ZW3D from Entry to Master Tutorial

3X Machining

ZW3D™ V2023 From Entry to Master CAM 3X Machining

Printed in the P. R. China.

ZWSOFT CO., LTD.(GUANGZHOU)

Room 01-08, 32/F, No.15, Zhujiang West Road,

Tianhe District, Guangzhou 510623, China

(8620)38289780

Foreword

In this tutorial, we provide various case studies, which are from easy to difficult and combine theory with practice. We hope to improve users' 3D CAD/CAM skills and techniques with ZW3D.

The tutorial bases on our technical engineers' years of experience in the industry and ZW3D, which is the fruit of a lot of efforts and wisdom. We sincerely hope that the tutorial will do help to you, and your precious advice on it is highly welcomed.

There are three series for this tutorial: ***Primary Tutorial***, ***From Entry to Master Tutorial***, and ***Advanced Tutorial***. From easy to difficult, they offer a step-by-step learning process that can meet different user needs.

Primary Tutorial series is for users who have little or no prior 3D CAD/CAM experience. If you are green hands of 3D CAD/CAM software, or if you are a new user of ZW3D, we recommend that you get started with this tutorial. Here you can learn the basic knowledge and concepts of ZW3D, rapidly master the simple operations and workflows of ZW3D, and practice simple cases.

From Entry to Master Tutorial series is for users with basic know-how of 3D CAD/CAM software. If you have experience in 3D CAD/CAM software and want to master common functions of ZW3D, we suggest that you start with this series. Here you can dig deeper into the functions and master more operations of ZW3D.

Advanced Tutorial series is for users with practical experience in 3D CAD/CAM software. If you hope to have a comprehensive command of ZW3D and get the complicated operations done independently, you can choose to learn this series. Here you can learn to use the software more flexibly and get rich experience to increase your efficiency.

What you are learning is ***ZW3D From Entry to Master CAM 3X Machining***, a master tutorial.

Thanks for being our user!

The ZW3D Team

Contents

This tutorial is used for mastering 3X milling. The most important and frequently used operations and their parameters will be introduced here. Besides, it will share some skills/tips to help you get familiar with ZW3D 3X machining.

Key Points:
- ✧ Roughing toolpaths generation & setting
- ✧ Rest roughing
- ✧ Finishing operations introduction & generation
- ✧ Tool Path Editor
- ✧ Efficiency tools

Notes:
1. *This tutorial is based on ZW3D 2021 version, some functions or icons may not match the current version.*
2. *All the tutorial models can be found in the installation folder: …\training\3X machining model*

1 3X Milling Introduction

Generally, when you start a new 3X milling programming, the first step is to evaluate the manufacturing model so that you will have a general idea on how to simplify a complicated model, and then define suitable roughing, rest roughing, semi-finishing, finishing operations to generate toolpaths. The toolpaths should be verified to avoid workpiece damage and to ensure high-quality machining. Finally, specify a suitable post processor to translate the toolpaths to GM code for manufacturing. The workflow in ZW3D 3X milling is shown below.

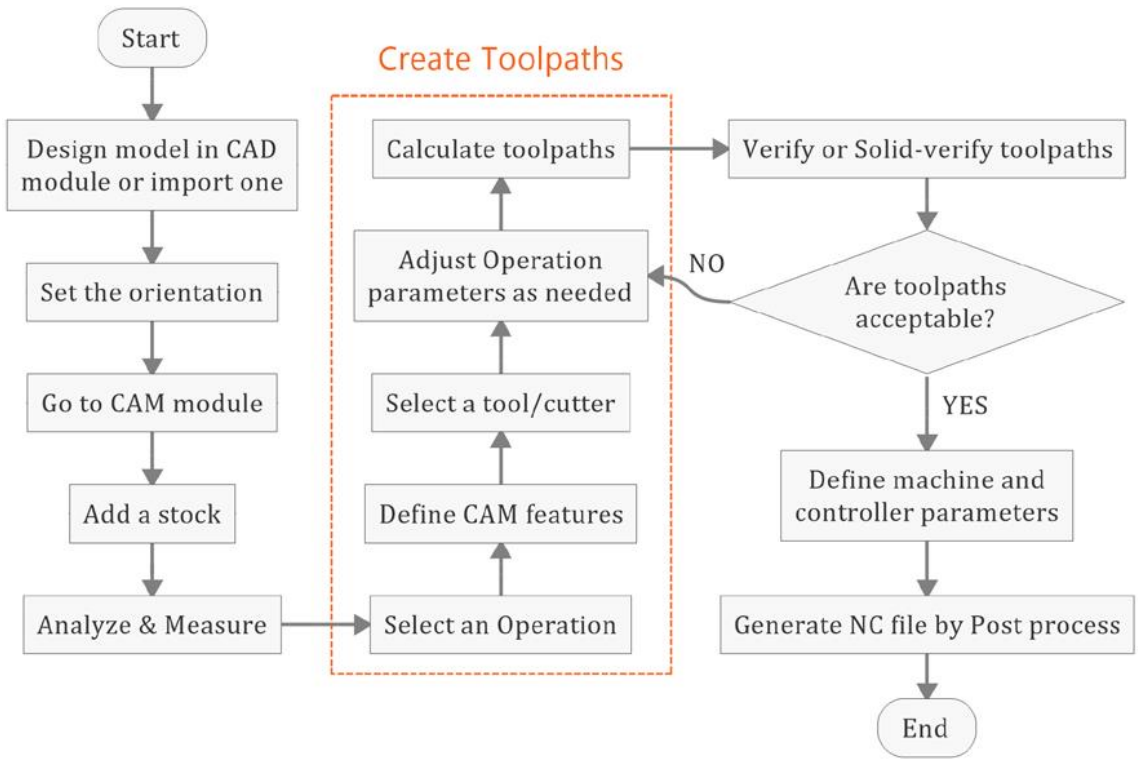


Figure 1 3X Workflow

2 Roughing

Roughing is used to quickly remove the material from stock to part, which sets a large stepdown with a large cutter to improve manufacturing efficiency. Of course, we need to leave a uniform rest material for the next process. ZW3D provides 4 types of roughing operations to satisfy different machining situations.

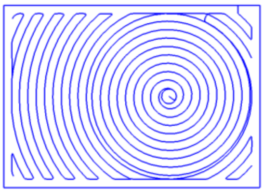
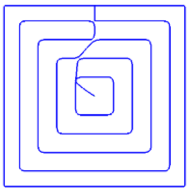
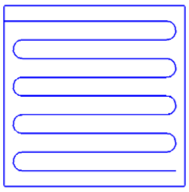
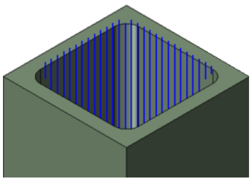
Operation	Pattern	Scenarios
		***Smooth Flow*** generates the toolpaths with a spiral curve from the center to the outer. It is suitable used to remove hard material, which generates uniform chip-load toolpaths as possible.
		***Offset 2D*** generates a set of offset patterns based on the contours of the stock, or part, or both in area clearance regions. It can be used for almost all kinds of materials.
		***Lace*** companies of a group of straight-line moves in the horizontal plane. It is usually used to remove soft material.
		***Plunge*** is an operation with a group of vertical plunging movements. It is used to generate toolpaths in some large models. ***Note:*** *It needs a specialized cutting tool and CNC machine to work with it.*

Figure 2 Roughing operation

2.1 The Requirement for Generating Roughing Toolpaths

To generate roughing toolpaths successfully, there are three essential steps:

- Select roughing operation
- Specify tool according to the size of the machinable feature
- Specify Feature (part, and stock or profile)

Task: Create roughing toolpaths

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

STEP 02 Click ***Setup Tab->Add Stock*** to add a stock of the part. (All the settings are defined by default)

STEP 03 Click ***Yes*** to hide the stock.

STEP 04 Click ***3x Quick Tab->Offset 2D*** in rough operation.

STEP 05 Double-click the ***Feature*** page to define a part and stock, then click ***OK*** to finish definition.

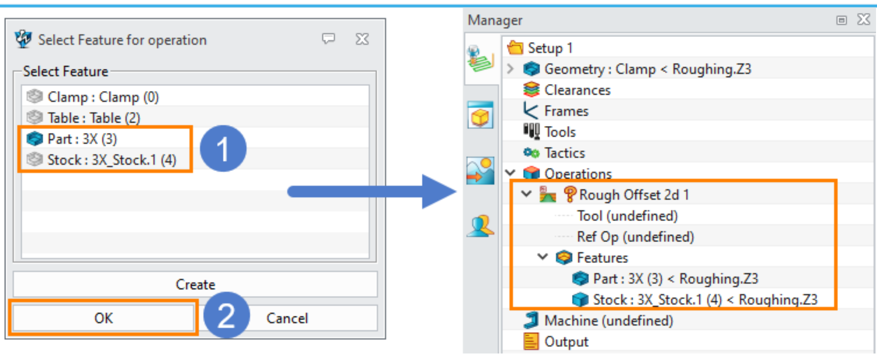


Figure 3 Define the machining features

STEP 06 Name **D10R0** and change the radius to **0** in the tool dialog box when it pops up.

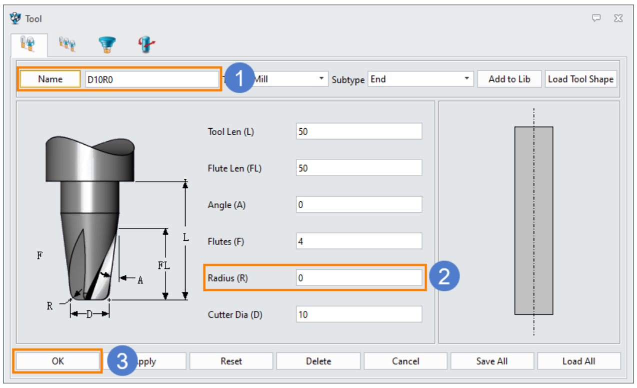


Figure 4 Tool definition

STEP 07 Click ***Yes*** to calculate the roughing toolpaths.

STEP 08 After calculation, the toolpaths will be generated as below.

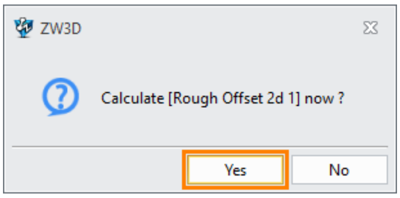


Figure 5 Toolpaths calculation

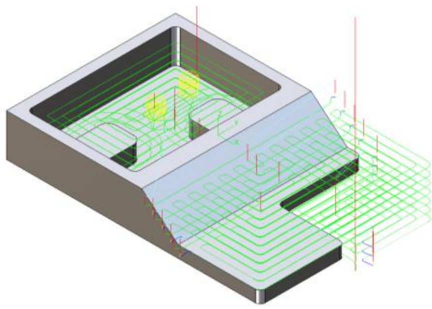


Figure 6 Offset 2D toolpaths

Notes:

1. *If the tool dialog box is not popped up, double-click **Tool (undefined)** in operation list to call it out.*
2. *If the calculation dialog box is not popped up, right-click the operation, then click **Calculate** in the context menu to generate the toolpaths.*

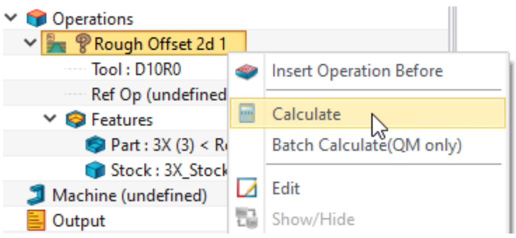


Figure 7 Callout toolpaths calculation

2.2 Detail Setting in 3X Roughing Operation

In section 2.1, the essential steps of toolpaths generation have been introduced. To get more suitable toolpaths in actual machining, ZW3D offers plenty of and easy to use parameters for users to achieve the goal.

Double-click a specific operation in the operations menu, a parameters dialog box will be displayed.

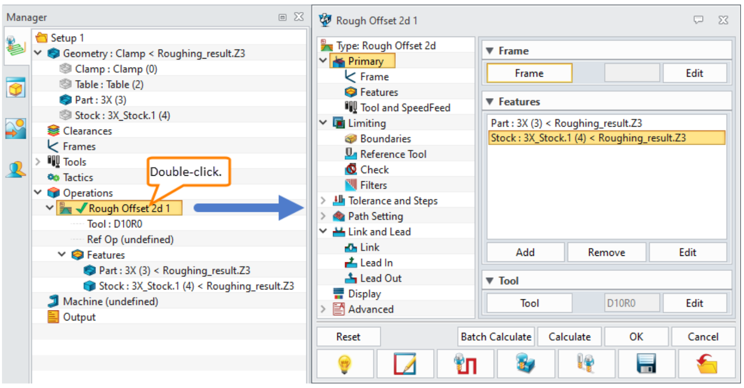


Figure 8 Parameters dialog box

2.2.1 Primary

1) Frame

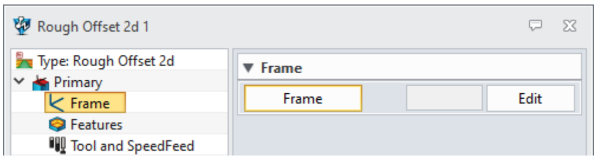


Figure 9 Frame parameters page

Frame: Specify a UCS (user coordinate system) to generate the toolpaths. The frame should be created before specifying. If it is blank, the global coordinate system will be used.

Edit: Create a user coordinate system.

Task: Create a user coordinate system and use it to generate the toolpaths.

Open the **User coordinate system.Z3** file and double-click the **3X_CAM** object to go into CAM space.

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

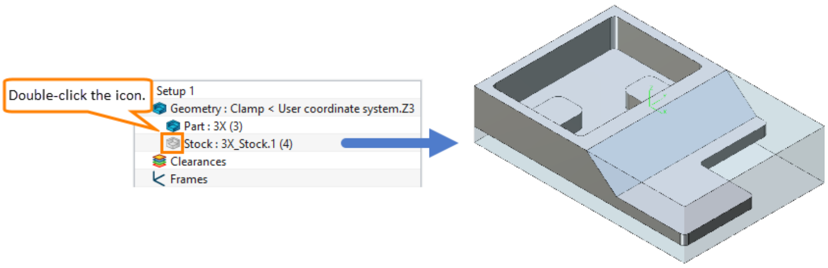


Figure 10 Display the stock

STEP 02 Create a new frame on the bottom side of the stock.

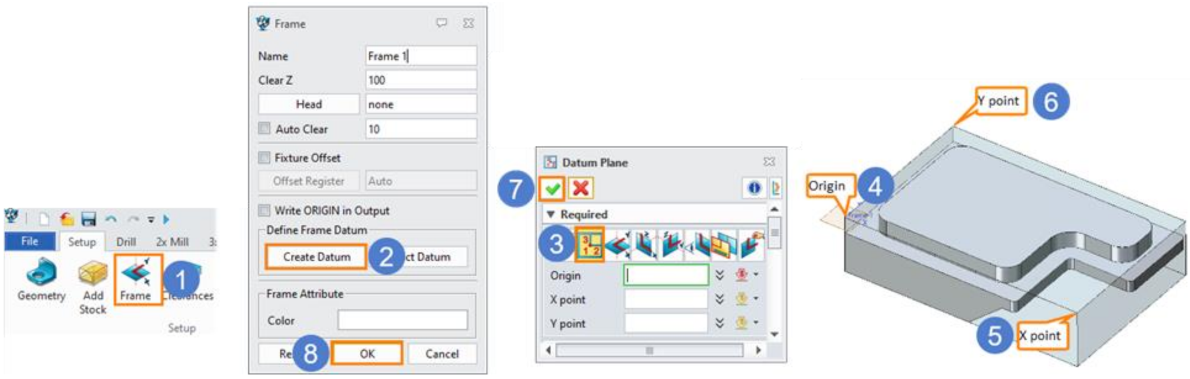


Figure 11 Create a frame

STEP 03 Right-click on **Rough offset 2d 1** operation and ***Duplicate*** it.

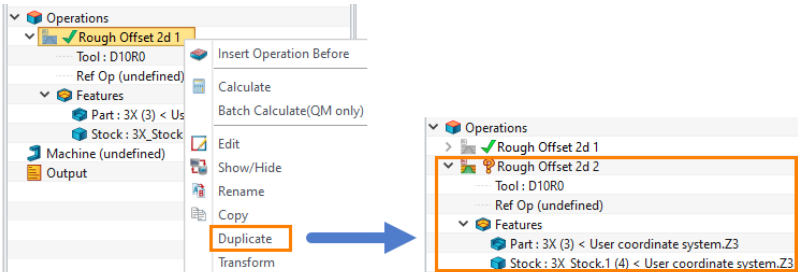


Figure 12 Duplicate operation

STEP 04 Double-click the copied operation, then select **Frame 1**.

STEP 05 Calculate the **Rough Offset 2d 2** operation and get the toolpaths as below.

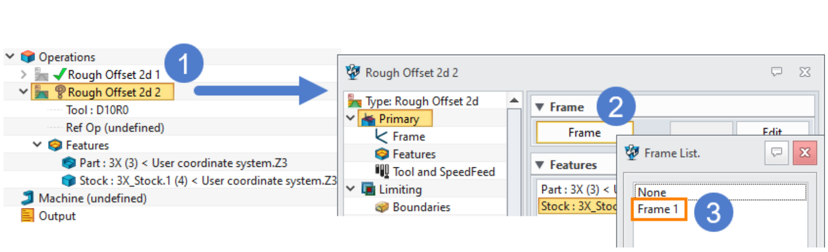


Figure 13 Specify a frame(UCS)

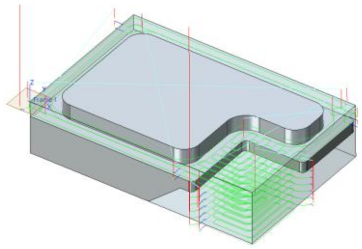


Figure 14 Generate toolpaths

In general, there are two usages for creating a frame.

- It is used in indexing machining, such as the undercut region in the world coordinate system.
- To output the NC code based on the coordinate system which is different from the toolpaths' local frame.

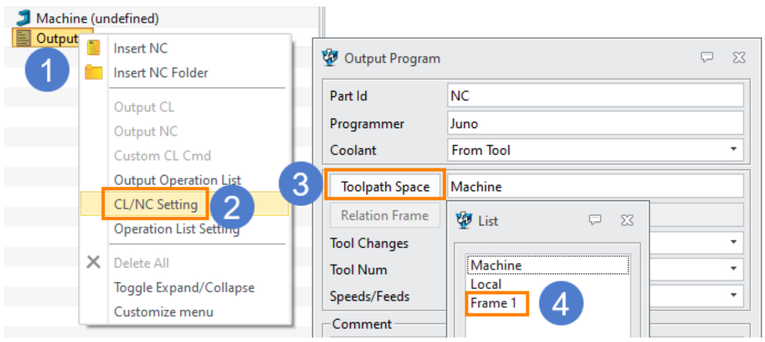


Figure 15 Output NC code with a specified frame

Tips:

1. Duplicate option can quickly create a new operation by referring to an existing one. All the parameters will be inherited.

2. The hints of the parameter will be popped up when the cursor is being held for a while on it.

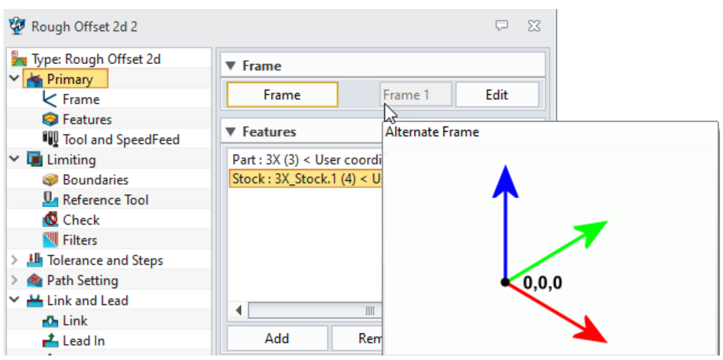


Figure 16 Show the illustration

2) Feature

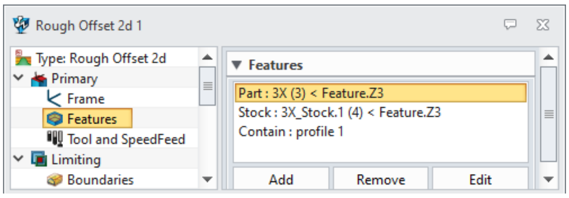


Figure 17 Feature parameters

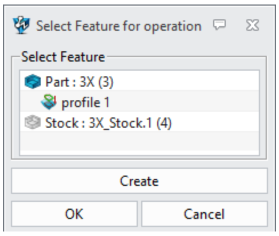


Figure 18 Feature dialog box

Add: Open a feature dialog box to select an existing feature or create a new feature.

Remove: Delete a selected feature above.

Edit: Redefine an existing feature above. And only the profile feature is available.

Tips:
*1. Double-click the **Features** page under the operation in the CAM tree can also open the feature dialog box.*
2. You can also right-click an existing feature to delete or redefine it on the context menu.

3) Tool and SpeedFeed

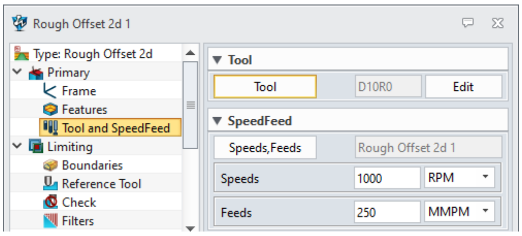


Figure 19 Tool and Speed/Feed parameters

Tool: Click this button to select a cutter in the tool list. Or open the tool definition dialog box if there is no existing tools in the list.

Edit: Redefine a selected cutter. This option is invalid if there is no cutter was selected.

Speeds, Feeds: Open the Speeds/Feeds dialog box to adjust each process parameter.

Speeds/Feeds: Adjust the speed/feed and its unit.

In the Speeds/Feeds dialog box, there are three options in each drop-down list. The following are the illustration of them.

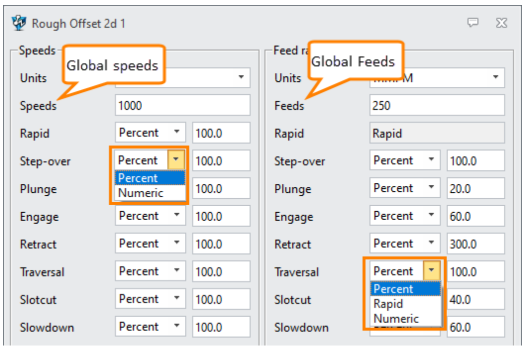


Figure 20 Speeds/Feeds parameters

Percent: Use a percentage(%) to calculate the speed of the process according to the global ***Speeds/Feeds***.

Numeric: Specify a fixed speed/feed for each process.

Rapid: Rapid movement in terms of the machine with G0 code.

Tips: *You can also double-click the **Tool** page under the operation to create or redefine a cutter of all its parameters.*

2.2.2 Limiting

1) Boundaries

It is a page to set the limitation of the horizontal(XY) and vertical(Z) machining region.

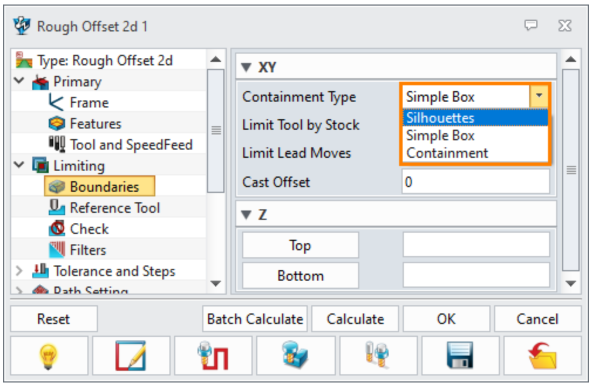


Figure 21 Boundaries page

Containment Type: There are three containment types list here including ***Silhouettes***, ***Simple Box,*** and ***Containment.***

- ***Silhouettes***: Create the maximum projection contour of the part as the limiting boundary.
- ***Simple Box***: Create a smallest cube surround the part, use its projection contour of the part as the limiting boundary.

In ZW3D, users can calculate the roughing toolpaths without defining a stock. Hence, when discussing the boundaries, we need to illustrate it in two aspects: feature with stock and feature without stock.

Feature with stock: When a stock is added in the machining feature, the ***Silhouettes*** and ***Simple Box*** options will be invalid. The toolpaths are generated within the stock. And if a profile is added, the toolpaths will be generated within the intersection region between stock and profile.

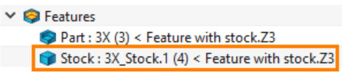
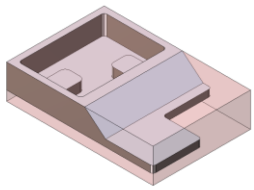
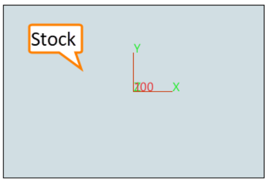
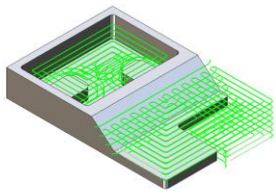
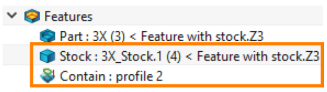
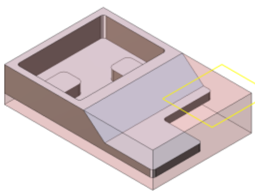
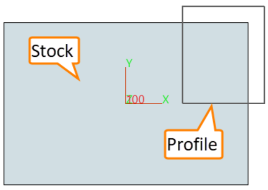
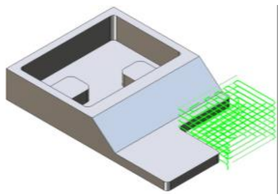
Boundary type	Boundary	Toolpaths
Stock 	 	
Stock and profile 	 	

Figure 22 Feature with stock

Feature without stock: When the stock is not added in the machining feature, ZW3D will set the limiting boundary of ***Silhouettes*** or ***Simple Box*** as the virtual stock to generated the toolpaths without a profile. After adding the profile, the ***Silhouettes*** and ***Simple Box*** options will be invalid, the toolpaths will be generated within the intersection region between the smallest cube of the part and profile.

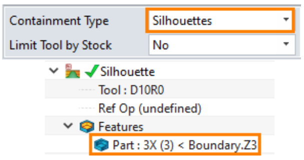
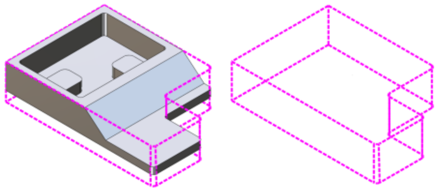
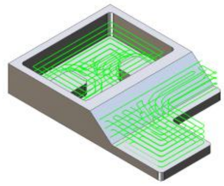
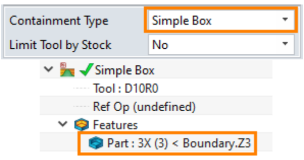
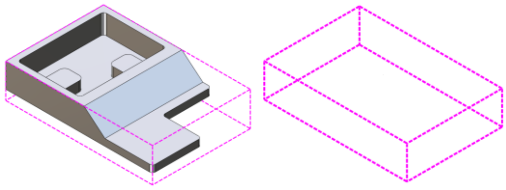
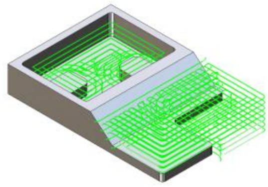
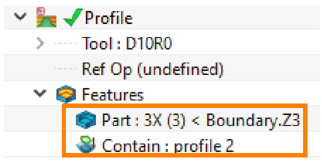
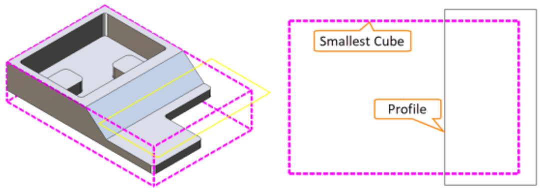
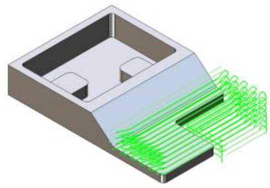
Boundary Type	Boundary	Toolpaths
Silhouette 		
Simple Box 		
Adding Profile 		

Figure 23 Feature without stock

Limit Tool by Stock: To define whether all the generated toolpaths are within the stock or not.

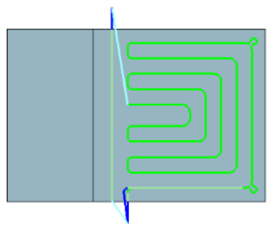
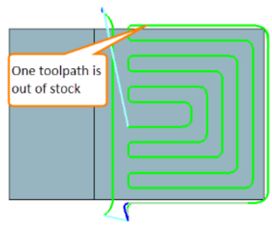
Limit Tool by Stock	Yes	No
Toolpaths		

Figure 24 Limit tool by stock

Cast Offset: Define a value, ZW3D will create a customed stock that has united allowance using the defined value, then generate the clearance toolpaths to remove the material from the stock to part. Keep it blank or set it 0, this option is invalid.

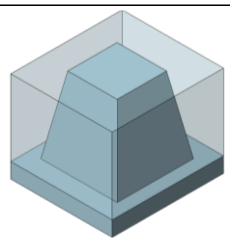
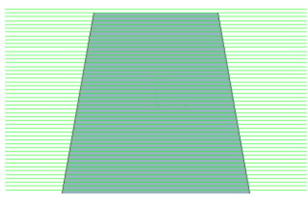
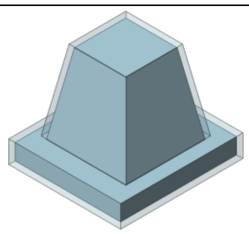
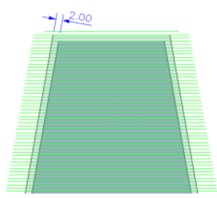
Limit Tool by Stock		Toolpaths
Cast Offset = **0**		
Cast Offset = **2**		

Figure 25 Cast offset

Top/Bottom: Top and bottom options are used to define the manufacturing region in the Z-Axis direction, which means the toolpaths will be limited from the top point to the bottom point in the vertical direction. And they could be defined via picking points on the part or specifying a value. Either of them could be blank, it will be limited by the boundary of stock.

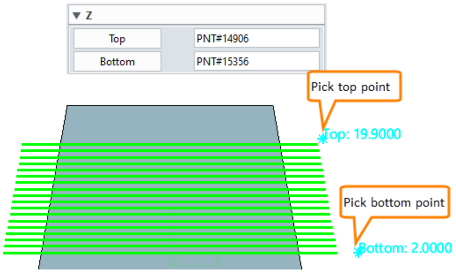
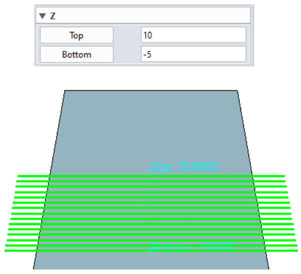
Top and Bottom	Pick points	Top: **10** Bottom: **-5**
Toolpaths		

Figure 26 Top and bottom limit

2) Reference tool

The parameters on this page are used to optimize the rest roughing. There are 3 ways for it in ZW3D.

- Set reference operation
- Specify rest material as stock
- Specify reference tool

And reference operation is recommended to be used in priority.

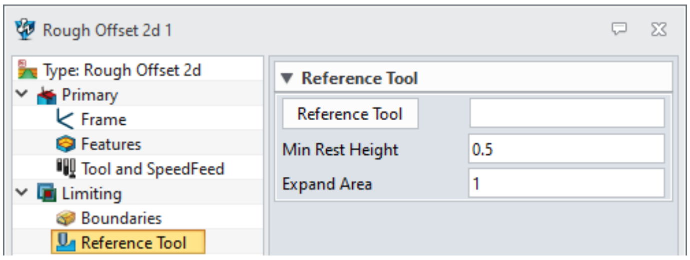


Figure 27 Reference Tool

Reference Tool: Specify a larger tool. Hypothetically, it was used for roughing in advance and the rest material will be left. Then, use the current tool to generate toolpaths to remove the rest material.

Min Rest Height: Enter a threshold value(0.5 mm).The calculation ignores rest material thinner than it(0.5 mm). It avoids a rest-cut of the negligible regions. This option is also available when defining a reference operation.

Expand Area: Enter a distance(1 mm) to expand the rest region(outlined in red), which will be extended along the surface according to it(1 mm). This is usually used with ***Min Rest Height*** to remove the undetected area(outlined in pink) by offsetting the rest region. This option is also available when defining a reference operation.

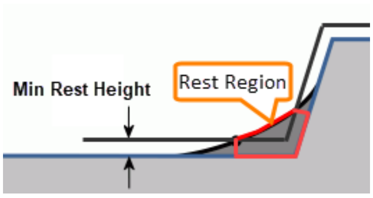
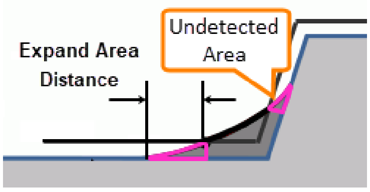
Reference tool	Min Rest Height	Expand Area
Illustration		

Figure 28 Reference tool parameters

3) Check

Detect the collision between the toolpaths and assistant equipment(such as the table, clamp, tool holder, etc), which ensures machining safely.

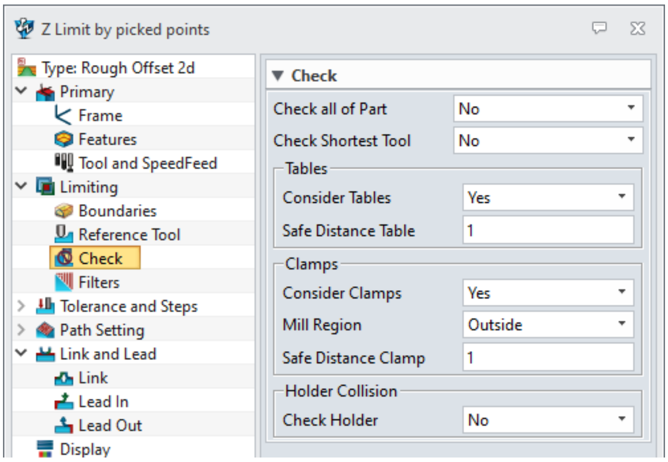


Figure 29 Check page

Check all of Part: When defining some surfaces of the part as the machining feature. Turn on this option(Yes) to avoid the tool crash into the part.

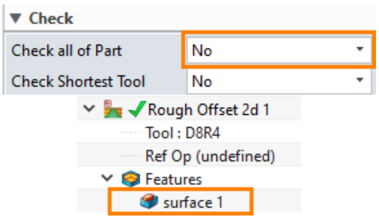
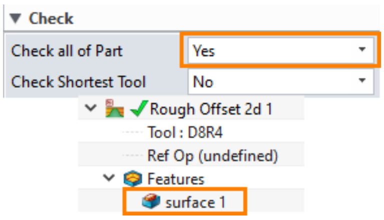
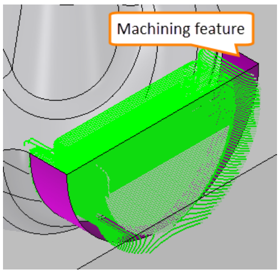
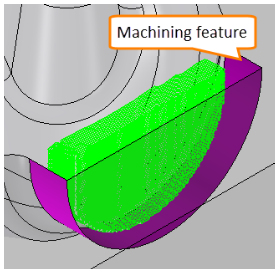
Check all of Part	No	Yes
Setting		
Toolpaths		

Figure 30 Check all of part

Check Shortest Tool: If a tool holder is defined in the tool manager, turn it on(Yes), ZW3D will calculate the minimum tool length to avoid the holder collision. The minimum length will be displayed on the Tool page in the CAM manager.

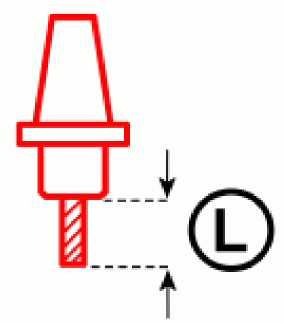


Figure 31 Minimum length

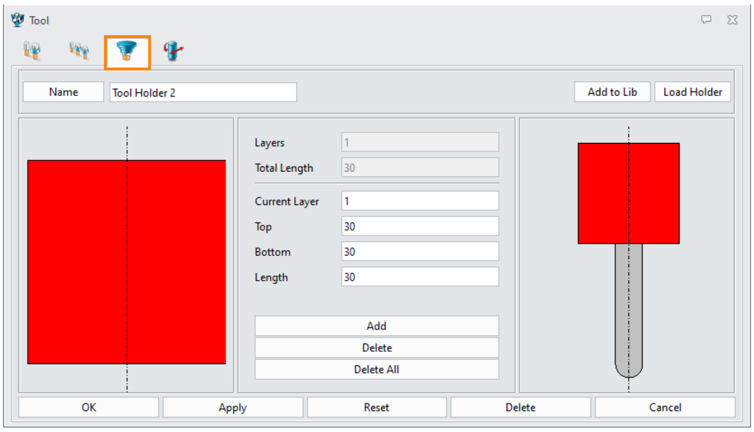


Figure 32 Tool holder definition

Tips:

1. The tool holder's definition is on the 3rd tab of the tool manager.

*2. You can right-click the **Setup** in the CAM manager to open the **Setting** dialog box. Check on the **Minimum** to display the minimum length.*

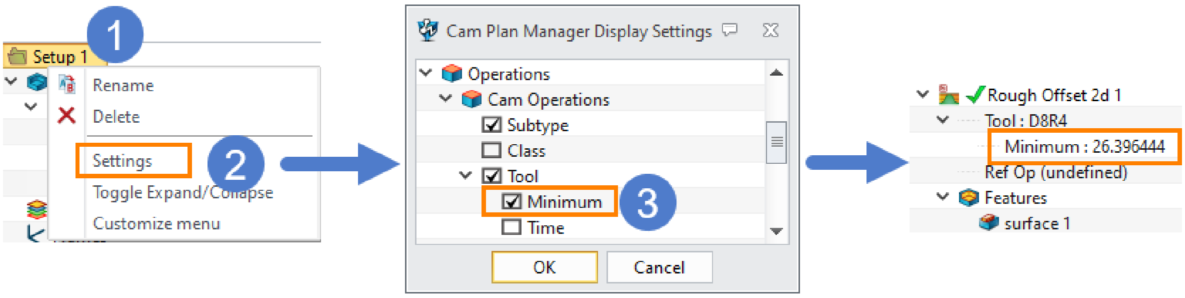


Figure 33 Display the minimum length

Consider Tables: Select Yes. ZW3D will generate the toolpaths according to the position of the table, which avoids the table to be cut by the tool.

Safe Distance Table: Set a safe distance, which limits the toolpaths to be generated close to the table.

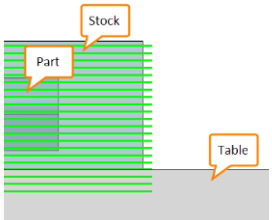
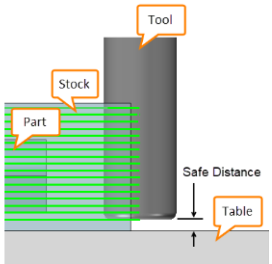
Table	Consider Table: **No**	Consider Table: **Yes** Safe Distance Table: **2**
Toolpaths		

Figure 34 Consider table and safe distance

Consider Clamps: Select Yes. ZW3D will generate toolpaths considering the clamps which have been defined and will not be manufactured within a safety region.

Mill Region: Change the direction of generating toolpaths.

Safe Distance Clamp: Enter a safe distance(2 mm), the cutter will maintain a distance(2 mm) from the clamps.

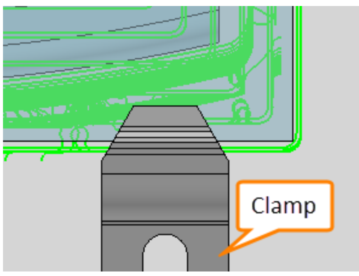
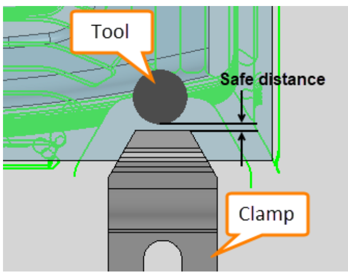
Clamps	Consider Clamps: **No**	Consider Clamps: **Yes** Safe Distance Clamp: **2**
Toolpaths		

Figure 35 Consider clamp and safe distance

Check Holder: Select Yes. The tool holder will be considered when generating the toolpaths, which avoids the holder crash to the product. Hence, the tool holder should be defined in advance.

Safe Distance Holder: Enter a safe distance(1 mm), the cutter will maintain a distance(1 mm) from the holder.

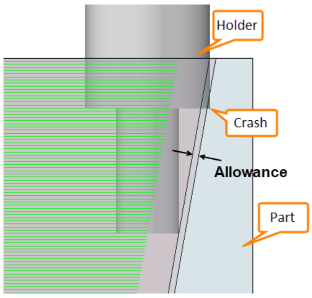
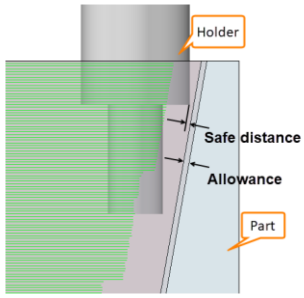
Holder	Consider Holder: **No**	Consider Holder: **Yes** Safe Distance Holder: **1**
Toolpaths		

Figure 36 Consider holder and safe distance

4) Filter

% Small Length/Area: Enter a value(10), which will be used to calculate the threshold value(10% *tool diameter). ZW3D will eliminate the toolpaths' level whose ratio between span and area is less than the threshold value to avoid the negligible cut.

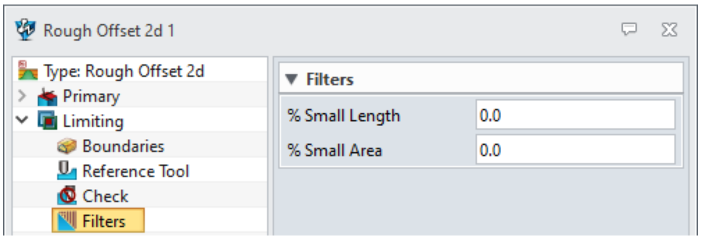


Figure 37 Filter

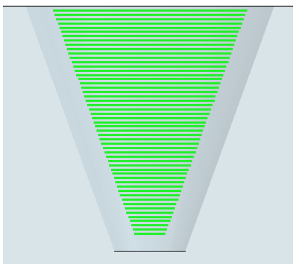
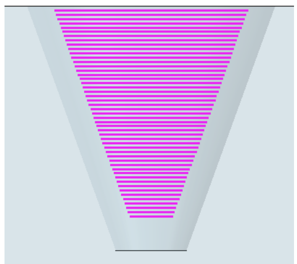
Filter	% Small Length: **0**, % Small Area: **0**	% Small Length: **80**, % Small Area: **80**
Toolpaths		

Figure 38 Filter setting

2.2.3Tolerance and Steps

1) Tolerance and Thick

It is a page to define tolerance and allowance.

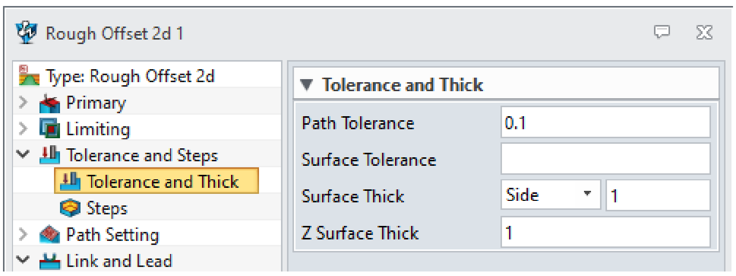


Figure 39 Tolerance and Thick

Path Tolerance: Specify a value to determine what accurately the toolpaths follow the contours of the machining model is. In the following image, the blue curve is an ideal toolpath, the polyline in black is a generated toolpath.

Surface Tolerance: It is a chord height tolerance, which is applied to surface/solid geometry to control the density of toolpaths points.

Surface Thick: Set the thickness of the material to be left on the model. If select the ***Side***, the radial thickness and axial thickness can be set separately. If select the ***Total***, the two thicknesses are the same, they are set in the current textbox.

Z Surface Thick: Set the axial thickness of the material to be left on the model. It is displayed when ***Side*** was selected in the ***Surface Thick***.

Figure 40 Path tolerance

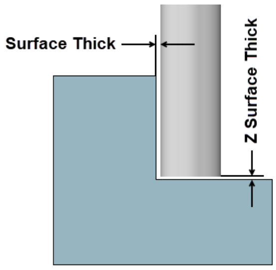


Figure 41 Surface thick

Tips: *The **Surface Thick** or **Z Surface Thick** options could be blank. And the following form shows three scenarios with different settings on them.*

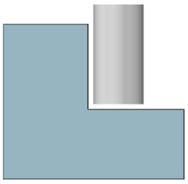
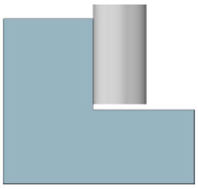
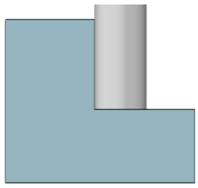
Thick	Surface Thick: **1** Z Surface Thick: **Blank**	Surface Thick: **Blank** Z Surface Thick: **1**	Surface Thick: **Blank** Z Surface Thick: **Blank**
Result			
Note	***Z Surface Thick*** will inherit the allowance from the ***Surface Thick***.	Keep ***Surface Thick*** blank, it will be considered as zero.	Keep both of them blank, they will be considered as zero.

Figure 42 Keep blank in thick

2) Steps

On this page, the stepover and stepdown can be defined.

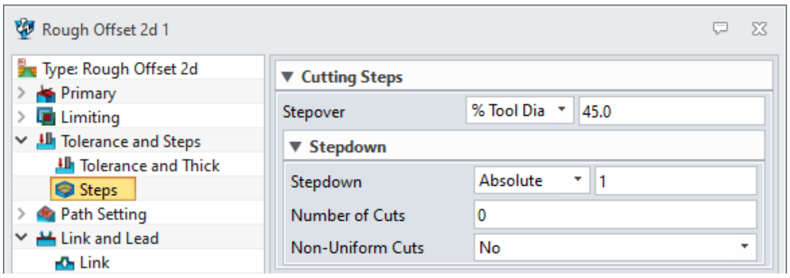


Figure 43 Steps

Stepover: Define a distance between two adjacent toolpaths. There are three ways to define it.

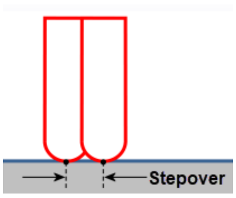
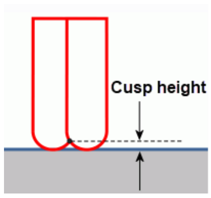
Stepover	Absolute & % Tool Dia	Scallop
Illustration		

Figure 44 Stepover

- ***Absolute***: Specify the stepover value directly with a fixed value.
- ***% Tool Dia***: Specify the stepover value according to the percentage of the tool diameter.
- ***Scallop***: Specify the stepover by the cusp height of the material left between Z height does not exceed the scallop.

Stepdown: Define the distance between successive toolpaths in the Z-Axis direction.

- ***Absolute***: Specify the stepdown value directly with a fixed value.
- ***% Flute Len***: Specify the stepdown value according to the percentage of the current cutter flute length.

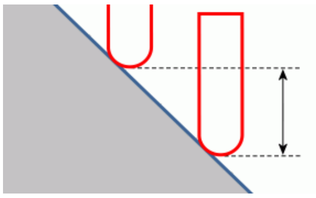


Figure 45 Stepdown

Number of Cuts: Specify the number of contour cuts between two roughing toolpaths, which can minimize terracing when create roughing toolpaths with a large stepdown. It rest-machines the terraces machining from the roughing toolpaths with the same tool in the same operation.

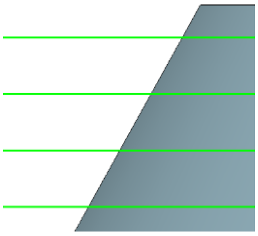
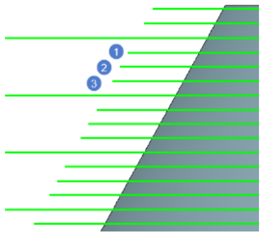
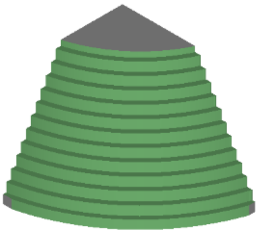
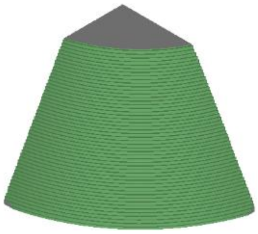
Number of Cuts	Standard rough toolpaths without setting the ***Number of Cuts*** (**0**)	The same toolpaths setting ***Number of Cuts*** (**3**)
Toolpaths		
Result		

Figure 46 Number of Cuts

- ***Z Intercut Progress***: When the ***Number of Cuts*** is defined, there are two option can be selected to determine the cutting sequence: Top to Bottom and Bottom to Top.

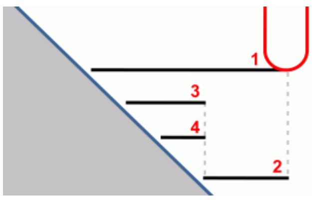
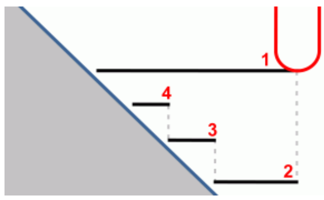
Z Intercut Progress	Top to Bottom	Bottom to Top
Result		

Figure 47 Z intercut progress

Non-Uniform Cuts: Select Yes to enable generate toolpaths with different stepdown in different regions.

The following options will be displayed when selecting ***Yes*** here.

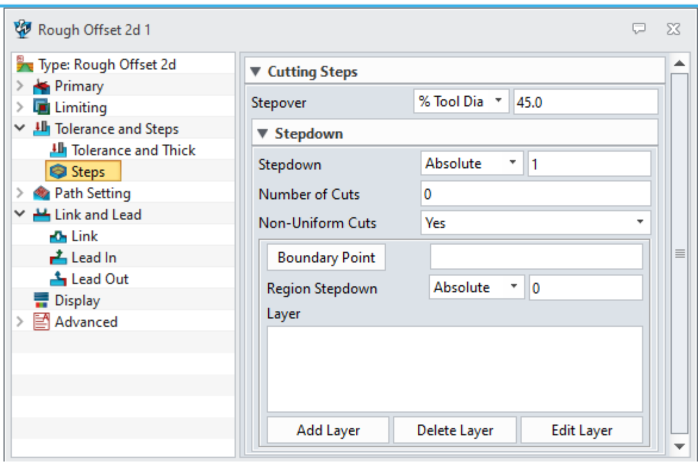


Figure 48 Non-Uniform Cuts

- ➢ ***Boundary Point***: Click this button to pick a point in the graphic area. It will divide the region into two layers and be able to define the different stepdown setting in the region under the boundary point via the ***Region Stepdown*** option.
- ➢ ***Region Stepdown***: Define the stepdown value of the layer under the boundary point.

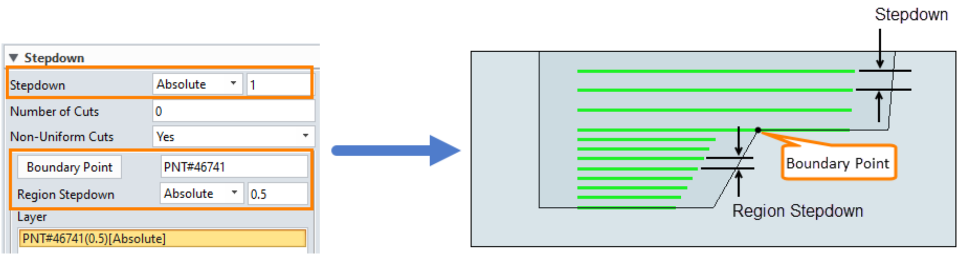


Figure 49 Region stepdown

- ➢ ***Add Layer***: Click it to confirm the layer setting and add it to the list.
- ➢ ***Delete Layer***: Delete the selected layer.
- ➢ ***Edit Layer***: After adjusting the boundary point or region stepdown value, click this option to save the modification.

2.2.4Path Setting

The cutting toolpaths can be controlled or optimized on this page.

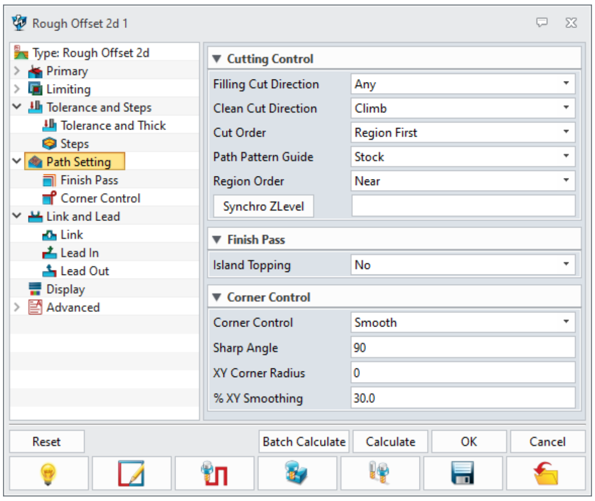


Figure 50 Path Setting

1) Cutting Control

Filling Cut Direction/Clean Cut Direction: Define the filling cut and clean cut toolpaths' direction. Clean cut toolpaths are the last toolpaths close to the feature boundary of each level. Filling cut toolpaths(within the imaginary lines below) are inside the clean cut toolpaths.

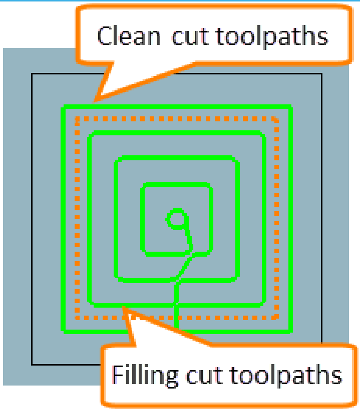


Figure 51 Filling clean cut toolpaths

- ***Climb***: Only use climb milling to generate the toolpaths. The tool is on the left of the machined edge when viewing in the tool movement direction.
- ***Conventional***: Only use conventional milling to generate toolpaths. The tool is on the right of the machined edge when viewing in the tool movement direction.

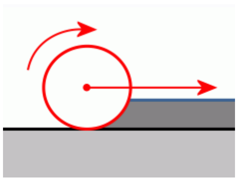
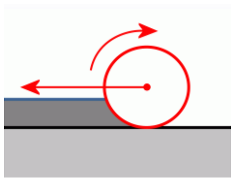
Direction	Climb	Conventional
Illustration		

Figure 52 Climb and conventional

- ***Any***: ZW3D will assign the toolpaths to climb or conventional to get Zigzag pattern toolpaths.
- ***Automatic***: It will follow the filling cut toolpaths.
- ***Cut Order***: Specify the cutting priority between cutting region and level.
- ***Region First***: It will completely process one region before going on to another one.
- ***Level First***: It will completely process one level of each region before going down to the next level.

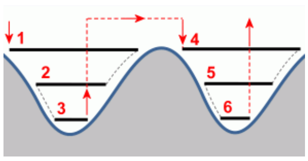
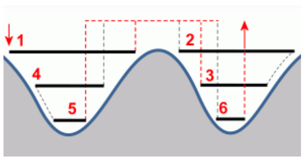
Cut Order	Region First	Level First
Illustration		

Figure 53 Cut order

Path Pattern Guide: The toolpaths will be generated according to part silhouette, stock silhouette, or both of them.

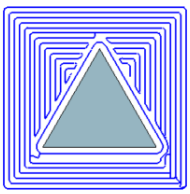
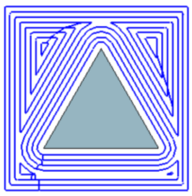
Path Pattern Guide		Part	Stock	Both
Part				
Stock				

Figure 54 Path pattern guide

Region Order: Specify the cutting sequence of each region.

- ***Near***: It will go on to the nearest region after finishing processing one region.

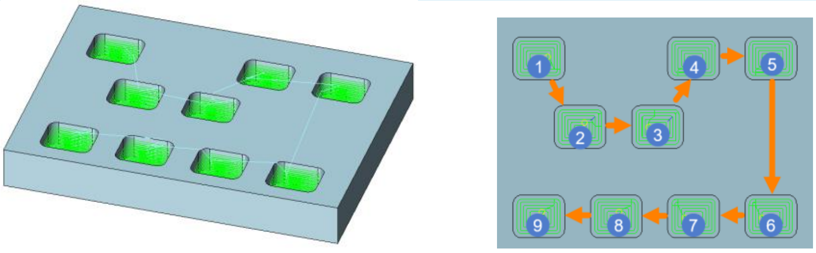


Figure 55 Region order by near

- ***X/Y Along***: Process the region along the X/Y direction, then go to the beginning of the next row/column to continue the process.
- ***X/Y ZigZag***: Process the region along the X/Y direction, then go to the nearest region of the next row/column to continue the process.

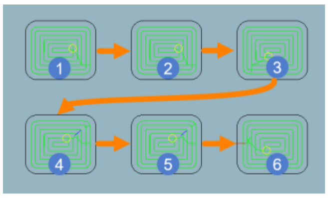
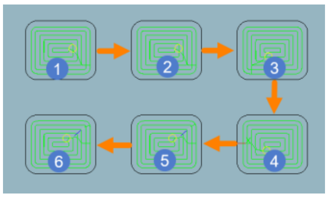
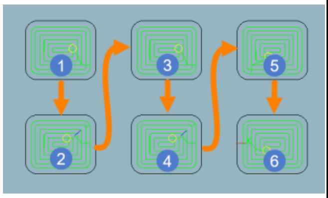
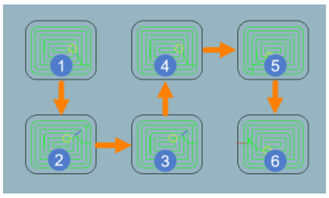
X Along	X ZigZag	Y Along	Y ZigZag
			

Figure 56 X/Y along and zigzag

Synchro ZLevel: Click this option and pick the point(s) in the graphic area. The toolpaths must be generated to pass the point(s). This option is used to maintain uniform allowance on the planner surfaces which selected from the part manually.

2) Finish Pass

Island Topping: Check on this option(Yes), another option(***%Island Area***) will be displayed The operation will automatically detect the planar face on the part. The toolpaths will be generated to machine the planar face to get a uniform rest material. It is a way to generate the ***Synchro ZLevel*** toolpaths without picking the sync point(s) manually.

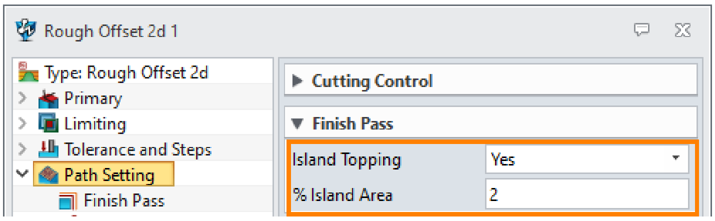


Figure 57 Finish Pass

- ***% Island Area***: Specify a value. It will calculate a threshold area according to the percentage of the value (Threshold area= Silhouettes area of the part * Value%). Only if the area of the planar face is equal to or greater than the threshold area, the planar will be considered for generating toolpaths.

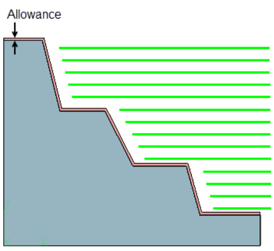
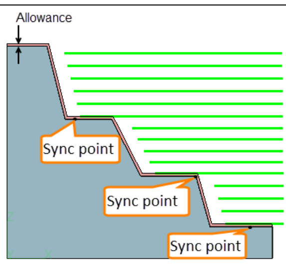
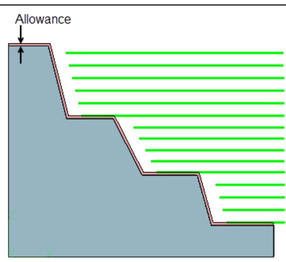
Status	Synchro Zlevel(**No**) Island Topping(**No**)	Synchro Zlevel(**Yes**)	Island Topping(**Yes**) % Island Area = **2**
Toolpaths			

Figure 58 Synchro ZLevel and Island Topping

3) Corner Control

To adjust the pattern of the toolpaths at the corner, setting the parameters here could get an expected result to satisfy different machining requirements.

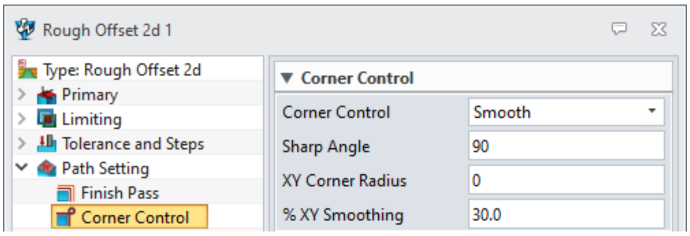


Figure 59 Corner Control

Corner Control: Specify a method to generate the toolpaths at the corner.

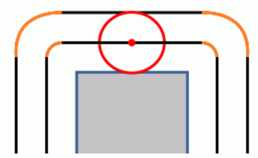
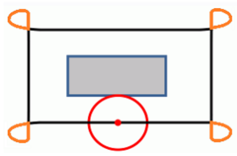
Corner Control	Smooth	D Loop
Toolpaths		

Figure 60 Corner control pattern

Sharp Angle: This option is valid only if ***D Loop*** was selected in ***Corner Control***. Specify a threshold value. If the corner angle of the toolpaths is equal to or less than it. A D Loop pattern will be generated.

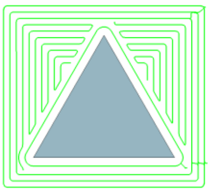
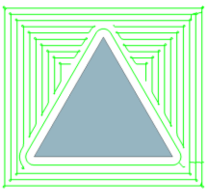
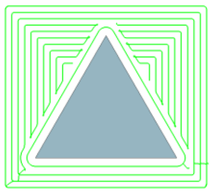
Sharp Angle	Smooth	Sharp Angle(**90**)	Sharp Angle(**60**)
Toolpaths			

Figure 61 Sharp angle

XY Corner Radius: Specify the fillet value for clean cut toolpaths.

% XY Smoothing: Specify a value. It will calculate a fillet value according to it with the ***Stepover*** value(Fillet value=Stepover value * value%). This fillet value will be used to generate the Filling cut toolpaths.

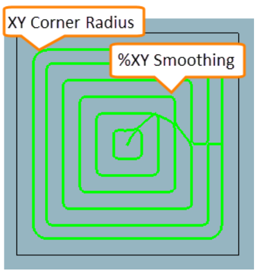


Figure 62 XY Corner Radius and %XY Smoothing

2.2.5 Link and Lead

In general, the toolpaths structure of ZW3D has ***lead in(engage)***, ***lead out(retract)***, ***cut***, ***plunge***, ***rapid*** and ***link*** segments. In some cases, to optimize the toolpaths, some short links will be treated as a cut movement. And each segment will be displayed with different colors.

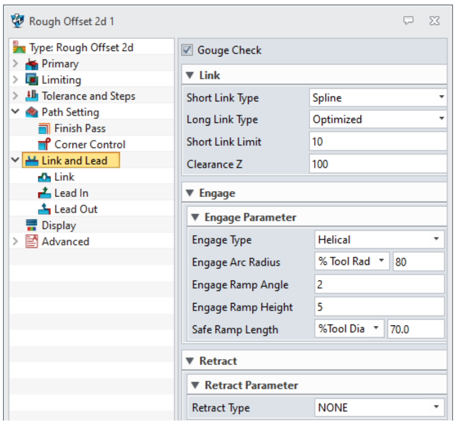


Figure 63 Link and Lead

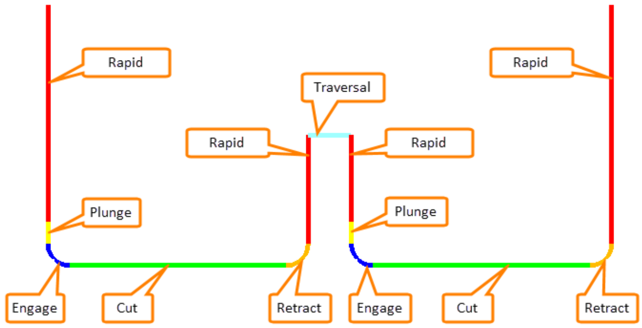


Figure 64 Tool movement segments

Gouge Check: This option is checked by default. It is used to check the gouge of toolpaths about lead and link. In most of the cases, it should be checked on, only very few special scenarios it could be checked off for achieving the wondering toolpaths, such as the extension in ***Tool Path Editor***.

1) Link

The link options are used to define how to connect the adjacent cutting movements. There are two types of links: short link, and long link. Users enable to set proper link types on them in terms of different machining scenarios.

Short Link Limit: The short link limit means the distance between the tail and the head of adjacent cutting movements. Specify a threshold value on this option to separate the links into two types. Links are greater than the threshold will be assigned as long links, otherwise, they will be short links.

Short Link Type: ZW3D provides 5 kinds of short links to fulfill different machining requirements. They are ***On surface***, ***Step***, ***Spline***, ***Optimized***, ***Clearance***.

Long Link Type: ZW3D provides 2 kinds of long link to satisfy different machining requirements. They are ***Optimized***, ***Clearance***.

Default Link Type: Select the type of link moves from the list, which provides ***optimized*** and ***clearance***, to be used if the constraint criteria for short links are not met, or if they gouge.

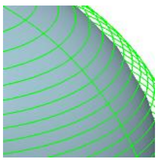
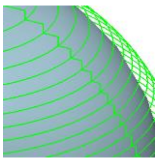
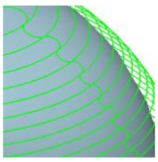
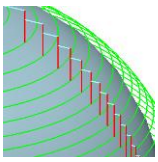
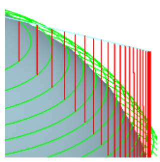
Link type	On surface	Step	Spline	Optimized	Clearance
Illustration	The links follow the surface of the model with a straight line and avoid gouging.	The links traverse to the next contact point horizontally and then vertically descend to reach the next contact point.	The links follow the surface of the model with spline and avoid gouge.	The links travel clear of the part by the specified incremental distance, descend at a rapid feed to the specified incremental distance above the contact point, and then plunge the remaining distance.	The tool descends at a rapid feed from the clearance plane to the specified incremental distance above the target point, then plunges the remaining distance.
Toolpaths					
Tool lift	No	No	No	Yes	Yes

Figure 65 Link type

Safe Distance: This value is added to the clearance that is automatically detected to avoid hitting the stock and machining part.

Clearance Z: This value is defined as the tool clearance distance measured from the origin of the Z-axis of the alternate frame**.**

Tips: *The default value of* ***Clearance Z*** *will inherit the global* ***Clearances*** *set in the CAM manager.*

Plunge Length: This is a distance for a plunge before the start of the actual cut. It is a plunge move added before each lead in for safer tool engagement.

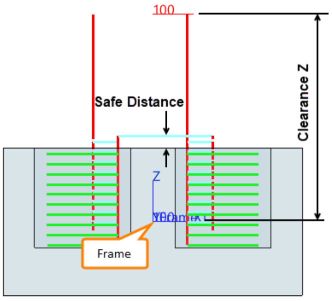


Figure 66 Safe Distance and Clearance Z

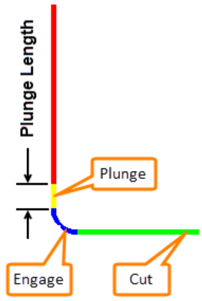


Figure 67 Plunge Length

Add Lead to Short Link: It is an option that controls whether to add leads (lead in and lead out) before short link.

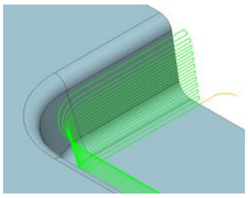
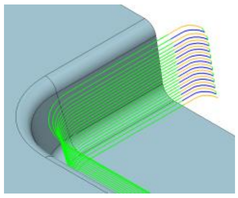
Add Lead to Short Link	No	Yes
Toolpaths		

Figure 68 Add Lead to Short Link

%Spline Elasticity: Specify a value to control the shape of the spline, which is only available for spline link moves. The larger value is set, the more convex spline will be generated.

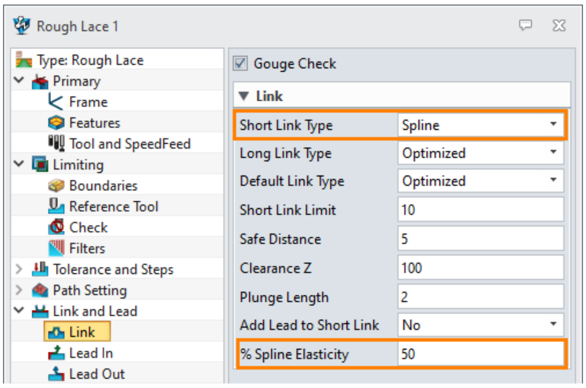


Figure 69 Sort Link Type and %Spline Elasticity

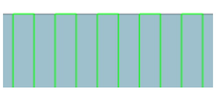
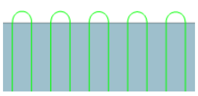
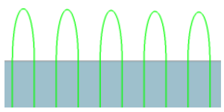
%Spline Elasticity value	**0**	**50**	**200**
Toolpaths			

Figure 70 %Spline Elasticity

2) Lead In

It is used to define the movement of the tool approaching the stock.

Engage Type: ZW3D provides 3 types of lead in toolpaths including ***Arc_Line***, ***Helical***, ***Along_Toolpath***.

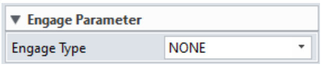
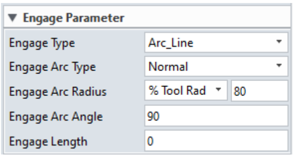
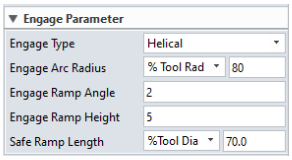
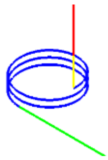
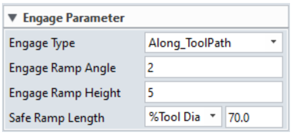
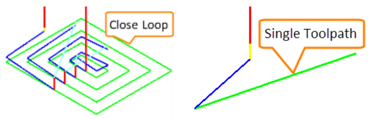
Lead type	Available Parameters	Description	Toolpaths
NONE		There are no leads in applied to the toolpaths.	
Arc_Line		It will insert a tangential arc move at the beginning of each cutting toolpath. This arc lies in the plane defined by the engage arc radius option.	
Helical		The ramp moves are circular. If the requested circle can not be fitted into the machining area, then the along_toolpath method is used automatically.	
Along_Toolpath		Ramps the tool into the model at a specified angle by following the cutting toolpath. If the cutting toolpath is not a close loop, the engage toolpath direction will be reversed.	

Figure 71 Lead type

Note: *When applying the **Helical** lead-in in roughing operations, and the requested circle can not be fitted into the machining area, then the **Along_Toolpath** method is used automatically. If **Along_Toolpath** doesn't fit either, **None** will be used automatically. Hence, the priority of them is: **Helical** >> **Along_Toolpath** >>**None.***

Engage Arc Type: Define the direction of the arc.

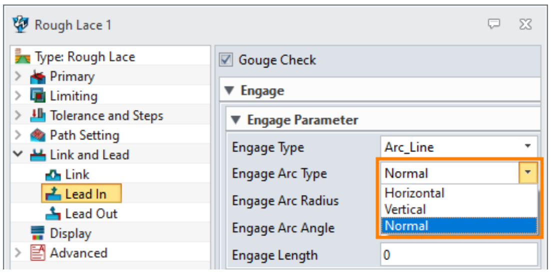


Figure 72 Engage Parameters

- ***Horizontal***: The arc will be generated on the plane which is parallel to the XY plane.
- ***Vertical***: The arc will be generated on the plane which is perpendicular to the XY plane.
- ***Normal***: The arc will be generated on the plane which is perpendicular to the machining surface.

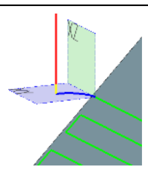
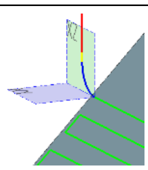
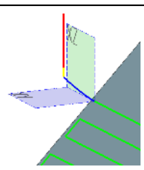
Engage Arc Type	Horizontal	Vertical	Normal
Toolpaths			

Figure 73 Engage Arc Type

Engage Arc Radius: Specify the arc radius.

- ***Absolute***: Specify a value to define the arc radius.
- ***%Tool Rad***: Specify a percentage of the tool radius to define the arc radius.

Engage Arc Angle: Specify a value to define the arc angle.

Engage Length: Specify a value to define the length.

Engage Ramp Angle: Specify the ramp angle value.

Engage Ramp Height: Specify the ramp height value.

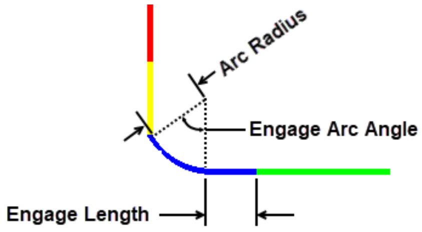


Figure 74 Engage Arc and Engage Length

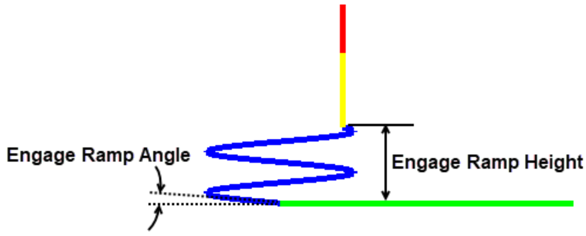


Figure 75 Engage Ramp

Safe Ramp Length: When using the non-center cutting tools for roughing. If cutting toolpaths are too small to cut all the material out in small regions, the rest material might have a collision with the cuter and cause the tool damage. So there is an option for helping to eliminate the small movements for safety manufacturing. In general, the minimum value is (Tool diameter- 2* blade width). For example, using a D40R2 cutter with a 4mm width blade, the minimum value will be (40-2*4)=32.

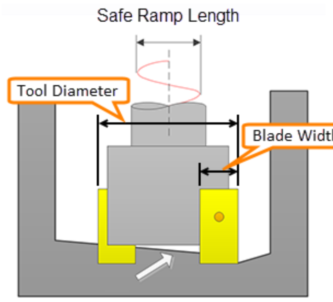


Figure 76 Safe Ramp Length

- ***%Tool Dia***: Specify a percentage of the tool diameter to define the threshold value.
- ***Absolute***: Specify a value to define a fixed threshold value directly.

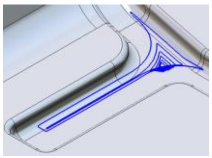
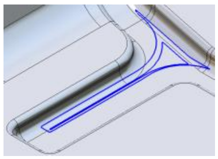
Safe Ramp Length Tool Diameter(20mm)	Absolute(**2**)	Absolute(**12**)
Toolpaths		

Figure 77 Safe Ramp Length result

Copy to Retract: Copy all the settings from the current page to ***Lead Out*** page.

Copy from Retract: Inherit all the settings from ***Lead Out*** page.

3) Lead out

The parameters on this page are the same as the ***Lead In*** page. Please refer to the instructions of lead in for more details.

2.2.6Display

The parameters on this page are used to define toolpaths display attributes including the style, width, and color.

Style: Specify the style of the toolpaths.

Width: Specify the width of the toolpaths.

Display Mode: Select a mode to define the color of the toolpaths for each segment. And check on/off the option below, it can display/hide the corresponding segments of the toolpaths. Change the color is available when clicking the color.

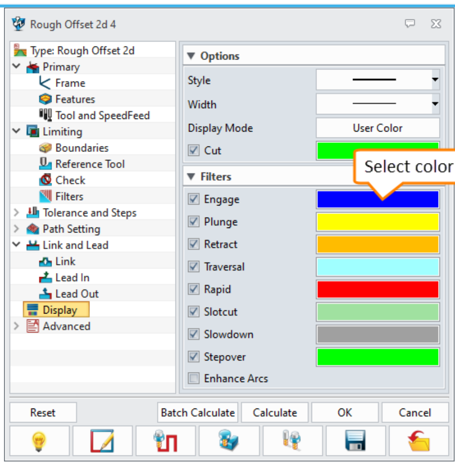


Figure 78 Display

2.2.7 Advanced

ZW3D provides some advanced parameters to help users get the expected result.

1) **Spiralize**

Enable Spiral: Turn this option on(Yes), more spiral parameters will be displayed.

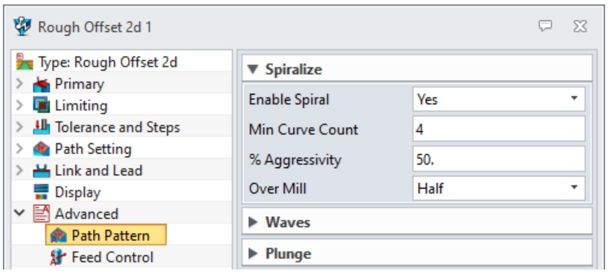


Figure 79 Spiral

➢ ***Min Curve Count***: Specify a threshold value, if the loop number of filling cut toolpaths is equal to or greater than it, the filling cut toolpaths will be converted to a spiral pattern.

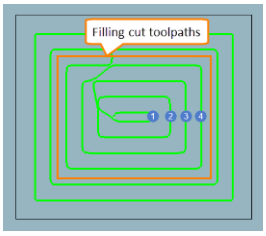
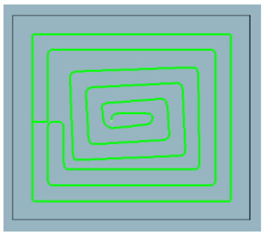
Spiral	Enable Spiral(**No**)	Enable Spiral(**Yes**) Min Curve Count(**4**)
Toolpaths		

Figure 80 Spiral comparison

➢ ***%Aggressivity***: This is a factor that determines how aggressive curves are fitted. A higher value means that curve fitting is more accurate at the cost of increased calculation time.

➢ ***Over Mill***: Add an extra spiral at the start or the end of the toolpaths.

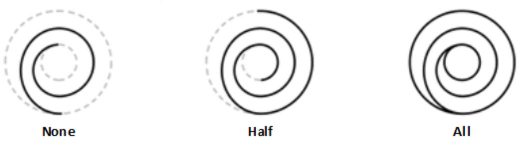


Figure 81 Over Mill

2) **Waves**

Enable Wave Path: Turn this option on(Yes), more wave parameters will be displayed.

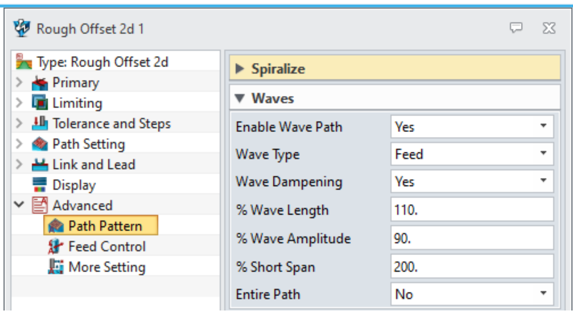


Figure 82 Wave

➢ ***Wave Type***: Specify the toolpaths' pattern of the wave.

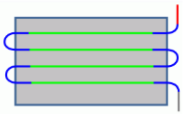
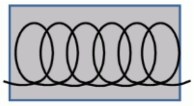
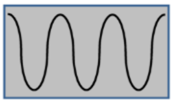
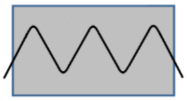
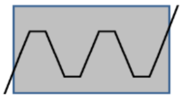
Feed	Trochoidal	Sinusoidal	Triangular	Trapezoidal
				

Figure 83 Wave Type

➢ ***Wave Dampening***: Turn this option on(Yes) to decrease the wave amplitude at the beginning and the tail of the cutting toolpaths.

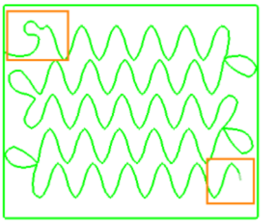
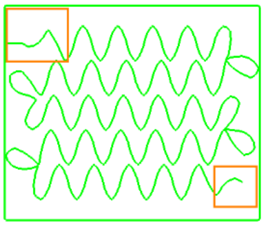
Wave Dampening	**No**	**Yes**
Toolpaths		

Figure 84 Wave Dampening

➢ ***%Wave Length***: Define a value as the percentage, which is used to calculate the wavelength according to the ***Stepover*** value(Wave Length = Stepover Value* %Wave Length).

➢ ***%Wave Amplitude***: Define a value as the percentage, which is used to calculate the amplitude distance according to the ***Stepover*** value(Wave Amplitude = Stepover Value* % Wave Amplitude).

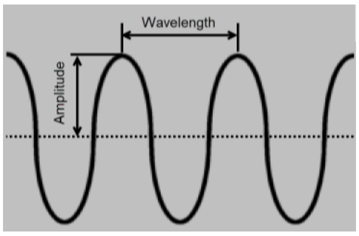


Figure 85 Wavelength and amplitude

➢ ***%Short Span***: Define a value as the percentage, a threshold will be calculated according to the ***Stepover*** value. If the wave span is less than it, the wave will not be generated.

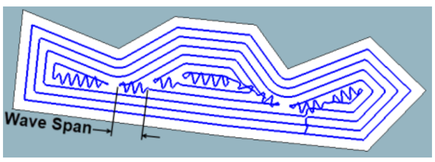
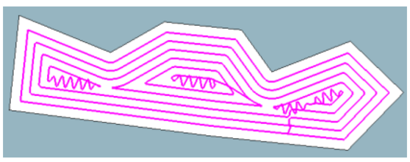
%Short Span	**100**	**300**
Toolpaths		

Figure 86 Short Span

- ***Entire Path***: Turn on this option (Yes), all the filling cut toolpaths will be generated in wave mode. Otherwise, only the first cut pattern which moves through the region will be in wave mode, the others will keep the normal cutting mode.

Note: *The* ***%Short Span*** *setting option will be invalid when turning on the* ***Entire Path*** *option.*

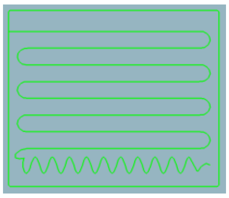
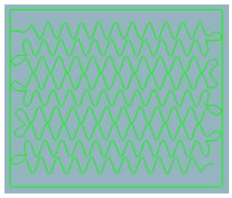
Spiral	Entire Path (**No**)	Entire Path (**Yes**)
Toolpaths		

Figure 87 Entire Path

3) Plunge

Enable Plunge Path: Turn this option on (Yes), the toolpaths will be converted to the plunge pattern.

- ***Plunge Step Type***: Specify a type to define the stepover value. Please refer to the ***Stepover*** on the ***Tolerance and Steps*** page to find the details.
- ***Plunge Step Value***: Specify a plunge stepover value.

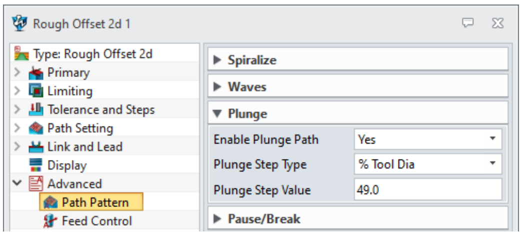


Figure 88 Plunge

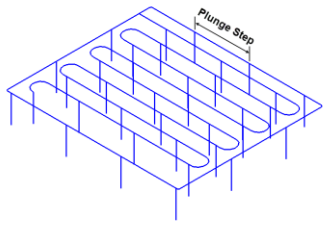


Figure 89 Plunge Step

4) Pause/Break

If users are not sure the tool-life could finish the machining of the whole toolpaths, particularly for the composite cutter. Here is the toolpaths breaks function that enables users to break the toolpaths in terms of the machining time or milling distance, for checking and tool damage or change the inserts.

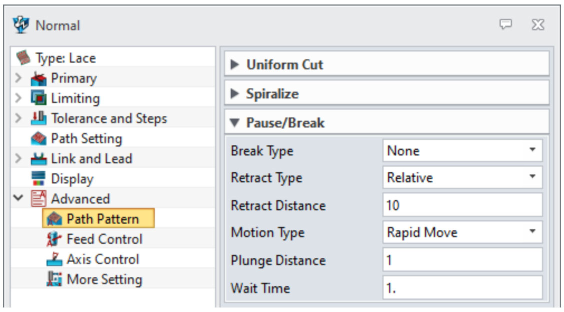


Figure 90 Pause/Break

Break Type: Define if the toolpaths breakpoint according to machining time or toolpaths' length.

- ***None***: It will not generate the breakpoints on current toolpaths.
- ***Time***: Generate the breakpoints in terms of the cutting time, its unit is minute.
- ***Length***: Generate the breakpoints in terms of the toolpaths' distance, its unit is millimeter.

Retract Type: Select the ***Absolute*** or ***Relative*** as the retract type.

- ***Retract Distance (Absolute)***: Specify a fixed Z value in the activated frame, which will be the retract plane of the breakpoints.
- ***Retract Distance (Relative)***: Specify the distance which is related to the breakpoints in the Z direction, which will be the retract points of the breakpoints.

Motion Type: Select ***Feed Move*** or ***Rapid Move*** as the break motion.

- ***Feed Move***: The additional break toolpaths' speed and feed rates will be the same as the cutting toolpaths'.
- ***Rapid Move***: The additional break toolpaths' speed and feed rates will be the same as the rapid toolpaths'.

Plunge Distance: Specify the plunge distance of the break toolpaths.

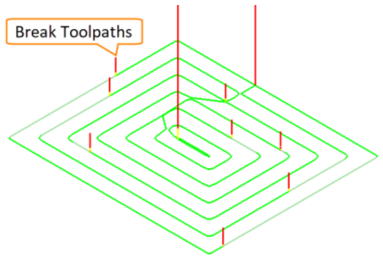


Figure 91 Break Toolpaths

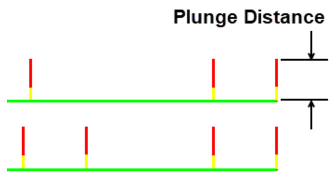


Figure 92 Plunge Distance

Wait time: Specify the waiting time(second) for each breakpoint.

5) Feed Control

AFC (Advanced Feed Control) is an alternative method of calculating QM feeds. It will automatically change the feed rates according to the chip-load, which smoothes the toolpaths to increase the tool life.

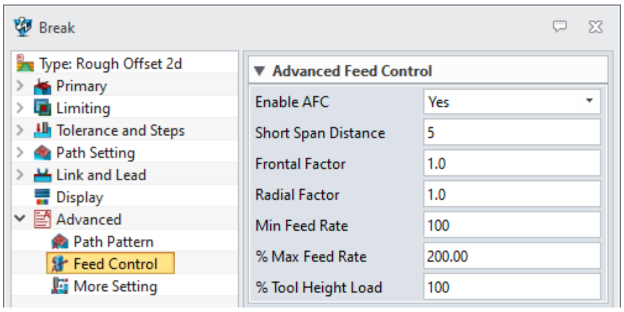


Figure 93 Feed Control

Enable AFC: Turn this option on(Yes), more AFC parameters will be displayed.

- ***Short Span Distance***: The value used to remove small spans with higher feed between long spans. A larger value will make the milling process smoother and slower.
- ***Frontal Factor***: A more rigid tool, with more flutes, has this coefficient higher. For the Frontal Factor, we divide the feed (rate) by this value. The valid range is from 1 to 20.
- ***Radial Factor***: A more rigid tool, with more flutes, has this coefficient lower.
- ***Min Feed Rate***: Specify the minimum feed rate.
- ***%Max Feed Rate***: Specify the maximum feed rate according to the percentage of the minimum feed rate.
- ***%Tool Height Load***: Tool loading as a % of tool height.

6) More Setting

Safe RAPID Traversal: Turn this option on(Yes), the traversal toolpaths will be separated into two segments, which moves ΔX and ΔY with the same feed rate.

Note: *In general, the CNC machine has two types of rapid movements, which could be defined in the configuration of the CNC controller. So the rapid movement of the toolpaths needs to match the machine setting, otherwise, it might cause the collision. And the Safe rapid traversal enables users to switch the rapid mode.*

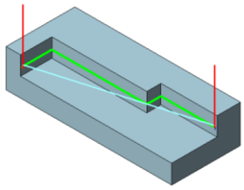
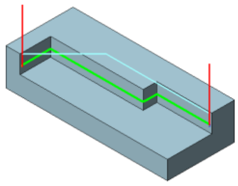
Safe RAPID Traversal	No	Yes
Toolpaths		

Figure 94 Safe RAPID Traversal

Limit Stepover: The stepover will be limited, and less than 90% of the tool diameter to avoid the material island left from each roughing layer. The default setting is "Yes".

Enable Arcs: Check on the options on this page, G02/G03 will be outputted in NC code if there are any arc toolpaths on the corresponding plan or movement.

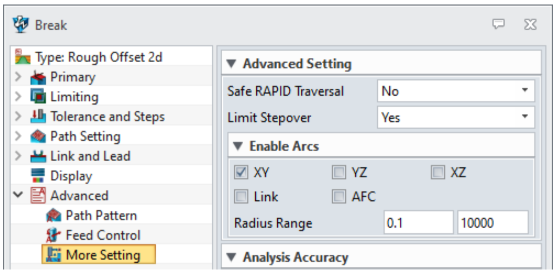


Figure 95 Advanced Setting and Enable Arcs

Radius Range: Define the minimum and maximum radius of the arc toolpaths that will be outputted G02/G03.

User Value: ZW3D enables users to adjust the accuracy of the model via this option for getting better toolpaths only when meeting toolpaths generation issue on the steep region or complicated model. There are two ways to define this option: set a specific value, or drag the scroll bar.

Note: *The toolpaths generating time will increase rapidly when adjusting this option for higher accuracy.*

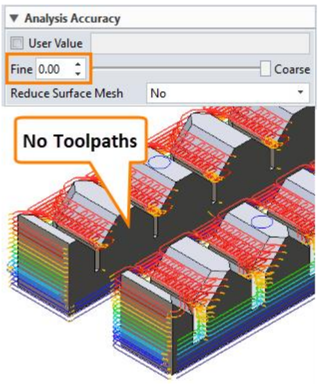
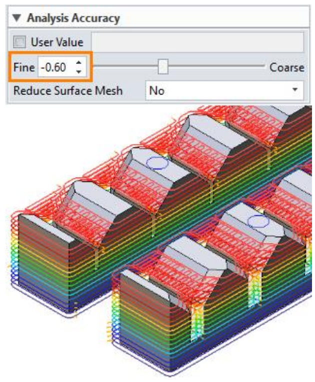
User Value	0	-0.6
Toolpaths		

Figure 96 User Value

Reduce Surface Mesh: Turn this option on(Yes), ZW3D will reduce the mesh of some surfaces to improve the toolpaths calculation efficiency.

2.3 Rest Roughing Operation

Rest roughing is using a smaller tool to eliminate the large terraces and to rough areas of the model that the large roughing cutting tool could not reach, such as pockets and corners. ZW3D offers 3 ways to achieve the rest roughing pattern, including reference operation, reference tool, and rest stock. The reference tool function has been introduced on the ***Reference Tool*** page in chapter 2.2.2, and reference operation and rest stock methods will be demonstrated below.

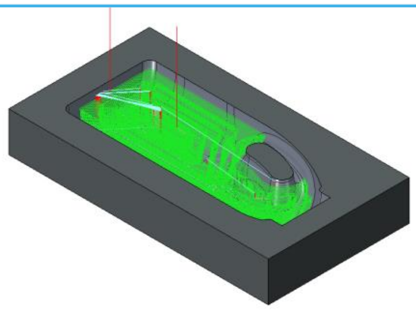


Figure 97 Roughing toolpaths

Task: Create a rest operation by reference operation.

Open the **Rest roughing operation.Z3** file and go into CAM space.

STEP 01 Right-click the **Rough Offset 2d 1** operation to duplicate a new operation for rest roughing.

Note: *Duplicate an existing operation will inherit all the parameters and setting from it.*

STEP 02 Select an existing one as the reference in the new operation.

Figure 98 Create a new operation

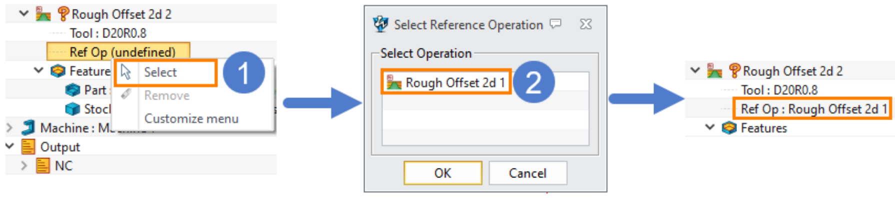


Figure 99 Select a reference operation

STEP 03 Specify a smaller tool, add the same machining features and define the parameters in the operation.

STEP 04 Calculate the operation.

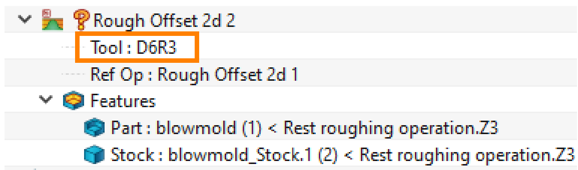


Figure 100 Define operation

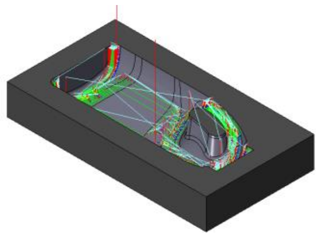


Figure 101 Rest roughing toolpaths by reference operation

Task: Create a rest operation by rest stock.

STEP 01 Execute ***Solid Verify*** on the **Rough Offset 2d 1** operation and create a rest stock.

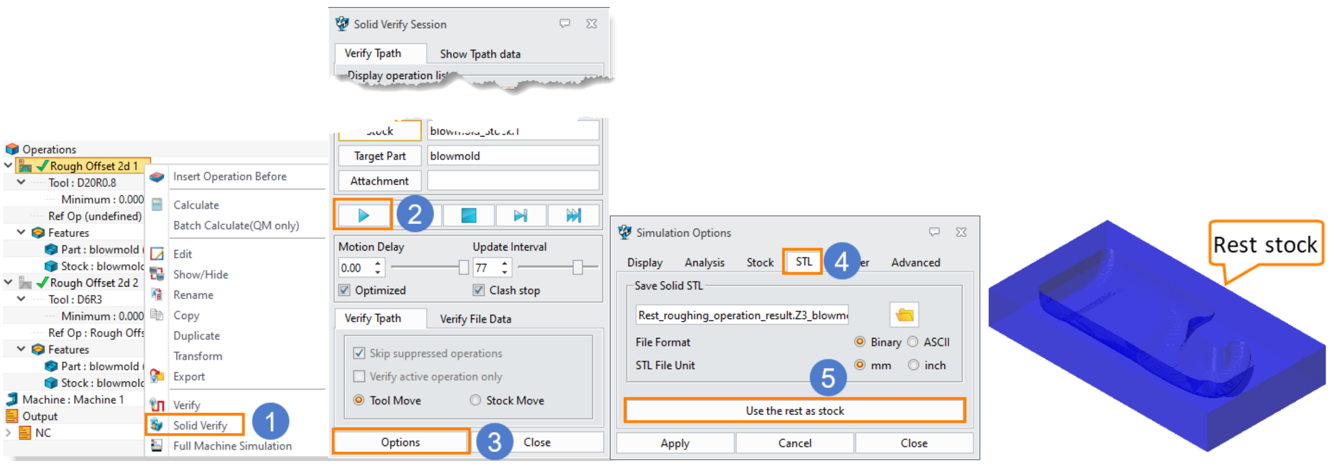


Figure 102 Create a rest stock

STEP 02 Duplicate the **Rough Offset 2d 1**operation, then add the rest stock instead of the existing stock in the new operation.

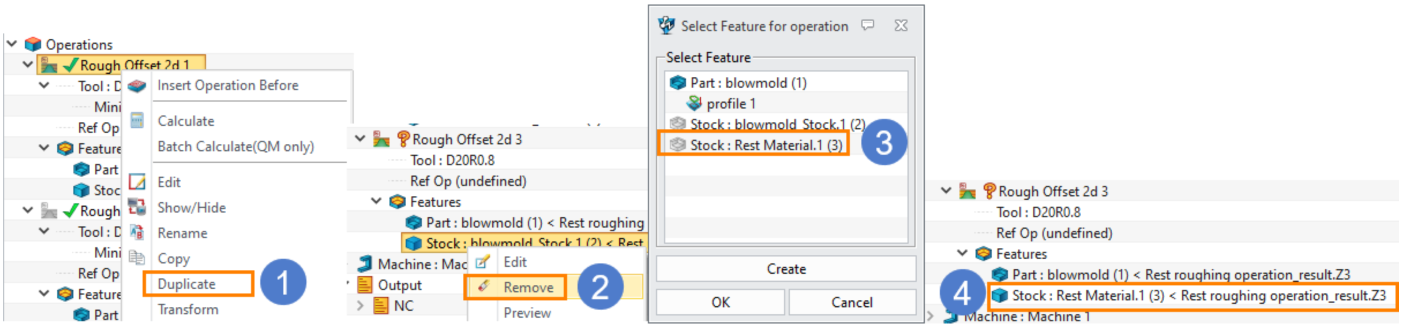


Figure 103 Add the rest stock

STEP 03 Change a smaller tool and calculate the operation.

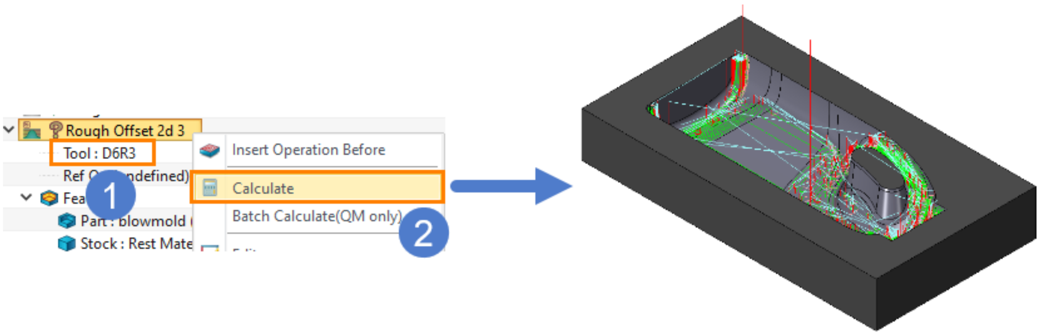


Figure 104 Rest roughing toolpaths by rest stock

3 Finishing

The purpose of finishing operation is different from roughing's. Roughing focuses on removing the excess material rapidly, so it will use a big step and tool. But finishing focuses on dimensional accuracy and surface quality, it will use a high speed, smaller step, and suitable tool to ensure a high-quality result.

Most parameters in finishing operation are the same as roughing's, so we will only introduce the different ones or the ones that only exist in it. In addition, each finishing operation has its own pattern generation theory and suitable application scenarios. Those will be introduced one by one.

***Note:** For illustration, the toolpath parameters in the following figures are different from the actual machining.*

3.1 Lace Operation

Lace operation will create a group of parallel toolpaths in XY plane first, and then project it to the surface of 3D model, which is suitable for the finishing of shallow regions.

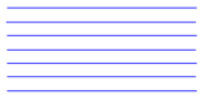


Figure 105 Lace Toolpaths pattern

In general, there are two requirements to create the toolpaths of Lace operation.

- Specify a tool
- Define the machine features (part or part with profile boundary)

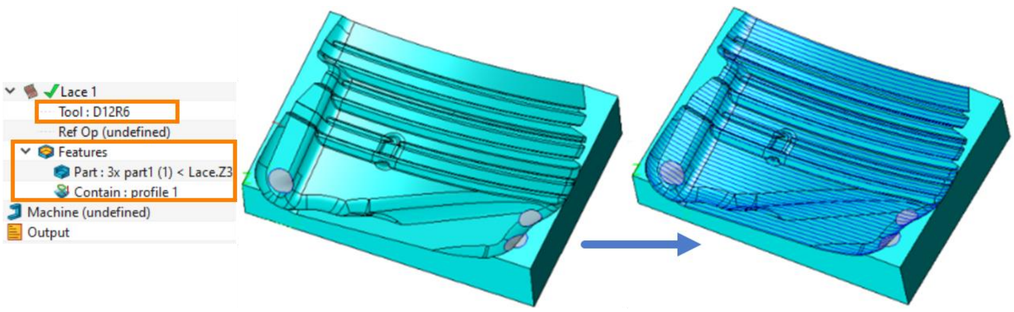


Figure 106 Lace operation

Some key parameters of all operations are introduced here, including some common parameters and unique options.

- **Limiting Parameters-> Boundaries**

Containment type: 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.

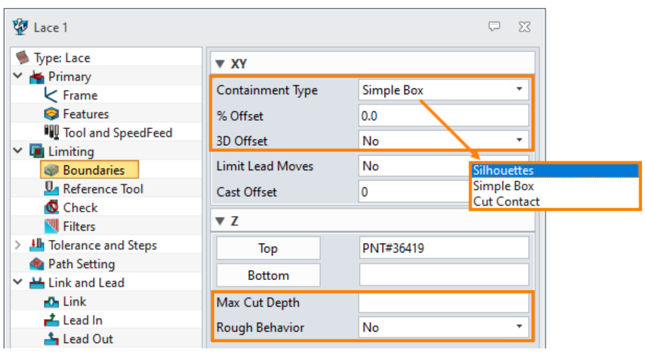


Figure 107 Limiting—boundaries

Simple box: Create a smallest cube surround the part, use its projection contour of the part as the limiting boundary.

Silhouette: Create the maximum projection contour of the part as the limiting boundary.

Cut contact: It is the boundary that limits the tool contact point rather than the tooltip position.

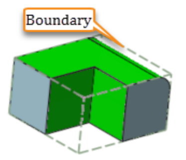
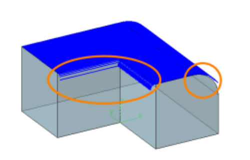
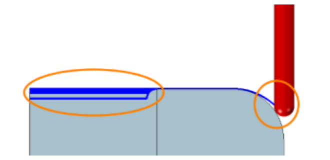
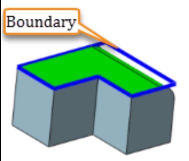
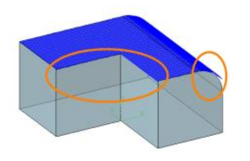
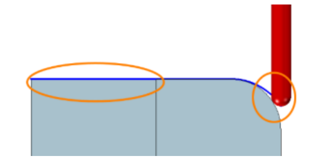
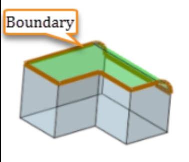
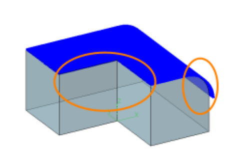
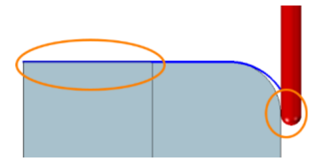
Type	Boundary	Toolpaths	
Simple box			
Silhouette			
Cut contact			

Figure 108 Boundaries

Notes: *The green faces will join the toolpaths generation.*

% Offset: Specify the containment offset value by the percentage of the tool diameter.

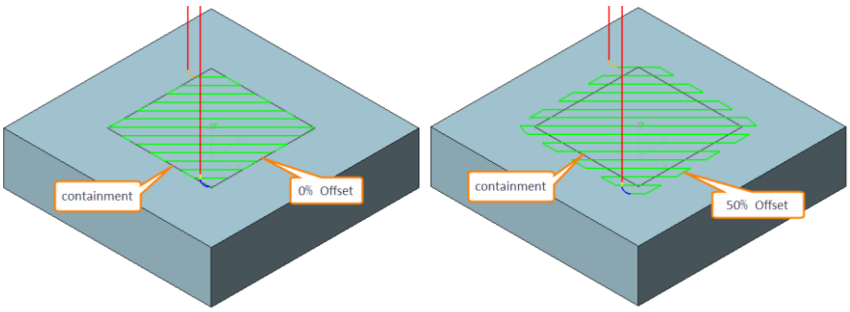


Figure 109 % Offset

3D offset: Specify whether the offset is 3D or 2D.

Rough Behavior: Specify whether to allow lace operation to generate multi-layer toolpaths for roughing (normally lace operation will generate only one layer for finishing).

Max Cut Depth: Specify the max cut depth when ***Rough Behavior*** option is yes.

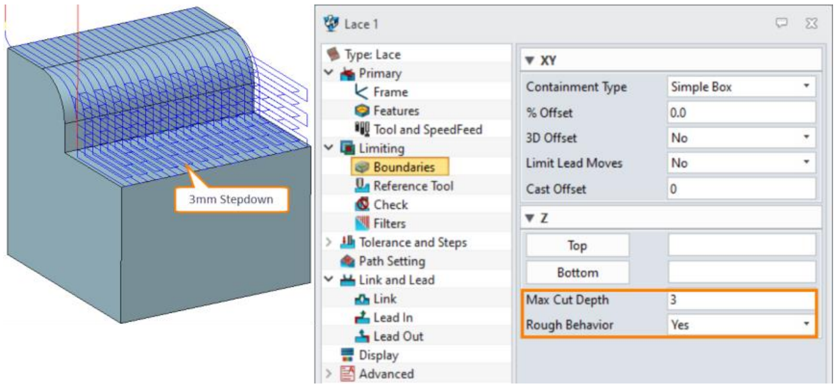


Figure 110 Rough Behavior and Max Cut Depth

- **Limiting Parameters -> Filters**

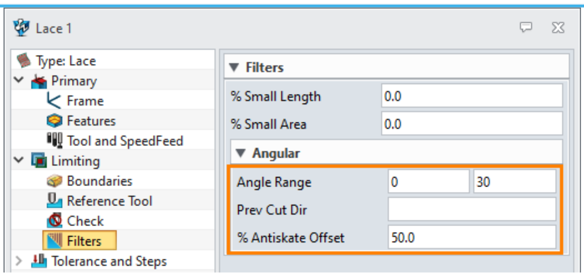


Figure 111 Filter

Angle Range: Specify the angle range to generate lace toolpaths (0~90°).

a) The angle range is defined as the angle between the surface normal and the XY plane normal.

b) In order to make full use of its advantage, lace operation is often used to process flat and shallow regions, the recommended angle range is 0-60 degrees.

Prev Cut Dir: To better understand this parameter, please refer to cases below and the summary after that.

a) First lace operation, Cut Angle=0, without setting Pre Cut Dir. To ensure surface quality, need to generate supplementary toolpaths on the Nonuniform toolpaths area.

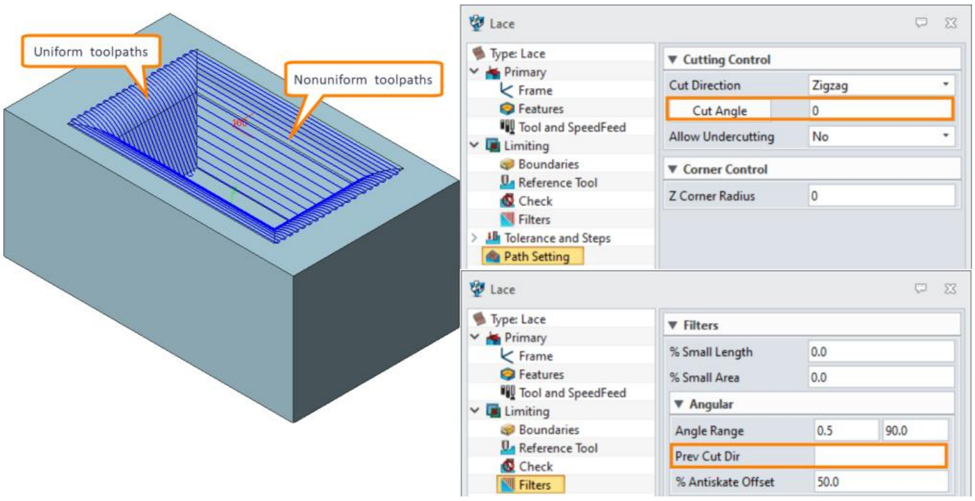


Figure 112 Cut Angle=0, without setting Pre Cut Dir

b) Create the second lace operation as supplementary toolpaths, Cut Angle=90, Pre Cut Dir=0. Please notice the green toolpaths, it will only generate on the Nonuniform toolpaths area of the first lace operation.

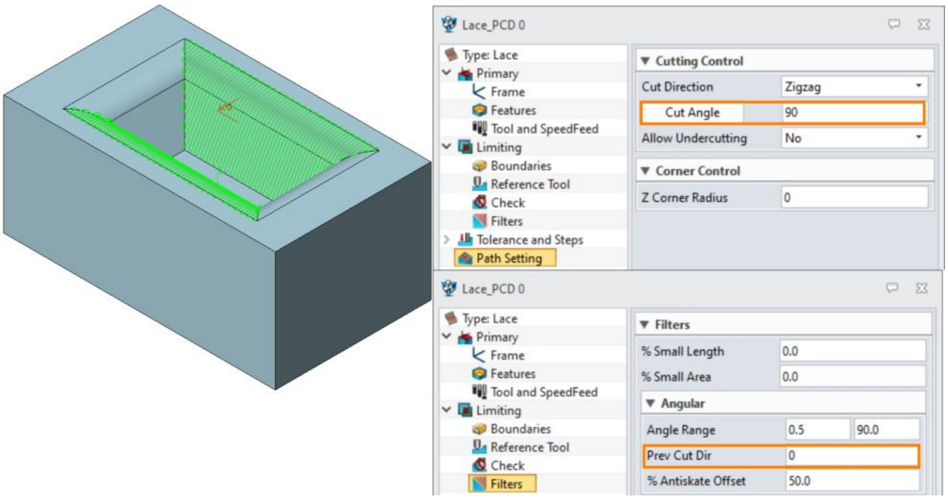


Figure 113 Cut Angle=90, Pre Cut Dir=0

c) Double click to show both operations.

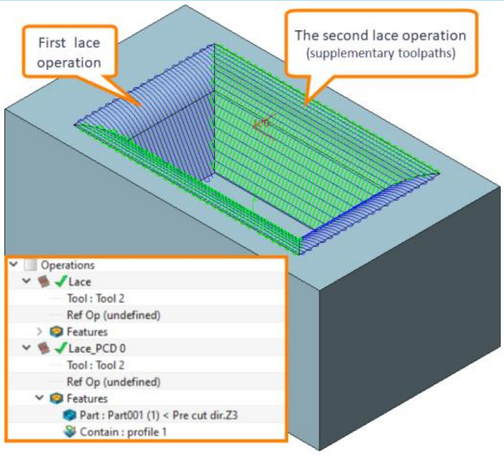


Figure 114 Combination of both toolpaths

Summary: For lace operation, it will generate uniform toolpaths on the surfaces that are perpendicular to the toolpaths cutting direction, but on surfaces that are parallel to the toolpaths cutting direction, it is nonuniform. To ensure surface quality, set ***Prev Cut Dir*** parameter to generate supplementary toolpaths on surfaces that are parallel to the toolpaths cutting direction. It needs to be specified according to ***Cut Angle*** parameter.

Tips: Cut Angle *is perpendicular to* ***Prev Cut Dir****, as mentioned above, Cut Angle=90, Pre Cut Dir=0.*

% Antiskate Offset: when it comes to the complex adjacent steep region, lace operation might generate skating toolpaths. Thus, specify a tiny offset value of the clean-up toolpaths in lace operation to avoid skating toolpaths.

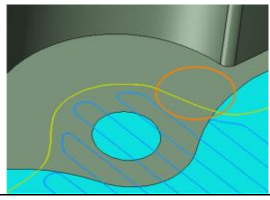
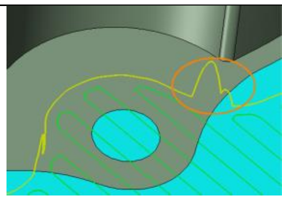
% Antiskate Offset	**120**	**20**
Toopaths		

Figure 115 % Antiskate Offset

- **Path setting**

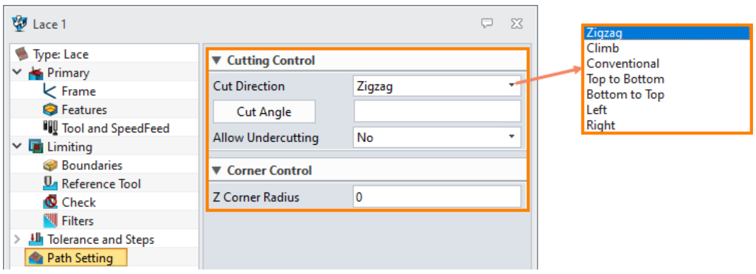


Figure 116 Path setting

Cut direction: ZW3D provides the following cut direction types.

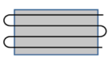
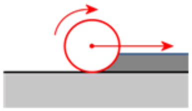
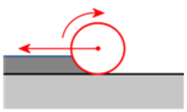
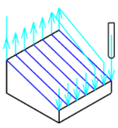
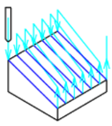
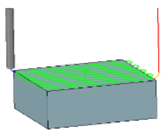
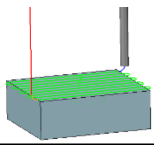
Type	Zigzag	Climb	Conventional	Bottom to top	Top to bottom	Left	Right
Pattern							

Figure 117 Cut direction

Notes: *The tool position in the above image is the start point.*

Cut angle: The cut angle is the angle between toolpaths and the positive direction of X-axis. After specifying the cut angle. ZW3D will automatically change the start point to generate continue toolpaths. Please refer to the figure below: when the cut angle is 0~90°, the start point is in the lower right corner of the workpiece.

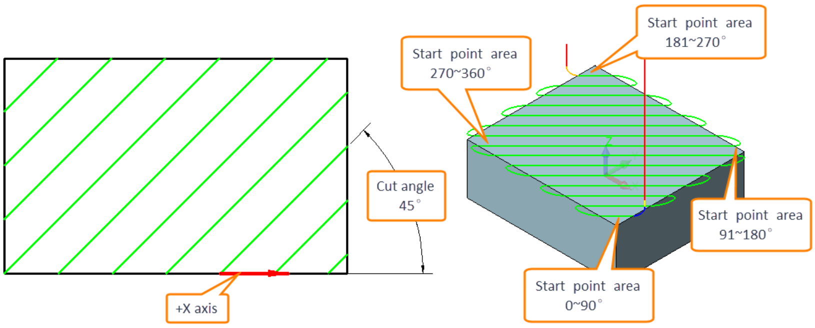


Figure 118 Cut angle

Allow undercutting: When the option is on and select a proper tool such as lollipop or wheel cutter, ZW3D will detect and generate the toolpaths on the back-off shape.

Z Corner Radius: Add a fillet in vertical toolpaths with a specific radius.

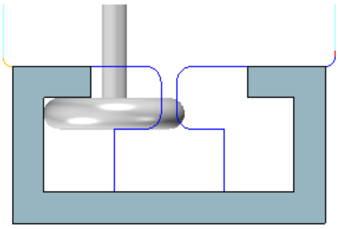


Figure 119 Allow undercut

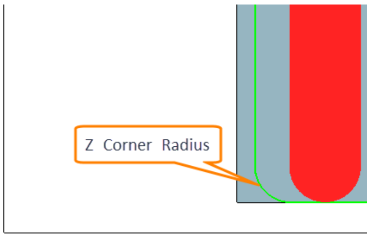


Figure 120 Z Corner radius

Task: Create Lace toolpaths

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

STEP 02 Select ***Lace*** finishing operation.

STEP 03 Specify features (part and profile 1) in CAM manager.

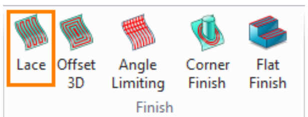


Figure 121 Lace operation

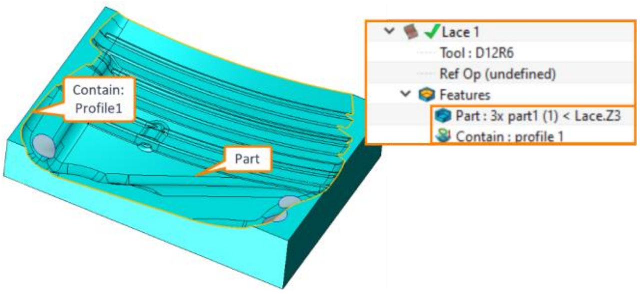


Figure 122 Specify features

STEP 04 Specify a tool(D12R6), then calculate the toolpaths with default settings.

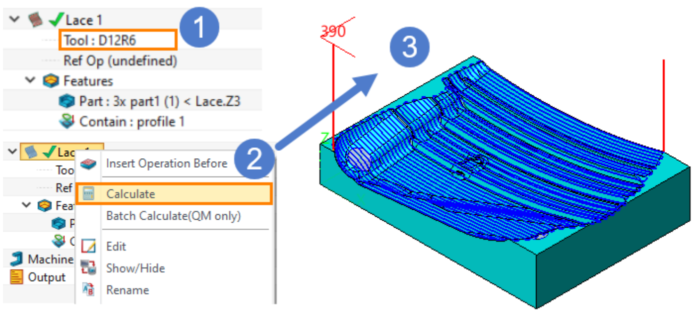


Figure 123 Specify tool and calculate

3.2 Offset 3D Operation

Offset 3D will generate toolpaths with united 3d stepover all over the part and following the silhouette or 3D boundary profile. If the boundary profile is not specified, ZW3D will use the part silhouette as a base to offset and generate the whole toolpaths.

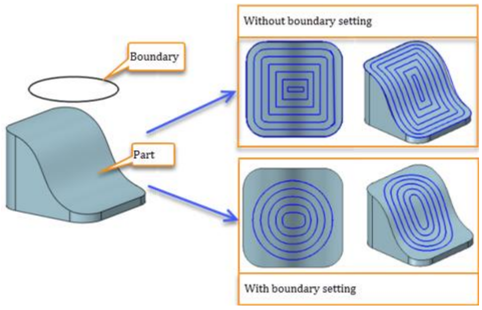


Figure 124 Offset 3D Toolpaths pattern

In general, there are two requirements to create the toolpaths of Offset 3D operation.

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

Some key parameters of all operations are introduced here, including some common parameters and unique options.

- **Tolerance and Steps**

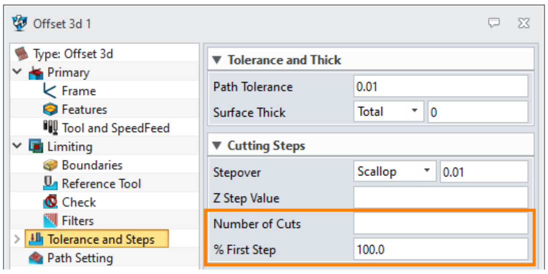


Figure 125 Offset 3D operation—Tolerance and Steps

Number of Cuts: Specify the number of toolpaths generated from the boundary. ZW3D will generate complete toolpaths if it is blank.

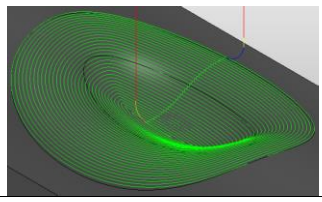
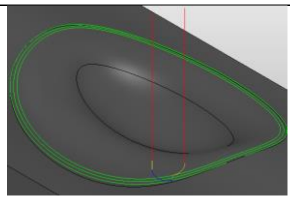
Number of Cuts	**0**	**3**
Toopaths		

Figure 126 Number of Cuts

% First Step: Specify the first stepover value by the percentage of the tool diameter.

- **Path Setting**

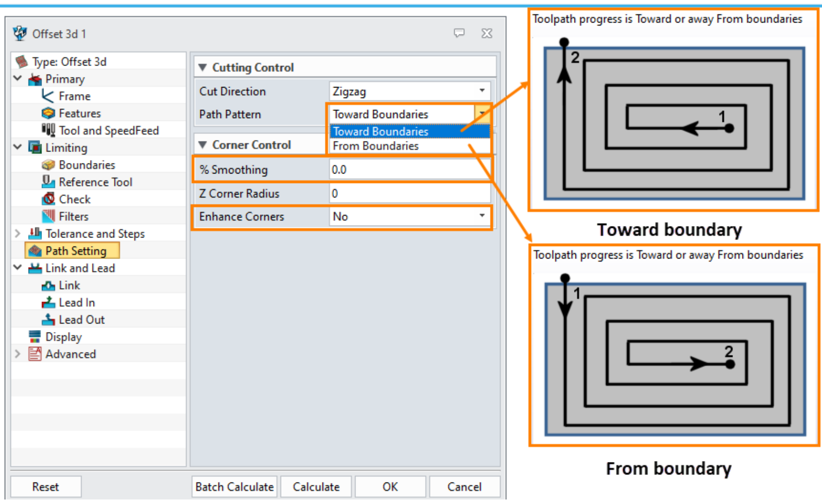


Figure 127 Offset 3D operation—Path Setting

Path pattern: Toward boundaries and from boundaries.
% Smoothing: Specify the smoothness of the toolpaths at the corners.

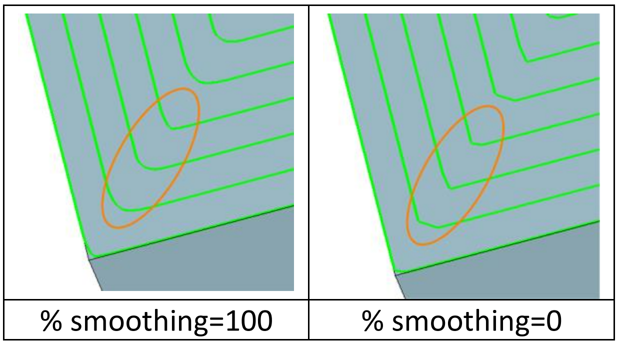


Figure 128 %Smoothing

Task: Create Offset 3D toolpaths

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

STEP 02 Select ***Offset 3D*** operation.

STEP 03 Specify features (part and profile 1) in CAM manager.

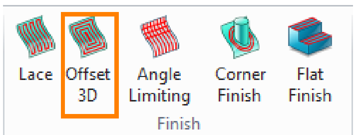


Figure 129 Offset 3D operation

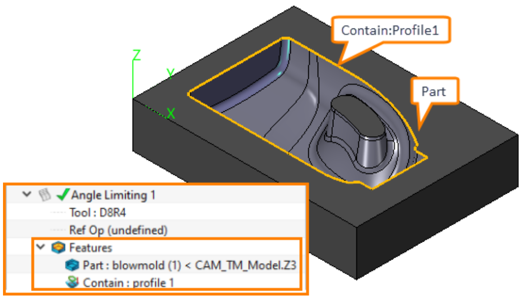


Figure 130 Specify features

STEP 04 Specify a tool(D8R4) in CAM manager and stepover to 5mm.

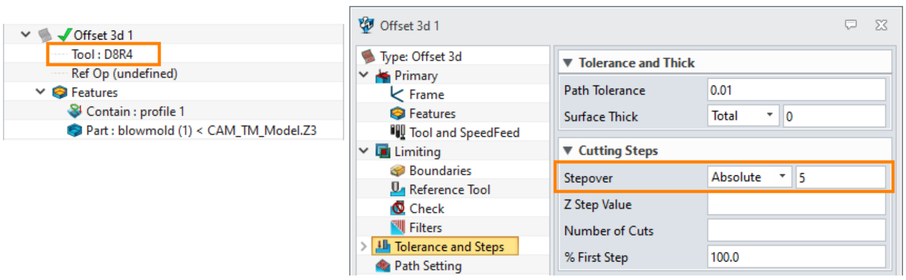


Figure 131 Specify tool and stepover

STEP 05 Calculate the toolpaths.

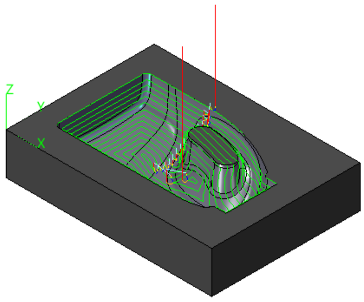


Figure 132 Toolpaths calculation

3.3 Drive Curve Operation

Drive curve operation will project the guiding profiles on the model to generate toolpaths, and it enables to offset the profiles in both sides to generate toolpaths by defining the number of cuts.

Note: *The guiding profiles can be opened or closed.*

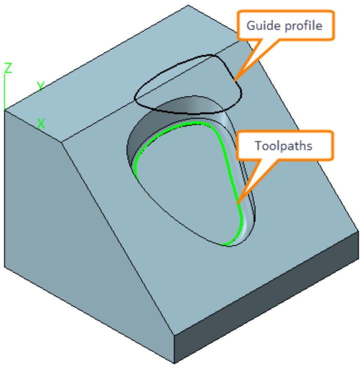


Figure 133 Drive Curve toolpaths pattern

In general, there are three requirements to create the toolpaths of Drive Curve operation.

- Specify tool
- Specify features (part or part with profile boundary)
- Specify drive curve (the profile type should be set as part)

Tips: *The drive curves can be closed or opened*

Some key parameters of all operations are introduced here, including some common parameters and unique options.

- **Tolerance and Steps**

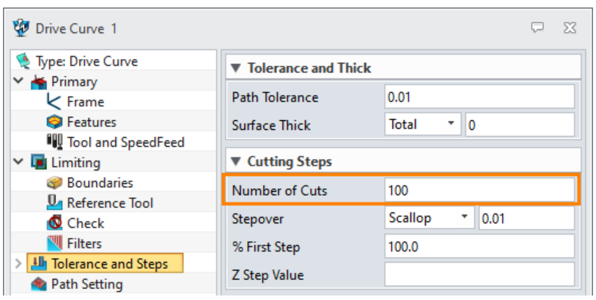


Figure 134 Drive Curve operation—Tolerance and steps

Number of Cuts: Specify the number of toolpaths generated by offsetting from guide profiles. If it is empty, ZW3D will generate the toolpaths along with the guide profile. Otherwise, it will generate toolpaths on both sides.

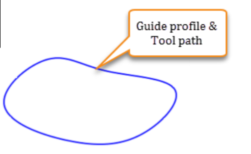
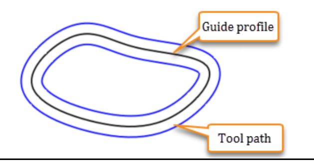
Number of cuts	0	1
Toolpaths pattern		

Figure 135 Drive Curve operation – Number of cuts

Tips: *To make toolpaths within the guide profile, users need to add a contain profile feature.*

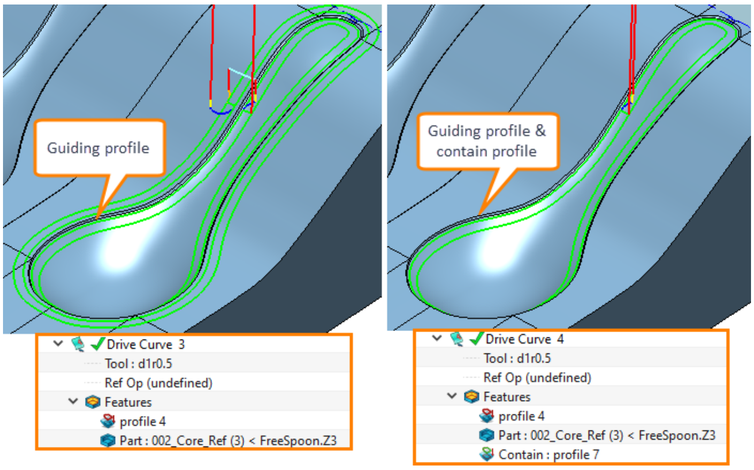


Figure 136 Tips case

- **Path setting**

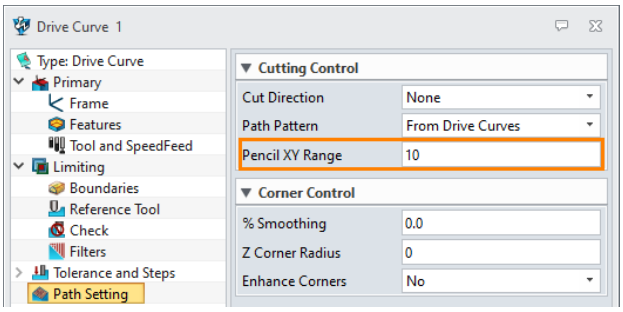


Figure 137 Drive Curve operation—Path Setting

Pencil XY Range: According to the generation theory of drive curve operation, it will project the guiding profiles on the model to generate toolpaths. However, in some cases, it can not generate toolpaths along with the profile since the distance between the profile and the walls of part is smaller than the cutter's radius (please refer to situation 1 below). Thus, specify a value in *Pencil XY Range* to detect the machining area to generate pencil toolpaths guided by the drive curve (please refer to situation 2).

a) Situation 1: Pencil XY Range=0, it cannot generate toolpaths along with the profile (drive curve).

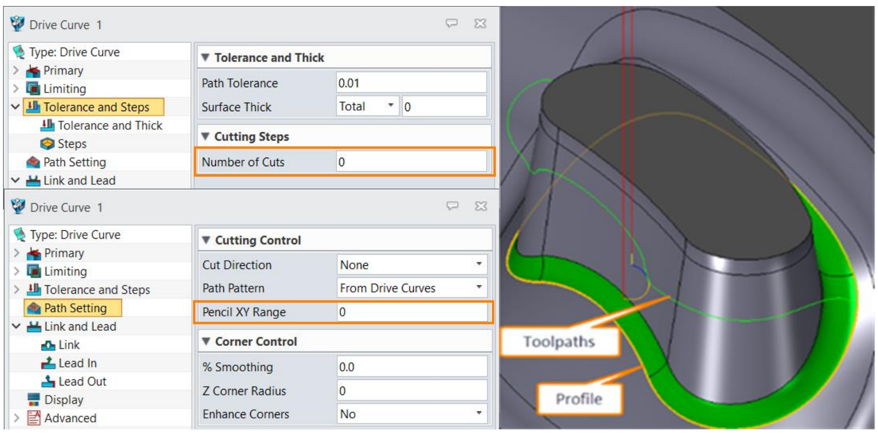


Figure 138 Pencil XY Range=0

b) Situation 2: Pencil XY Range=10, it will extend the detection area and generate pencil toolpaths that guided by the drive curve within the given pencil XY range value.

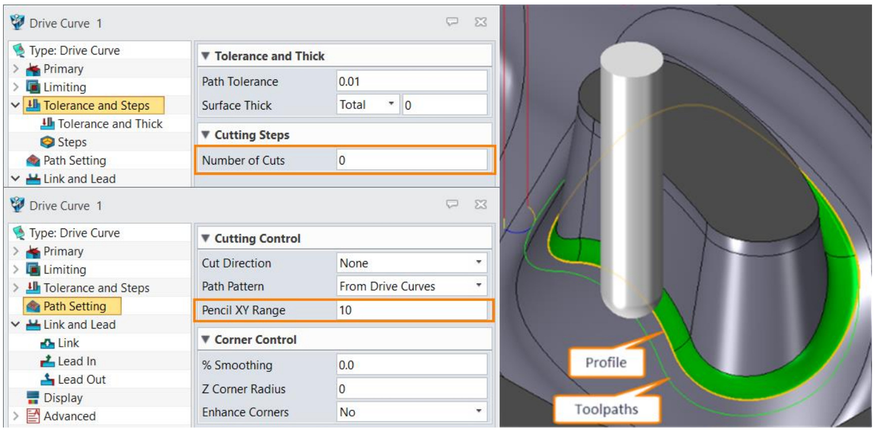


Figure 139 Pencil XY Range=10

Tips: *The corner surface with green color and the tool have the same radius, R=5mm.*

Task: Create Drive Curve toolpaths

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

STEP 02 Select ***Drive Curve*** operation.

STEP 03 Specify features (part) in CAM manager.

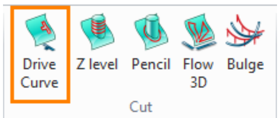


Figure 140 Drive Curve operation

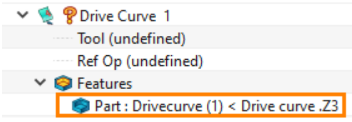


Figure 141 Specify features—part

STEP 04 Specify features (profile) as the drive curve.

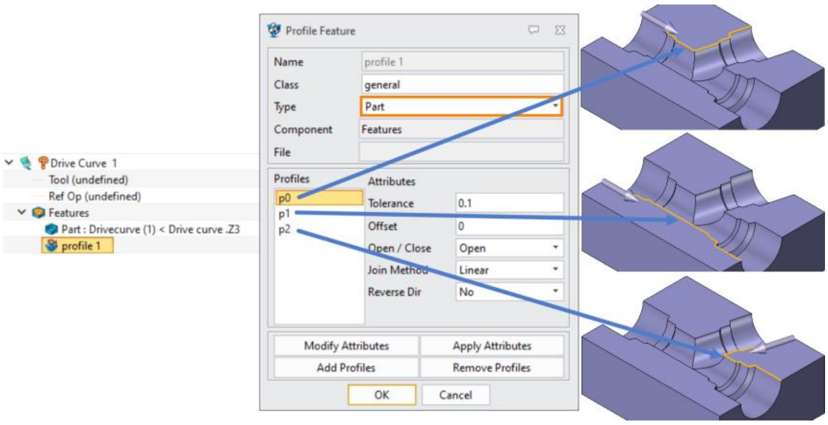


Figure 142 Specify features—drive curves

STEP 05 Specify features (profile) as containment.

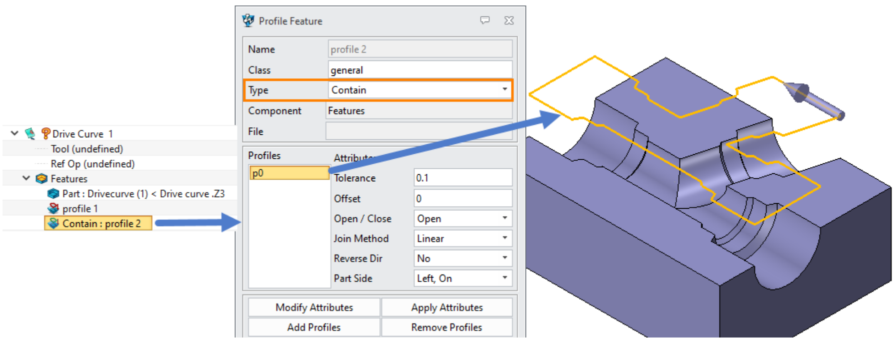


Figure 143 Specify features—containment

STEP 06 Specify tool(D18R9) and related parameters.

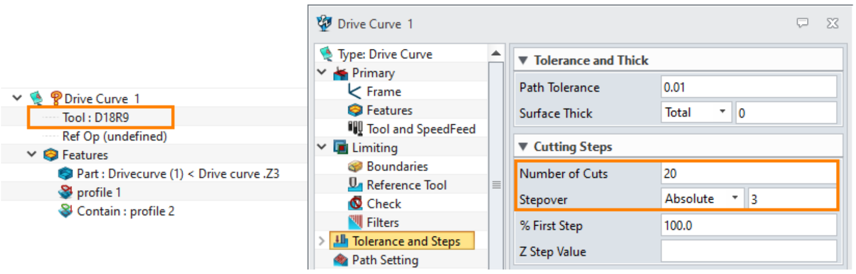


Figure 144 Specify tool and related parameters

STEP 07 Calculate the toolpaths.

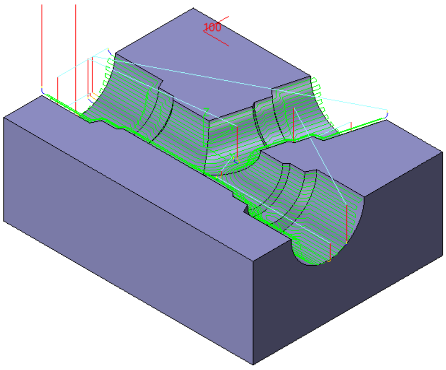


Figure 145 Toolpaths calculation

3.4 Flow 3D Operation

Flow 3D operation will morph a pair of guide profiles to generate a set of toolpaths which has a uniform 3D stepover. The guide profiles can be opened or closed.

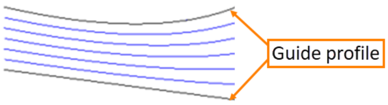


Figure 146 Flow 3D toolpaths pattern

Notes:

1. *The start point and direction of guide profiles should be coincident, otherwise, it will generate twisted toolpaths.*
2. *Two guide profiles should be separated.*

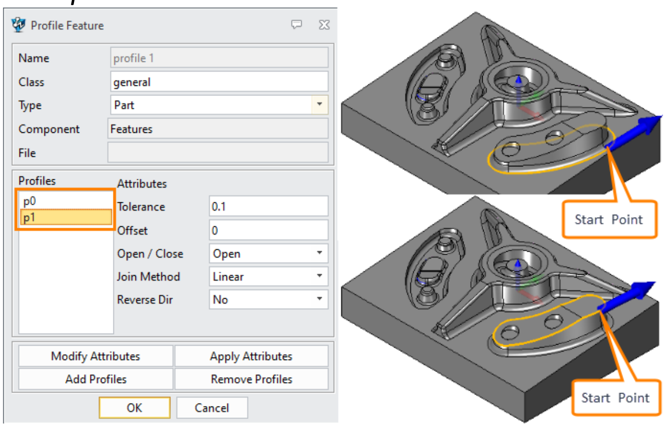


Figure 147 Separated profiles, coincident direction and start points

In general, there are three requirements to create the toolpaths of Flow 3D operation.

- Specify tool
- Specify features (part or part with profile boundary)
- Specify a pair of guide profiles (the profile type should be part)

Task: Create Flow 3D toolpaths

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

STEP 02 Select ***Flow 3D*** operation.

STEP 03 Specify features(part) in CAM manager.

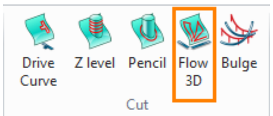


Figure 148 Flow 3D operation

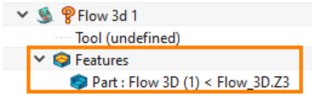


Figure 149 Specify features—part

STEP 04 Specify features(guide profiles).

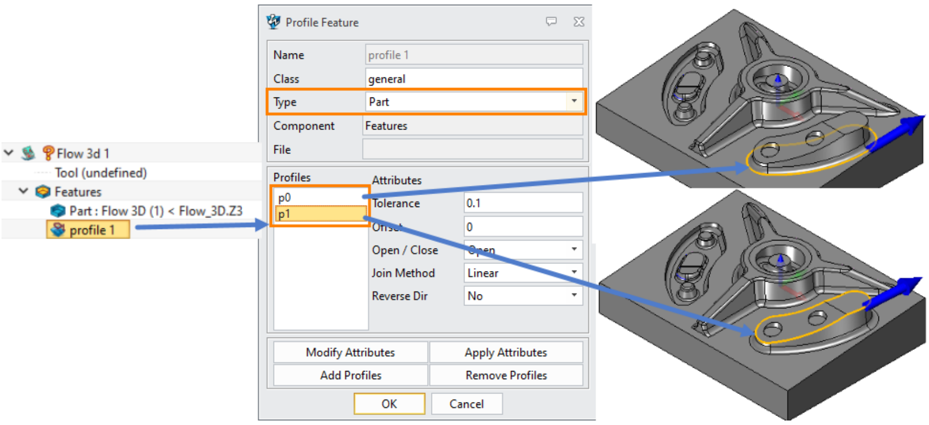


Figure 150 Specify features—guide profiles

STEP 06 Specify tool(2mm Ball Endmill).

STEP 07 Calculate the toolpaths.

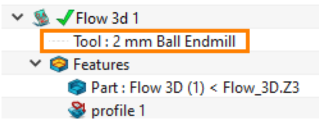


Figure 151 Specify tool

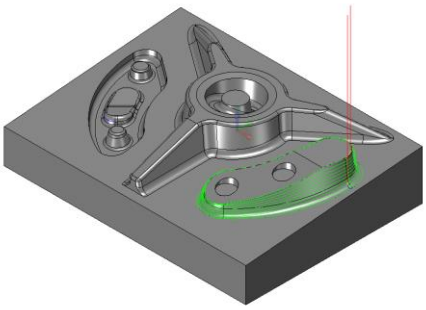


Figure 152 Toolpaths calculation

3.5 Z Level Operation

Z level operation will generate a set of contour toolpaths in Z-axis direction which is suitable for the finishing of the steep region.

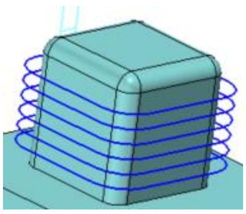


Figure 153 Z Level toolpaths pattern

In general, there are two requirements to create the toolpaths of Z Level operation.

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

Some key parameters of all operations are introduced here, including some common parameters and unique options.

- **Boundaries**

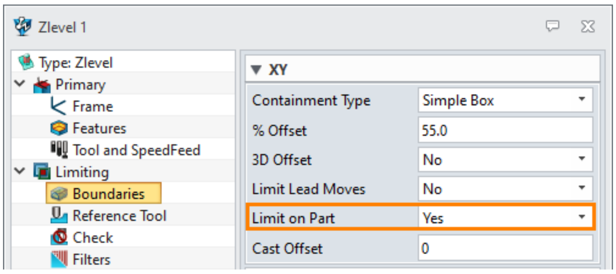


Figure 154 Z Level—Boundaries

Limit on Part: Limit toolpaths on the part.

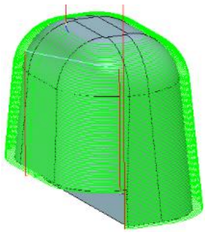
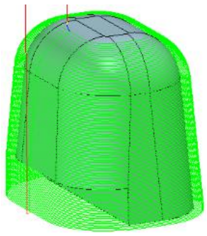
Limit on Part	**Yes**	**No**
Toopaths		

Figure 155 Limit on part

- **Tolerance and Steps**

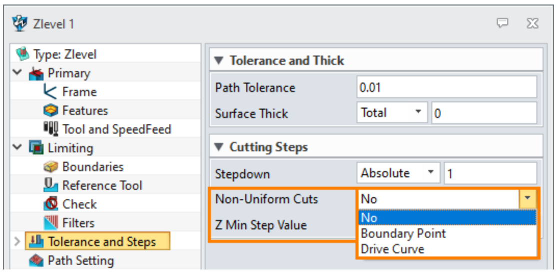


Figure 156 Z Level operation—Tolerance and Steps

Non-uniform cuts: When facing a part which has different draft angled surfaces, the Non-uniform cuts function enable users to define different stepdown setting on regions or use a guiding profile to generate toolpaths to achieve uniform surface finishing on all regions in one operation.

a) ***No:*** ZW3D will generate Z-Level toolpaths with uniform stepdown value.
b) ***Boundary point:*** Divide the geometry into different regions by specifying boundary points, then users can set different stepdown in each region. Please refer to the following steps.

STEP 01 Specify a boundary point on geometry, the region under the specified boundary point can use a different stepdown.

STEP 02 Specify region stepdown for each layer.

STEP 03 Add a layer to the list.

STEP 04 Calculate.

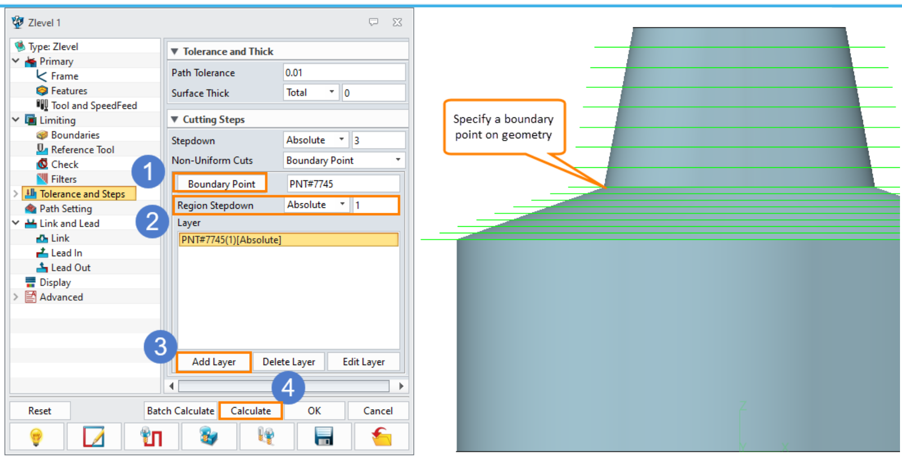


Figure 157 Non-uniform Cuts—Boundary Point

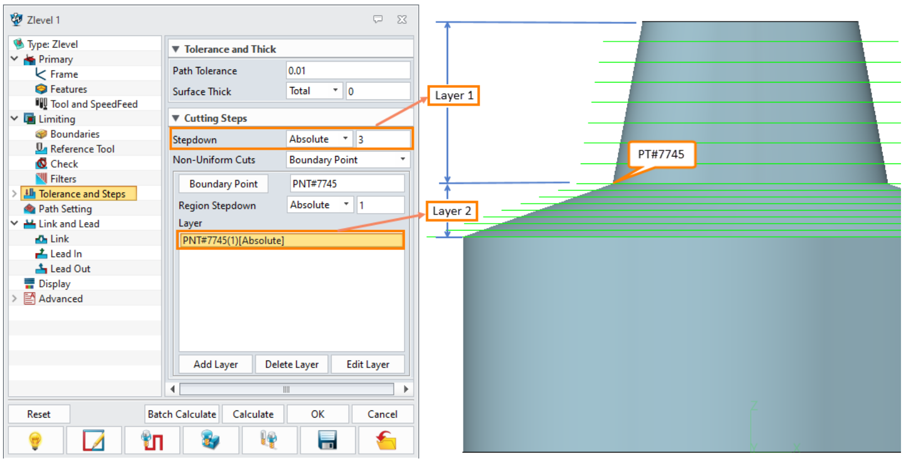


Figure 158 Calculation result

Tips: *To delete or edit layer, users need to select the corresponding layer first.*

c) Drive curve: ZW3D will generate toolpaths according to the given 3D step on the drive curve.

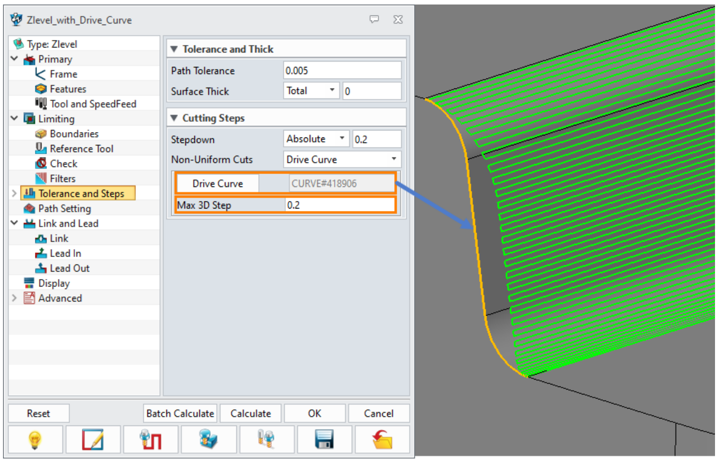


Figure 159 Non-uniform Cuts—Drive Curve

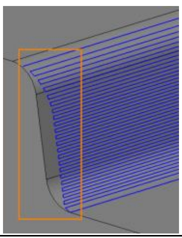
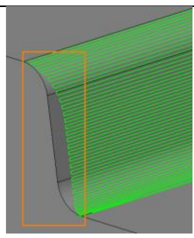
Max 3D step	**Blank**	**0.2**
Toopaths		

Figure 160 Difference between toolpaths with/without drive curve

Z Min Step Value: Specify a minimum step value in the Z direction in addition to the given stepdown value. It will supplement toolpaths in the shallow region to get high-quality surfaces.

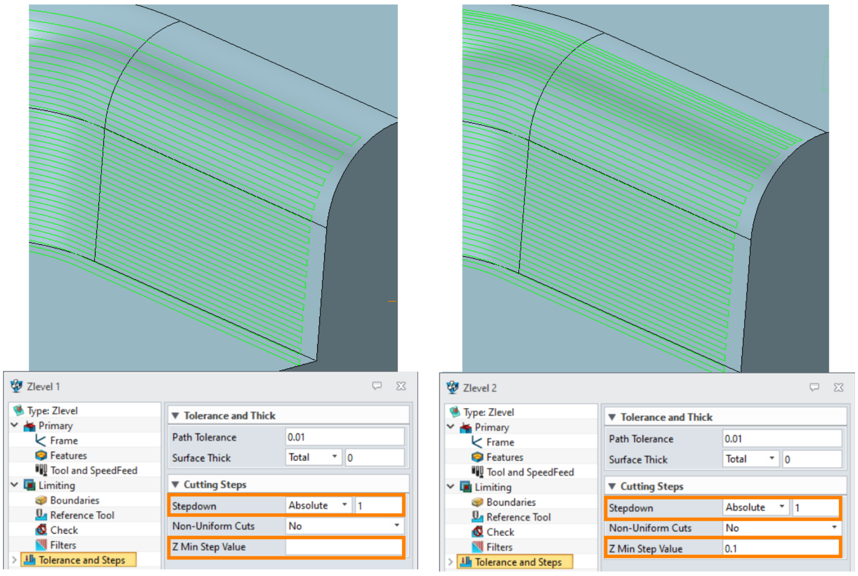


Figure 161 Z Min Step Value

➢ **Path Setting**

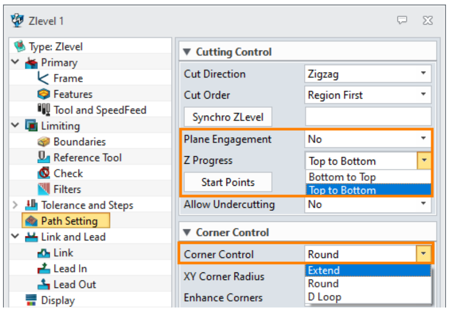


Figure 162 Z Level operation—Path Setting

Plane Engagement: Enforce tool engagement at each planar face.

Z Progress: Specify the machine order, top to bottom, or bottom to top.

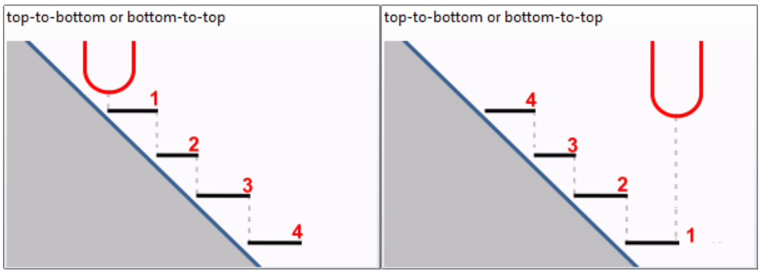


Figure 163 Z Progress

Start Points: Specify the lead in point.

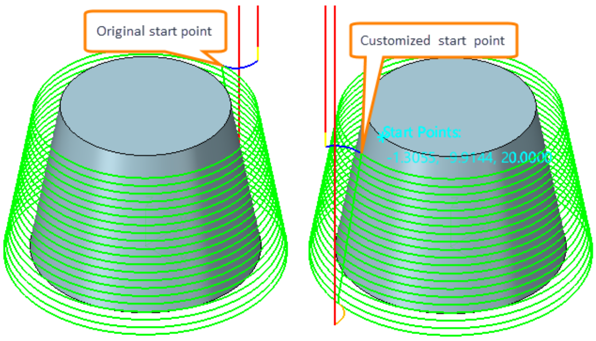


Figure 164 Start point

Corner Control: Specify the toolpaths pattern at the corners.

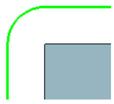
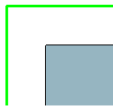
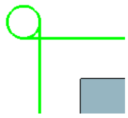
Corner Control	Round	Extend	D Loop
Toolpaths patterns			

Figure 165　Z Level operation—Corner Control

Task: Create Z Level toolpaths

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

STEP 02 Select ***Z level*** operation.

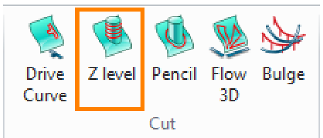


Figure 166　Z Level operation

STEP 03 Specify features (part and profile 1) and tool(D8R4) in CAM manager.

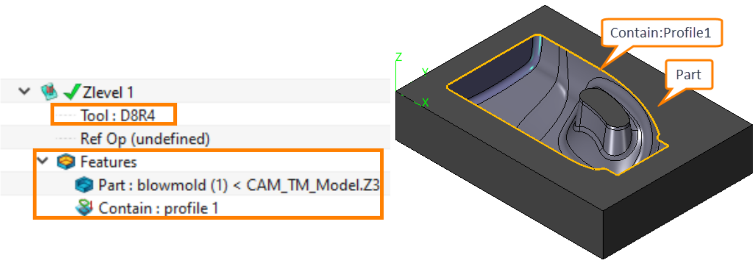


Figure 167　Specify features and tool

STEP 04 Specify angle range of steep regions for generating Z level toolpaths.

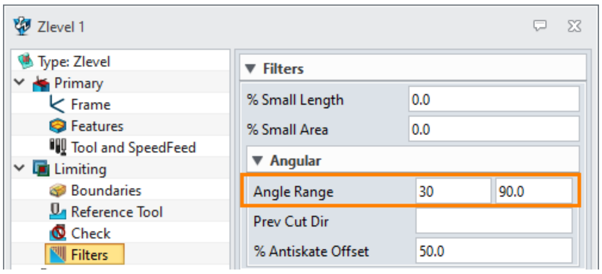


Figure 168　Specify angle range

STEP 05 Calculate the toolpaths.

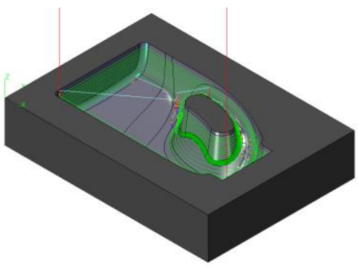


Figure 169　Toolpaths calculation

3.6 Angle Limiting Operation

Angle limiting operation is a composite program. It will distinguish where are the shallow and steep regions in terms of the steep angle value, which set by users, then assign proper toolpaths mode to those regions. ***Lace***, ***Offset 3D,*** or ***HSM Lace*** operation is used for flat areas and ***Z Level*** is used for steep regions, for getting uniform surface finishing on the part with one operation.

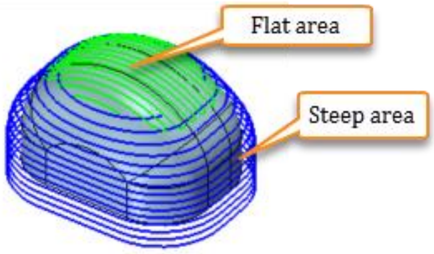


Figure 170 Angle Limiting toolpaths pattern

In general, there are two requirements to create the toolpaths of Angle Limiting operation.

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

Some key parameters of all operations are introduced here, including some common parameters and unique options.

- **Tolerance and Steps**

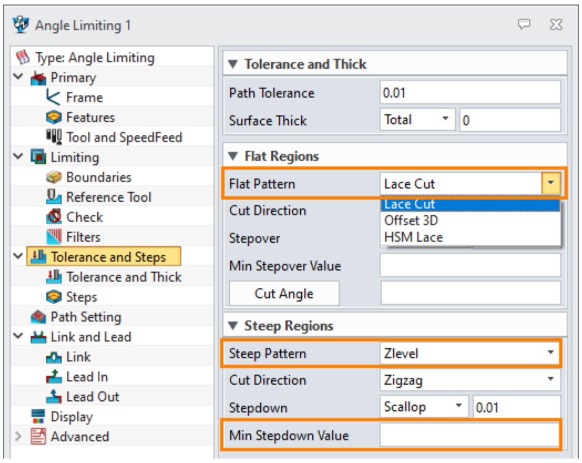


Figure 171 Angle Limiting operation—Tolerance and Steps

Flat Pattern: Specify toolpaths pattern for the flat regions, ***Lace Cut*** or ***Offset 3D*** is recommended.
Steep Pattern: Specify toolpaths pattern (***Z level***) for the steep regions.
Min Stepdown Value: Specify a minimum step value in Z direction in addition to the given stepdown value to get a better surface quality. It is similar to ***Z Min Step Value*** in ***Z Level*** operation.

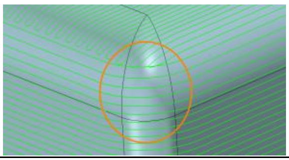
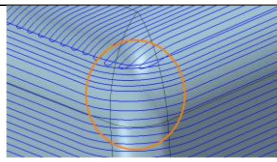
Min Stepdown Value	Stepdown: **1** (absolute) Min Stepdown Value: **Blank**	Stepdown: **1** (absolute) Min Stepdown Value: **0.2**
Toopaths		

Figure 172 Min Stepdown Value

- **Path Setting**

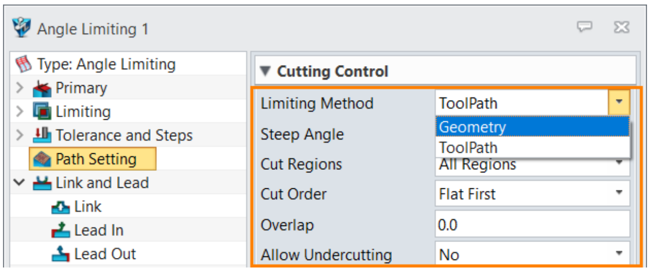


Figure 173 Angle Limiting operation-—Path Setting

Limiting Method: Specify a method to detect the steep regions.
a) Geometry: Detect the steep regions based on the angle between the normal of the geometric surface and the normal of the XY plane.
b) ToolPath: Detect the steep regions according to the angle between the normal of the outer contour of the toolpaths and the normal of the XY plane.
Steep Angle: Specify an angle value to distinguish the steep regions and flat regions. Features where the angle between the normal and the XY plane normal greater than this value will be defined as the steep regions. For example, set *Steep Angle* as 60, then the features where the angle between the plane normal and the XY plane normal is 0-60 degrees will be the flat regions. Otherwise, they will be the steep regions.

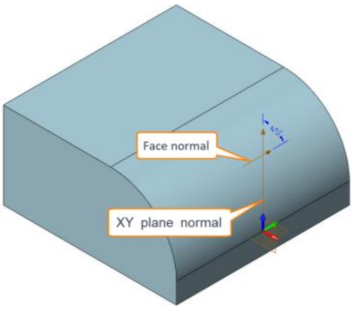


Figure 174 Steep Angle

Cut Regions: Specify regions to generate toolpaths, all regions, flat regions, or steep regions.

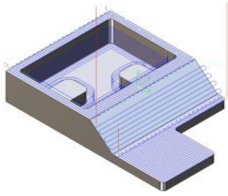
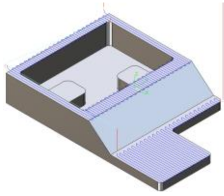
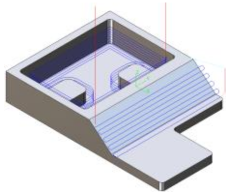
All regions	Flat regions	Steep regions
		

Figure 175 Cut regions

Cut Order: Specify cut order, flat first, or steep first.
Overlap: Specify the overlap size of toolpaths between flat area and steep area.

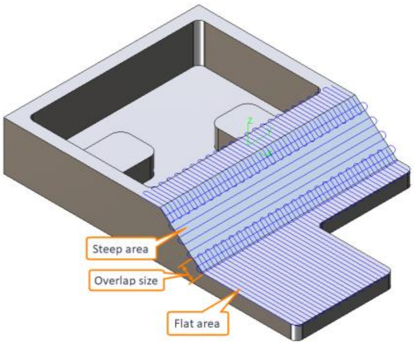


Figure 176 Overlap

Task: Create Angle Limiting toolpaths

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

STEP 02 Select ***Angle limiting*** operation.

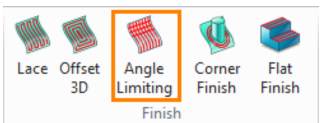


Figure 177 Angle Limiting operation

STEP 03 Specify features (part and profile 1) and tool(D8R4) in CAM manager.

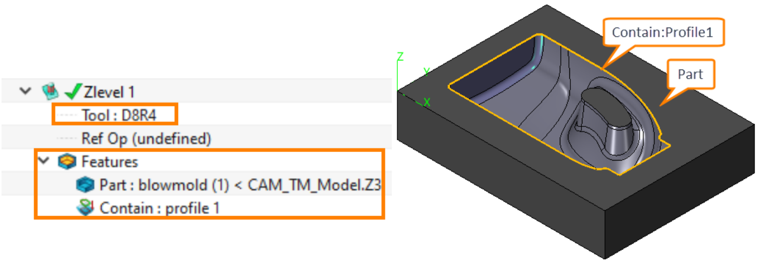


Figure 178 Specify features and tool

STEP 04 Specify toolpaths pattern for the flat regions and steep regions, then specify steep angle.

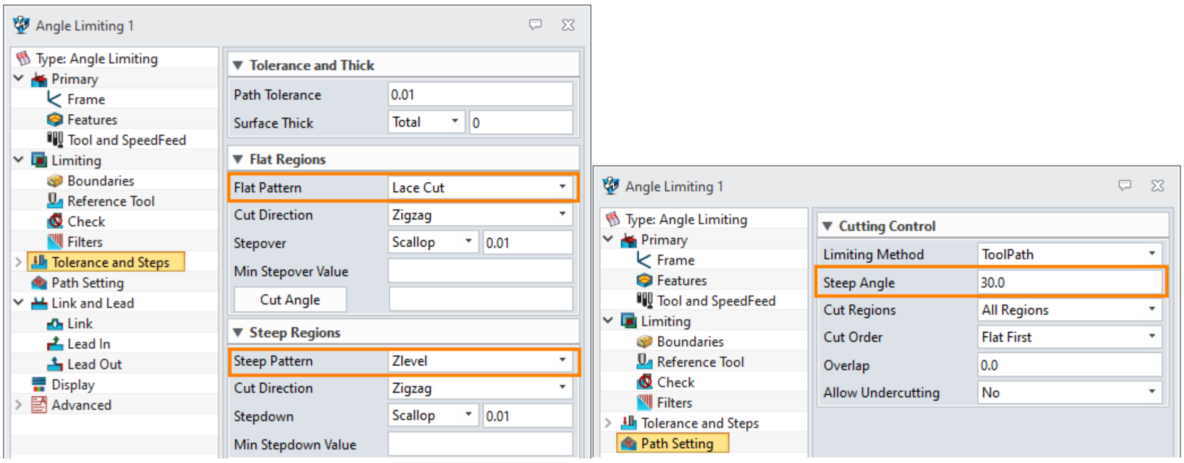


Figure 179 Specify toolpaths pattern and steep angle

STEP 05 Calculate the toolpaths.

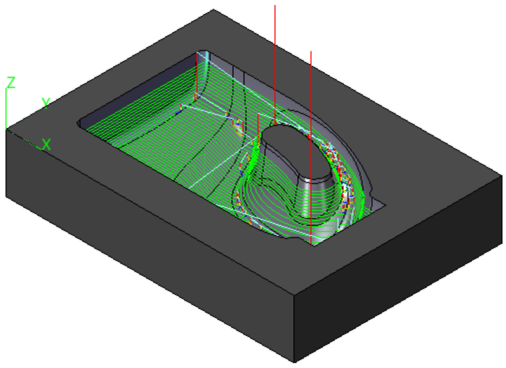


Figure 180 Toolpaths calculation

3.7 Flat Finish Operation

Flat finish operation is used to generate area clearance toolpaths (Lace or Offset 2D) on the planar surfaces of the manufacturing feature.

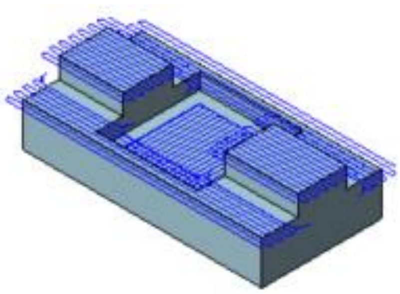


Figure 181 Flat Finish toolpaths pattern

In general, there are two requirements to create the toolpaths of Flat Finish operation.

- Specify tool

- Specify features (part or flat region feature)

Some key parameters of all operations are introduced here, including some common parameters and unique options.

- **Path Setting**

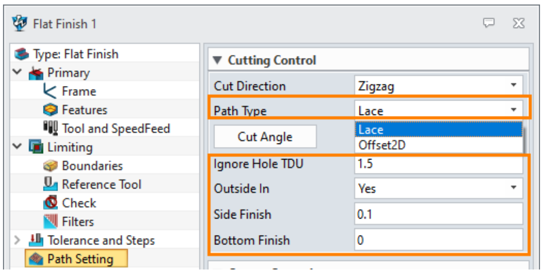


Figure 182 Flat finish operation—Path Setting

Path Type: Specify toolpaths pattern, ***Lace*** or ***Offset 2D***.

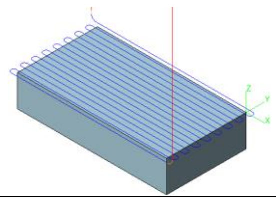
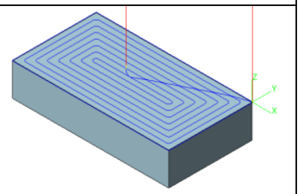
Path Type	Lace	Offset 2D
Toopaths		

Figure 183 Flat finish operation—Path type

Ignore Hole TDU: Set a threshold to define which holes to be ignored, if the ratio between hole diameter and tool diameter <the given *Ignore Hole TDU* value, ZW3D will ignore the hole and generate continuous toolpaths. Otherwise, the hole will be respected to generate toolpaths exclude the hole area for minimizing cutting movement.

Note: $Ratio = \frac{hole\ diameter}{tool\ diameter}$

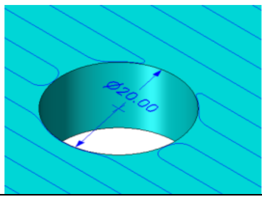
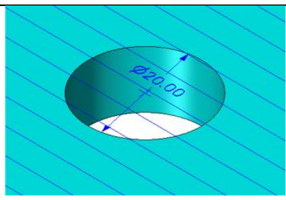
Ignore Hole TDU	Tool diameter=10mm Hole diameter=20mm Ignore Hole TDU=**1.8**	Tool diameter=10mm Hole diameter=20mm Ignore Hole TDU=**2.5**
Toopaths		

Figure 184 Ignore Hole TDU

Outside In: Specify whether the toolpaths are allowed to past the boundary.

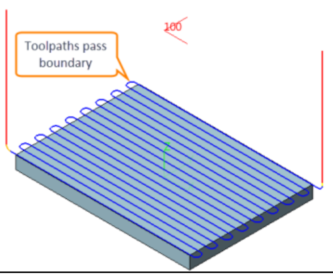
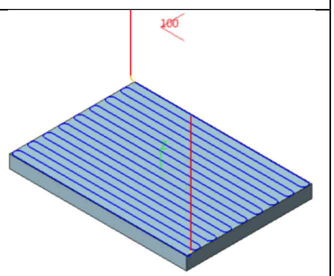
Outside In	**Yes**	**No**
Toopaths		

Figure 185 Outside in

Side Finish: Specify the distance between cleanup toolpaths and filling toolpaths.

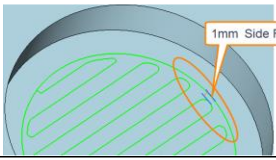
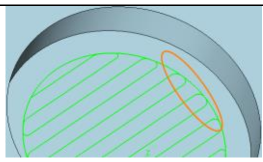
Side Finish	1	0
Toopaths		

Figure 186 Side Finish

Bottom Finish: Specify a bottom rest material, ZW3D will generate two layers of toolpaths, the first layer to get the specified rest material, then the second layer to finish the final clean up.

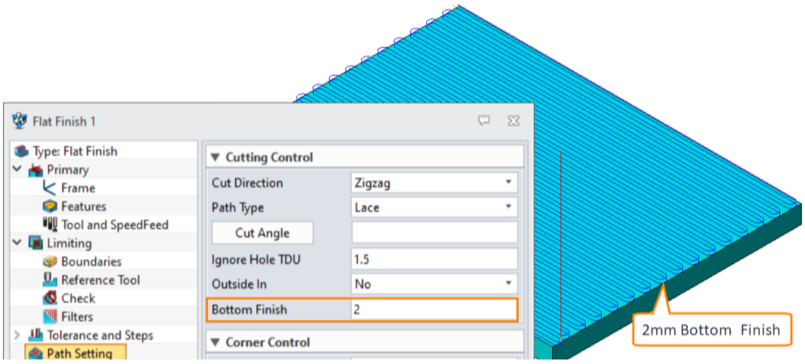


Figure 187 Bottom Finish

Task: Create Flat Finish toolpaths

STEP 01 Open **Flat Finishing.Z3** file and go into CAM space.

STEP 02 Select ***Flat Finish*** operation.

STEP 03 Specify features (part) and tool(D10R0) in CAM manager.

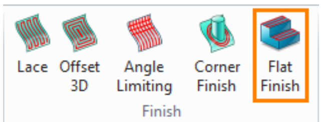


Figure 188 Flat Finish operation

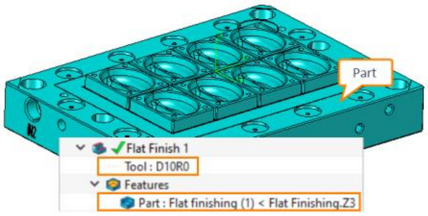


Figure 189 Specify features and tool

STEP 04 Specify toolpaths pattern and other parameters for flat finish operation.

STEP 05 Calculate the toolpaths.

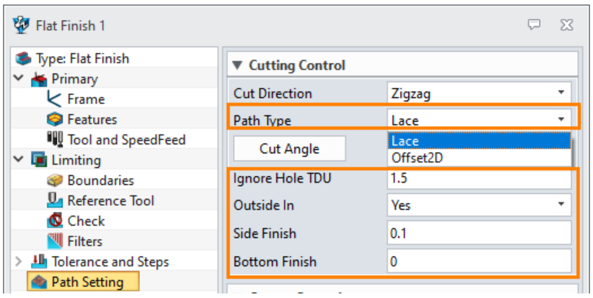


Figure 190 Specify toolpaths pattern

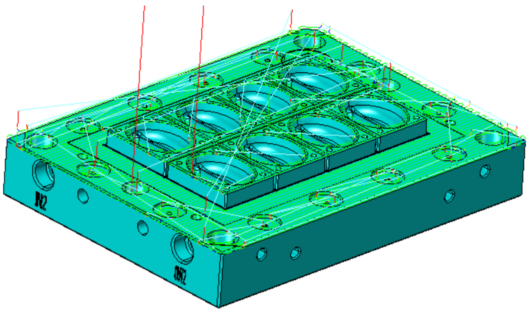


Figure 191 Toolpaths calculation

3.8 Corner Finish Operation

Corner finish operation is a corner cleanup strategy, it detects the rest material in corners left from the reference tool specified by the user, then generate multiple trace toolpaths to remove those rest materials.

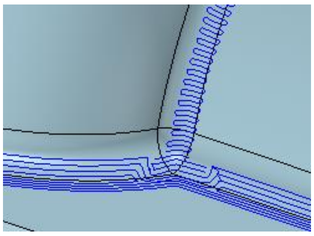


Figure 192 Corner Finish toolpaths pattern

In general, there are two requirements to create the toolpaths of Corner Finish operation.

- Specify tool
- Specify a reference tool
- Specify features (part or part with profile boundary)

Some key parameters of all operations are introduced here, including some common parameters and unique options.

- **Tolerance and Steps**

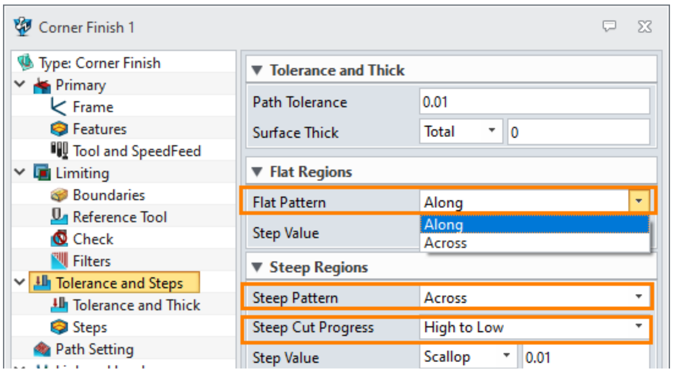


Figure 193 Flat Finish operation—Path Setting

Flat Pattern: Specify toolpaths pattern for the flat regions, along or across.
Steep Pattern: Specify toolpaths pattern for the steep regions, along or across.

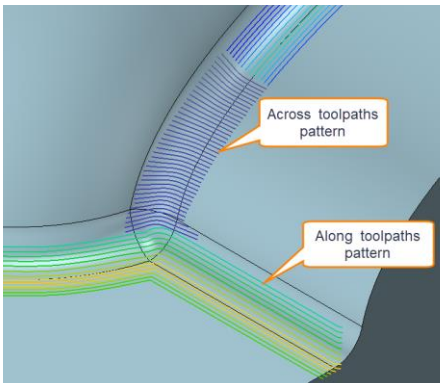


Figure 194 Toolpaths pattern—Along and Across

Steep Cut Progress: Specify the cutting order for the steep regions, high to low or low to high.

Steep Cut Progress	High to Low	Low to High

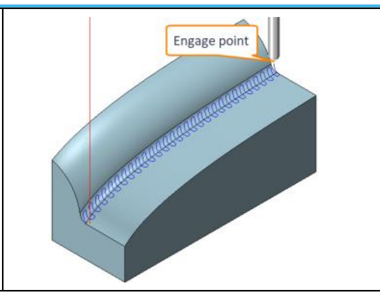
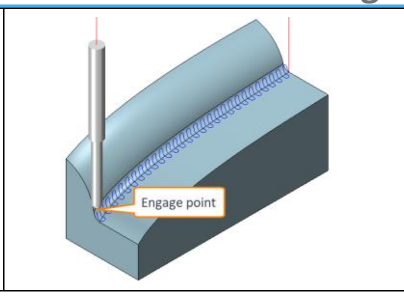
Toopaths		

Figure 195 Steep Cut Progress

Task: Create Corner Finish toolpaths

STEP 01 Open **Mouse_end.Z3** file and go into CAM space.

STEP 02 Select ***Corner Finish*** operation.

STEP 03 Specify features (part) and tool(D4R2) in CAM manager.

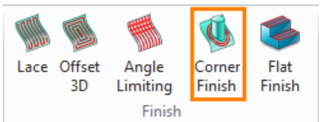


Figure 196 Corner Finish operation

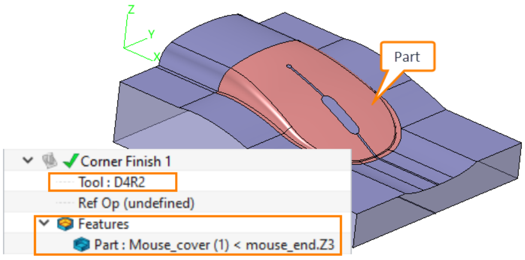


Figure 197 Specify features and tool

STEP 04 Specify toolpaths pattern for the flat and steep regions.

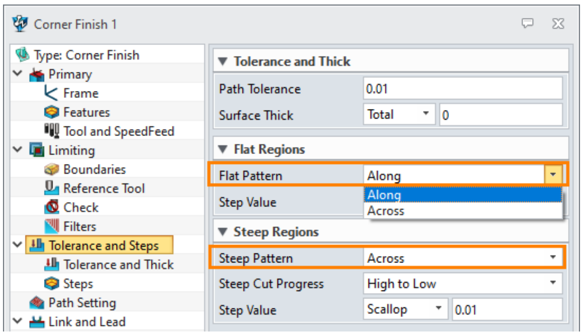


Figure 198 Specify toolpath pattern

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

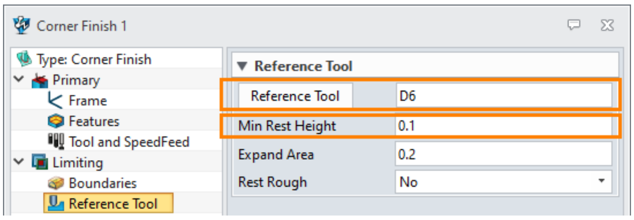


Figure 199 Specify reference tool and min rest height

STEP 06 Calculate the toolpaths.

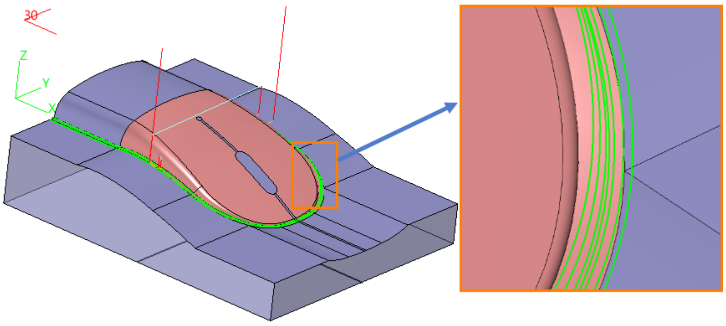


Figure 200 Toolpaths calculation

3.9 Pencil Operation

Pencil operation will detect all the corners when its radius equal to or less than the radius of the tool, and generate a single-trace toolpath to clean up the corners.

***Note:** users can specify the **Number of Cuts** value to generate multiple toolpaths.*

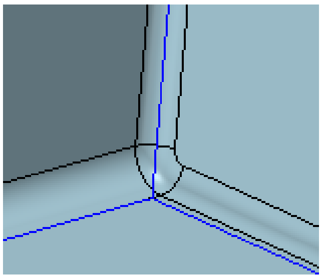


Figure 201 Pencil operation toolpaths pattern

In general, there are two requirements to create the toolpaths of Pencil operation.

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

Task: Create Pencil operation toolpaths

STEP 01 Open **Mouse_end.Z3** file and go into CAM space.

STEP 02 Select ***Pencil*** operation.

STEP 03 Specify features(part) and tool(D4R2) in CAM manager.

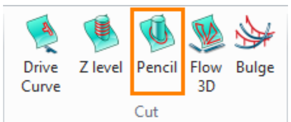


Figure 202 Pencil operation

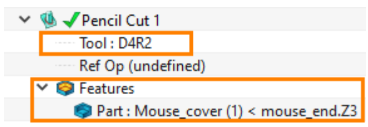


Figure 203 Specify features and tool

STEP 04 Specify related parameters, for example, ***Number of Cuts***.

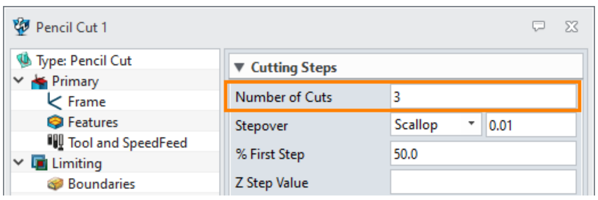


Figure 204 Specify Number of Cuts

STEP 05 Calculate the toolpaths.

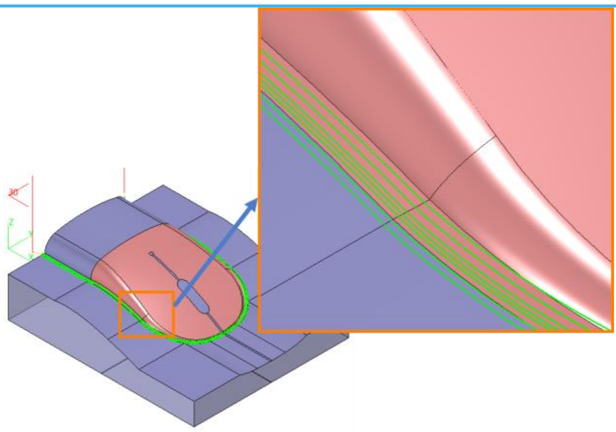


Figure 205 Toolpaths calculation

3.10 Bulge Operation

Bulge operation generates a network of bulges on the surface with two intersecting curves, one is defined as the drive curve (Directrix) and the other is defined as the generator curve (Generatrix). The Generatrix is used as a pattern generator which is guided by the Directrix.

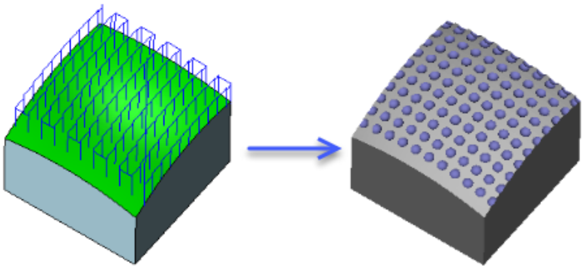


Figure 206 Bulge operation toolpaths pattern

Note:

1. *The first profile under the features page will be defined as Directrix, and the second one will be defined as Generatrix.*

2. The interaction point of Directrix & Generatrix will be defined as the origin to array the toolpaths.

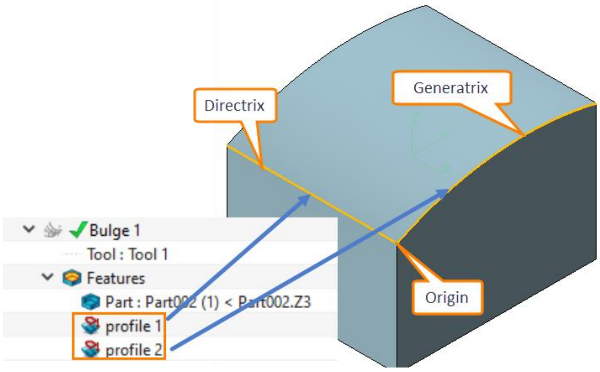


Figure 207 Bulge operation—Note

In general, there are three requirements to create the toolpaths of Bulge operation.

- Specify tool
- Specify features (part or part with profile boundary)
- Specify array guide curve (Directrix & Generatrix)

Some key parameters of all operations are introduced here, including some common parameters and unique options.

- **Tolerance and Steps**

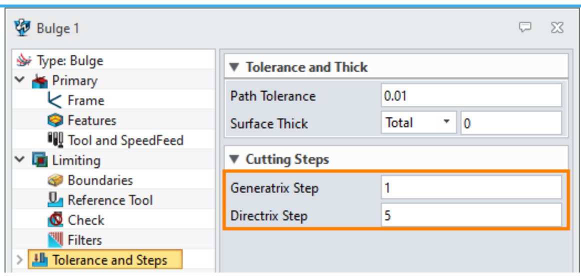


Figure 208 Bulge operation—Tolerance and Steps

Generatrix & Directrix Step: Specify the stepover value along Generatrix & Directrix direction.

➢ **Path setting**

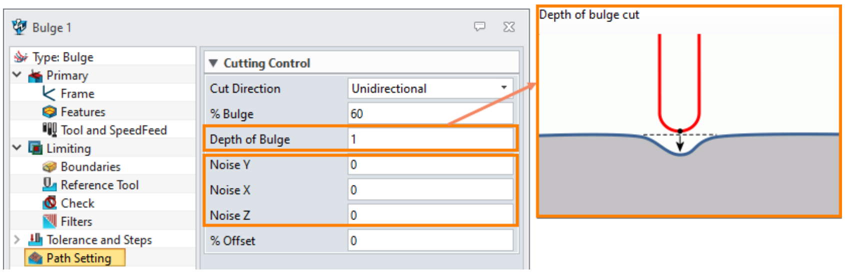


Figure 209 Bulge operation—Path setting

Depth of Bulge: Specify the depth of every bulge cut.

Noise Y, X, Z: Specify the noise value in X, Y, Z direction, and the toolpaths will offset the corresponding value randomly.

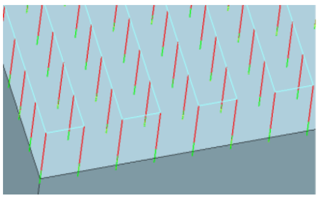
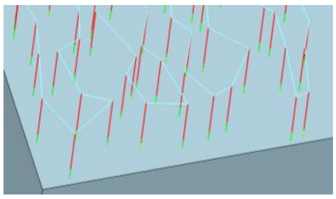
Noise Y, X, Z	Noise X=**0**, Noise Y=**0**	Noise X=**5**, Noise Y=**5**
Toopaths		

Figure 210 Bulge operation—Noise X, Y, Z

Task: Create Bulge operation toolpaths

STEP 01 Open **bulge.Z3** file and go into CAM space.

STEP 02 Select ***Bulge*** operation.

STEP 03 Specify features (part, Generatrix and Directrix) in CAM manager.

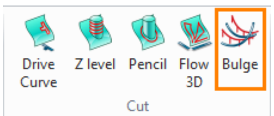


Figure 211 Bulge operation

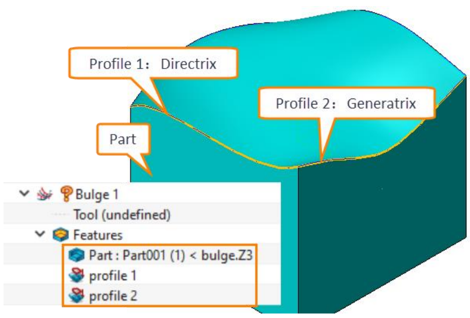


Figure 212 Specify features

STEP 04 Specify a proper tool in CAM manager.

STEP 05 Specify the related parameters, for example, ***Generatrix*** & ***Directrix Step, Depth of Bulge***.

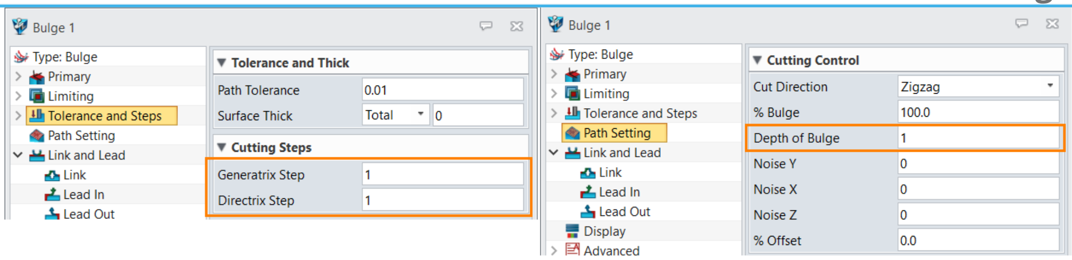


Figure 213 Specify related parameters

STEP 06 Calculate the toolpaths.

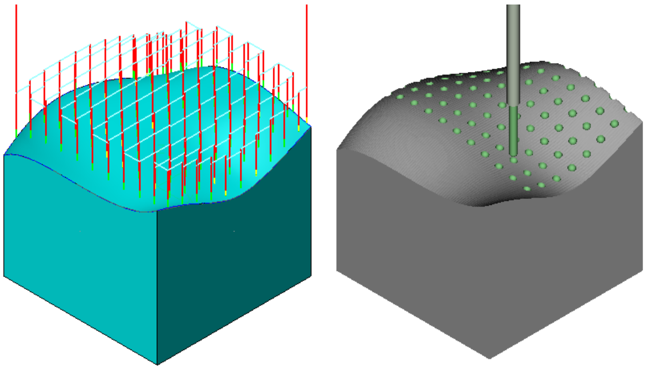


Figure 214 Toolpaths calculation

3.11 Engraving

This chapter will introduce how to generate engraving toolpaths. It contains Engrave 2D operation and Surface Engraving operation.

3.11.1 Engrave 2D Operation

In general, there are two requirements to create the toolpaths of Engrave 2D operation.

- Specify tool (Taper Ball End mill tool is recommended)
- Specify features (Part and text profile)

Task: Create Engrave 2D toolpaths

STEP 01 Open **Engraving 2D.Z3** file and go to CAM space.

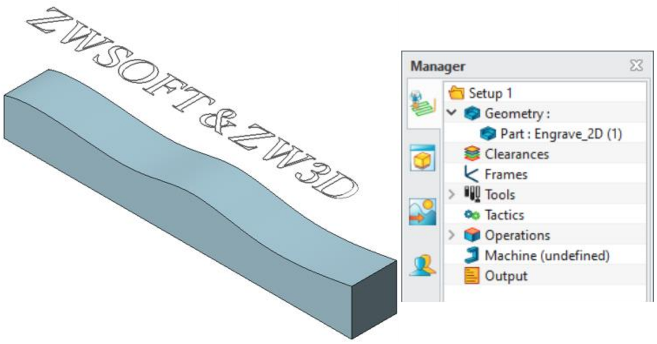


Figure 215 Engrave 2D operation—CAM space

STEP 02 Select ***Engrave 2D*** operation.

STEP 03 Specify features(part and profile) in CAM manager.

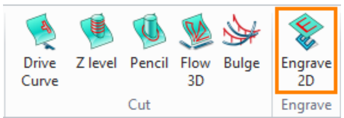


Figure 216 Engrave 2D operation

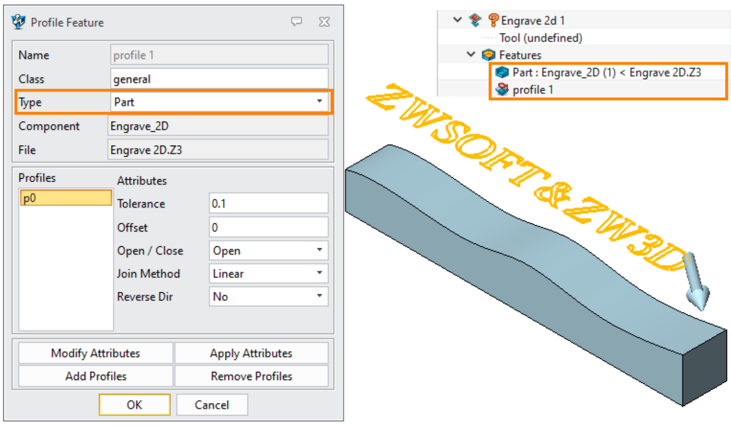


Figure 217 Specify features

STEP 04 Specify a Taper Ball End tool in CAM manager.

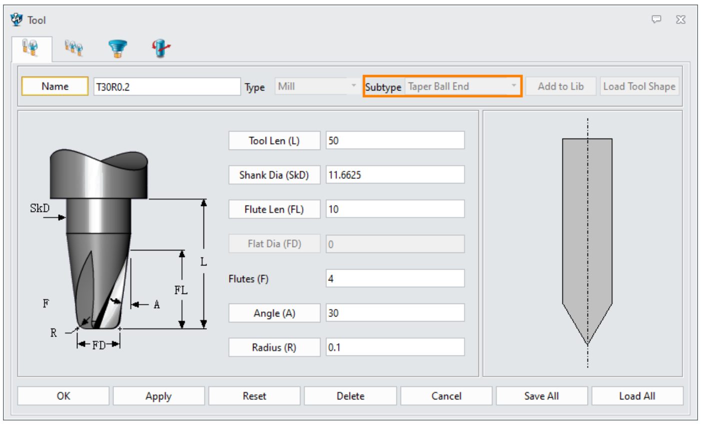


Figure 218 Specify tool

STEP 05 Calculate the toolpaths and solid verify.

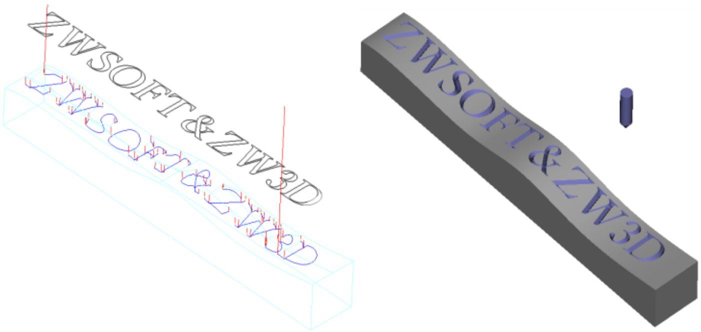


Figure 219 Toolpaths calculation and solid verify

3.11.2 Surface Engraving Operation

Surface Engraving operation is used to create a toolpath with the contact point defined by an embedded profile. This is similar to the drive curve operation.

In general, there are two requirements to create the toolpaths of Surface Engraving operation.

- Specify tool
- Specify features (Part and profile)

Task: Create Surface Engraving toolpaths

STEP 01 Open **Engraving.Z3** file and go to CAM space.

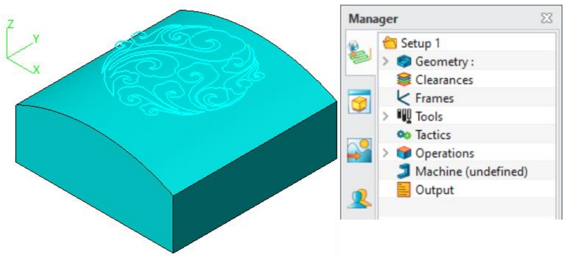


Figure 220 Surface Engraving operation—CAM space

STEP 02 Select ***Surface Engraving*** operation.

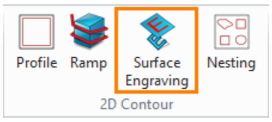


Figure 221 Surface Engraving operation

STEP 03 Specify features (part and profile) in CAM manager.

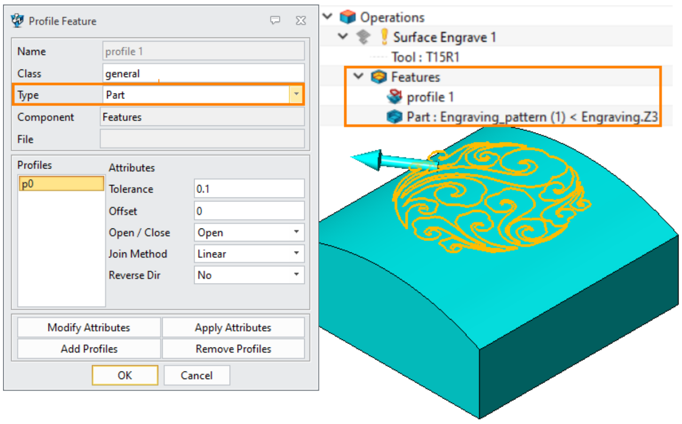


Figure 222 Specify features

STEP 04 Specify a Taper Ball end tool (T15R1) in CAM manager.

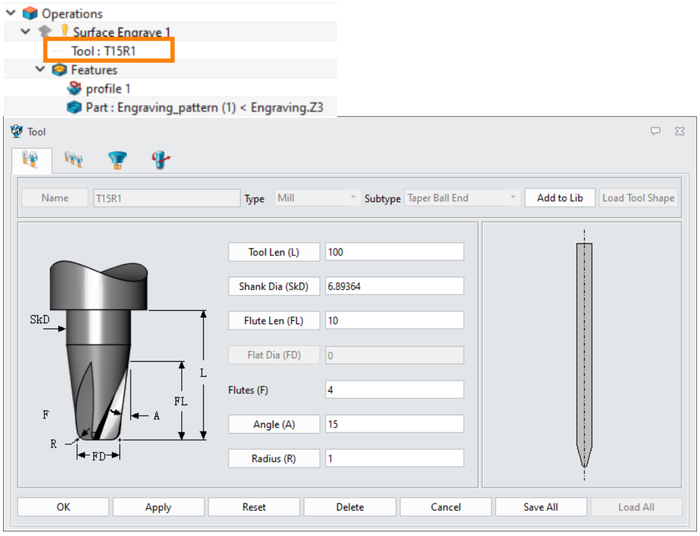


Figure 223 Specify tool

STEP 05 Calculate the toolpaths and solid verify.

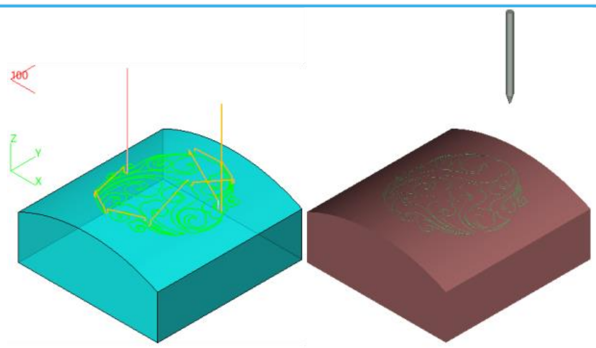


Figure 224 Toolpaths calculation and solid verify

4 Tool Path Editor

Tool path editor is an intuitive and simple tool, it can save a lot of CAM programming time. In ZW3D, tool path editor has modification functions and transform functions.

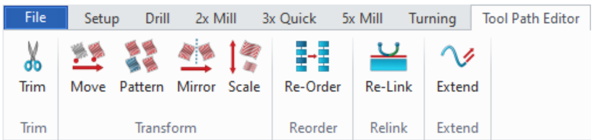


Figure 225 Tool Path Editor

Tips: *Users can also access* **Tool Path Editor** *via right-click on the operation in CAM manager.*

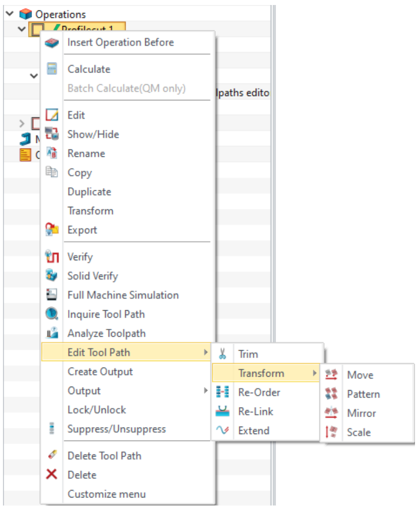


Figure 226 Access Tool Path Editor in CAM manager

4.1 Toolpaths Modification

Toolpaths modification can trim the unnecessary or improper toolpaths, relink the toolpaths, extend the toolpaths, and re-order the toolpaths.

Note:

1. The toolpaths modification will be overwritten if users recalculate the toolpaths.

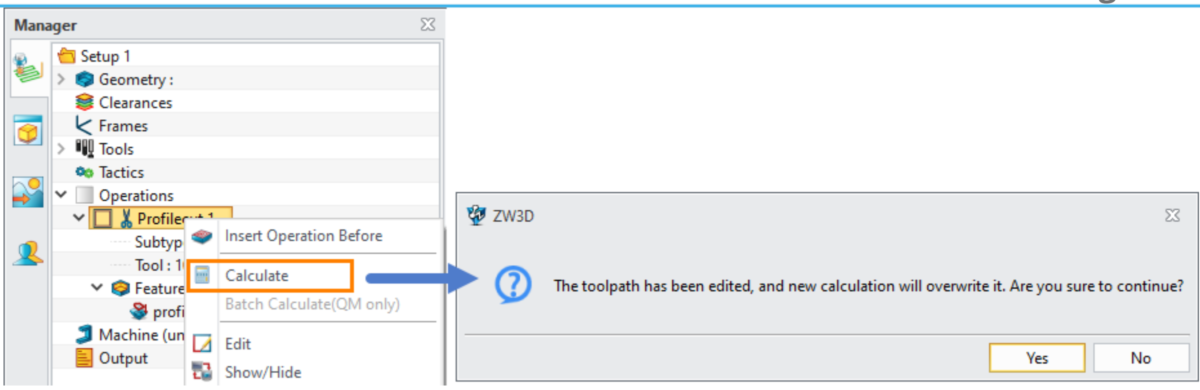


Figure 227 Overwrite toolpaths

2. All the operations (Trim, Relink, Extend and Reorder) should select the target operation first, then use the corresponding functions.
3. There will be a scissor icon in front of the operation after toolpaths modification.

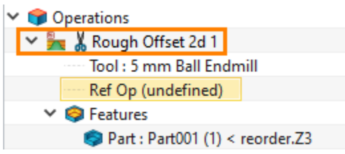


Figure 228 Scissor icon

4.1.1 Trim

Trim function is used to trim the toolpaths of the selected operation. There are 3 trim types in ZW3D.

Tips: *To use the trim function, users need to select the operation first.*

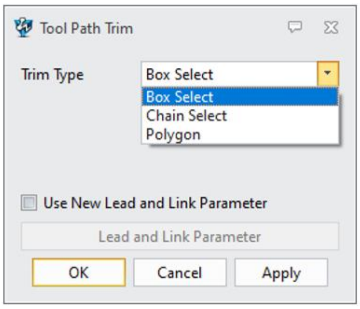


Figure 229 Trim Type

Box Select: Users can use box select then the selected toolpaths will be highlighted. Click ***Apply*** to trim the selected toolpaths.

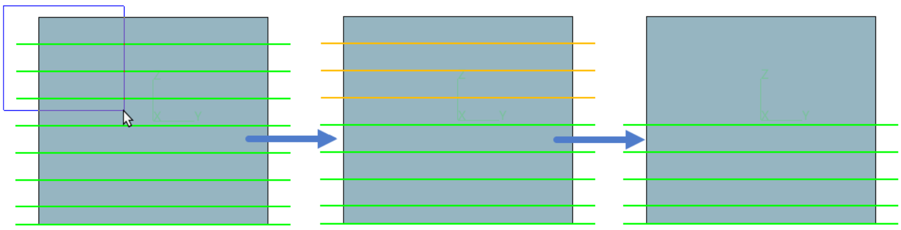


Figure 230 Trim—Box Select

Chain Select: Users can hold the ***Shift*** key and click any segment on the toolpaths to chain select the toolpath. Then click ***Apply*** to trim the selected toolpath.

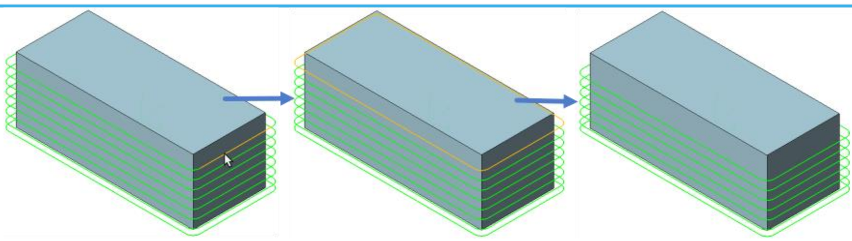


Figure 231 Trim—Chain Select

Polygon: Users can create a boundary with polygon, only toolpaths inside the polygon will be trimmed. After creating the polygon, users need to click the middle button of the mouse to trim the toolpaths.

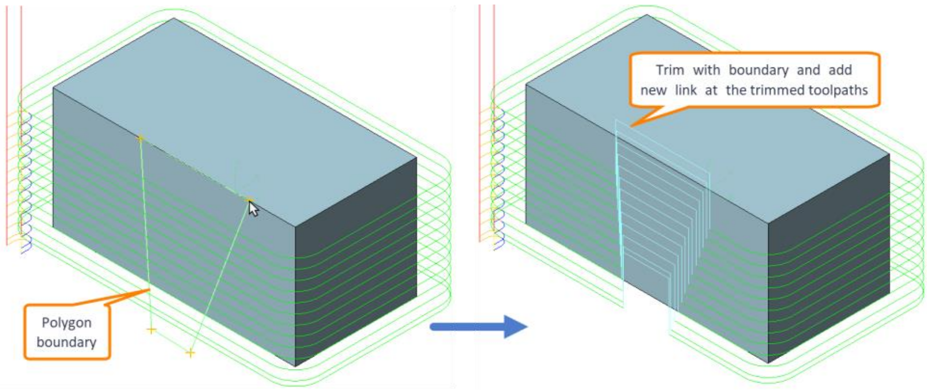


Figure 232 Trim—Polygon

Use New Lead and Link Parameter: Check this option to set a local lead and link parameters for the trimmed toolpaths. If this option is off, ZW3D will use the same lead and link parameters of the selected operation to calculate the trimmed toolpaths.

Note: *For the detailed lead and link parameters setting, please refer to the corresponding instruction in the roughing operation chapter.*

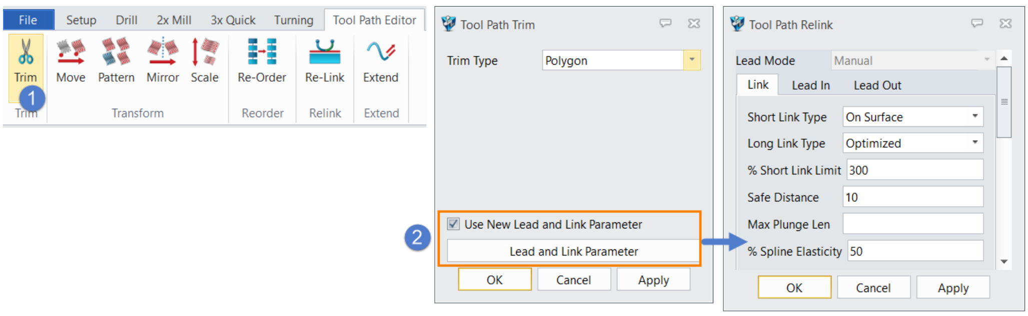


Figure 233 Use New Lead and Link Parameter

4.1.2 Re-link

Re-link function will change all the lead and link parameters of the selected pattern by setting new parameters.

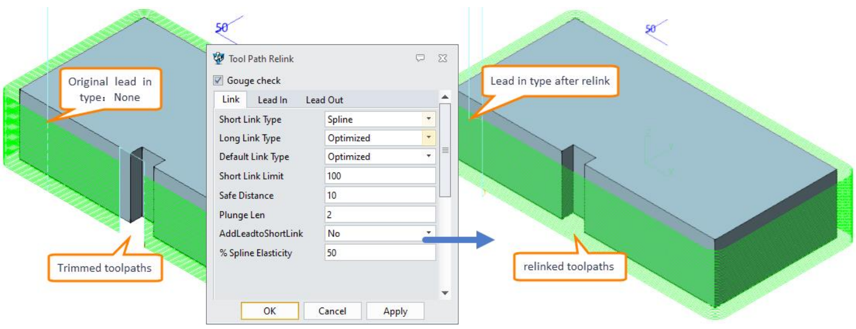


Figure 234 relink

4.1.3 Extend

Extend the selected patterns by window selection with a specified length. For example, it can be used to ensure complete machining if the stock is bigger than the workpiece.

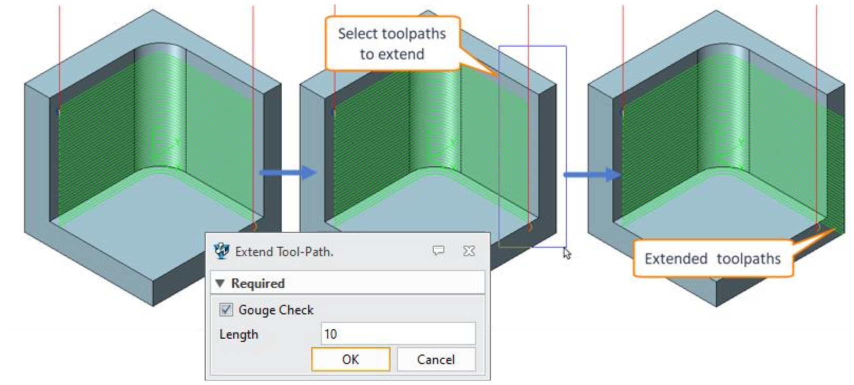


Figure 235 Extend

4.1.4 Re-order

Re-order enables to change the order and direction of toolpath segments. The dialog below displays all the toolpaths segments and enables you to reorder them, altering the machining order.

Note: *Selecting segments within the list also displays the corresponding patterns in yellow in the graphic area.*

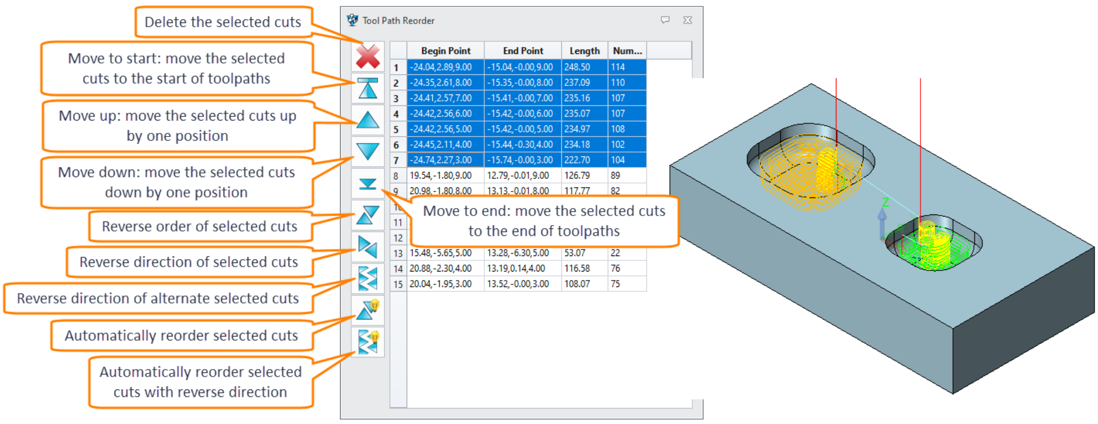


Figure 236 Reorder

4.2 Toolpaths Transform

Toolpaths transform provides a method to quickly create a serial of similar toolpaths by move, pattern, and mirror, which can significantly improve work efficiency.

4.2.1 Move

Move the selected toolpaths from point to point, along a direction, by aligning frames, or rotate the toolpaths around a direction. Take move toolpaths along a direction for example.

CASE: Move toolpaths along a direction

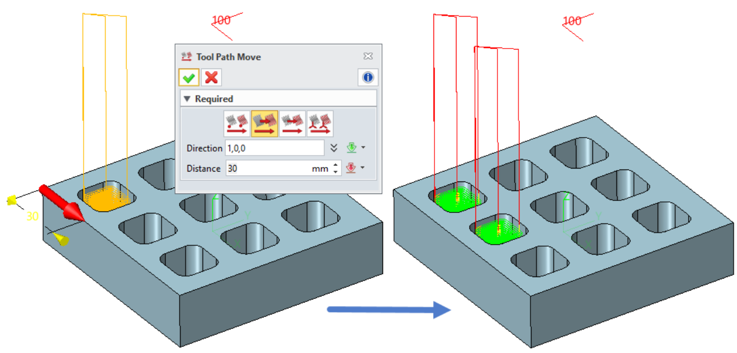


Figure 237 Case: move toolpaths along a direction

4.2.2 Pattern

Array the selected toolpaths by linear or circular pattern.

Linear: users can array the toolpaths along the specified direction and specify the machine order for the arrayed toolpaths.

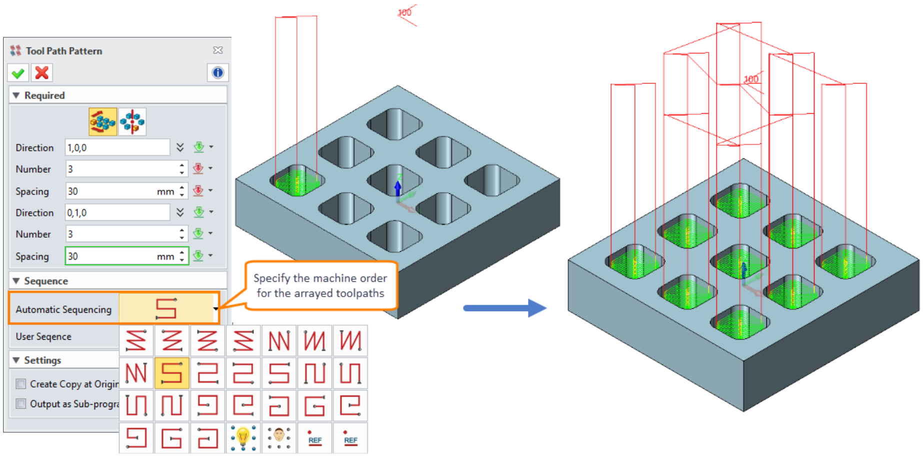


Figure 238 Linear pattern

Circular: users can array the toolpaths around a specified origin and direction.

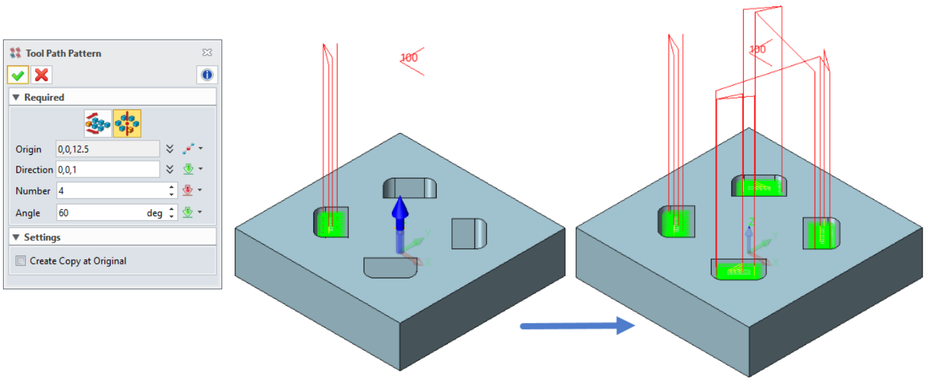


Figure 239 Circular pattern

4.2.3 Mirror

Mirror the selected toolpaths by specifying a plane or datum defined by a point with a plane normal.

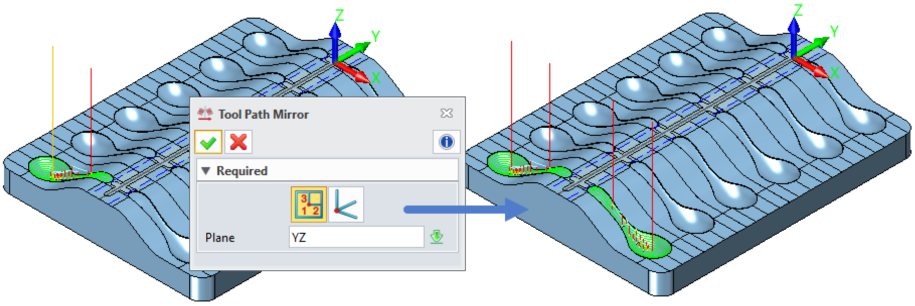


Figure 240 Mirror

5 Others

This chapter will introduce some tools or methods to speed up work efficiencies, such as the QuickMill batch calculation, CAM template, and operation library.

5.1 Quick Mill Tools

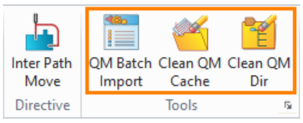


Figure 241 Quick Mill Tools

5.1.1 Quick Mill Batch Calculate

For some complicated cases, toolpaths calculation will be a time-consuming process and users need to wait until the calculation is completed. But with batch calculation method, users can continue to work with ZW3D during the toolpaths calculation and just need to import to the corresponding operation when it is done, which will significantly improve the work efficiency.

Tips: *The batch calculation is only valid in QuickMill operations.*

Task: Calculate toolpaths with Quick Mill Batch calculation

STEP 01 Open **Batch calculation.Z3** file and go to CAM space.

STEP 02 Create an ***Offset 2D*** roughing operation, then specify the tool and features.

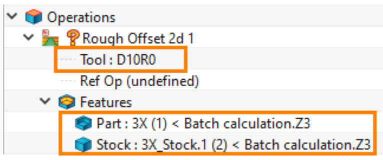


Figure 242 Create offset 2D operation

STEP 03 Right-click the operation then click ***Bath Calculate*** in the context menu, it will automatically pop up the Qm Tracker dialog box to show the calculation progress.

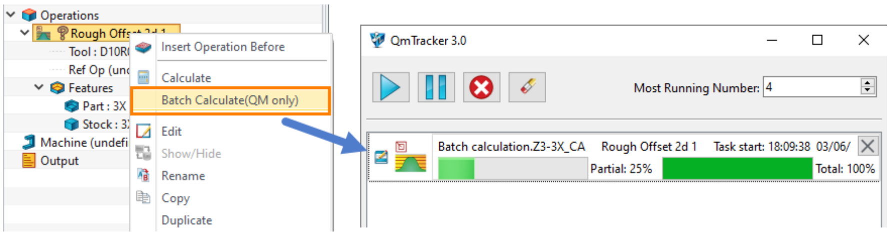


Figure 243 Batch Calculate

STEP 04 When the calculation is completed, import the toolpaths into operation.

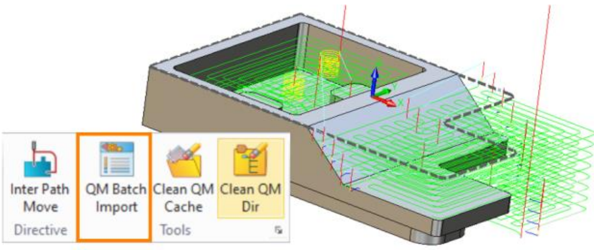


Figure 244 Import toolpaths

5.1.2 Clean QM Cache & Clean QM Dir

To ensure the calculation efficiency, ZW3D Quick Mill uses advanced cache control, which includes cache files and STL data. Cache files are used in the incremental calculation and STL data is the basis of Quick Mill calculation. But a long period of use, ZW3D will generate a lot of cache files. Users can use ***Clean QM Cache***

& Clean QM Dir to clean these 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 cache folder which is under the user folder. Thus all the cache files and STL data will be deleted.

5.2 CAM Template

When dealing with some similar workpieces, the programming process, parameters, and tools might be the same or very close. Users can use CAM Template to avoid repetitive work to slash a lot workload.

Note: *In the following introduction, we will use the same model to demonstrate how to create and reuse the CAM template.*

Task 1: Create a CAM template

STEP 01 Create a whole CAM plan with all necessary operations and parameters, then set the machine.

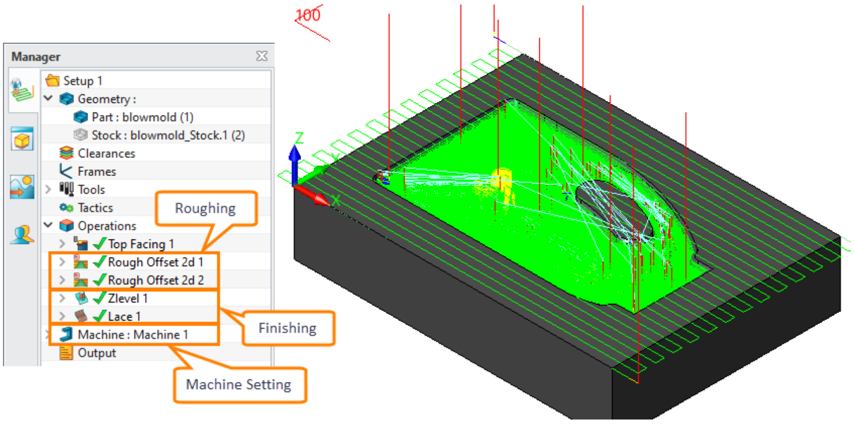


Figure 245 Create CAM Template-Step 1

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

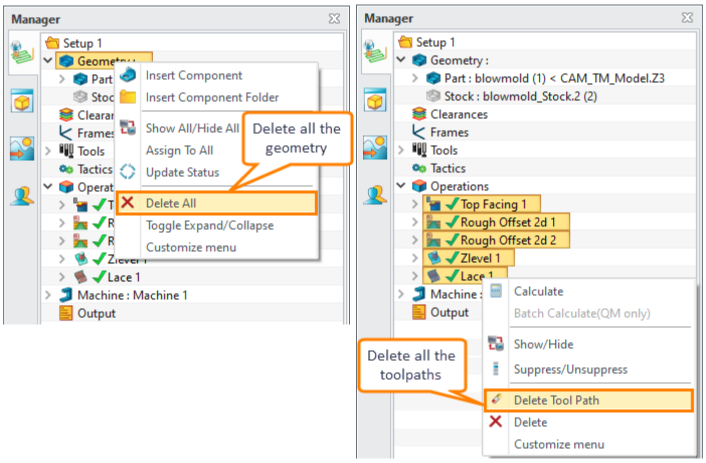


Figure 246 Create CAM Template-Step 2

STEP 03 Open the Template file.

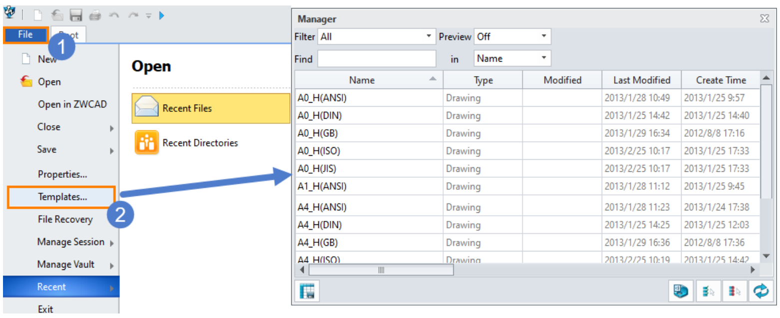


Figure 247 Create CAM Template-Step 3

STEP 04 Copy the CAM plan then paste to the template file.

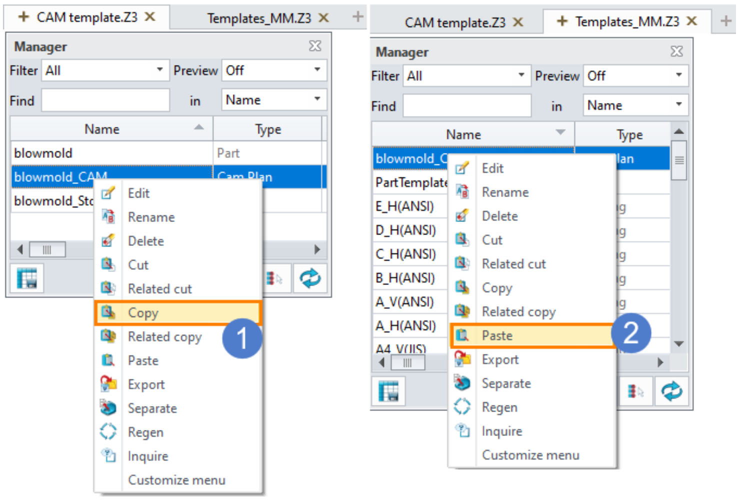


Figure 248 Create CAM Template-Step 4

Task 2: Reuse the CAM template

STEP 01 Open the model which needs to be machined (For illustration, we will use the same model).

STEP 02 Go to CAM space and select the corresponding template.

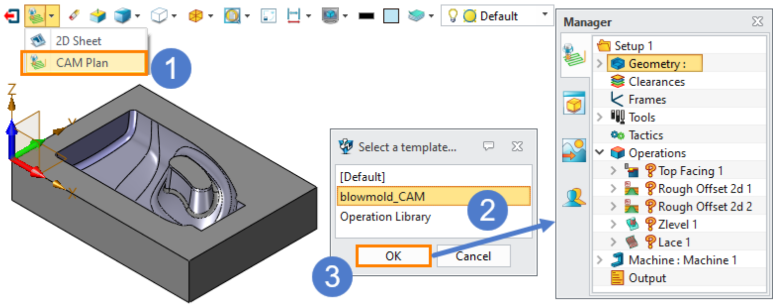


Figure 249 Reuse CAM Template-Step 2

STEP 03 Add a stock.

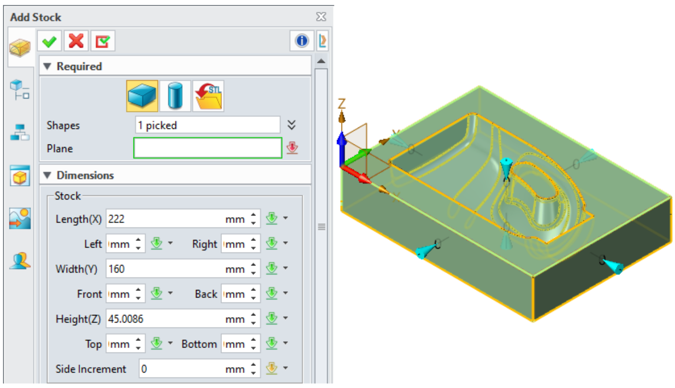


Figure 250 Reuse CAM Template-Step 3

STEP 04 Specify features for all operations and calculate all.

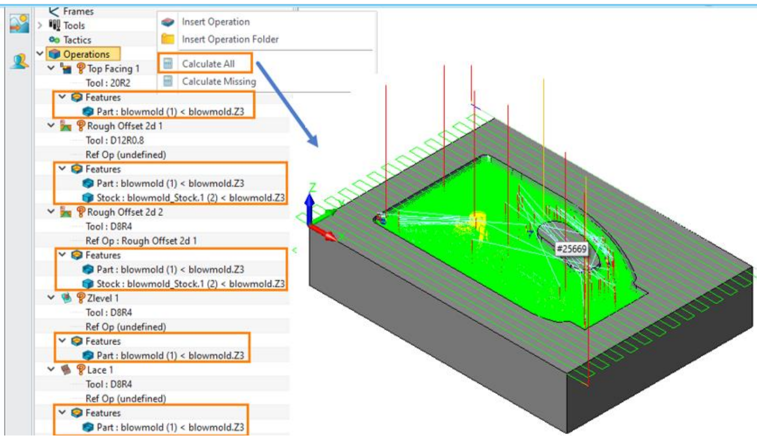


Figure 251 Reuse CAM Template-Step 4

5.3 Operation Library

In ZW3D, users can create their own operation library to improve work efficiency.

Task 1: Create an operation library

STEP 01 Create a new CAM Plan in the template file, after creation (click "ok" button), it will automatically open the file.

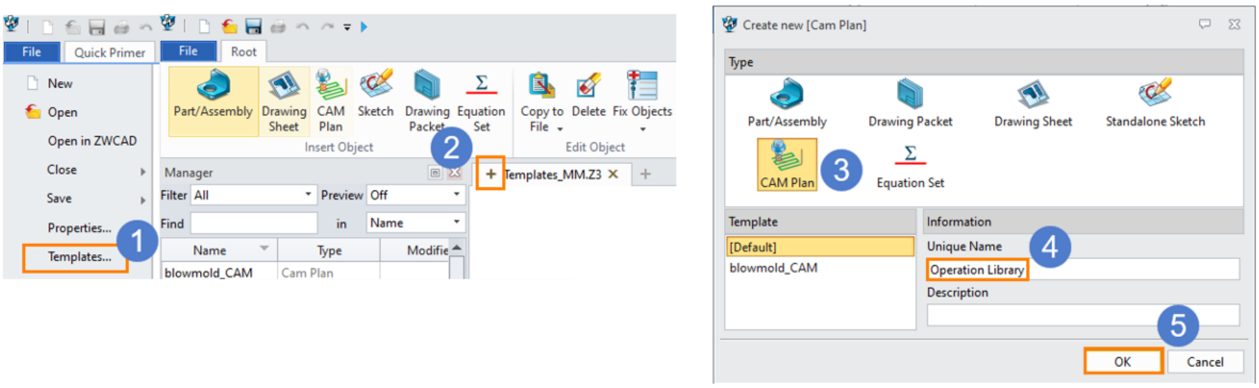


Figure 252 Create a new CAM Plan

STEP 02 Copy all the operations that you want from other CAM plan and paste to s this CAM plan, all the parameters like tool, speed & feed rate, stepover, stepdown can be saved.

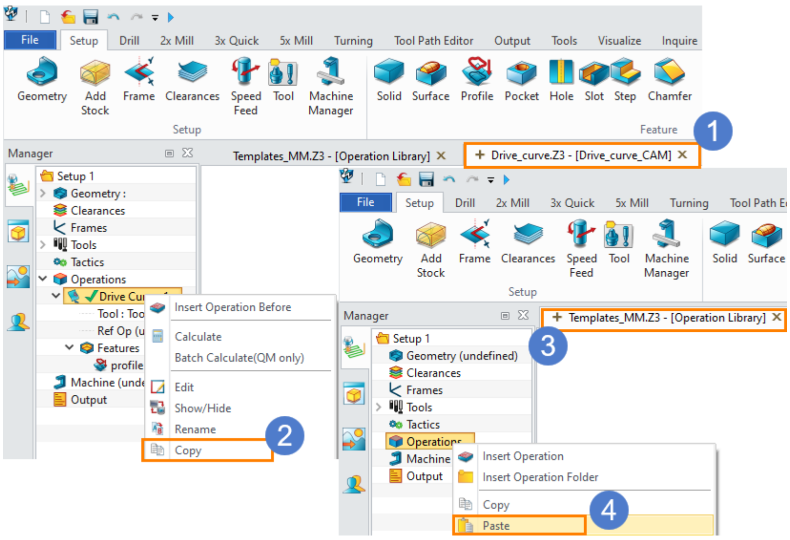


Figure 253 Copy and paste all the operation to CAM Plan

STEP 03 Save the template file.

Task 2: Reuse the operation library

STEP 01 Go to configuration and specify the saved operation library. It should be under the installation path as shown below (...\ZW3D 2021\languages\en_US\resource).

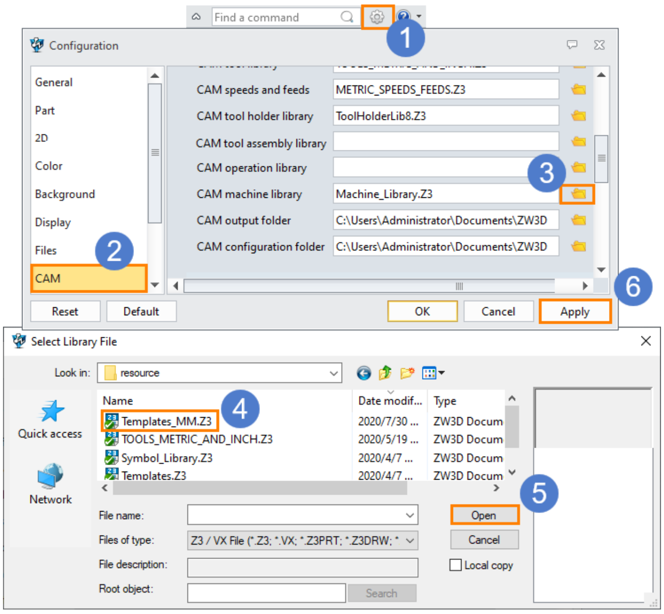


Figure 254 Configuration setting

STEP 02 Create a new CAM Plan, right-click on the operation to import the operations from operation library.

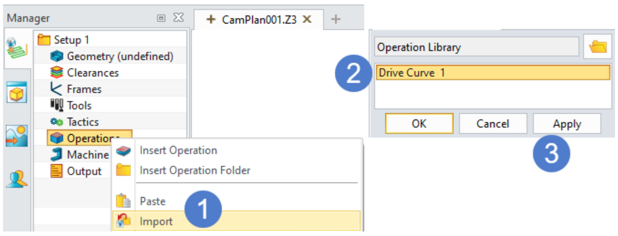


Figure 255 Import the operations

STEP 03 Check the imported operation, all the parameters are the same as the original ones.

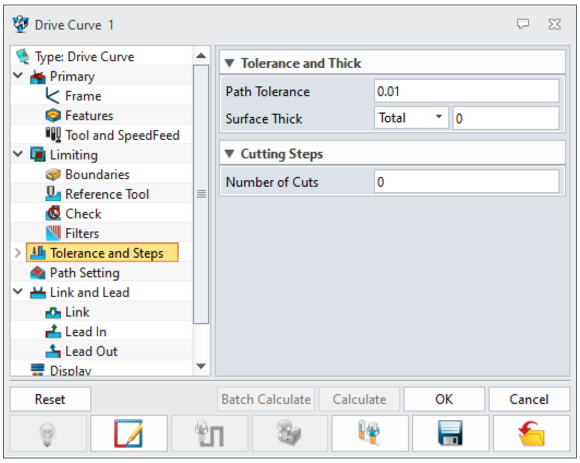


Figure 256 Check the parameters

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 2021, 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 Support Team: sales@zwsoft.com