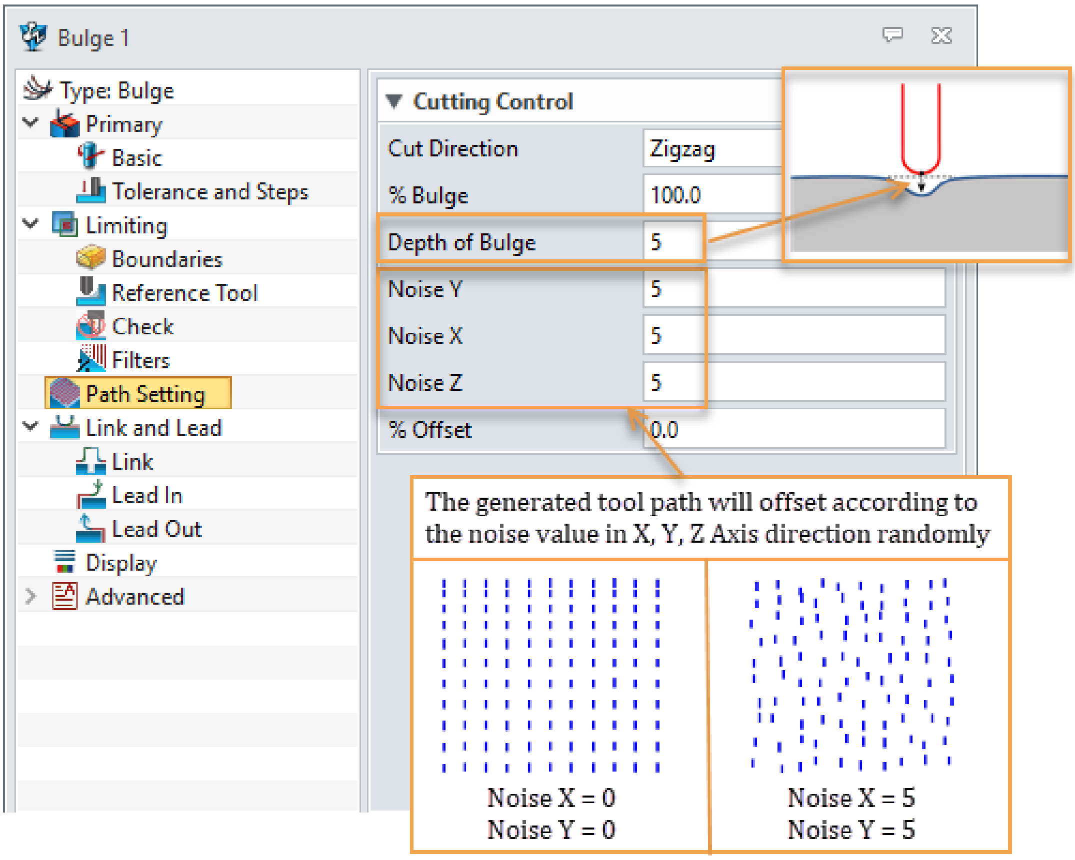


Figure134 Bulge operation – Path setting

1.4 Engraving

Except creating tool path to machine product, company logo, produce specification code or some other design pattern are also needed to add into the product. This chapter will introduce how to create those engraving tool path.

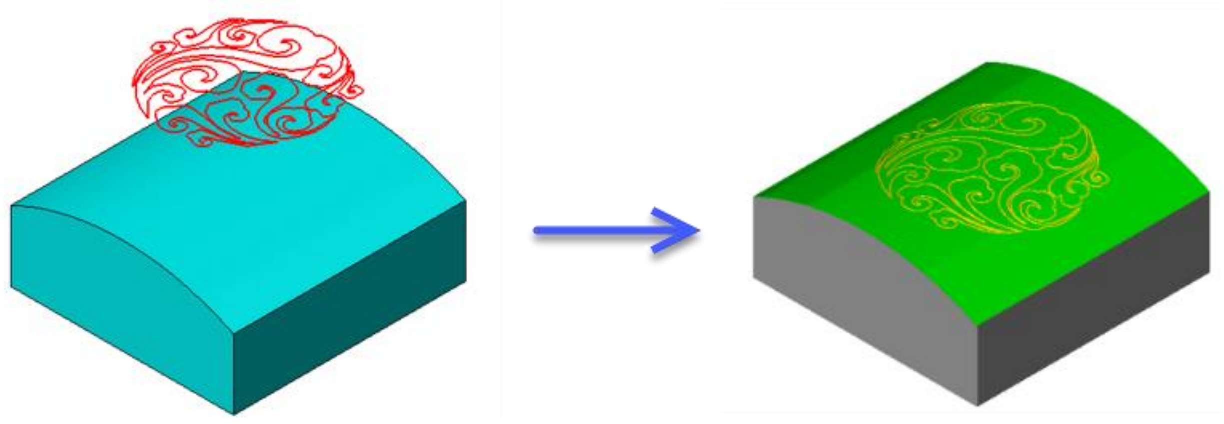


Figure135 Engraving

1.4.1 Engraving for design pattern

For design pattern engraving, ZW3D provides several methods to assist users create the tool path easily and efficiently. Such as using surface engraving, spiral finishing, drive curve operation. Here we will use 3 examples to introduce the engraving tool path generation.

- Filling tool path into design pattern

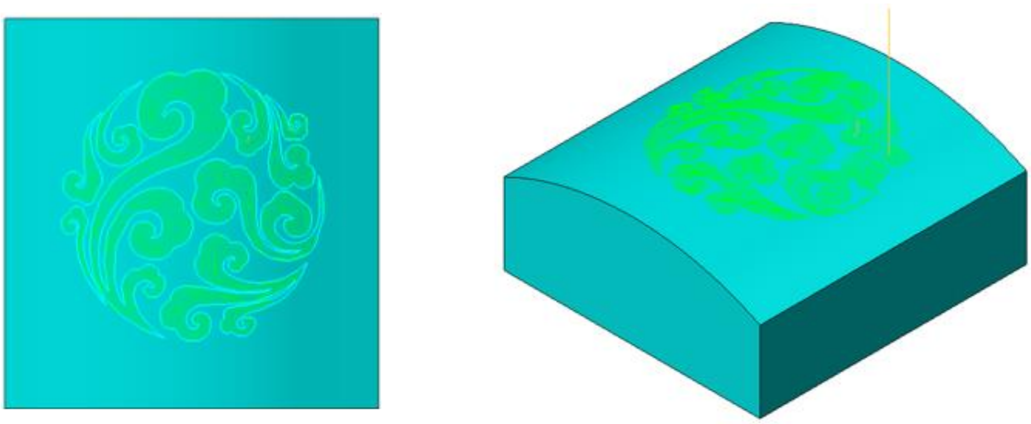


Figure136 Filling tool path into design pattern

For this kind of pattern, spiral finishing operation is recommended to use. To fill the tool path into the design pattern, which requests the profile to be closed. If the profile is an open one, we need to use CAD function to modify it first.

STEP 01 Open “Engraving.Z3”file and go into the CAM space.

STEP 02 Call out the 3X Nurbs ribbon menu first.

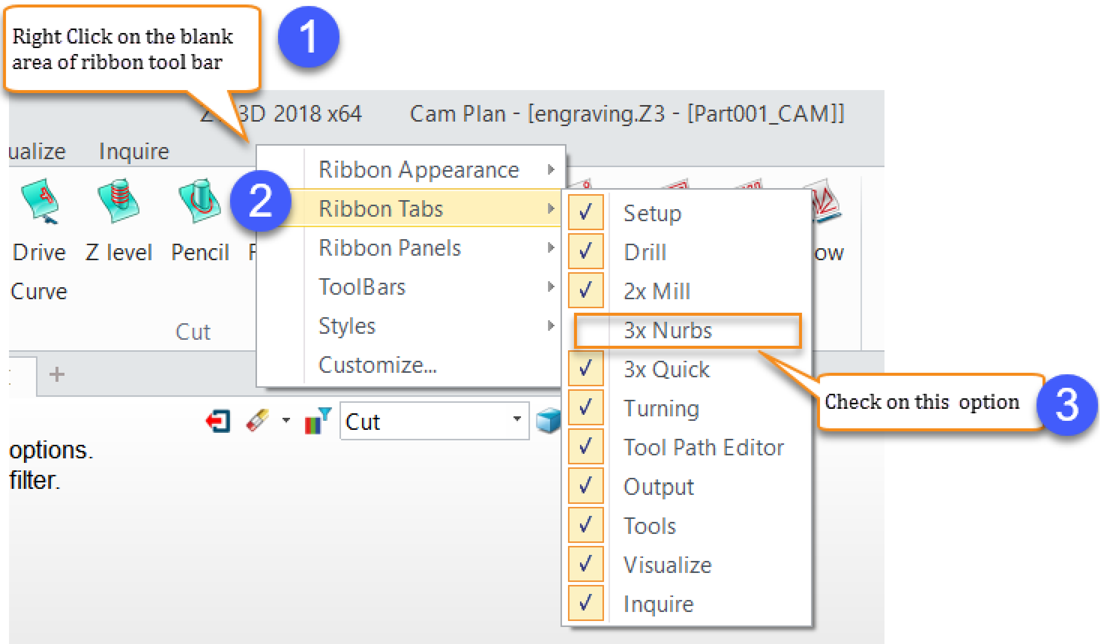


Figure137 Callout 3D Nurbs ribbon menu

STEP 03 Select the spiral finishing operation.

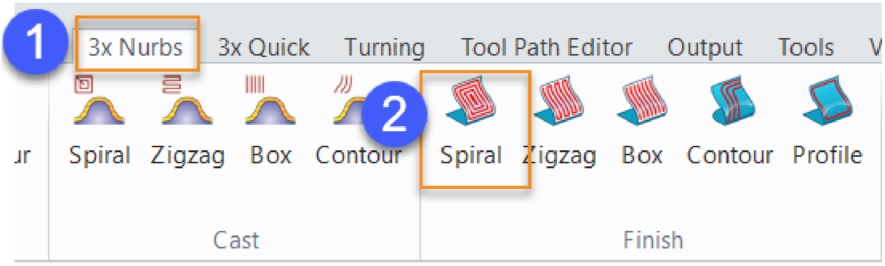


Figure138 Spiral operation

STEP 04 Select all the curves and define it as part profile and add the part to feature.

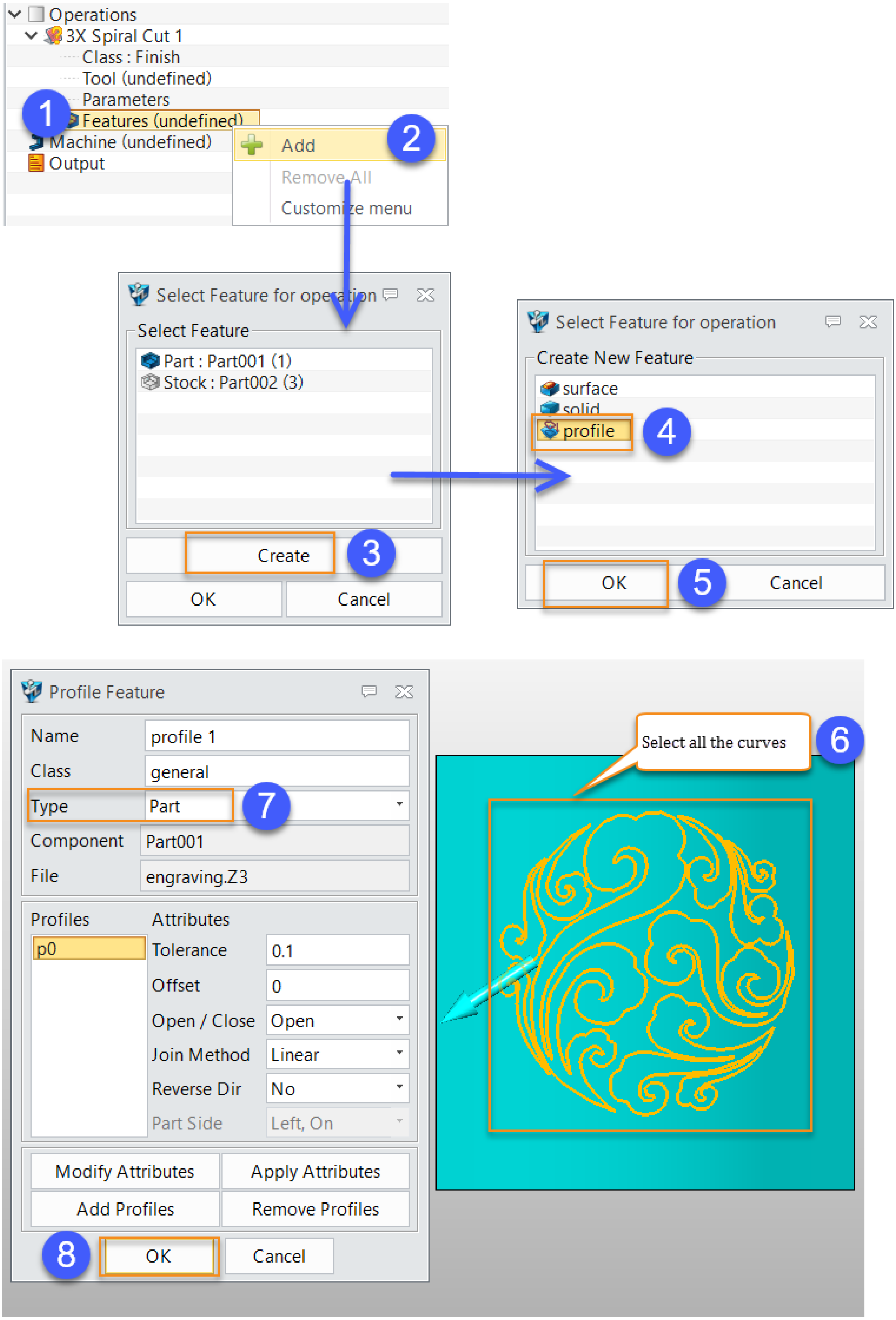


Figure139 Define operation feature

STEP 05 Specify a 1mm ball endmill.

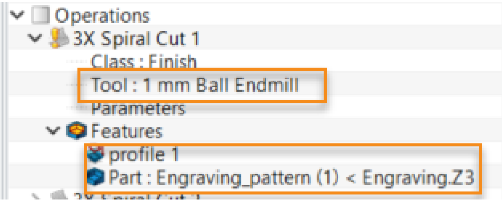


Figure140 Specify a 1mm ball end tool

STEP 06 Set engraving depth, step over value, the other option use default setting.

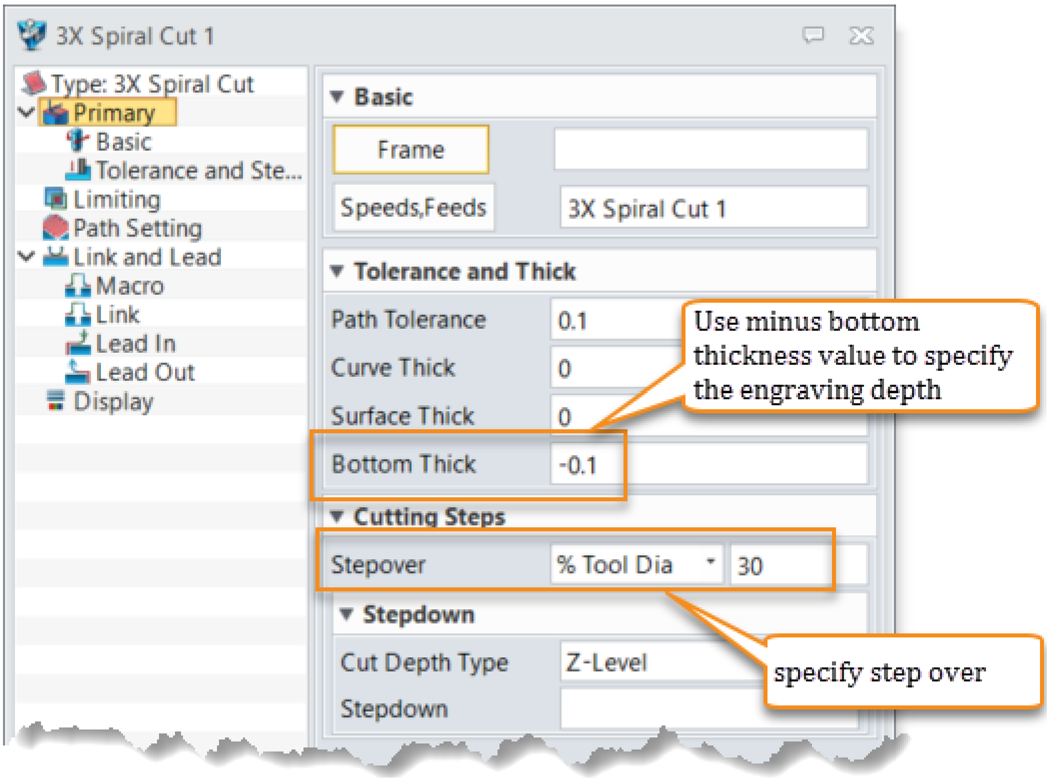


Figure141 Specify maching parameter

STEP 07 Calculate the tool path.(The green lines are the tool path)

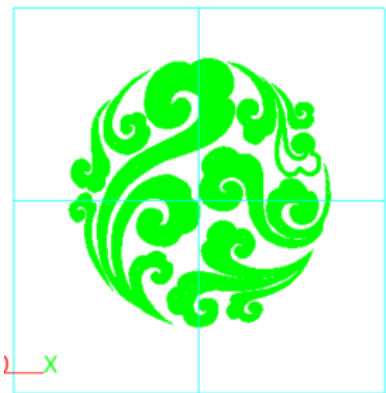


Figure142 Final result of engraving 1

Summary: Tool path generation requirement for spiral finishing operation

- Specify a tool
- Specify the boundary profile and set the profile type to "part"
- Add the part to feature
- Specify the engraving depth and step over size.

- Filling tool path outside the design pattern

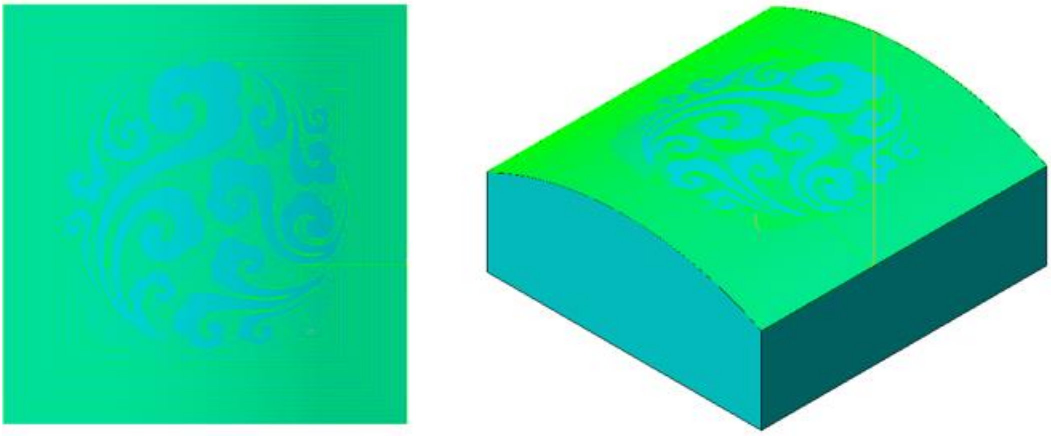


Figure143 Filling tool path outside the design pattern

The method of calculating the tool path outside the boundary is very similar to "Fill tool path into design pattern", the only difference is the boundary profile.

STEP 01 Duplicate the operation that we just created and remove profile 1

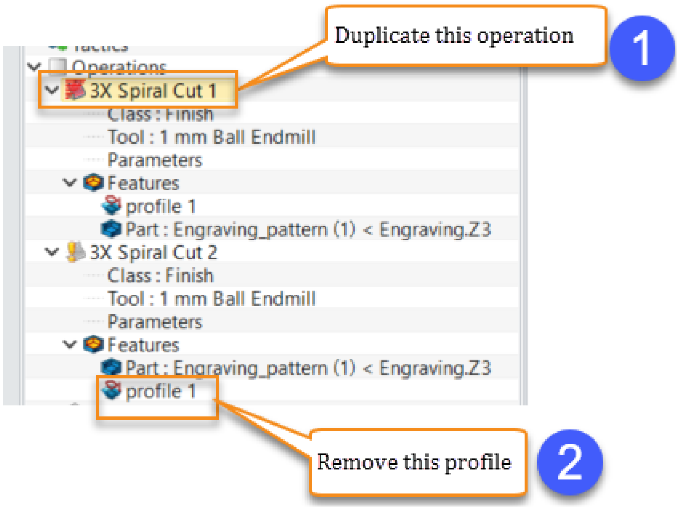


Figure144 Filling tool path outside the design pattern example step1

STEP 02 Select all the design pattern and the boundary of the part as profile 2.

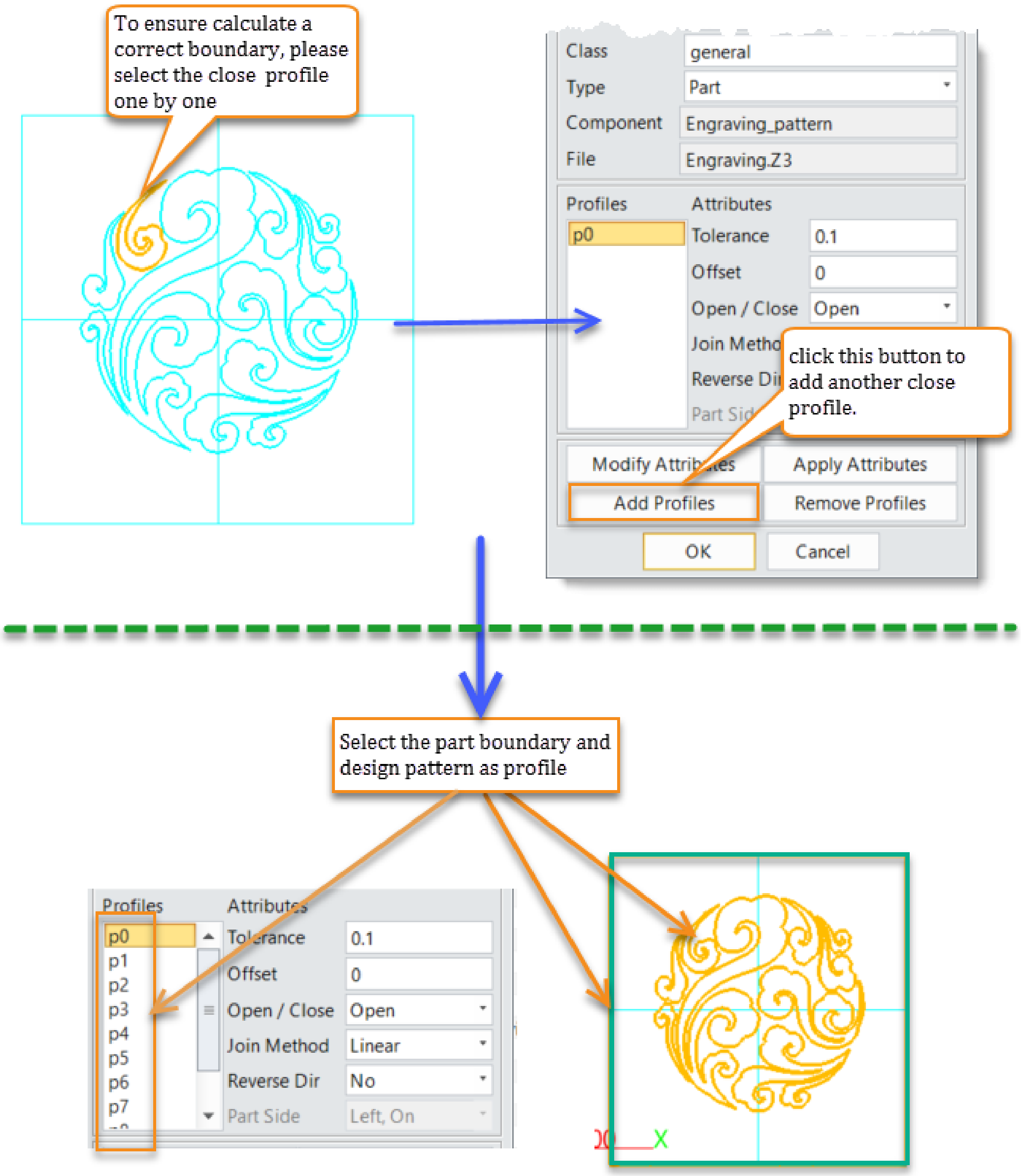


Figure145 Filling tool path outside the design pattern example step2

STEP 03 Don't modify associate parameters and calculate the tool path directory.

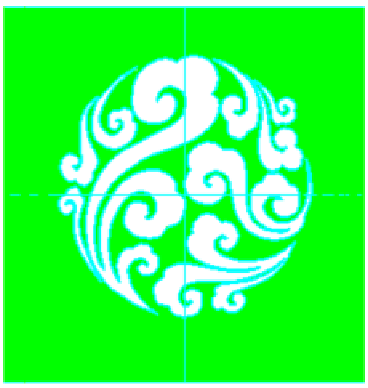


Figure146 Filling tool path outside the design pattern example step3

- Create tool path on the design pattern

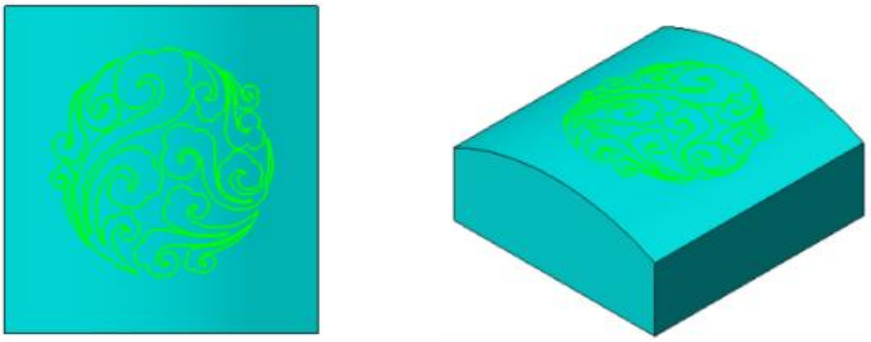


Figure147 Create tool path on the design pattern

To create tool path on design pattern, ZW3D provides two methods to do it. One is using surface engrave operation, the other is using drive curve operation.

○ Step of create tool path by surface engrave operation

STEP 01 Select surface engrave operation

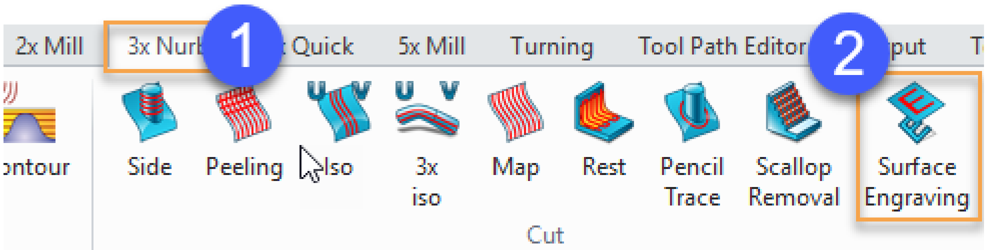


Figure148 Create tool path on the design pattern example step1

STEP 02 Select all the design pattern as profile and select part to feature. We can reuse the profile feature which are created in spiral finishing operation.

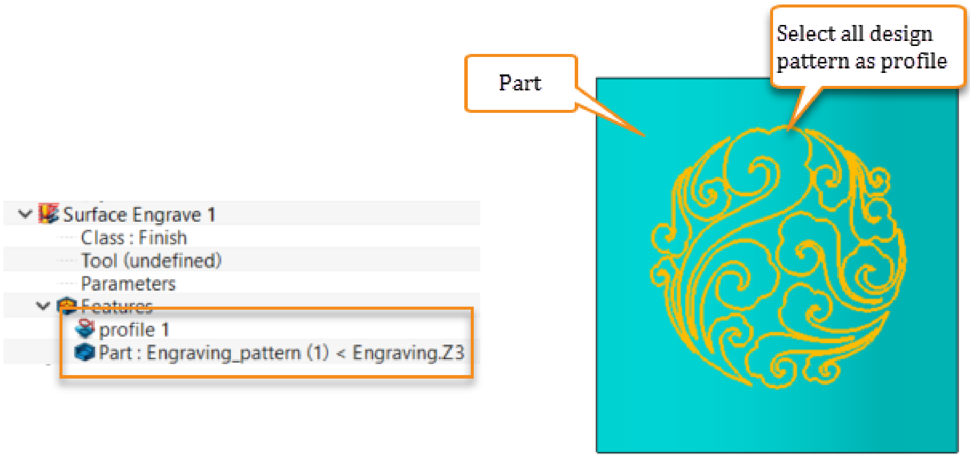


Figure149 Create tool path on the design pattern example step2

STEP 03 Select 1mm ball end tool

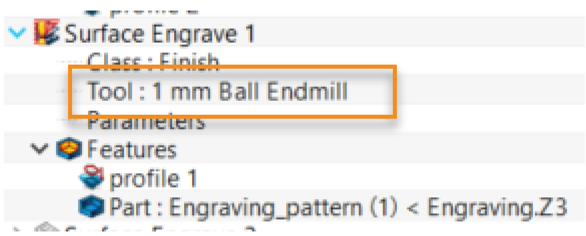


Figure150 Create tool path on the design pattern example step3

STEP 04 Specify the engraving depth by setting bottom thickness

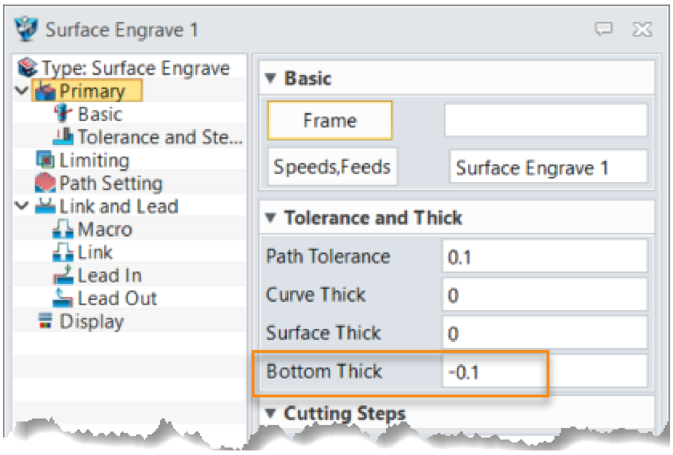


Figure151 Create tool path on the design pattern example step4

STEP 05 Calculate tool path by keeping associate parameters with default value.

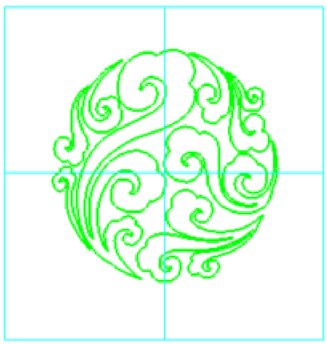


Figure152 Create tool path on the design pattern example step5

○ Steps of creating tool path by drive curve operation

STEP 01 Select drive curve engrave operation

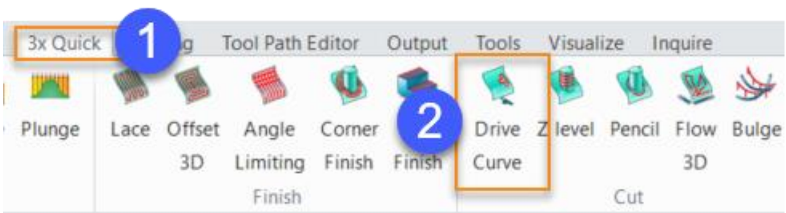


Figure153 Drive curve engraving example step1

STEP 02 Select all the design pattern as profile and select part to feature.

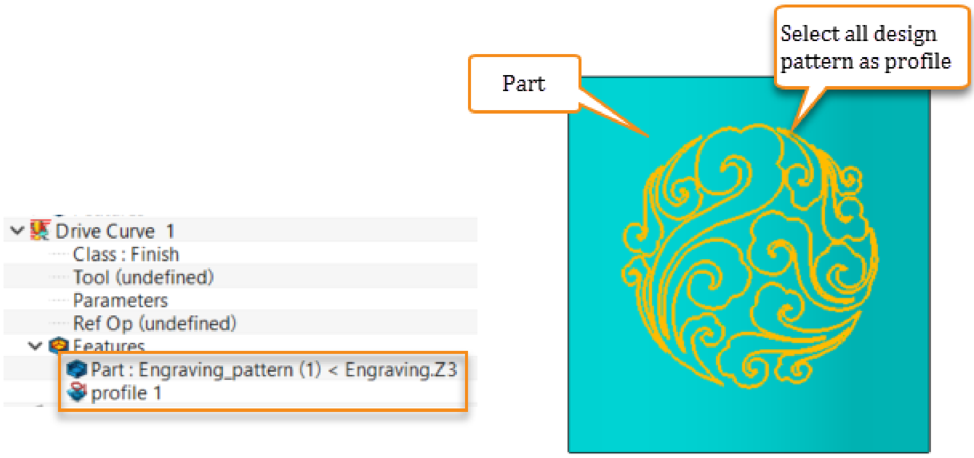


Figure154 Drive curve engraving example step2

STEP 03 Select a 1mm ball end tool

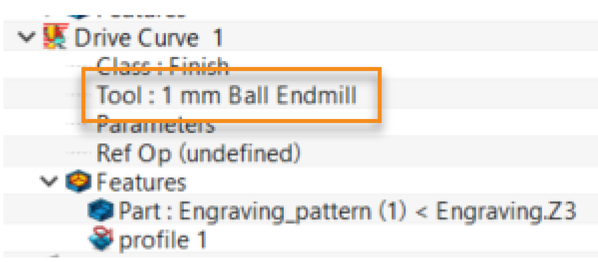


Figure155 Drive curve engraving example step3

STEP 04 Specify the engraving depth by setting Z Surface Thick.

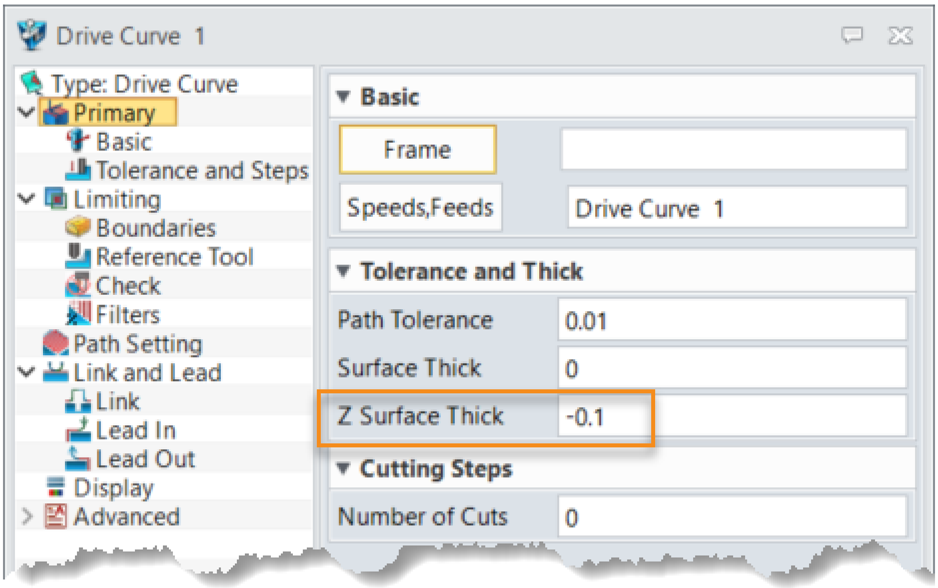


Figure156 Drive curve engraving example step4

STEP 05 Calculate tool path by keeping associate parameters with default value.

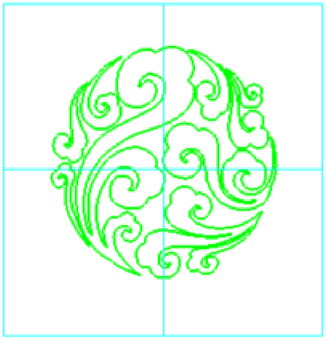


Figure157 Drive curve engraving example step5

Summary: Tool path generation requirement for surface engrave or drive curve operation

- Specify a tool
- Specify the boundary profile and set the profile type to "part"
- Add the part to feature
- Specify the engraving depth.

Difference: The tool path created by surface engraving or drive curve methods seems to the same, but also has its usage scene. Since we need to create a deep engraving and the tool can bear the depth in one time, we need to divide the depth to several times which are under the tool safe load. In this situation, we suggest using surface engraving operation because it can set step down value. But in other scenes, the depth is proper, we just want a much bigger size engraving. Of course we can get the result by changing a much bigger tool. Or we can use drive curve operation by offset the tool path in XY direction. Normally, without special request, surface engrave operation is recommended.

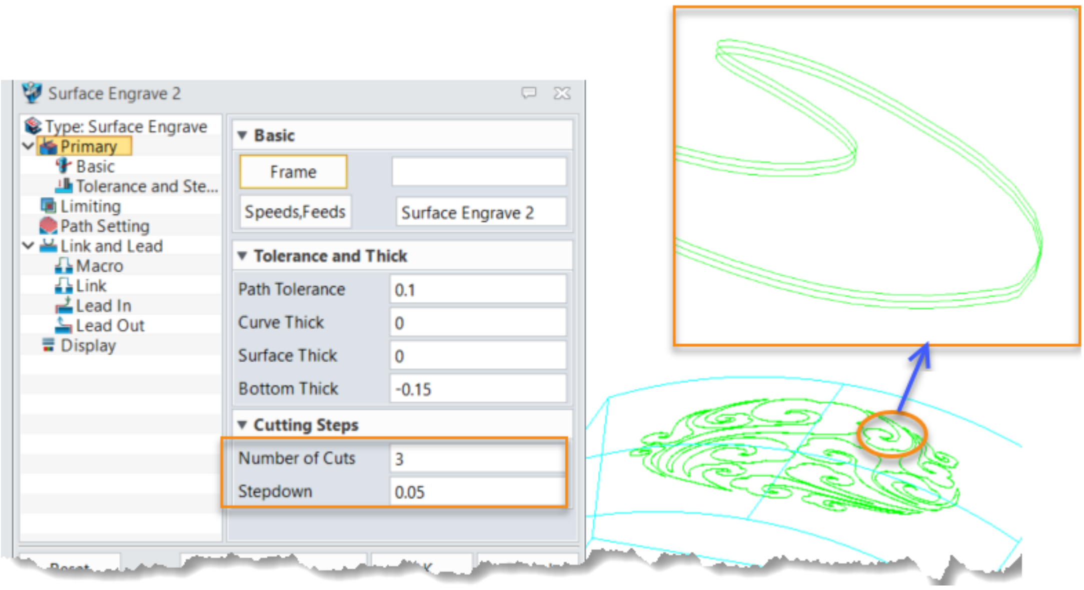


Figure158 Surface engraving

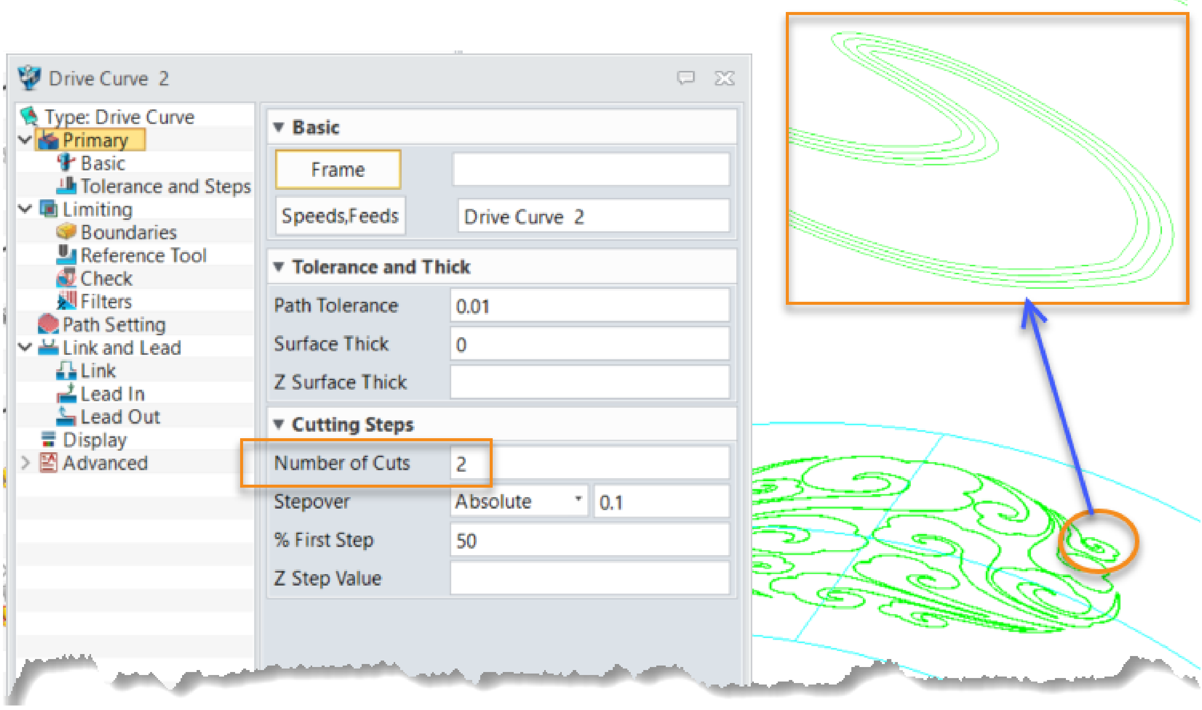


Figure159　　Drive curve engraving

1.4.2 Engraving for text

The method introduced in last section will get a uniform engraving depth, the method is also suitable for creating engraving for text, just as below:

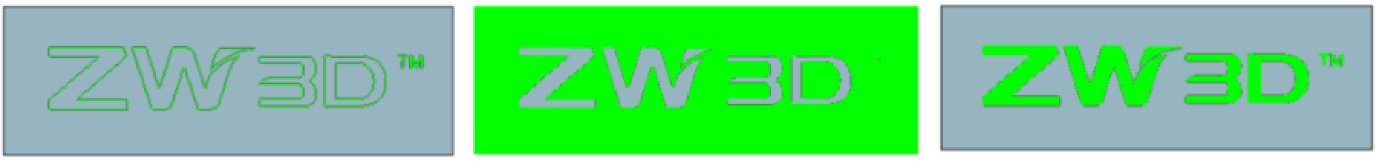


Figure160　　Text engraving

This section will introduce another method to create the engraving tool path for text, the engraving depth is nonuniform which will assign the depth, according to the gap of outline letters. The theory is to make the tool always tangent to the outline of letter.

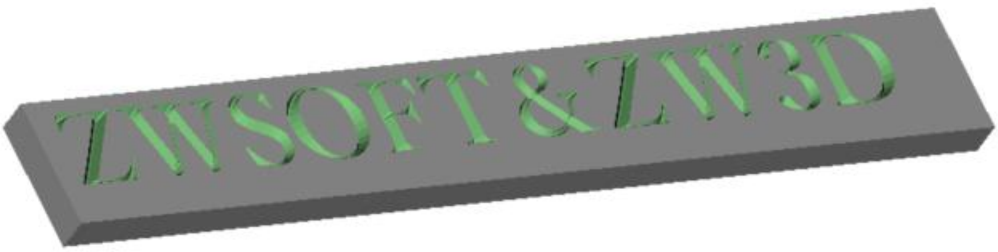


Figure161　　Engrave 2D operation

Steps for create engraving tool path by engrave 2D operation.

STEP 01 Open “Engrave 2D.Z3” file and go into “Engrave_2D_CAM” environment.

STEP 02 Select Engrave 2D operation

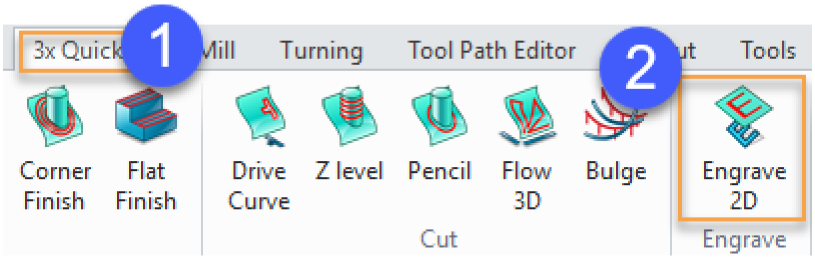


Figure162 Engrave 2D operation example step2

STEP 03 Select all the text curve as the profile and set the type to “part”.

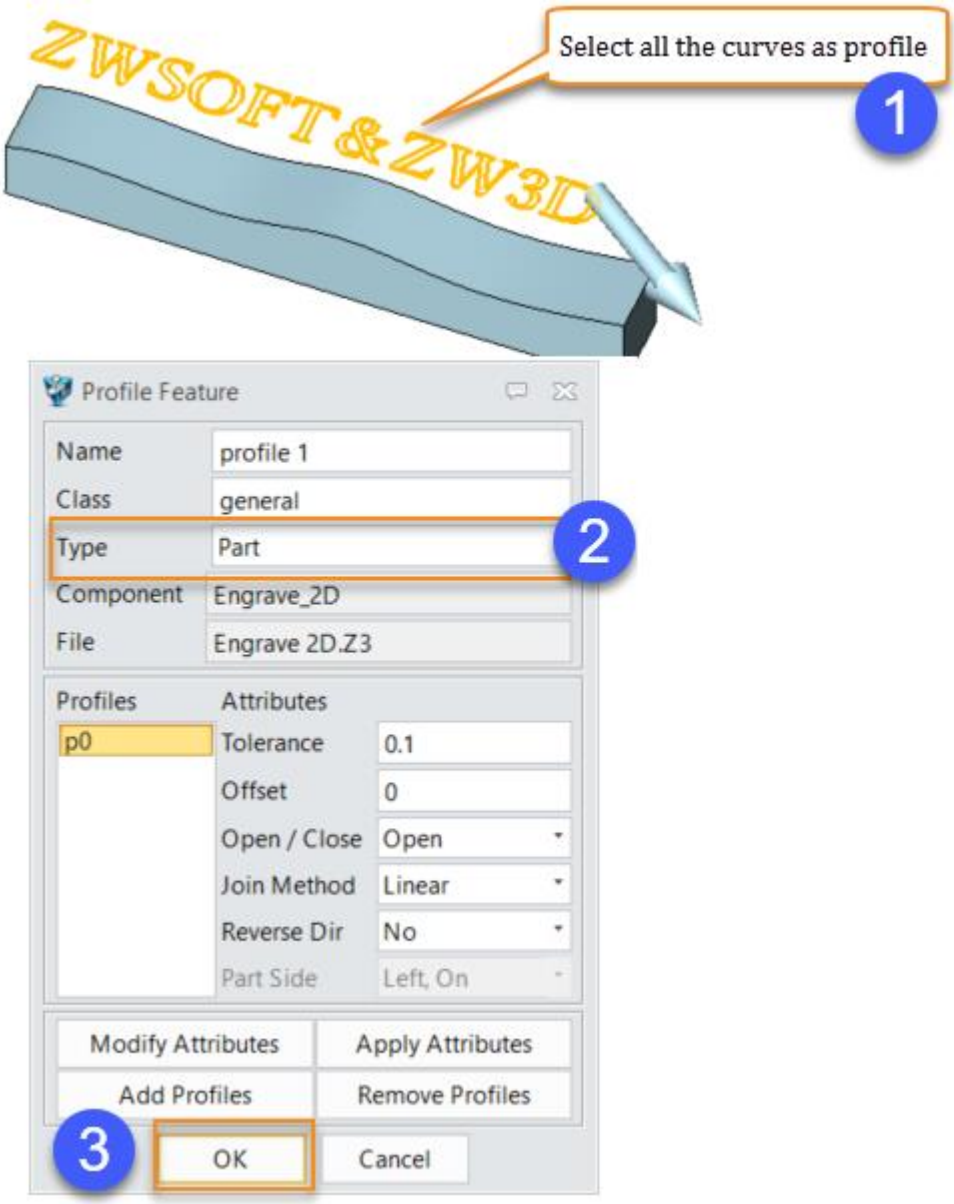


Figure163 Engrave 2D operation example step3

STEP 04 Add the part to the feature

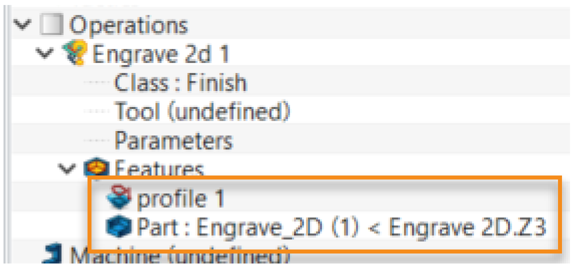


Figure164 Engrave 2D operation example step4

STEP 05 Create a Taper ball end tool, the detailed parameters are as below:

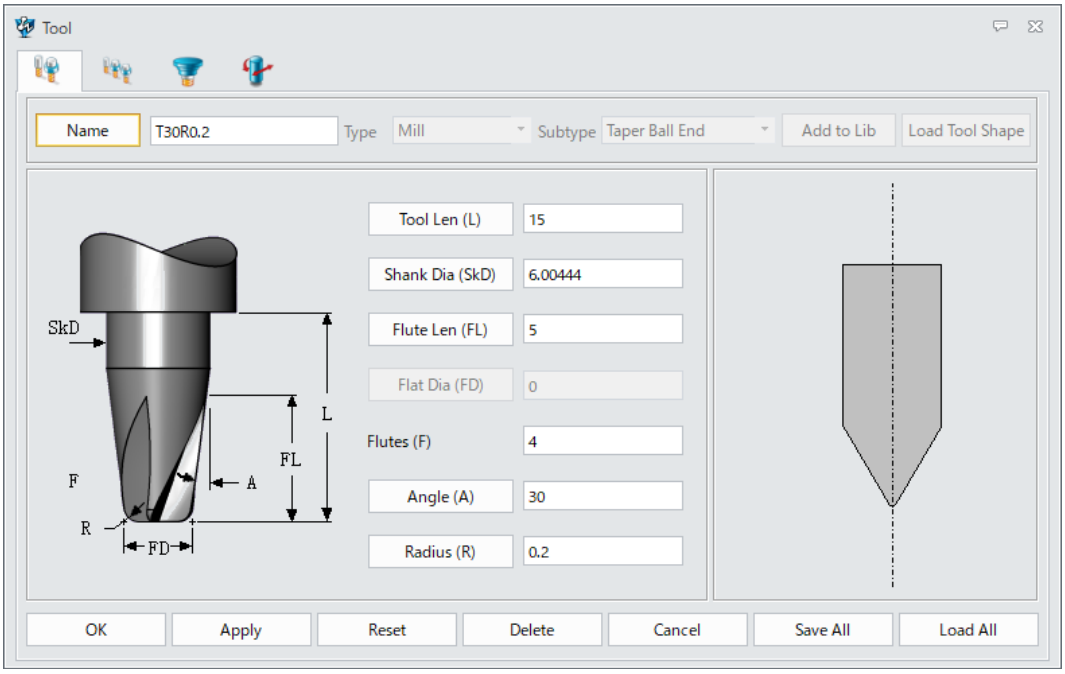


Figure165 Engrave 2D operation example step4

STEP 06 Calculate the tool path (The engraving depth will be decided by system according to the gap of outline letter)

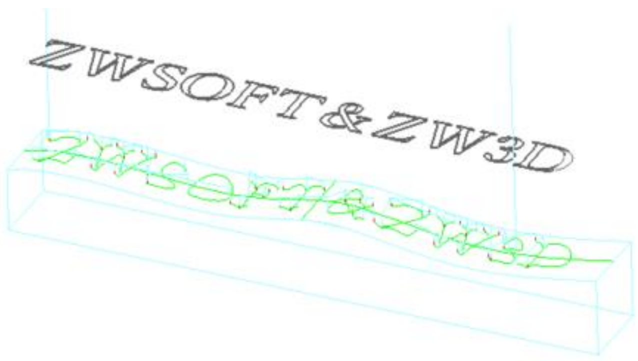


Figure166 Engrave 2D operation example step5

Summary: Tool path generation requirement for engraving 2D operation

- Specify a tool (taper ball end mill is recommended)
- Specify the boundary profile and set the profile type to "part". (The text should be hollow font)
- Add the part to feature

1.5 Tool Path Editor

During machining, perfect part always requires great manufacturing skills and experience to generate proper tool path. But generating proper or accurate tool path as we expected is no easy work, and will take quite a time to adjust the parameter, boundary, auxiliary face/line to get the result. Tool path editor is a more intuitive and easier to use tool, which can save quite a lot of CAM programming time.

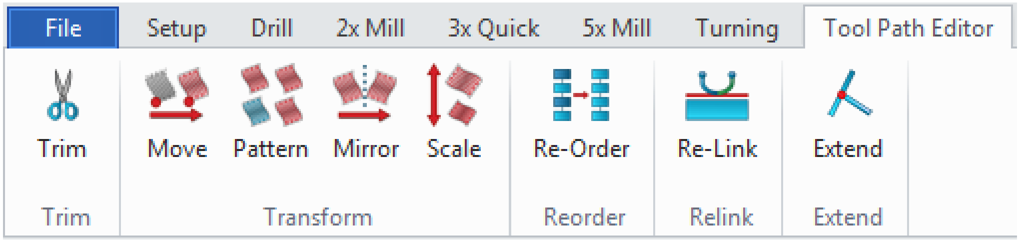
In ZW3D, the tool path editor functions have been classified into modification functions and transfer functions

1.5.1 Tool path modification

Tool path modification will trim unnecessary or improper tool path, relink the tool path, extend tool path and re-order tool path.

- Trim

  Trim function is used to trim the tool path of activated operation. ZW3D provides 3 methods to select tool path for trimming.



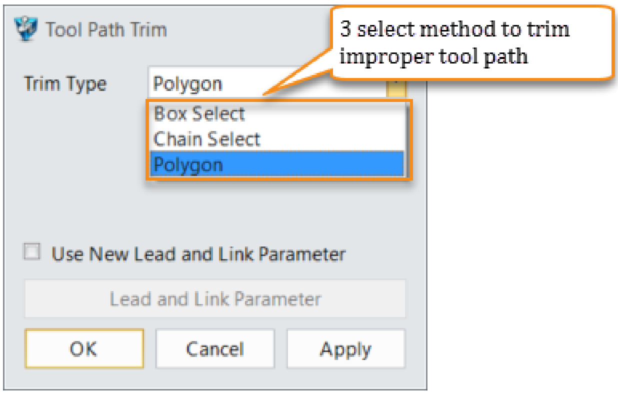


Figure167 Tool path editor -- Trim

Tips: To use trim function, users should activate (select) operation which to trim first, then select trim function.

○ Box select

User can hold mouse left key to drawing a box, all the tool path segments inside or cross by the box will be selected and deleted.

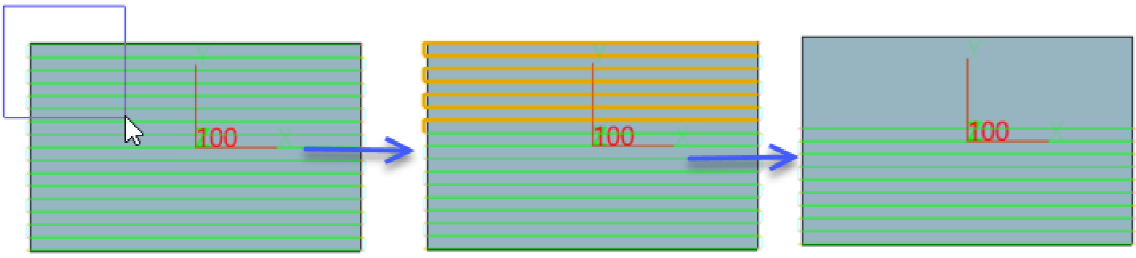


Figure168 Tool path editor – Trim (Box select)

○ Chain select

User can hold “shift” key to select the chain tool path segments to delete.

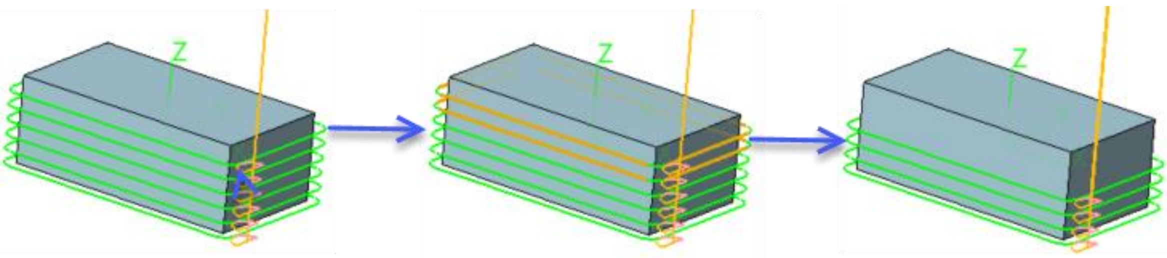


Figure169 Tool path editor – Trim (Chain select)

○ Polygon

This method is different from other two methods, it doesn’t delete the tool path segments. Only the tool path inside the polygon will be trimmed.

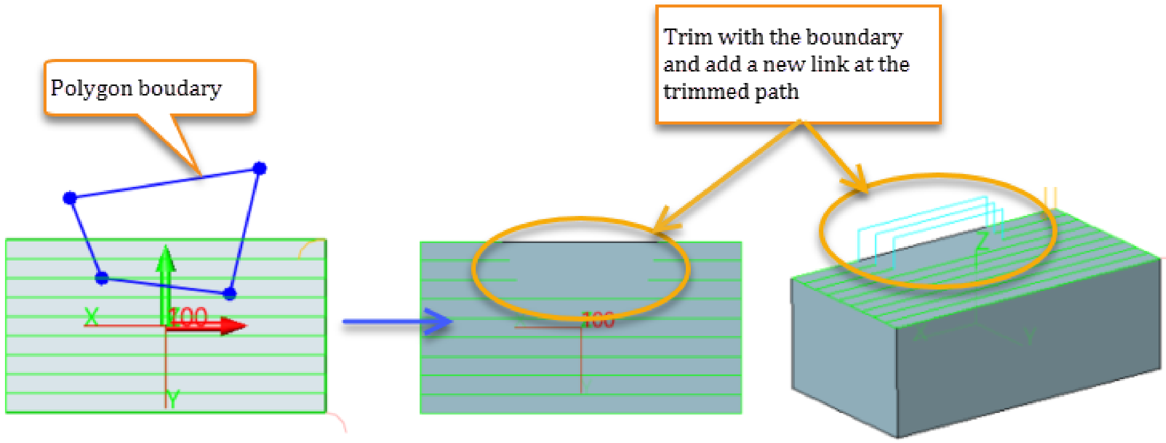


Figure170 Tool path editor – Trim (Polygon)

- Use new lead & link parameter

Check this option to set new lead & link parameters. The parameters are temporary, only available for adding lead & link at trimmed tool path. If this option is off, system will use the same lead and link parameter of activated operation to add new lead and link at trimmed tool path.

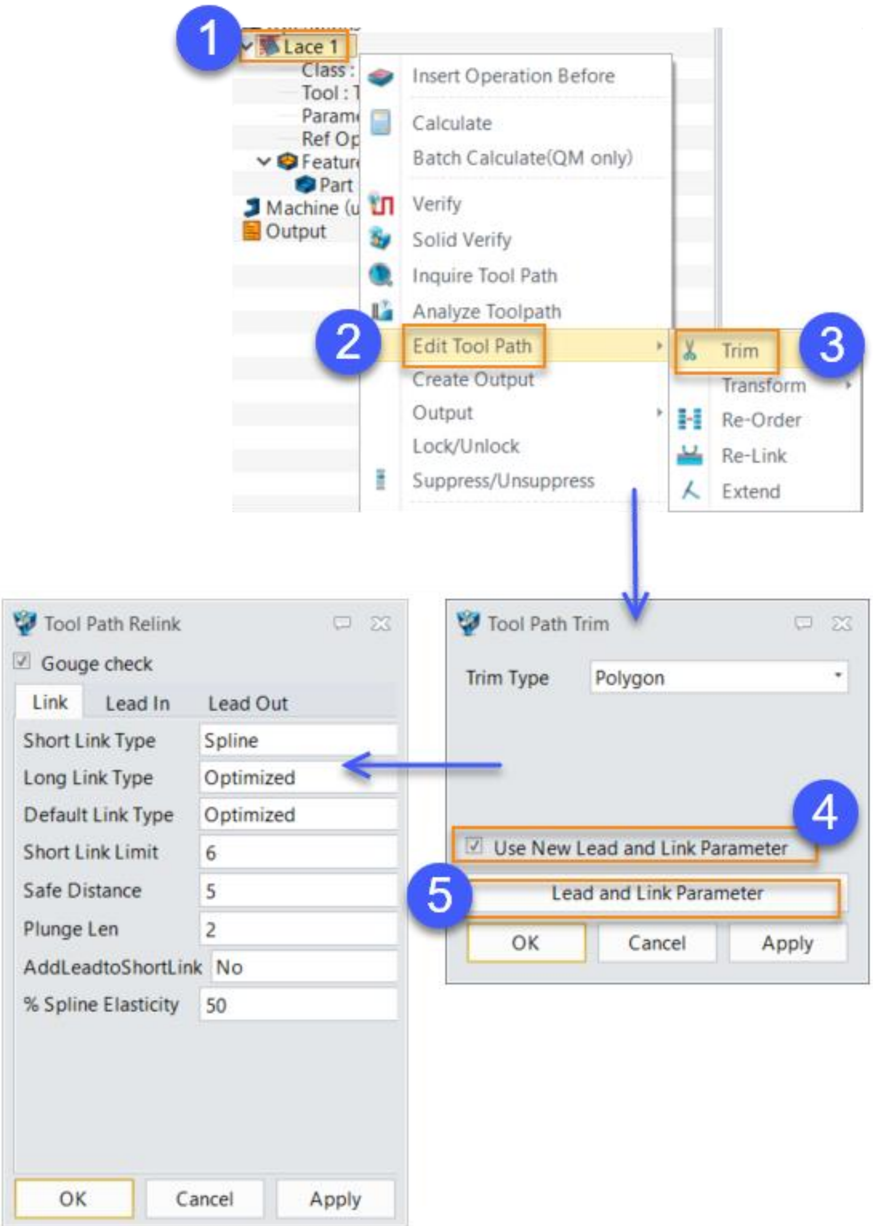


Figure171 Use new lead and link parameter

- Relink

Change all the lead and link of activated operation with new set parameter. Here, ZW3D have 3 places to change the lead and link parameter, below are the differences.

  - In tool path edit > trim

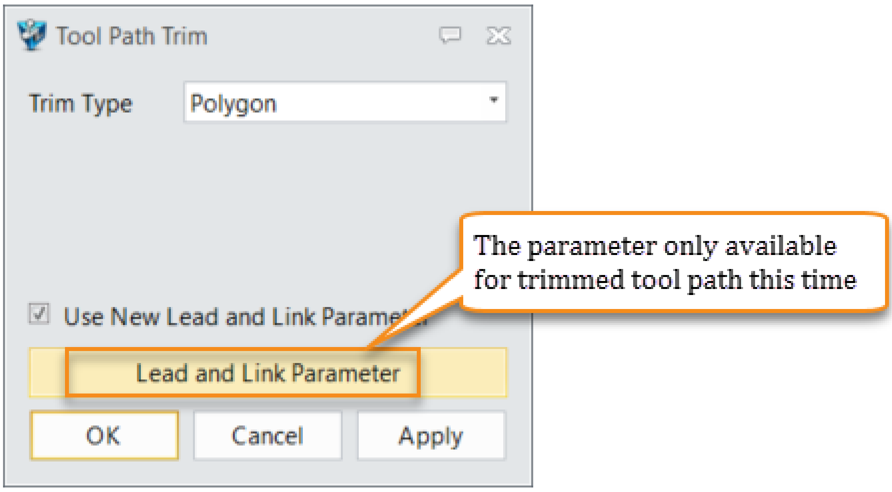


Figure172 Relink in trim

- In relink

It will use the new lead and link parameter to change lead and link in selected operation including the lead & link created by trim function. It would not re-calculate the operation but only change the lead and link

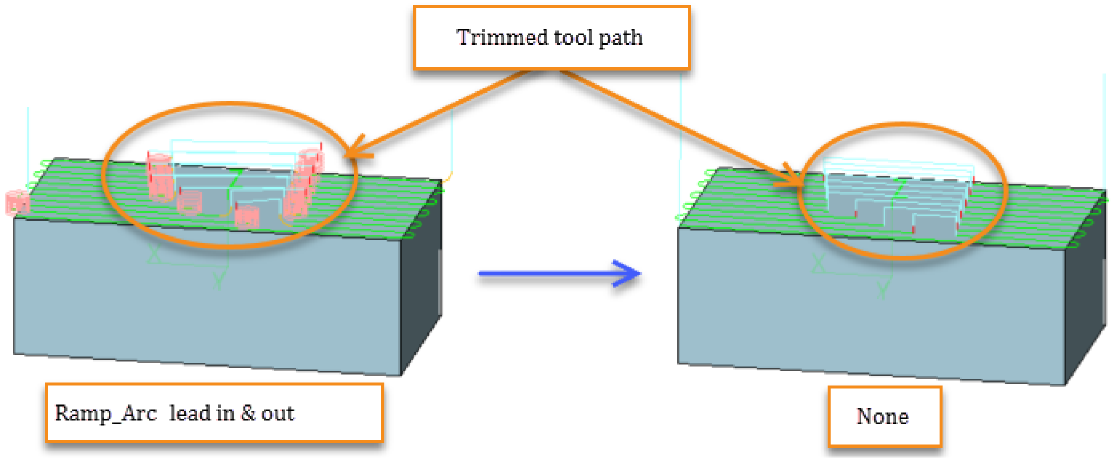


Figure173 Tool path editor -- Relink

- In operation parameters

It will delete all the tool path and recalculate the tool path again according to current parameter.

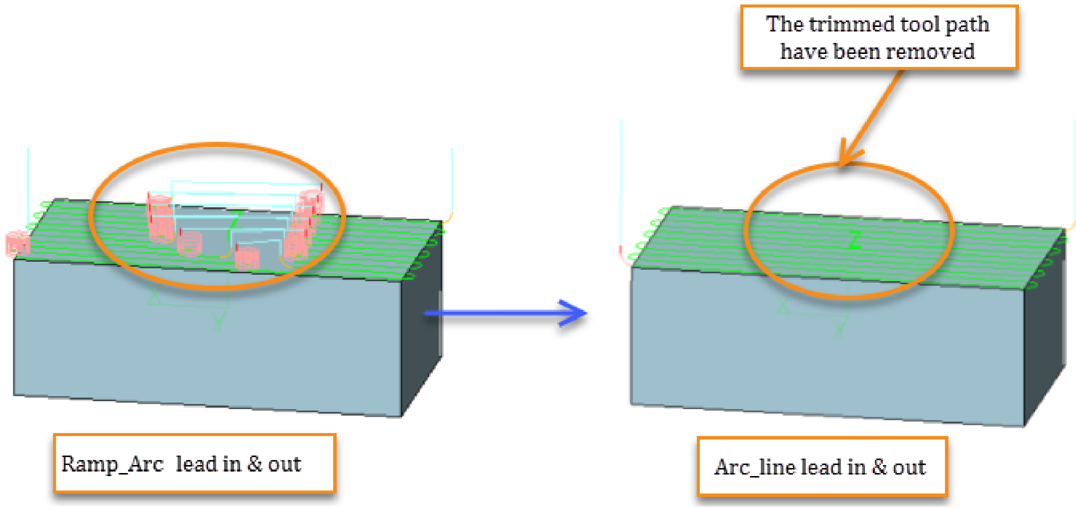


Figure174 Change in operation parameters

- Extend

Extend the tool path with a specified distance by selecting tool path and its short link.

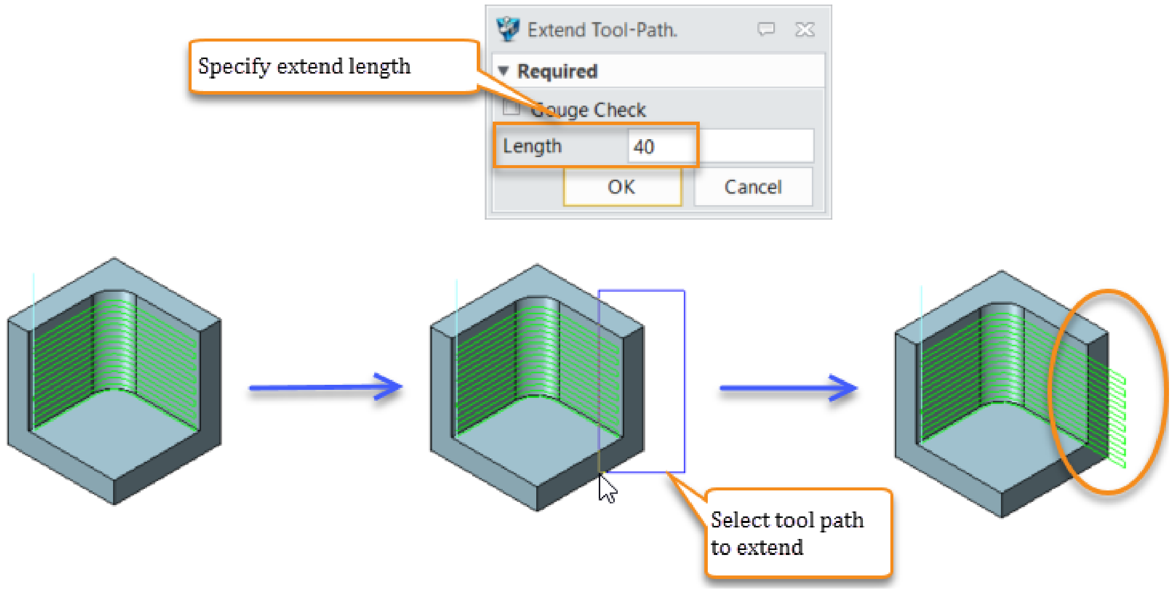


Figure175 Tool path editor -- Extend

Tips: Extend the tool path can be used to ensure a completed machining and protect some sharp edge, or the stock is bigger than part.

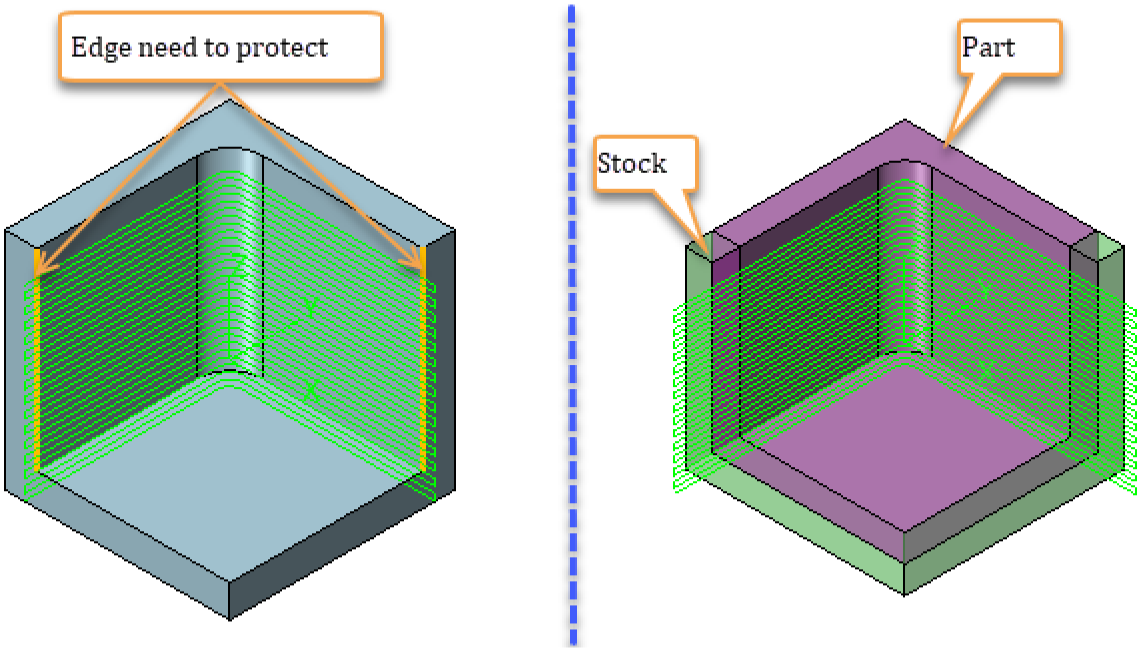


Figure176 Application scenario of Extend

- Re-order

  Tool paths are made up with cuts and Re-Order will list all cuts in the sheet, allowing users to select the target cuts to edit. Re-Order methods include delete cuts, change the order in a region or between regions. We don't recommend new users without rich machine experience to use this function, which will cause tool damage or machine accident.

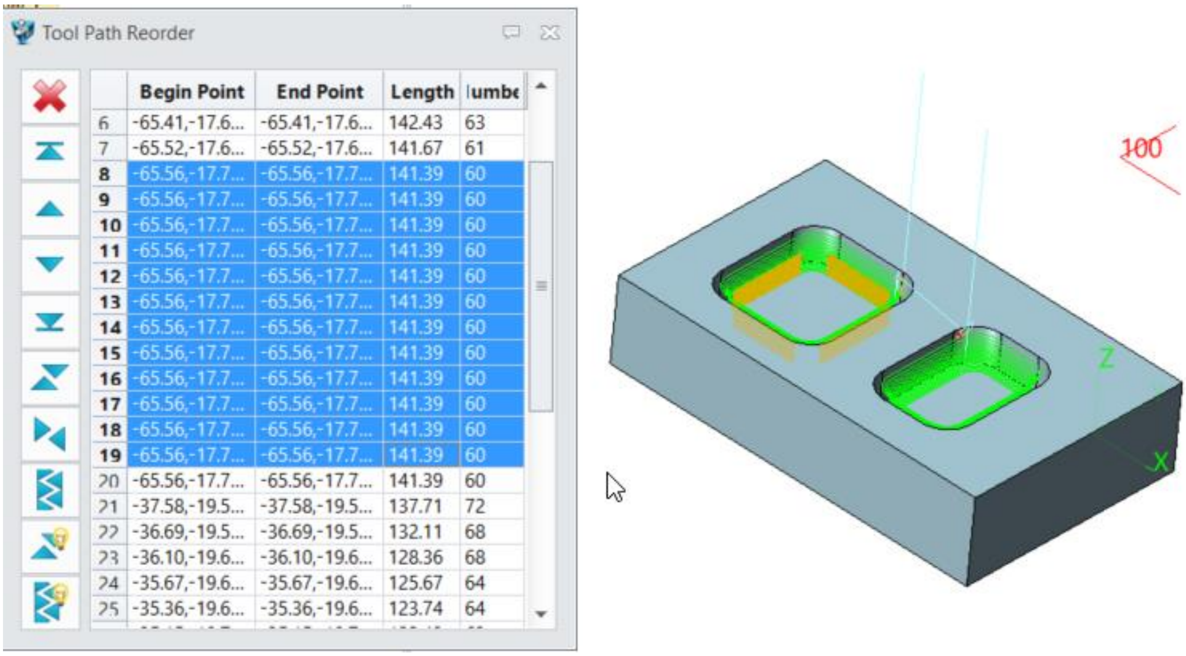


Figure177 Tool path editor -- reorder

Tips:

1) All of those operation (trim, relink, extend and re-order) only effect to current existed tool path, if users re-calculate the operation, the effect of trim, relink, extend and re-order will disappear. Please pay attention.

2) All of those operation (trim, relink, extend and re-order) should select target operation first, then specify the tool path edit function. Otherwise it can’t calculate the tool path.

1.5.2 Tool path transfer

Tool path transfer provides a method to quickly create a serial of similar tool path by move, pattern, mirror and scale. Which will highly speed up our daily work efficiency.

There are two methods to call out tool path transfer function.

1) Create transfer tool path by right menu

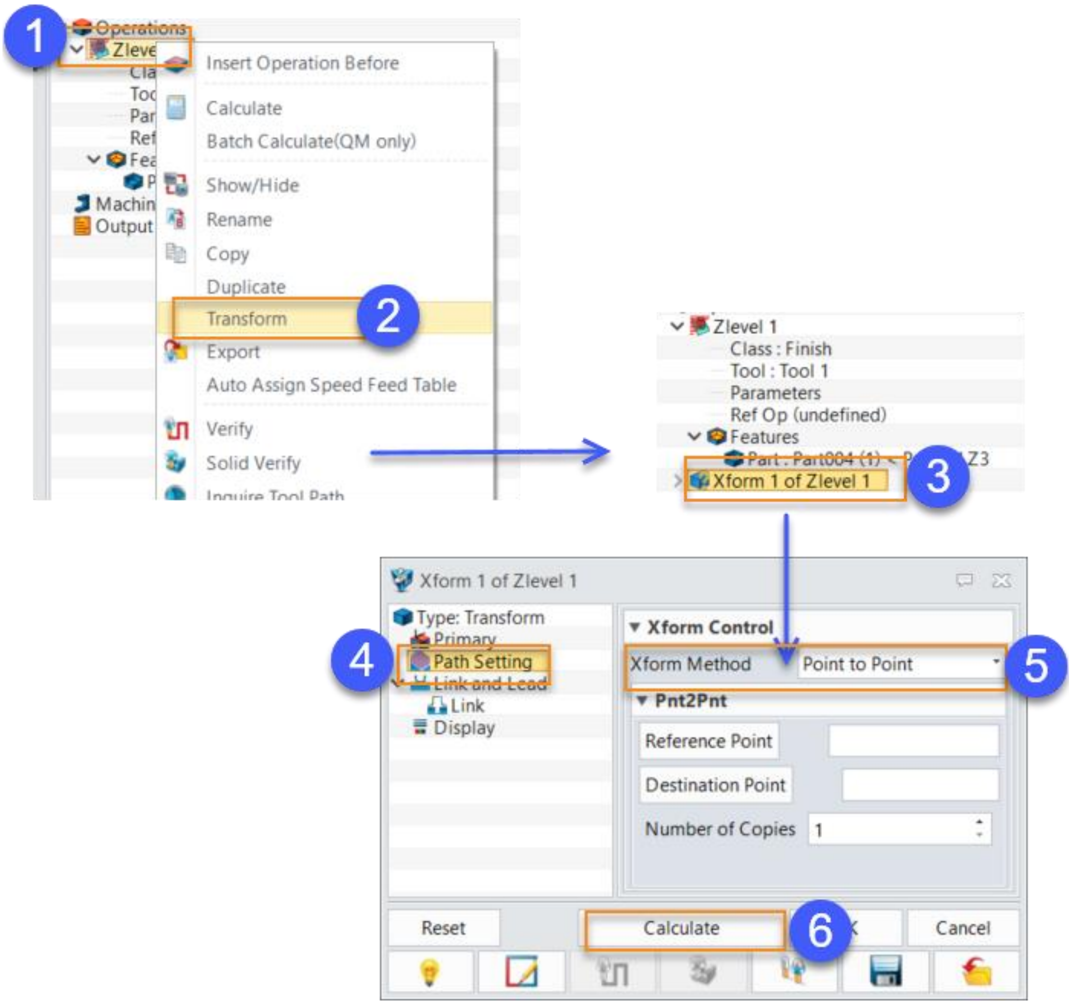


Figure178 Create transfer tool path by right menu

2) Create transfer tool path by ribbon menu

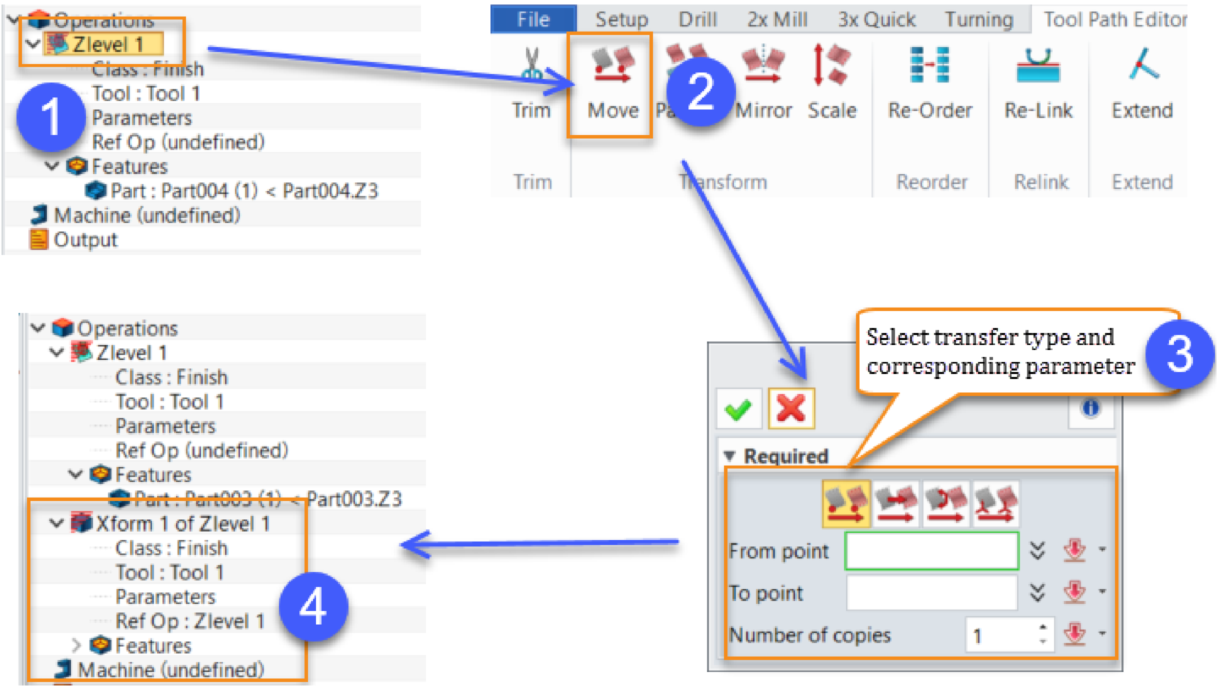


Figure179 Create transfer tool path by ribbon menu

- Move

Move the target tool path point to point, a direction, rotate around a direction or align frame to another frame. Copy number can be specified

  ○ Point to point

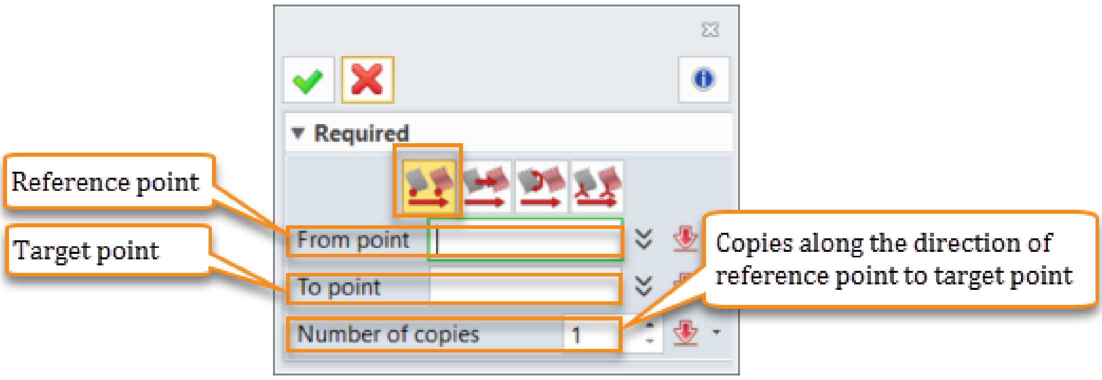


Figure180 Tool path transfer – Move (point to point)

○ Along a direction

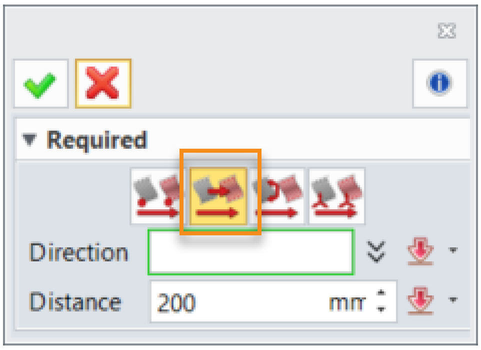


Figure181 Tool path transfer – Move (along a direction)

○ Rotate

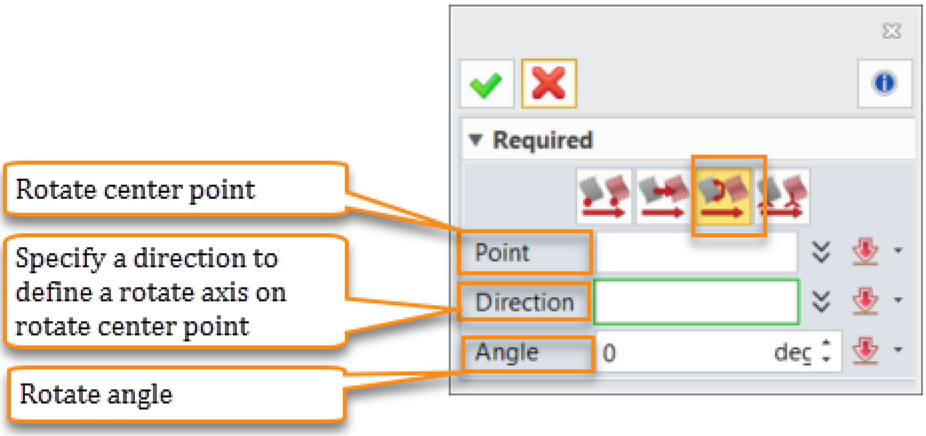


Figure182 Tool path transfer – Move (rotate)

○ Frame to frame

- Plane to plane

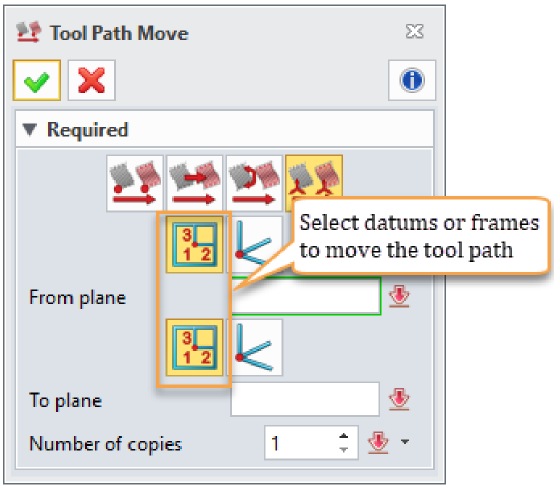


Figure183 Tool path transfer – Move (plane to plane)

- Point and Axis

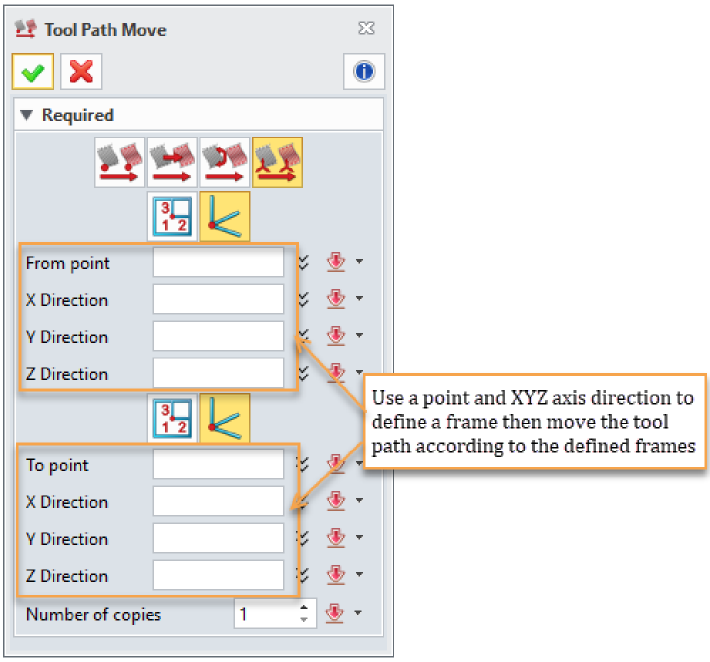


Figure184 Tool path transfer – Move (point and Axis)

- Pattern

Array target tool path by linear pattern or circle pattern

○ Linear pattern

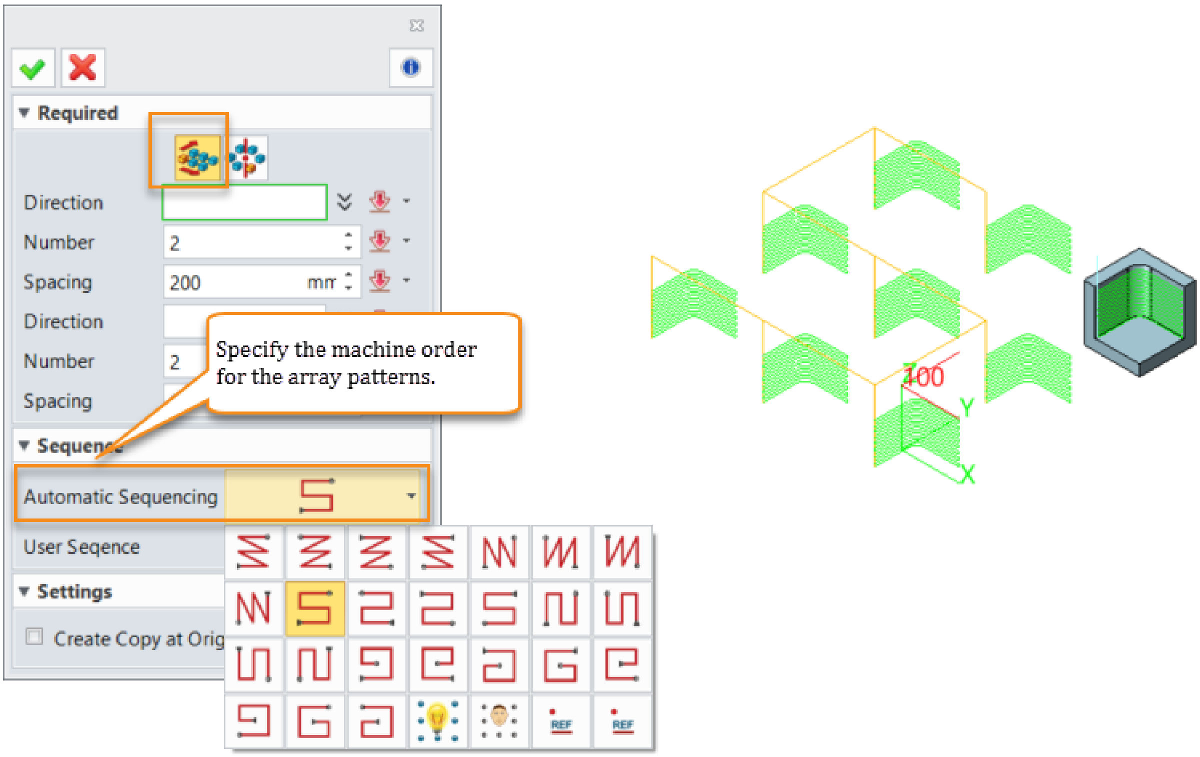


Figure185 Tool path transfer – Linear pattern

○ Circle pattern

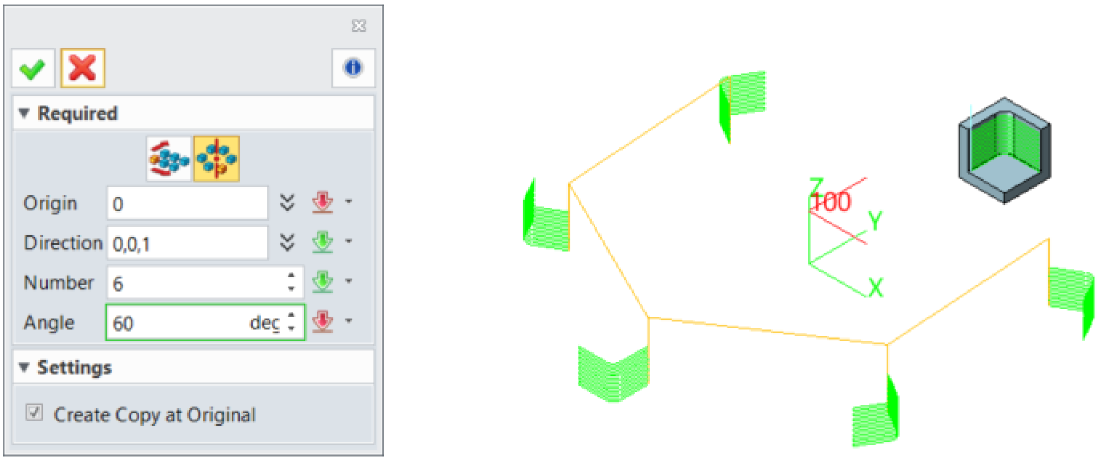


Figure186 Tool path transfer – Circle pattern

- Mirror

  Mirror the target tool path by a selected plane or datum defined by a point with a normal.

- Scale

  Expand or shrink the target tool path by %Scale value.

  ○ By datum

  If the orientation of scaled tool path is the same as the reference tool path, "By datum" is recommended, which is needed to specify a reference datum and a target point.

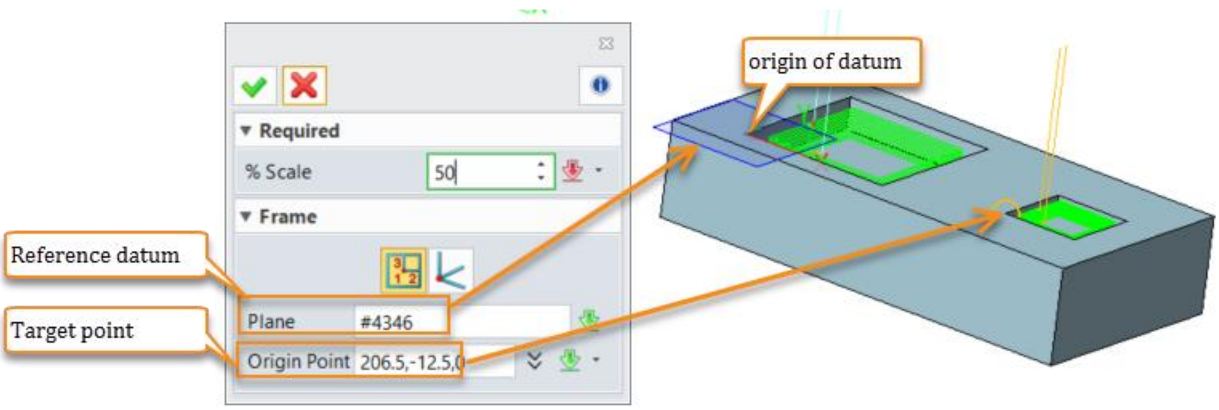


Figure187 Tool path transfer – Scale by datum

  ○ By frame

  If the orientation of scaled tool path is different from the reference tool path, "by frame" is recommended. While the orientation can be defined in fly.

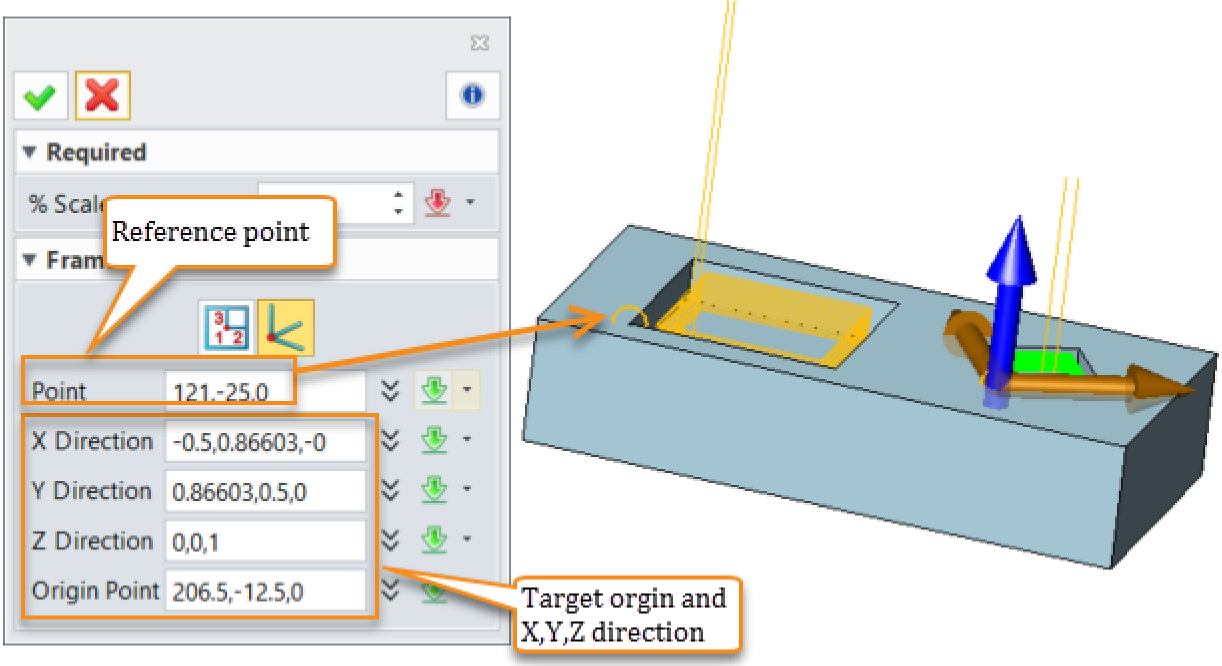


Figure188 Tool path transfer – Scale by frame

Tips : The X direction and Y direction is up down side at the moment, please pay attention to it. And we will enhance it in next version.

1.6 Others

In this chapter, we will introduce some tools or methods to speed up our work efficiency. Such as the QuickMill batch calculation, CAM template and operation library.

1.6.1 QuickMill batch calculation

In some complicated cases, the tool path calculation will be quite a time-consuming work. The normal tool path calculation method will freeze the operation window of ZW3D until the calculation is completed. So ZW3D provides batch calculation method to deal with the complicated tool path, so the users can continue his work without the operation window frozen. When the calculation is finished, users then import to the corresponding operation, which will highly improve our work efficiency.

Tips: The batch calculation only works for QucikMill operation.

Here is the work flow for QuickMill batch calculation.

STEP 01 Open the “Batch calculation.Z3” file and go to CAM space.

STEP 02 Create an offset 2D roughing operation without calculating the tool path, the detailed setting is as below.

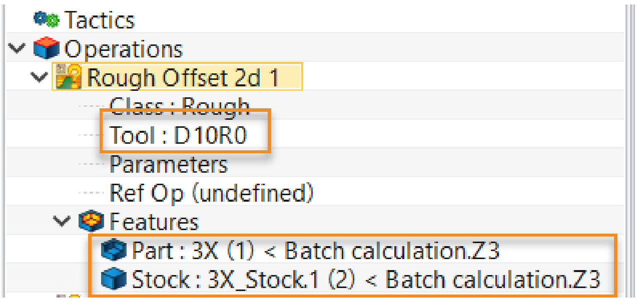


Figure189 QuickMill batch calculation step2

STEP 03 Select this operation and call out the “Batch Calculation” with right menu

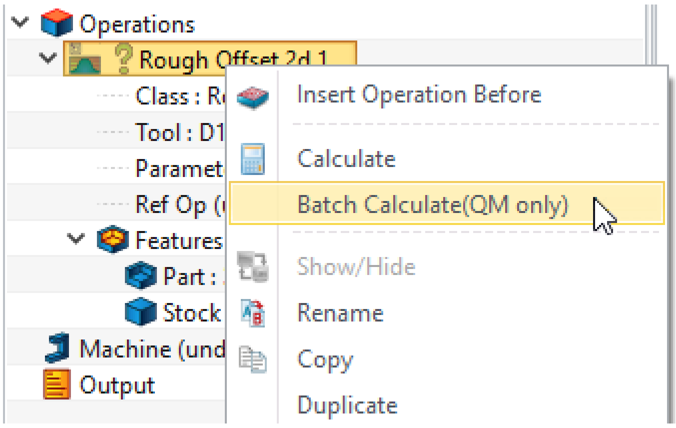


Figure190 QuickMill batch calculation step3

STEP 04 Check the calculation status by 3X Quick > “QM Batch Mgr”. If all the tasks have been completed, import the tool path into operation.

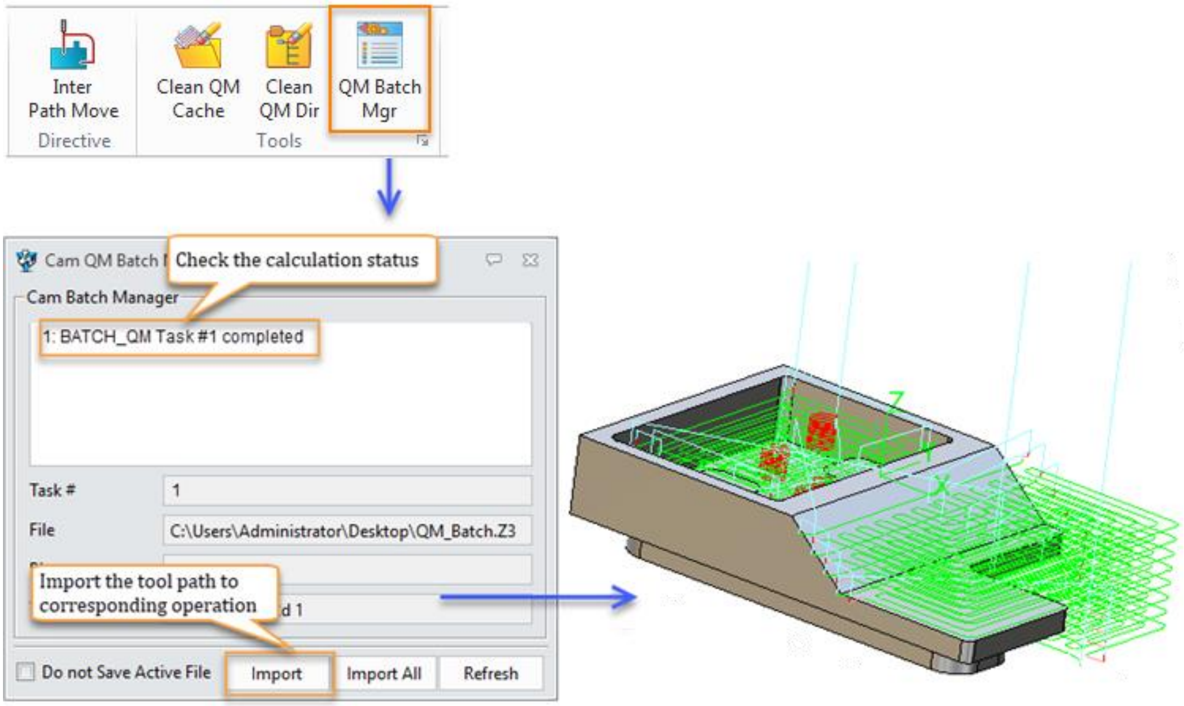


Figure191 QuickMill batch calculation step4

1.6.2 QuickMill Tools

QuickMill use incremental steps to generate tool path, most of the tool path calculation will create some caching files which will be reused in further tool path generation by ZW3D QuickMilling's advanced caching control. The caching control includes tool path generation caching files and STL data (ZW3D QM calculate the tool path base on the STL data), which will save in User folder. But after many calculations, ZW3D may create quite a lot of caching files, User can use “Clean QM Cache” and “Clean QM Dir” to clear those caching files.

- Clean QM Cache

  Clean QM Cache will clean all the caching files except STL data.

- Clean QM Dir

  Clean QM Dir will delete the whole caching folder which is in user folder. Thus all the caching files including the STL data will be deleted.

1.6.3 Cam Template

For some routine machining, the machining condition including the used operation, the parameters in corresponding operations and the tools, can be similar. Only the products have a few differences. In this case, we can create some CAM template to speed up the work efficiency.

How to create a CAM template?

STEP 01 Create a whole CAM plan with all necessary operation have been created and set the machine parameter.

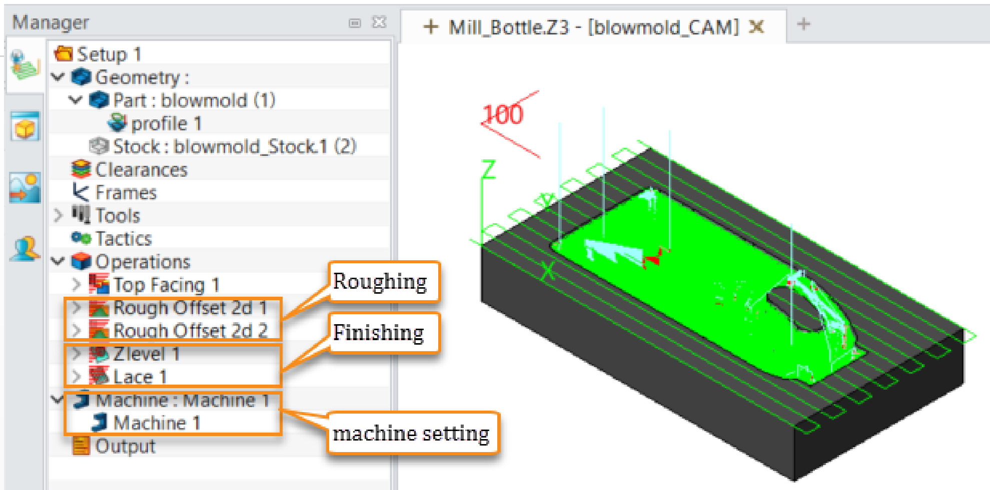


Figure192　　CAM template step1

STEP 02 Delete all the geometry and tool path, then save the file.

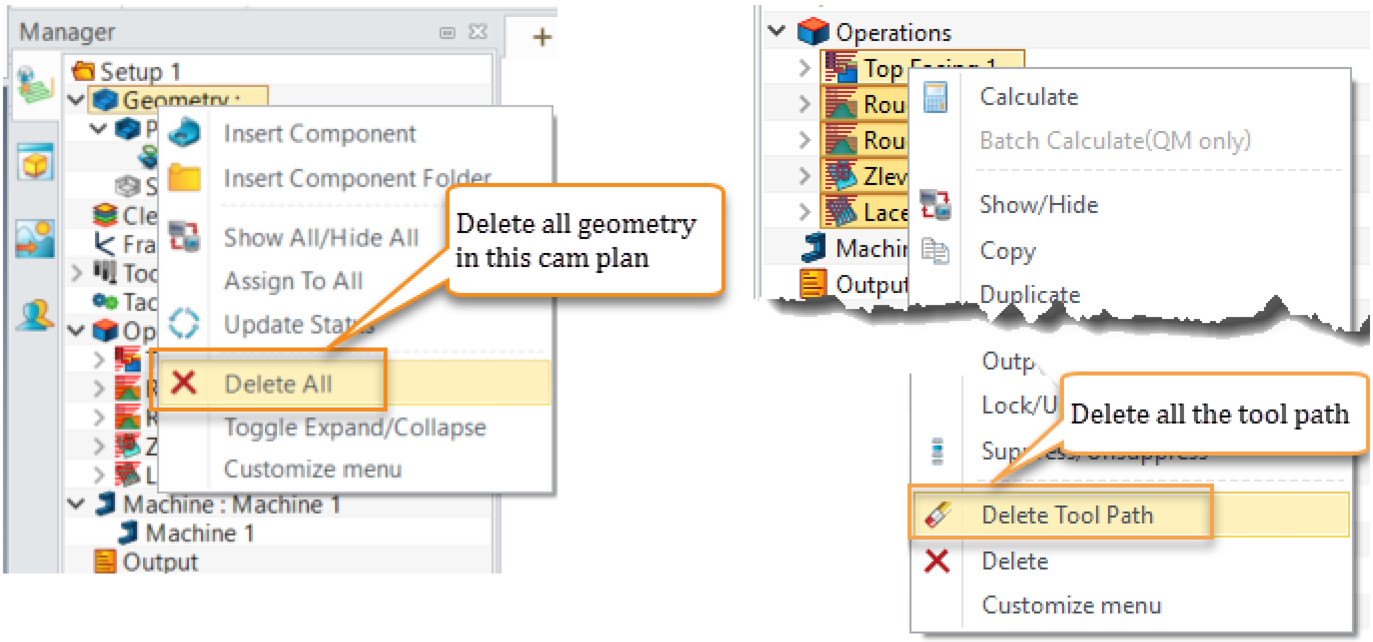


Figure193　　CAM template step2

STEP 03 Open the template library file

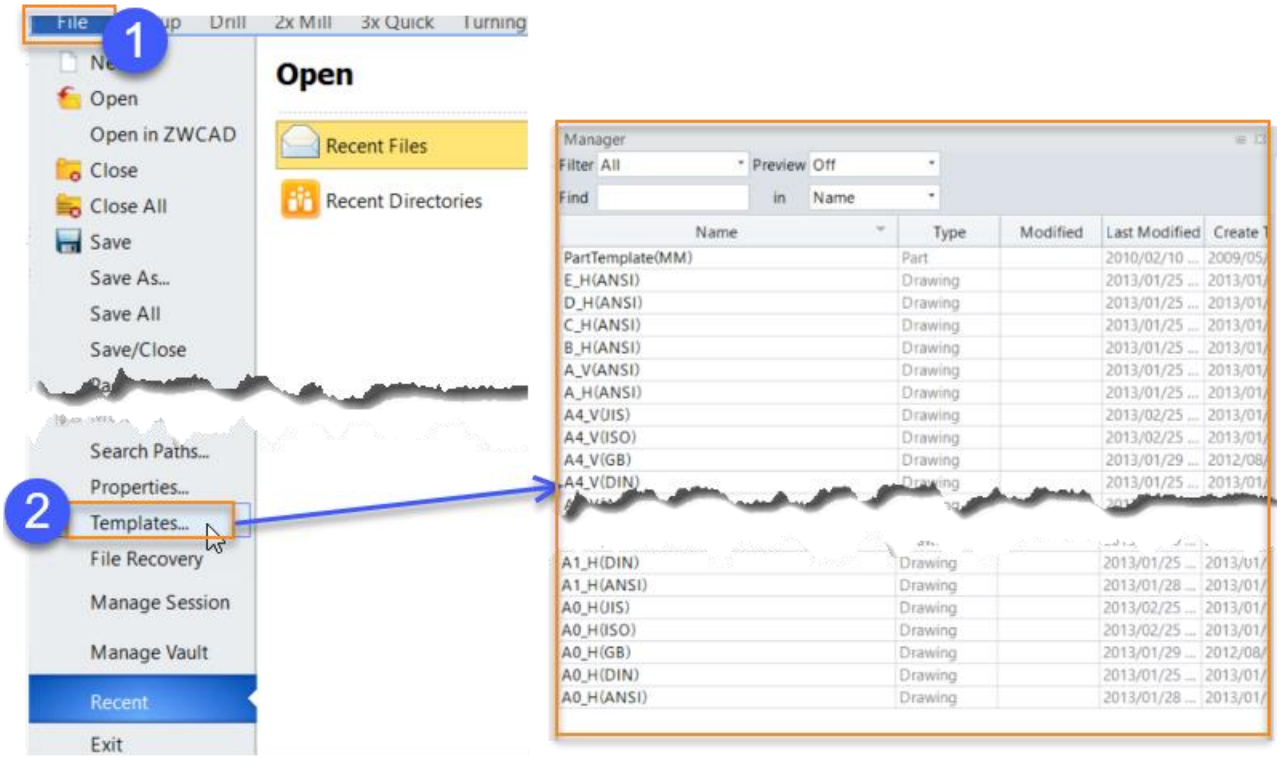


Figure194 CAM template step3

STEP 04 Copy the CAM plan file

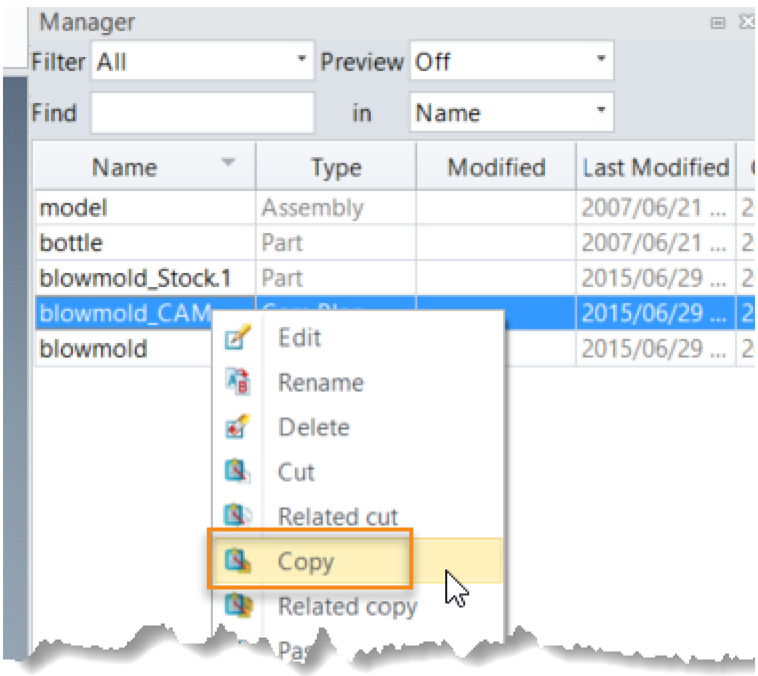


Figure195 CAM template step4

STEP 05 Paste the CAM plan file to template library file by “Ctrl +V”, and save the library file.

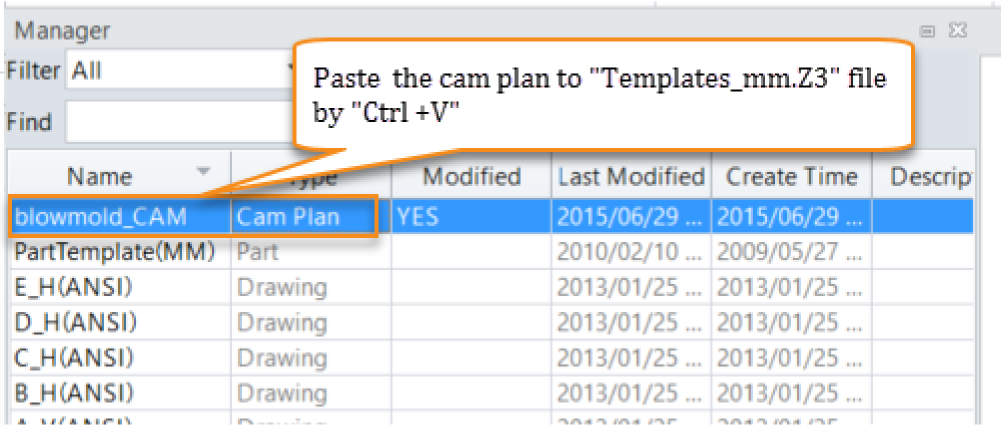


Figure196 CAM template step5

How to reuse the CAM template

STEP 01 Open the model which is needed to create CAM programming.(CAM_TM_Model.Z3)

STEP 02 Click the entry to go to the CAM space and select the corresponding template.

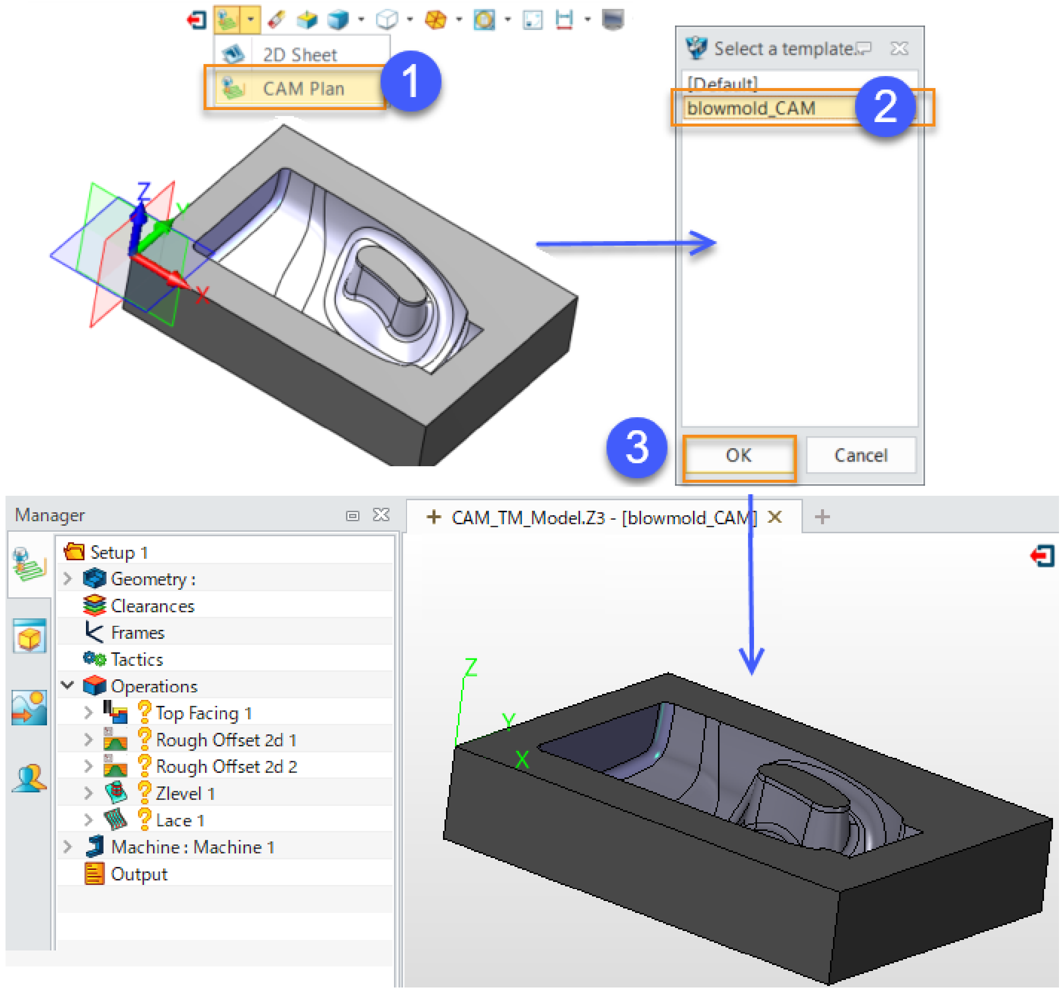


Figure197 Reuse cam template step2

STEP 03 Add stock for this model

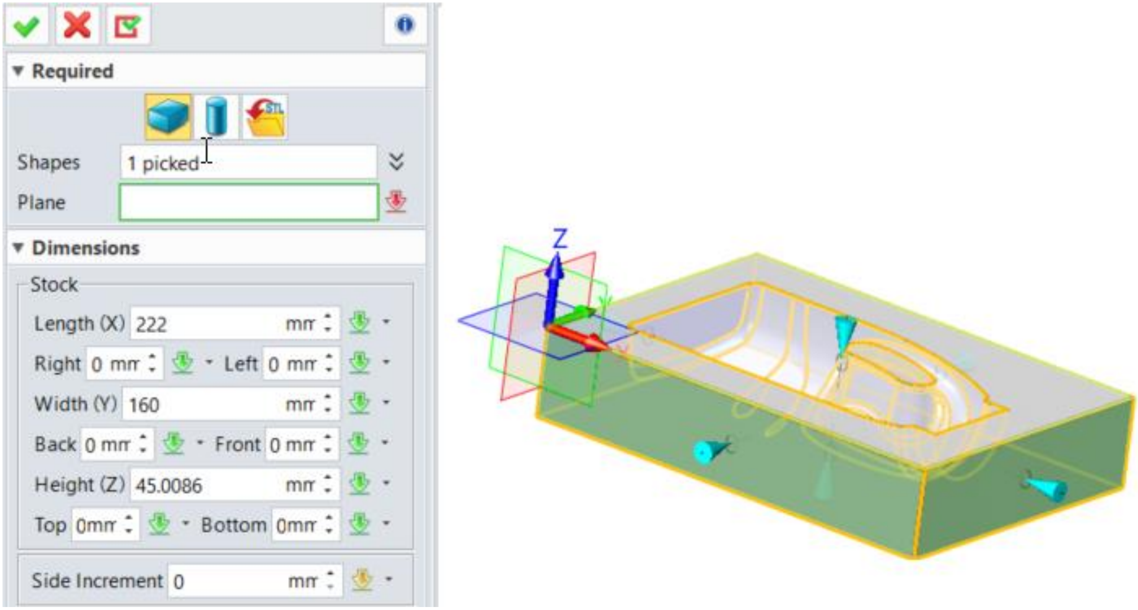


Figure198 Reuse cam template step3

STEP 04 Redefine the feature in all operations

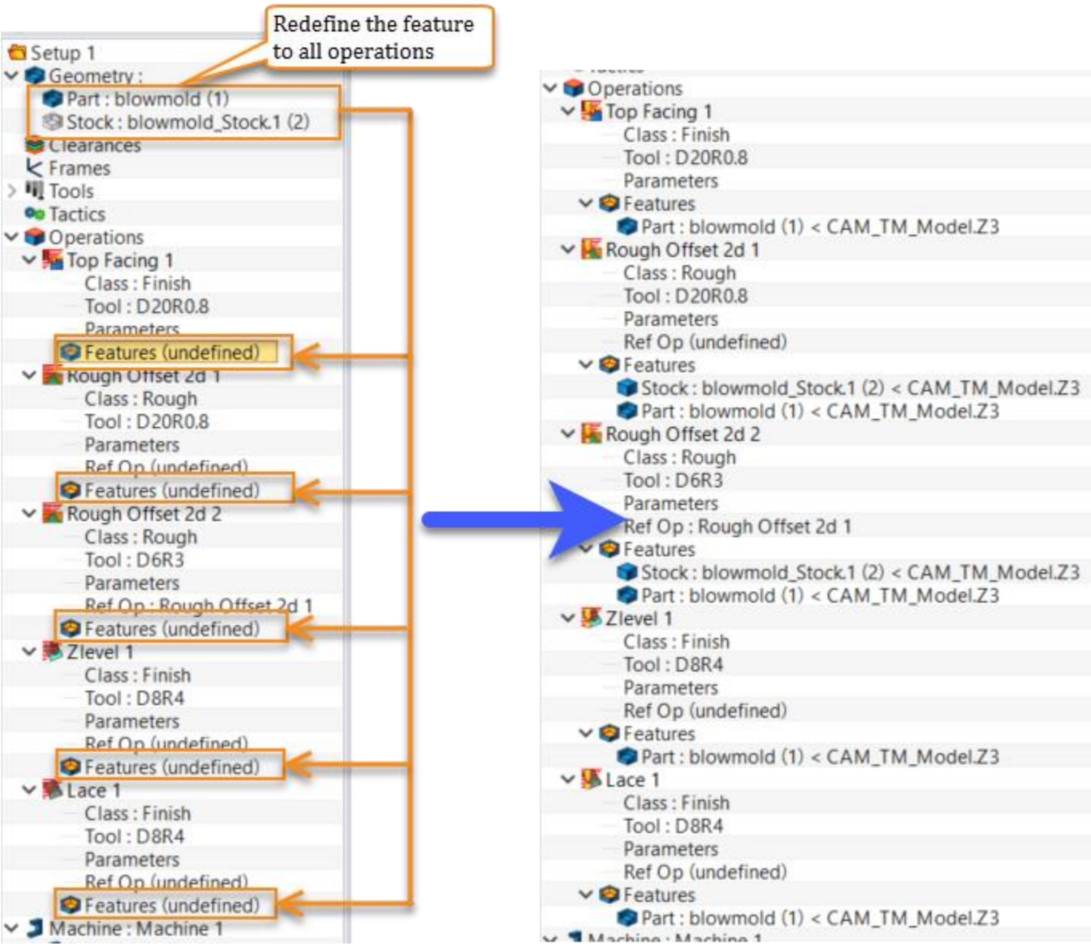


Figure199 Reuse CAM template step4

STEP 05 Recalculate all the operations, then all works are done.

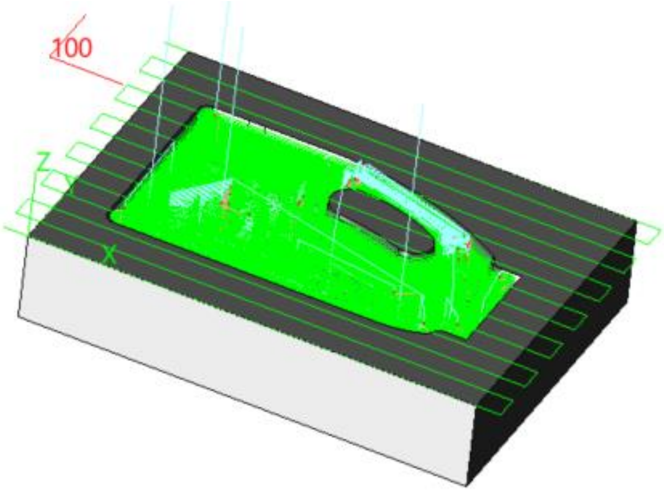


Figure200 Reuse cam template step5

1.6.4 Operation library

Since ZW3D allows users to create their own operation library to improve our work efficiency.

How to create an operation library？

STEP 01 Create a new CAM plan file in template library file.

STEP 02 Open the newly created CAM plan file

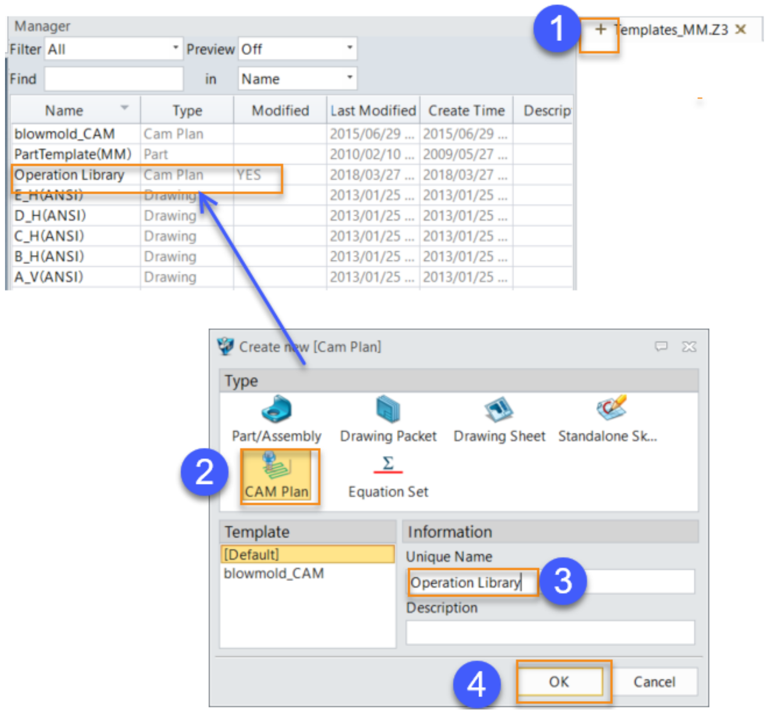


Figure201 Cam operation library step2

STEP 03 Paste all the operations you want to save to this CAM plan file. Except features (machining object or boundary) all the machining parameters can be saved, including the tool, spindle, feed rate, step over, step down and so on.

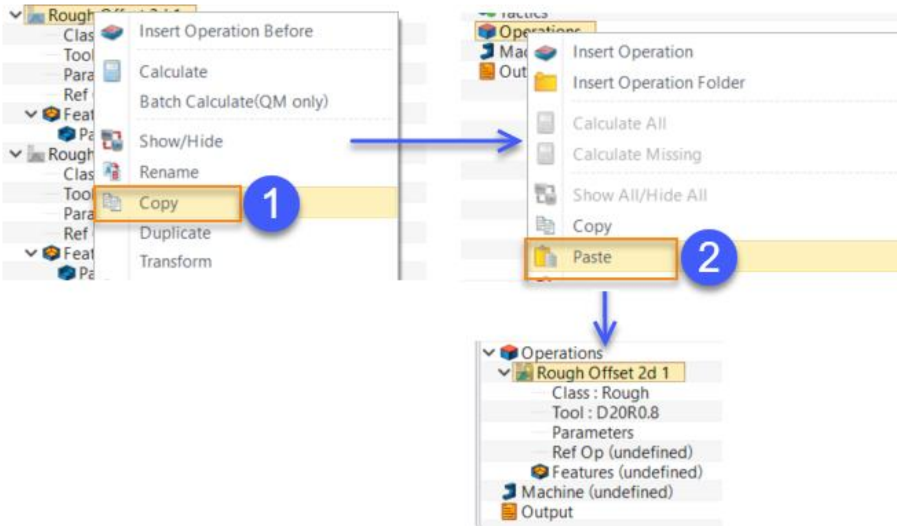


Figure202 Cam operation library step3

STEP 04 Save the template library file

How to reuse the operation library？

STEP 01 Open ZW3D and go to the configuration to specify the saved operation library.

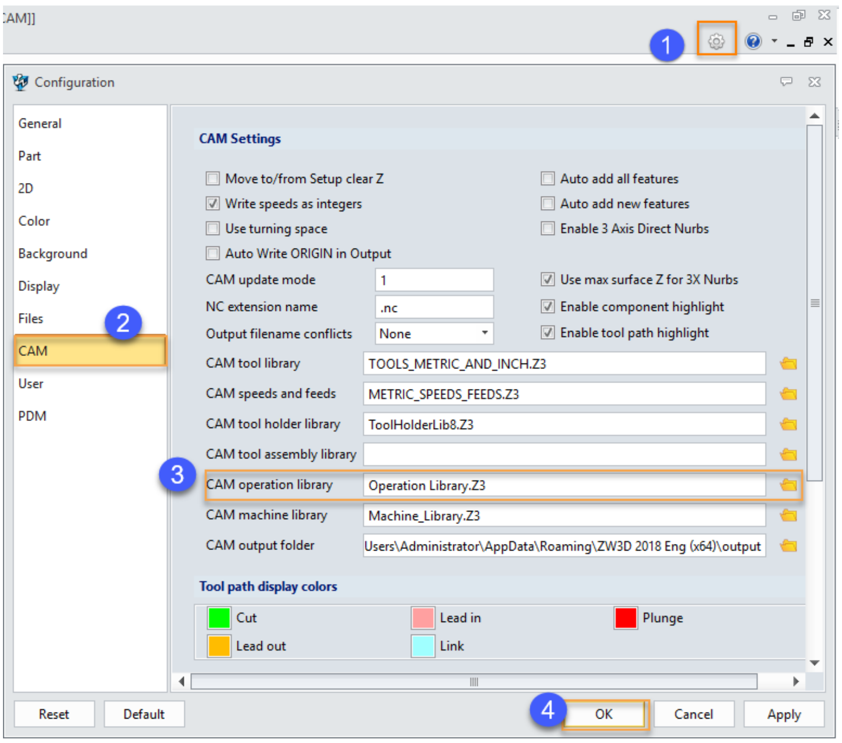


Figure203 Reuse CAM operation library step1

STEP 02 Right click on the operation tree and use import function to call out the operation.

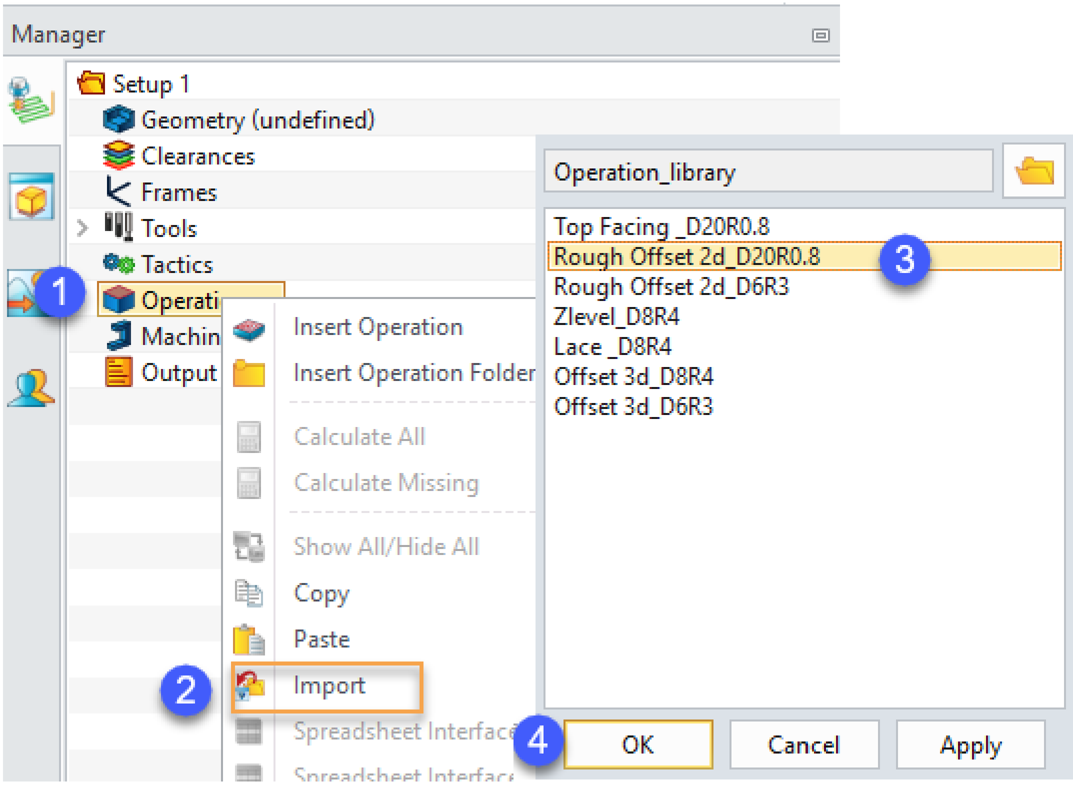


Figure204 Reuse CAM operation library step2

STEP 03 All parameters which have been saved in the library will be called out.

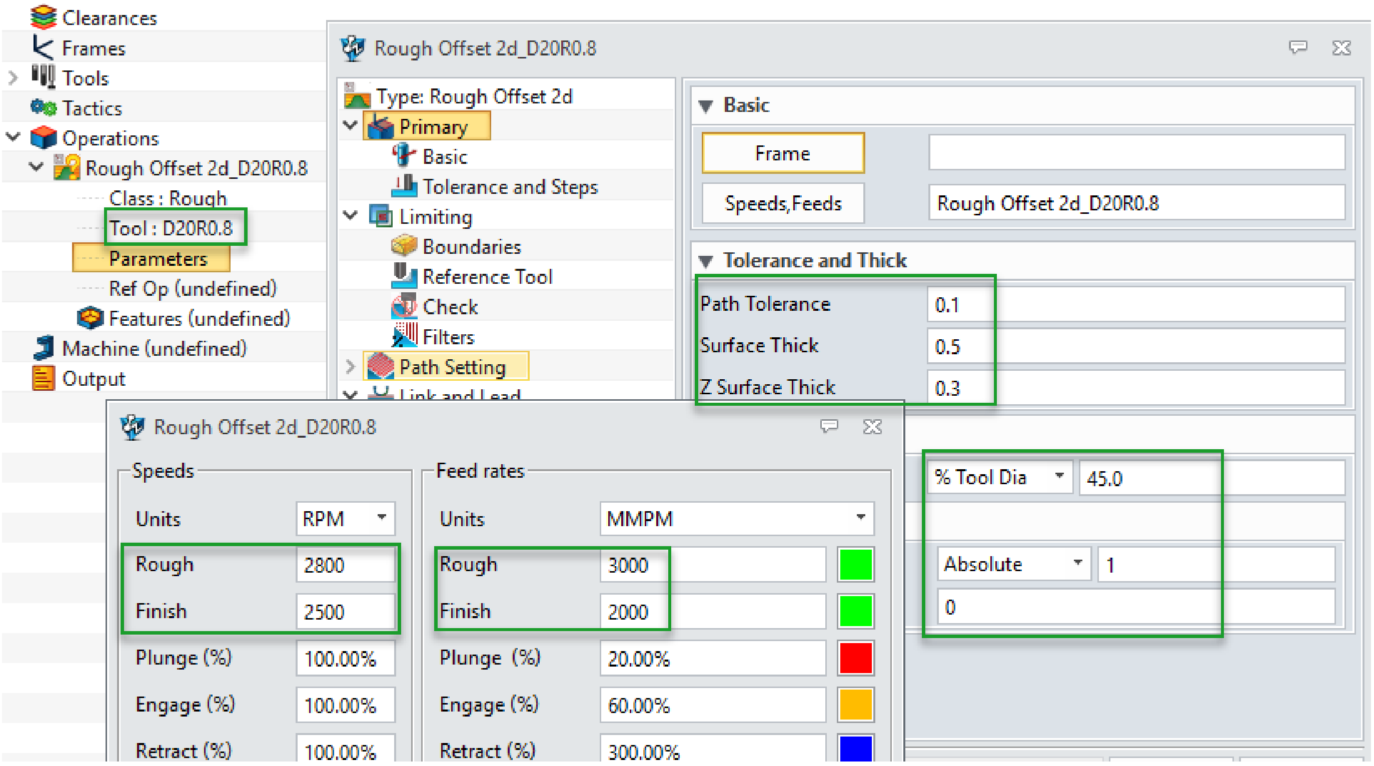


Figure205 Reuse CAM operation library step3

Epilogue

Thank you for your valuable time.

In this tutorial, we've shown you how to create 3X toolpaths with the key parameters explained in detail. We hope this tutorial can help you understand the way to apply 3X machining in ZW3D.

Notice: This tutorial is based on version ZW3D 2019, some functions or icons may not match the current version. If you have any suggestions or questions about this tutorial, please contact us at

ZW3D Global Website: https://www.zwsoft.com/zw3d/

ZW3D Forum: https://www.zwsoft.com/forum/forum-18.html

ZW3D Support Team: zw3d@zwsoft.com