ZW3D CAM

4-5X Machining

Contents

4-5X Machining

Key Points:

- ✧ 4X index milling
- ✧ 5X index milling
- ✧ 5X simultaneous movement operations

ZW3D CAM 5X module provides user 4X&5X indexing milling function and simultaneous movement function. With the indexing milling function user can easily mill multiple sides of a complicated part by one-time clamping. With the 4X/5X simultaneous movement operations user can deal with such tough jobs as the titled surface, undercut surface etc.

Notice:

1) This tutorial is based on ZW3D 2019 version, some functions or icons may not match the current version.
2) All the tutorial models can be found in installation folder: …training\5X machining model

1.1 4X Indexing Milling

Indexing milling function can position the tool axis in any direction by machine. Normally it includes 2 modes: one is for 4X indexing milling called as "3+1" mode. This means 3 linear axis plus 1 rotary axis, while the rotary axis is for positioning ; the other is for 5X indexing milling called "3+2" mode which means 3 linear axis plus 2 rotary axis, also the 2 rotary axis is for posisitioning. Open the practice file "**4X_Indexing.Z3**" as follows:

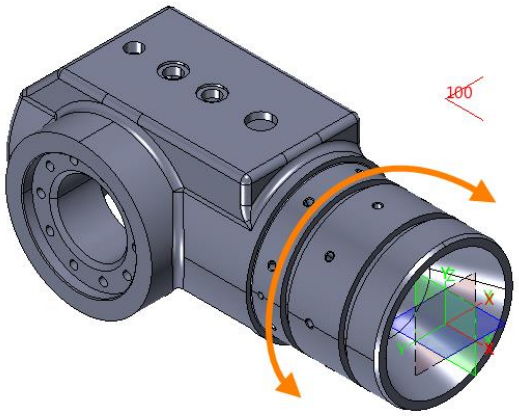


Figure1 4X_Indexing milling part

Analysis: It is apparent that if we use the 3X milling tool, we are not able to finish all sides by one-time clamping. From the structure of the part it is easy to see that rotating the part around the cylinderical center line(as indicated in Figure1) it is able to mill all sides by one-time clamping. So one rotary positioning axis is enough for this case, here let's suppose that the 4X machine 's rotating axis is A whichis rotating around X axis and the part is a casting part. Then let's see how to achieve the 4X indexing milling in ZW3D CAM.

1.1.1 Align The Part

In order to make use of the A axis here we will align the part to make the default X axis colinear with the cylindrical center line as follows:

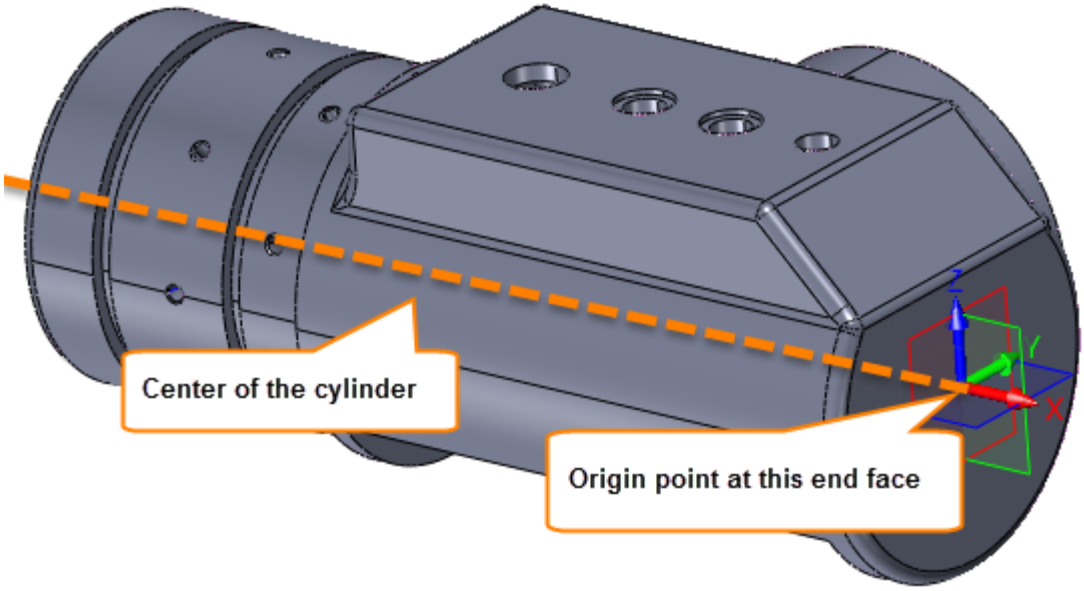


Figure2 Align the part

1.1.2 Create Sub Frame

After enter into CAM let'screate the sub frame as follows:

STEP 01 Create Sub frame for right side

I. Right click on Frames tab then we can get the Insert Frame command as follows:

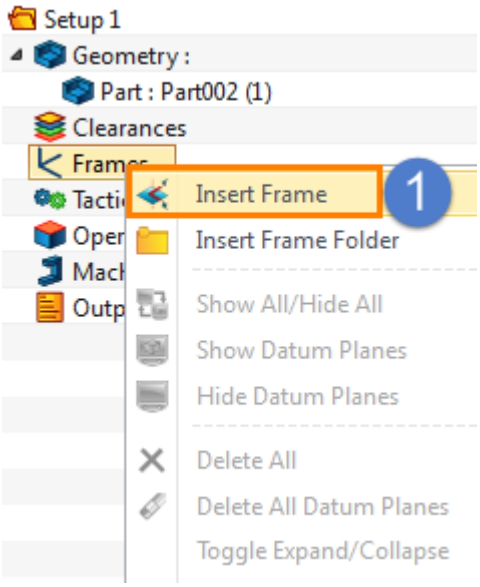


Figure3 Insert frame command

II. Click on Insert Frame command then the definition form for Frame will pop up as follows

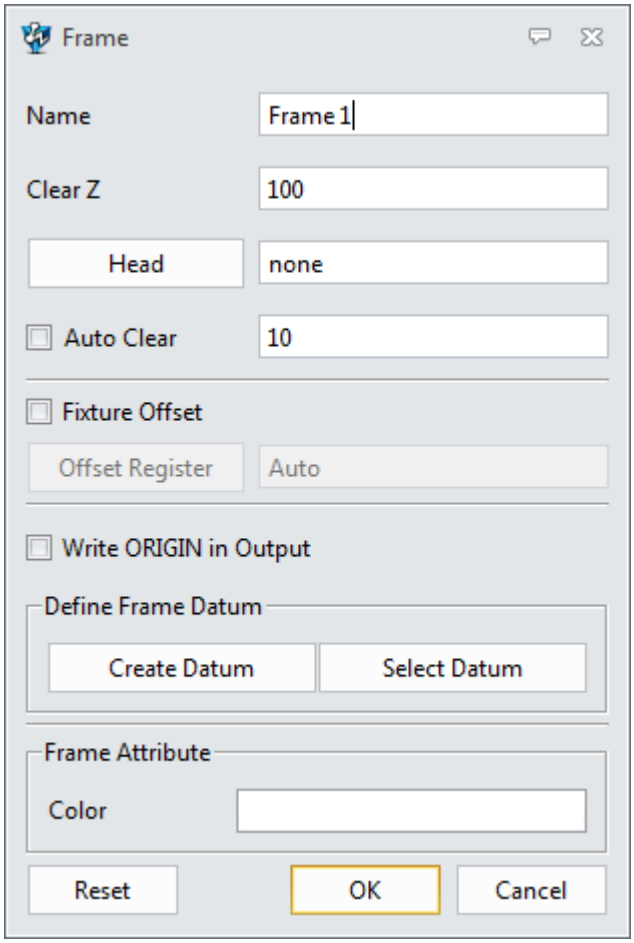


Figure4 Frame dinition form

III. Then we can click “create datum” to create new frame as follows

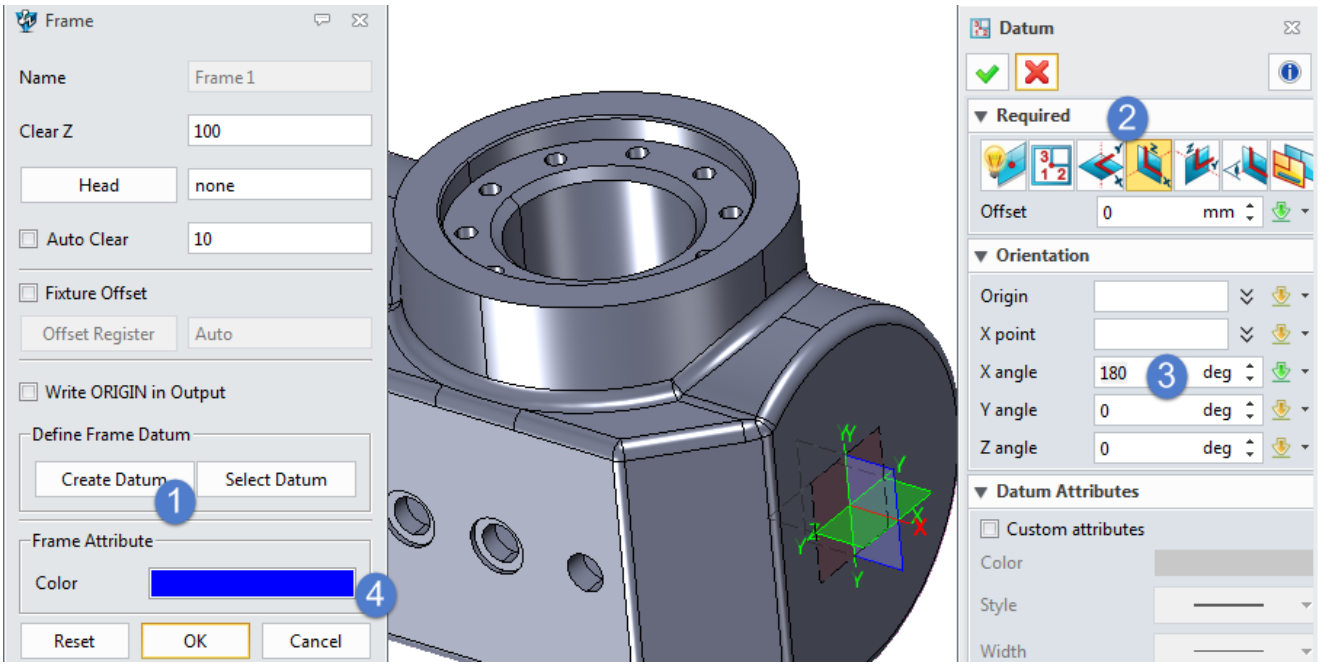


Figure5 Create frame based on XZ plane

Note: This process is the same as the process of creating datum in CAD level.

IV. Newly created frame will be as follows:

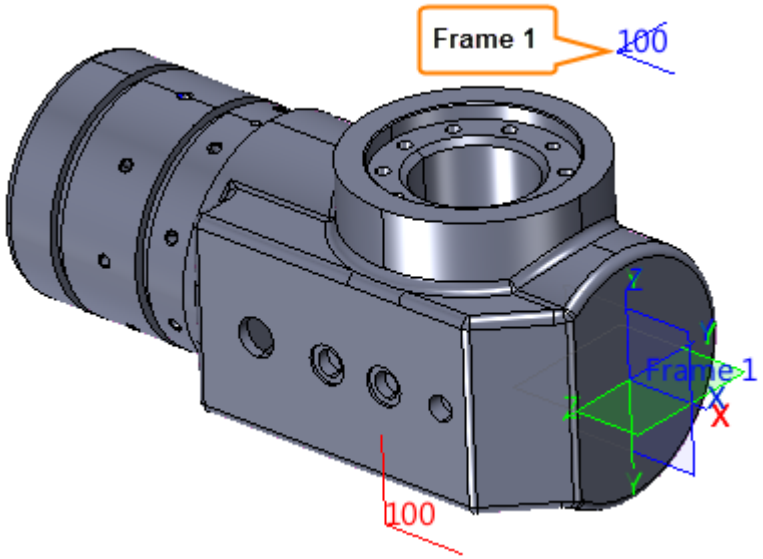


Figure6 Finished Frame 1 for right side

STEP 02 Then create another frame for the bottom side as follow:

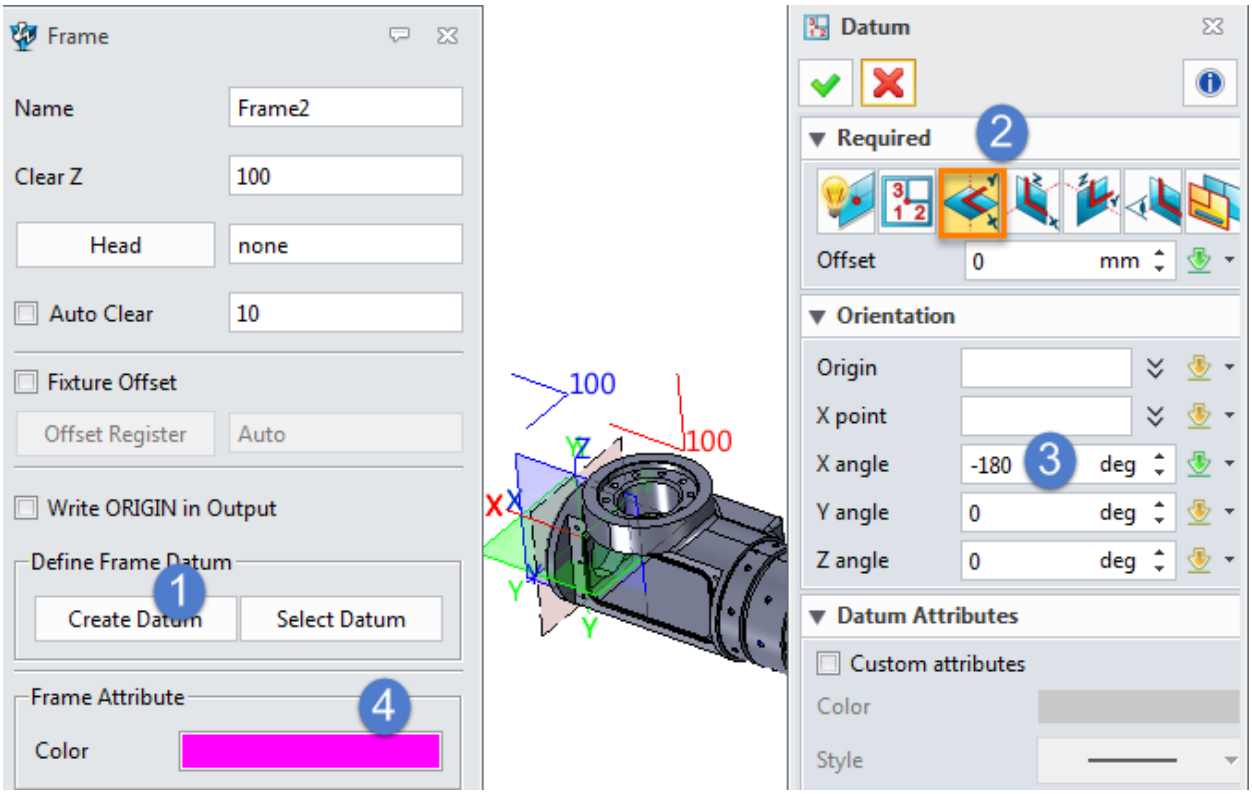


Figure7 Create frame on Bottom side

Tips: In order to distinguish different frame especially in the same position you can choose different color to tell the differences.

Then we can get the following result with 2 newly created frames:

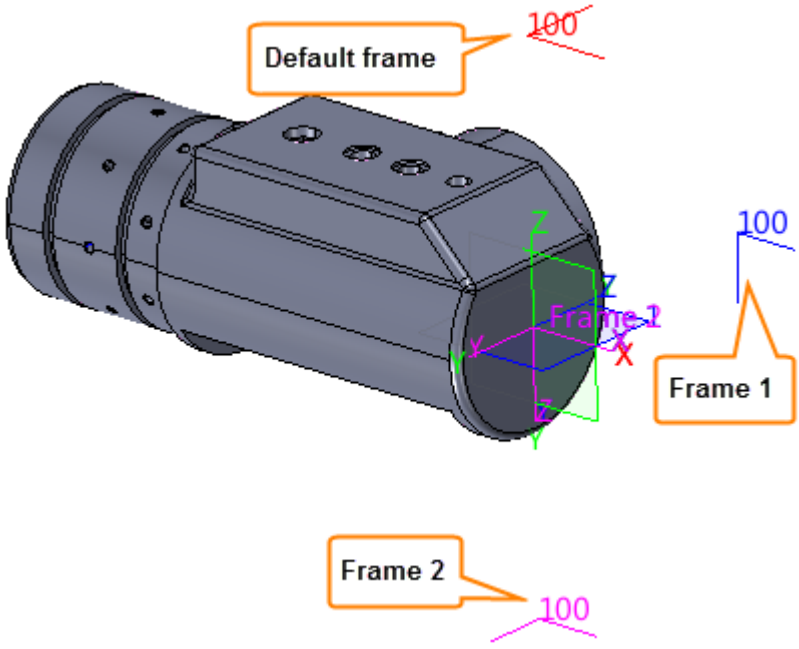


Figure8 Finished local frames and default frame

Next we can create toolpath based on these frames.

1.1.3 Create Toolpath Based On Each Frame

STEP 01 Create toolpath on top side: finish the top face based on default frame.

I. Let's choose 3X Flat Finish operation and choose the whole part as feature as follows.Then choose D10 flat end tool.

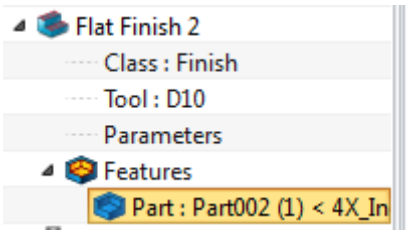


Figure9 Choose part for flat finish

II. Set up frame for it : open the parameter form to set up which frame to be based on as follows:

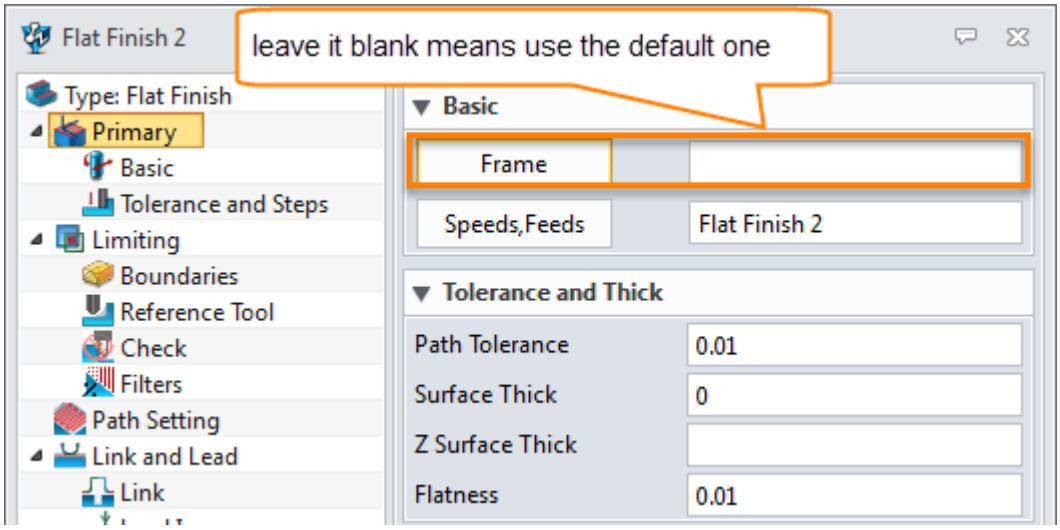


Figure10 Choose default frame

III. Set up the rest necessary parameters and then calculate the toolpath as follows:

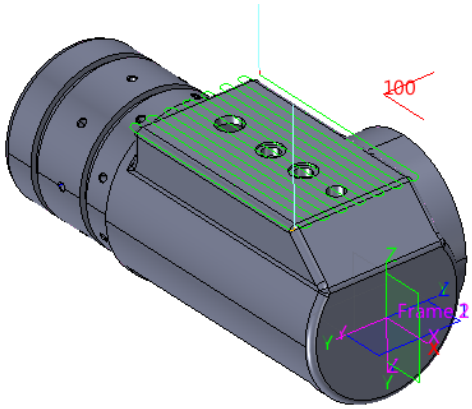


Figure11 Toolpath for top side

Note: since this process is the same as using 3X milling operation so here we will skip the details for creating the toolpath. And here we just use one opeartion to show indexing milling, for the rest necessary operations please finish by yourself.

Tips: In order to better manage the operations on each side , it is helpful to create operation folder and name it clearly. For example here we can name the folder for the operations in topside as "topside" and put all of the operations into this folder as follows:

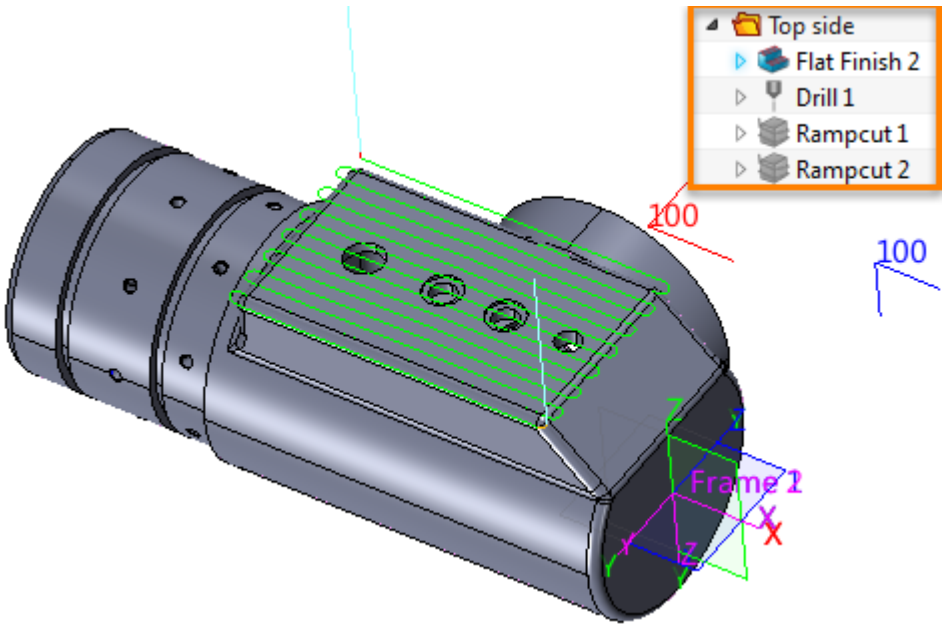


Figure12 Operation folder for top side

STEP 02 Create toolpath based on Frame 1 on right side as follows:

I. In order to get a clear idea about the toolpath on the left side we can firstly rotate the part as follows:

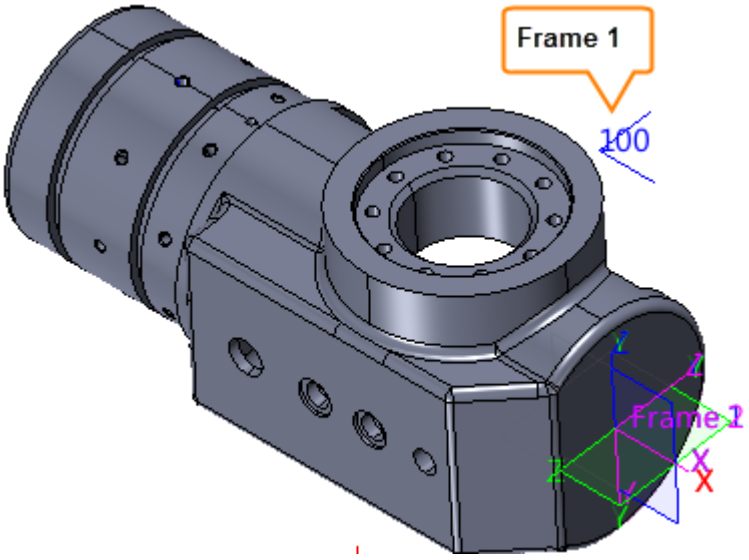


Figure13 Rotated part

II. Create a folder and name it as “Right side” and then go to add operations.

III. Pick 3X “Flat Finish” operation. Then Open the parameter form to set up the work frame as follows:

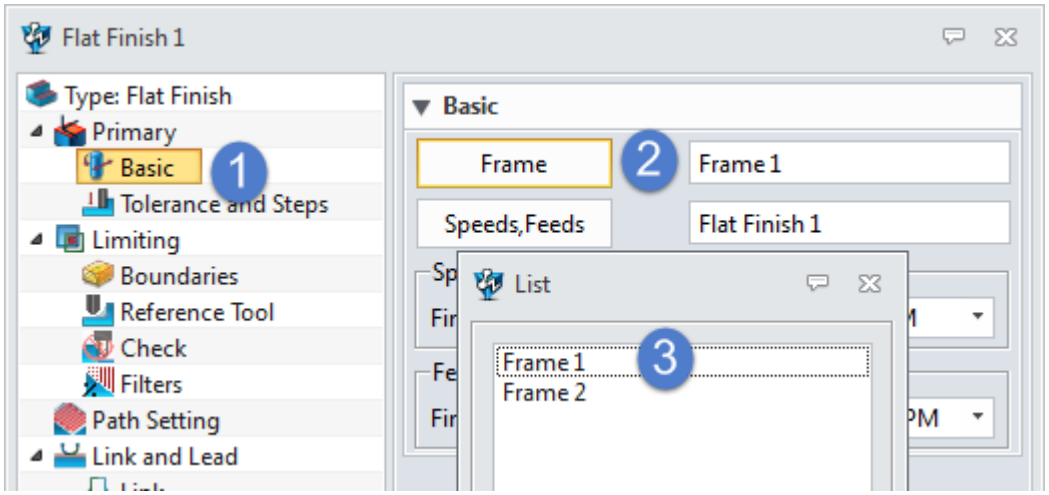


Figure14 Set up work frame for Flat finish operation

IV. Then set up the Boundaries as follows:

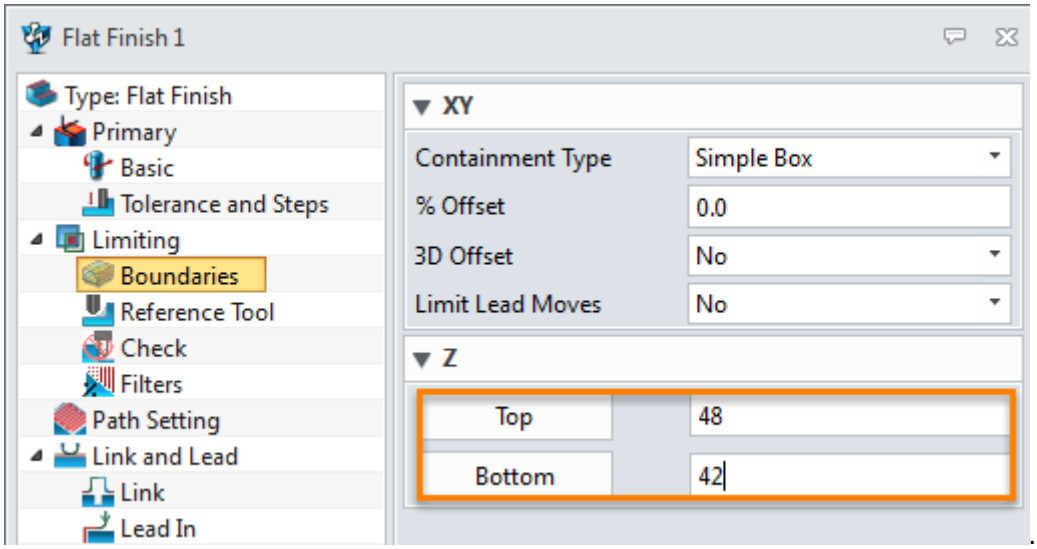


Figure15 Set up bottom boundary

V. Add the whole part as feature and then calculate (other parameters are default) then we can get the toolpath as follows:

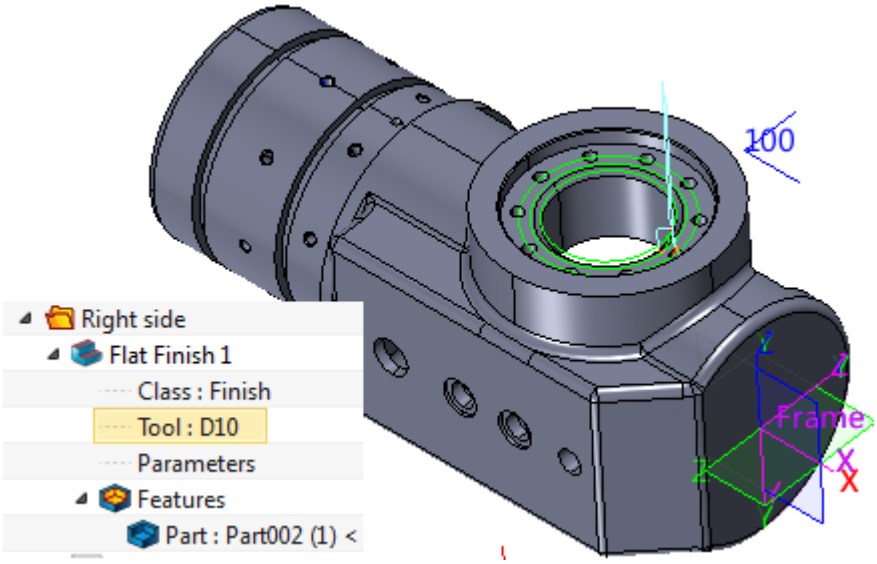


Figure16 Flat finish based on Frame1

Next, use the same process to choose other operations to finish the inner wall as follows:

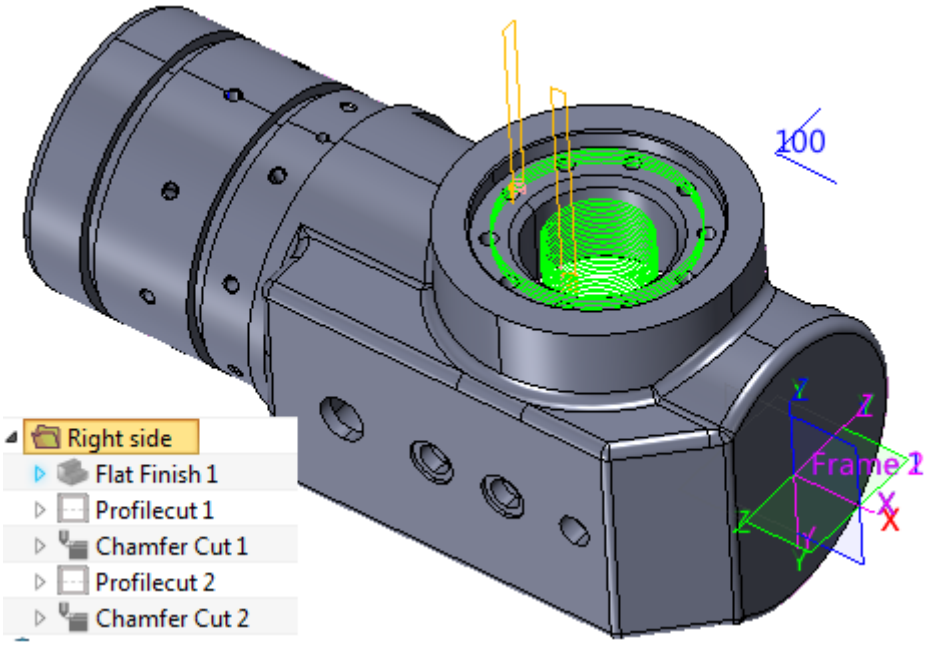


Figure17 Finish toolpath based on frame 1

So far we have created tool path in different frames, next we can go to verify the toolpath in frame1 as follows:

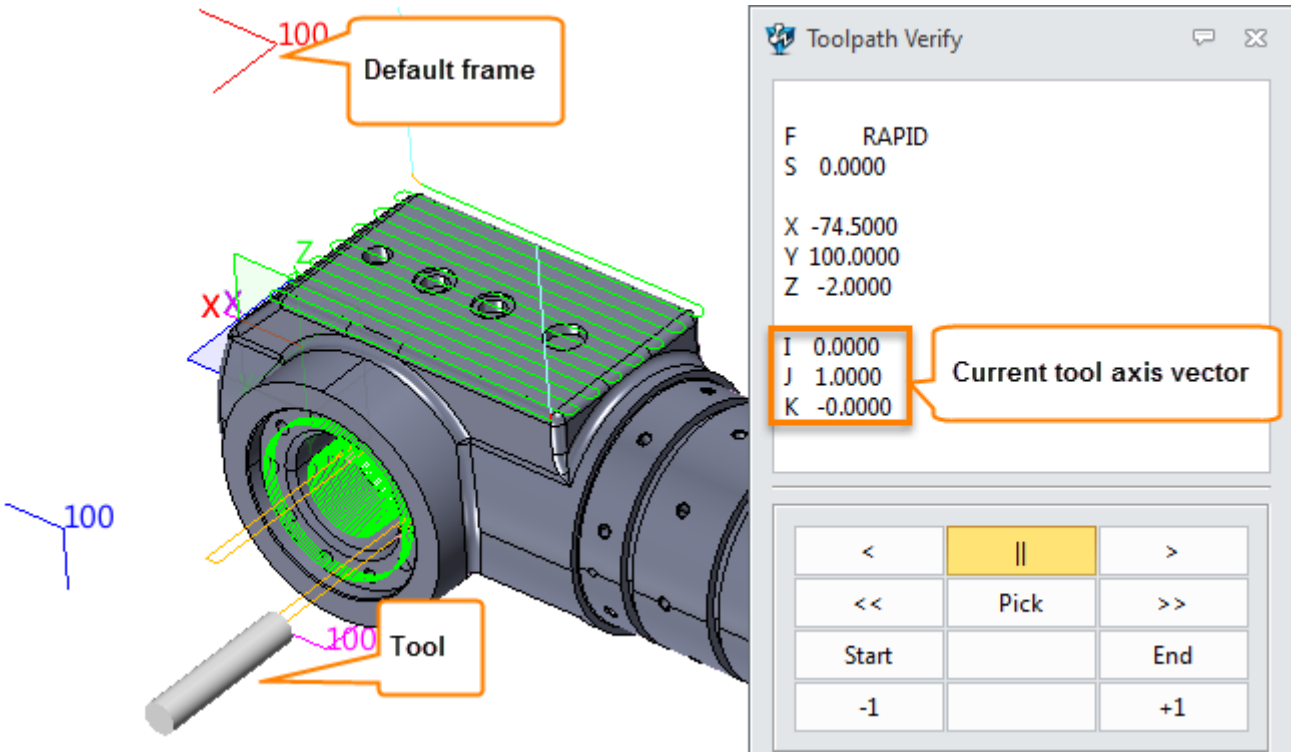


Figure18 Verify the tool and toolpath

From the verified result we can find that now the tool axis has been tranformed. If we check the tool axis's vector we can find it is not (0,0,1) which is default Z axis, butit is (0,1,0), that means the tool has been rotated around X axis 90 degree.

So far the indexing milling from default frame to frame1 is almost finsihed.

But there still is a probem confusing us: How is the tool transformed to this position? And we always want to track this process. Actually the "Inter Path Move" operation can achieve this function. It will connect the end point of the last operation in previous operations and the start point of the first operation in next frame.

STEP 03 Create link toolpath by Interpath operation:

I. Insert Interpath operation between Top side and Right side as follows:

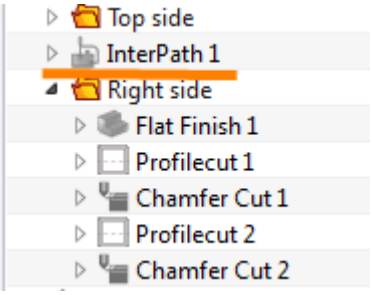


Figure19 Insert Inter Path Move operaion

II. And then set up the interpath operation as follows:

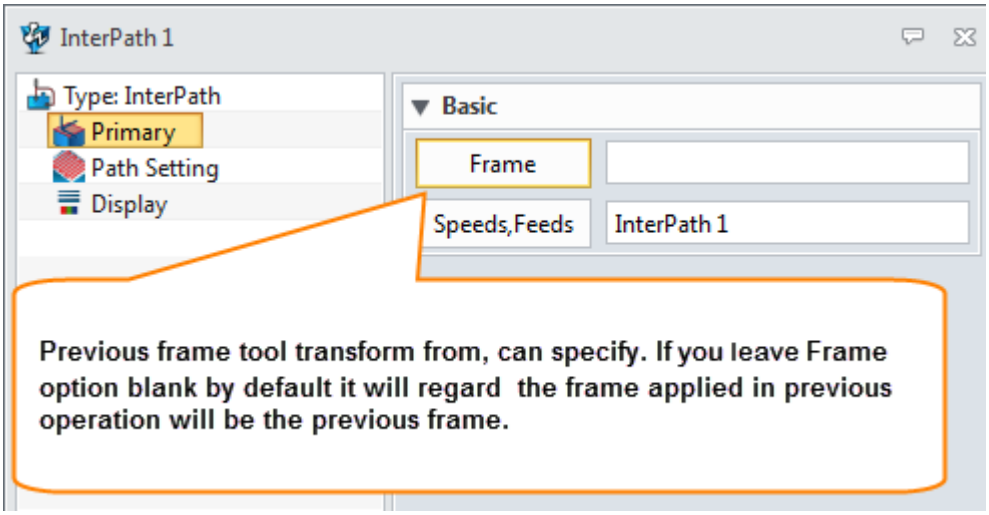


Figure20 Set up the previous frame

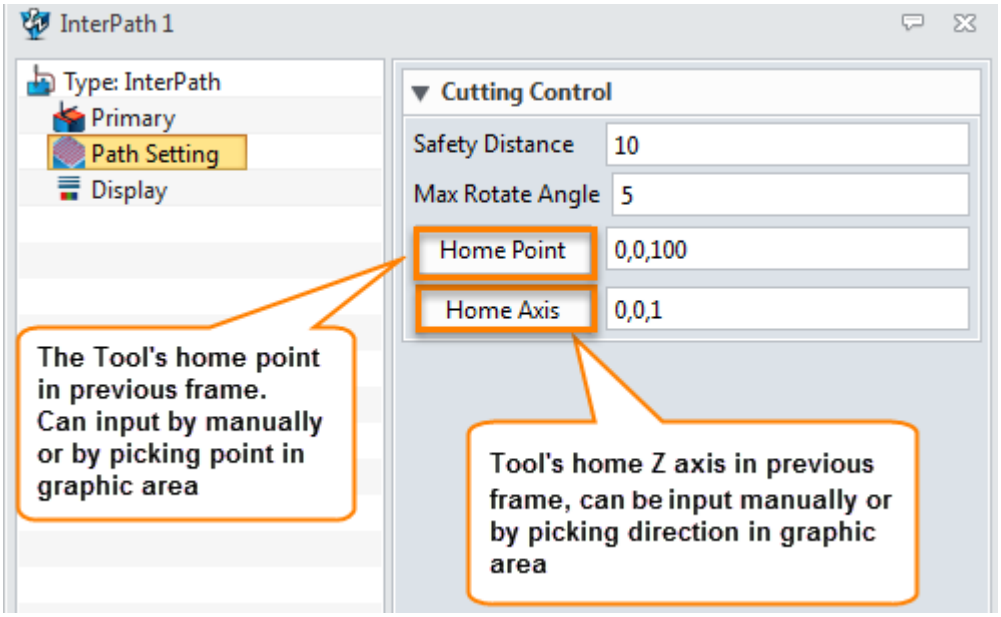


Figure21 Set up the Tool's information in previous frame

III. After calculating the InterPath operation we can get the following link toolpath :

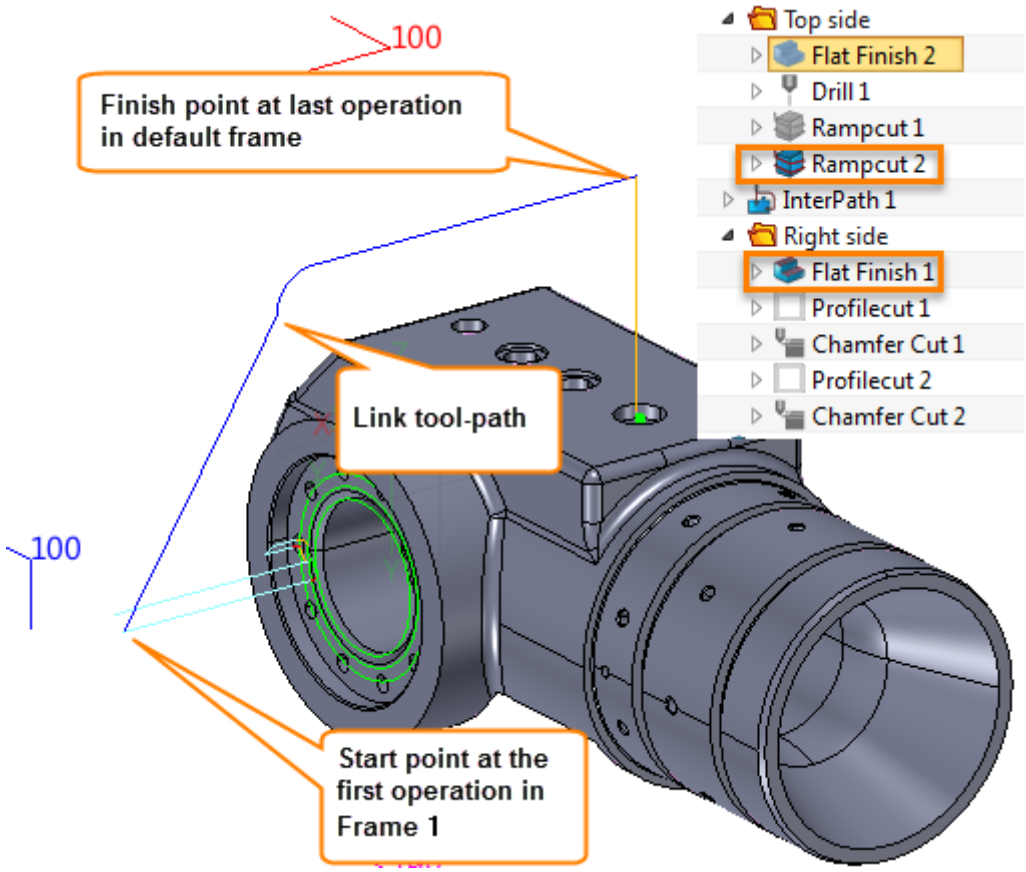


Figure22 Link tool path for transforming

The link toolpath can clearly show how the tool transform from the default frame to Frame 1, so it is helpful to check if it is safe or not.

IV. Verify the InterPath as follows:

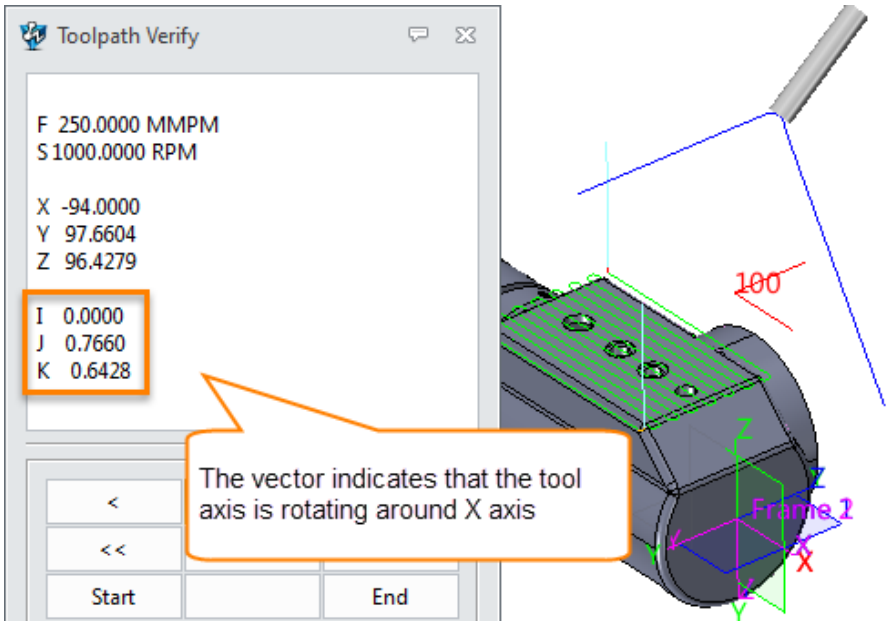


Figure23 Verify link path

Note: The Interpath operation's tool should be the same as the tool used by the first operation in next frame as follows:

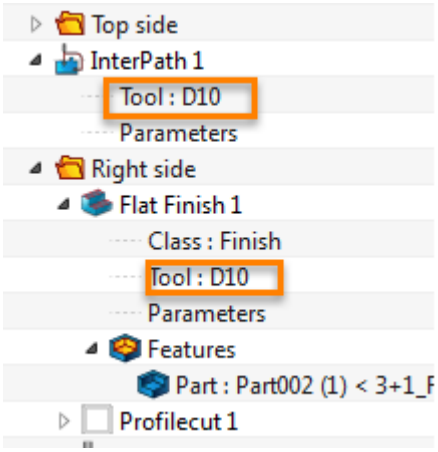


Figure24 Tool for InterPath operation

STEP 04 Create the toolpath on the bottom side as follows:

I. Create toolpath based on the frame 2 as follows:

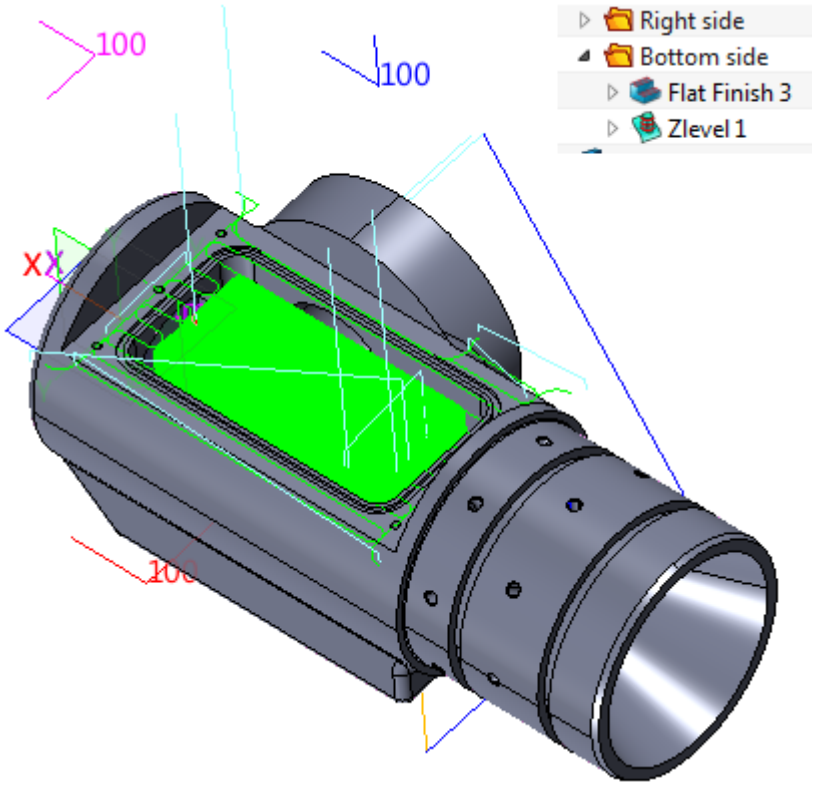


Figure25 Tool path on bottom side

II. Insert Interpath operation and set up as follows:

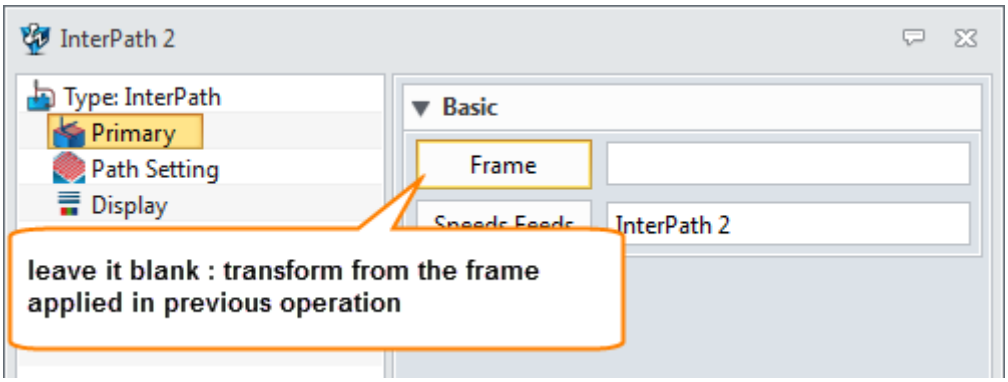


Figure26 Set up Interpah operation

III.Calculate the Interpath operation we can get the link tool-path between right side and bottom side as follows:

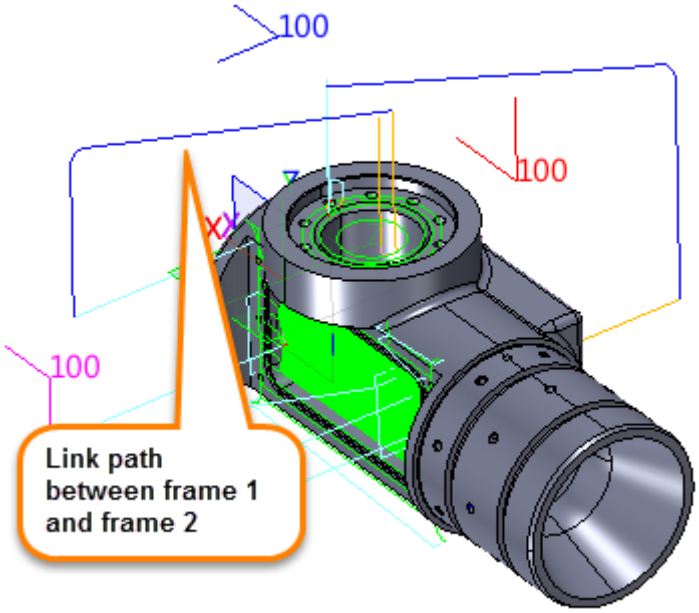


Figure27 Link path beween right side and bottom side

So far we have finished the 4X indexing milling process for this part. Next we need to output the NC code for it. Before we output the NC code, we have to check some necessary setting such as machine type, controller, output space etc.

1.1.4 Set Up The Machine

Double click on the machine tab to activate the Machine Manager and set up as follows:

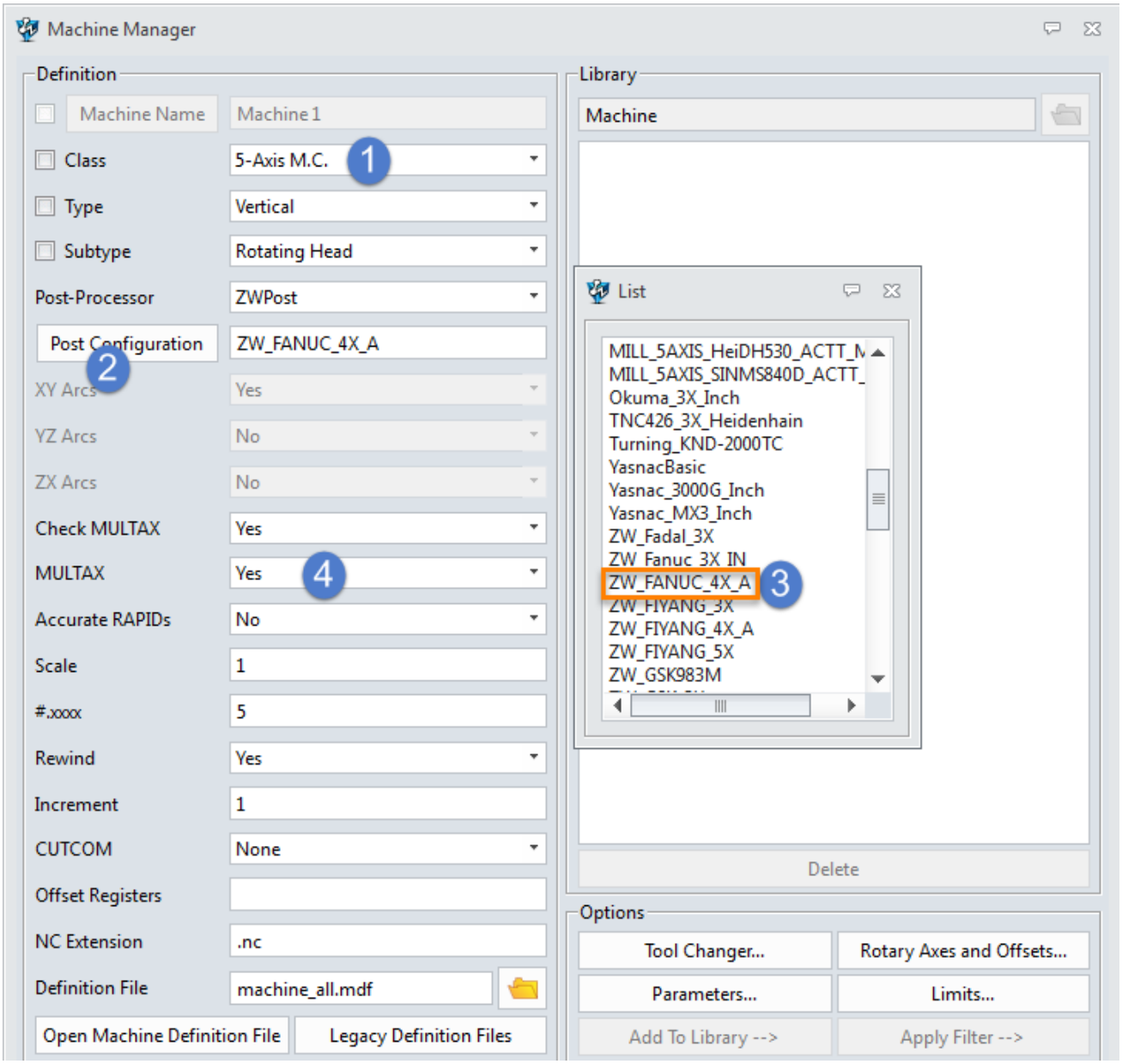


Figure28 Set up machine

Here let's choose the "**ZW_FUNAC_4X_A**" as an example to output.

1.1.5 Set Up Output Space

Since here we have already defined different frames,it is necessary for us to specify on which frame to output the NC code.

STEP 01 Insert a new NC file as follows:

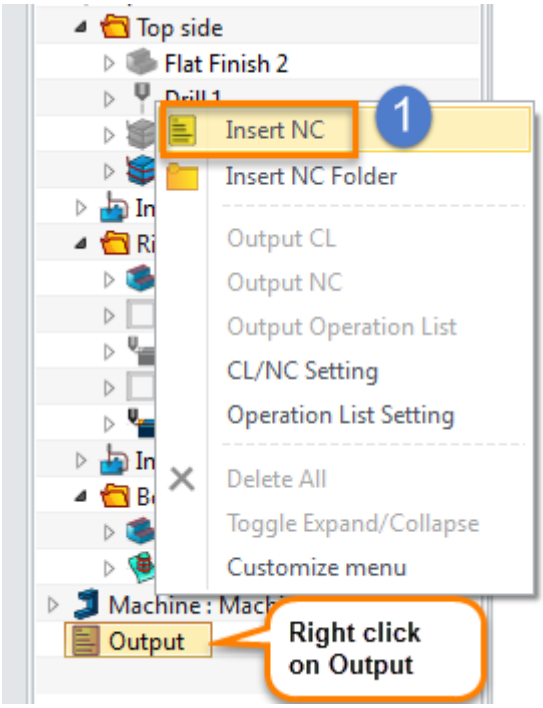


Figure29 Insert new NC file

STEP 02 Add operation for output as follows:

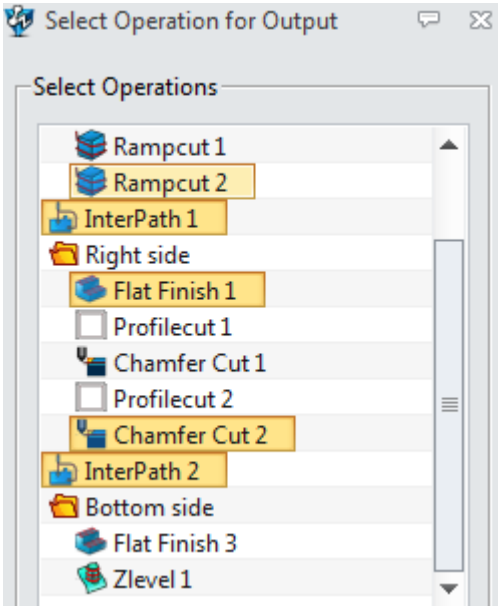


Figure30 Add operations for output

Here we just want to verify if the NC code can correctly output the rotating angle, so we can just pick some connected operation as test.

STEP 03 Set up output space as follows:

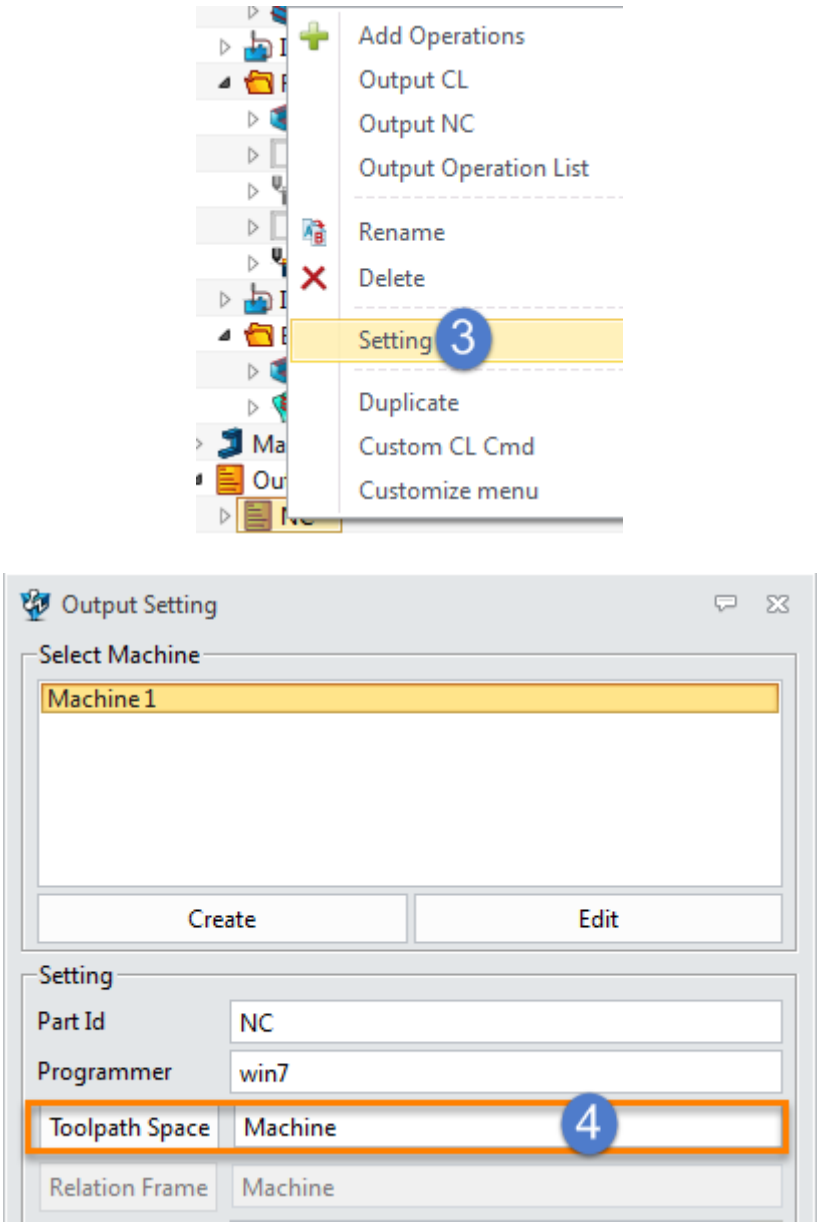


Figure31 Set up output space

1.1.6 Output NC Code:

Choose the first Inter-Path operation as an example to show:

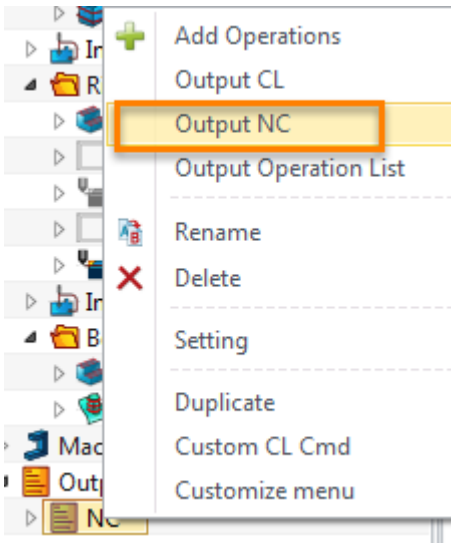


Figure32 Output NC

Output NC code as follows, it shows how the tool rotates around X axis from the default frame to frame1.

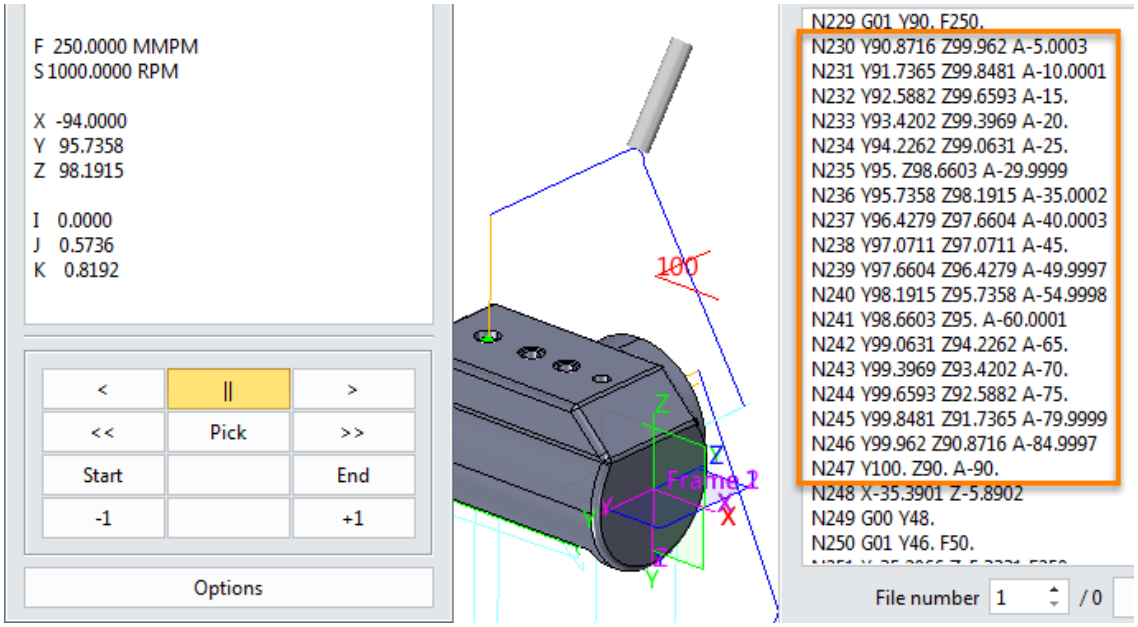


Figure33 Output NC code

If you want to check more, please go through the whole NC code by yourself.

So far, we have finished the sample case for 4X indexing milling. From this process we can find that the key points for the indexing milling are as follows:

1. Align the part according to the machine structure
2. Create sub frame
3. Set up 4&5 X machine type
4. Set up the output Space

1.2 5X Indexing Milling

Last case we finished the 4X indexing milling. Now we can use the same way to finish the 5X indexing milling case. Following is a 5X indexing milling case we are going to finish:

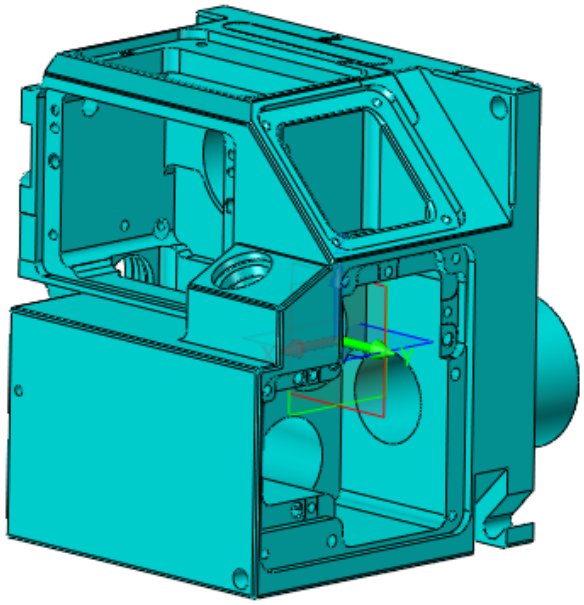


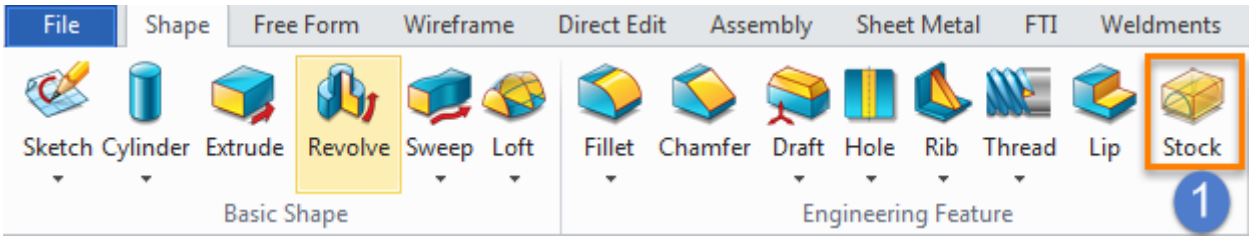
Figure34 5X indexing milling case

Analysis: From this part we can find that if we want to finish all sides by one clamping, the machine needs to rotate the tool axis around different axis. Here let's suppose the 5X machine is AC type, which means rotating around X, Z. Besides here we just take the finished operation as an example to show how to achieve in ZW3D CAM.

1.2.1 Align Part

Just as what we mentioned above, in order to make use of the default frame we can align the part in CAD level as follows, here the origin is located at the center of top.

STEP 01 Create a assistive stock for positioning part as follows:



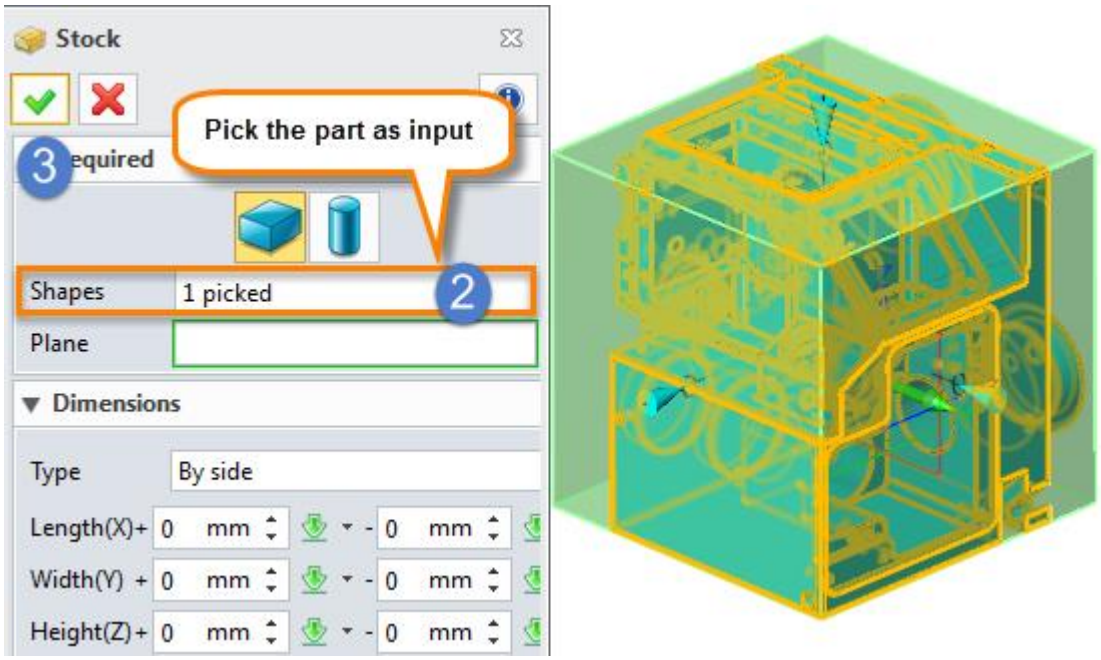


Figure35 Create stock in CAD level

STEP 02 Move the part as follows:

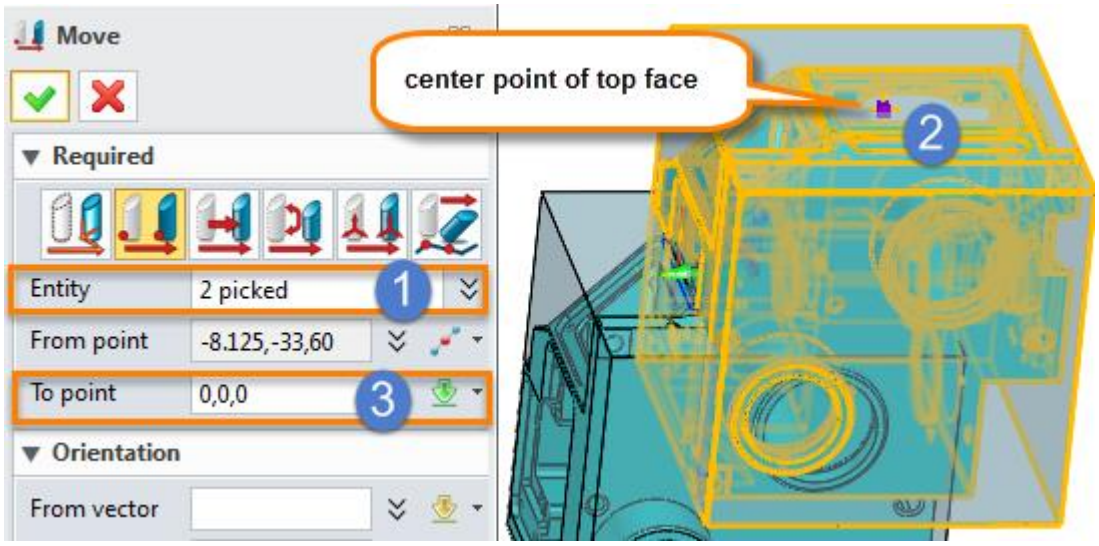


Figure36 Move the part and stock to origin point

STEP 03 Result will be as follows:

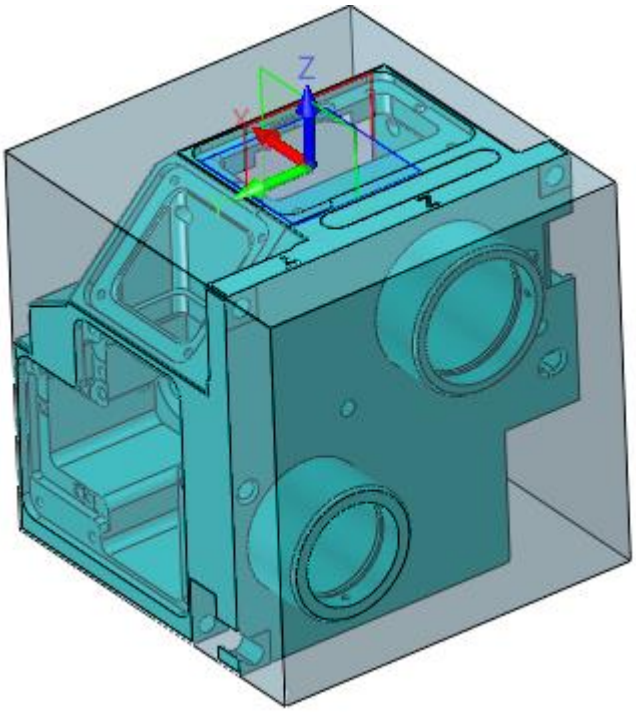


Figure37 Aligned Part

Then we can delete or hide the stock.

1.2.2 Create Sub-Frame For Multiple Faces

Here we will choose some faces as examples to show this detailed process which is as follows:

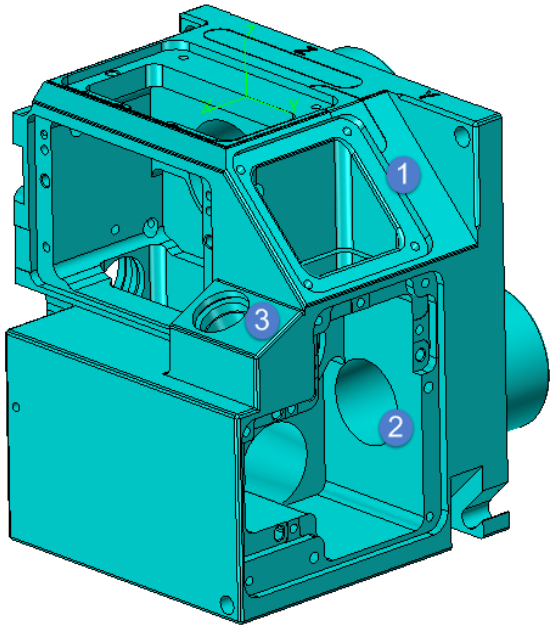


Figure38 Indexing milling area

STEP 01 Create the first frame as follows:

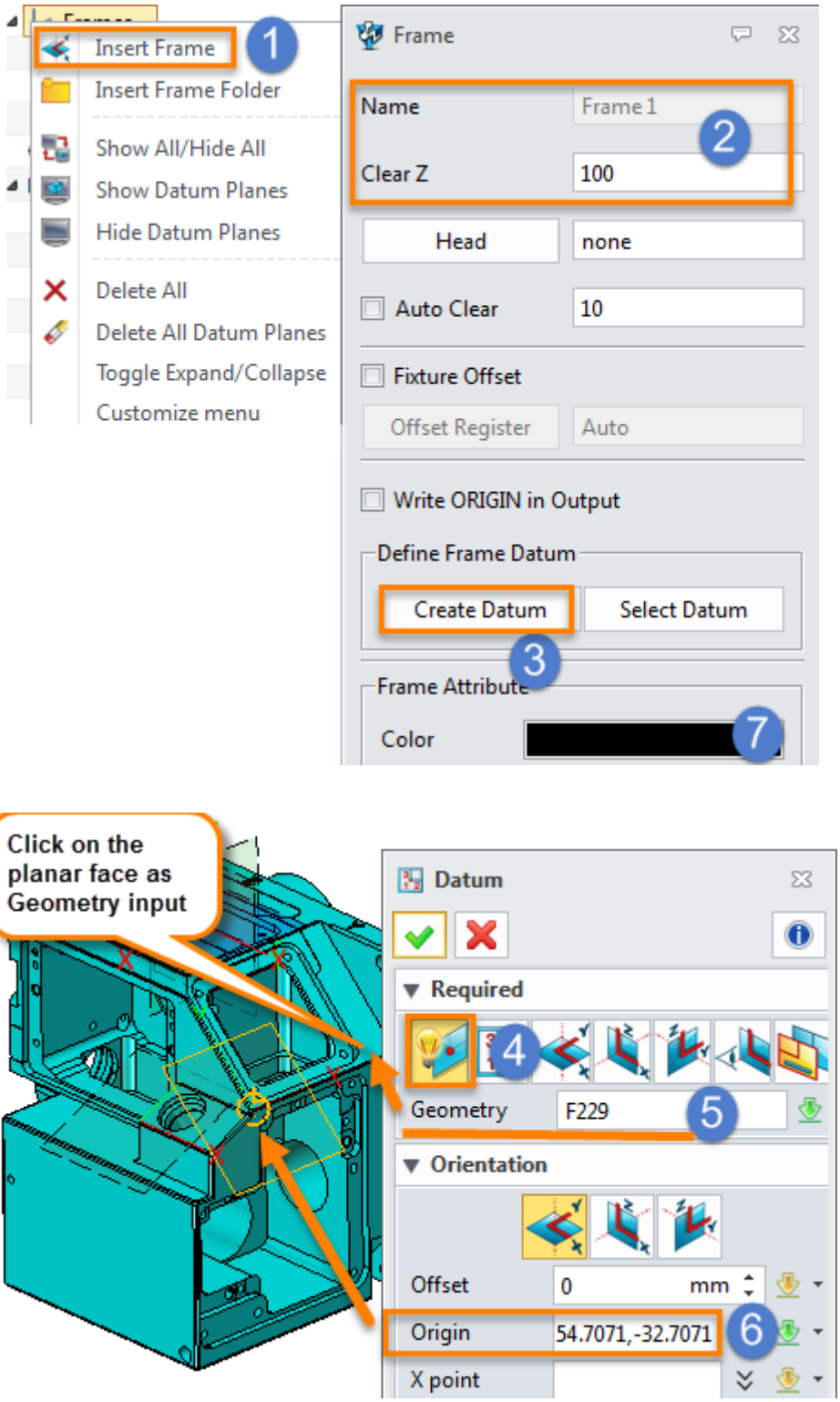


Figure39 Create the first sub frame

Finished result is as follows:

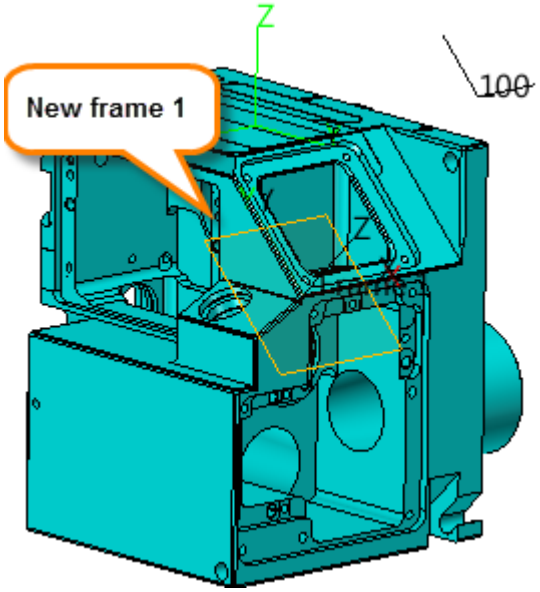


Figure40 Created new frame 1

Note : Here the sub frame is only used to define the tool axis's direction, so the sub frame's origin point can be located at any position. In order to show it clearly,here we chose an obvious position mentioned above.

STEP 02 Create other frames on area 2 and area 3 as follows:

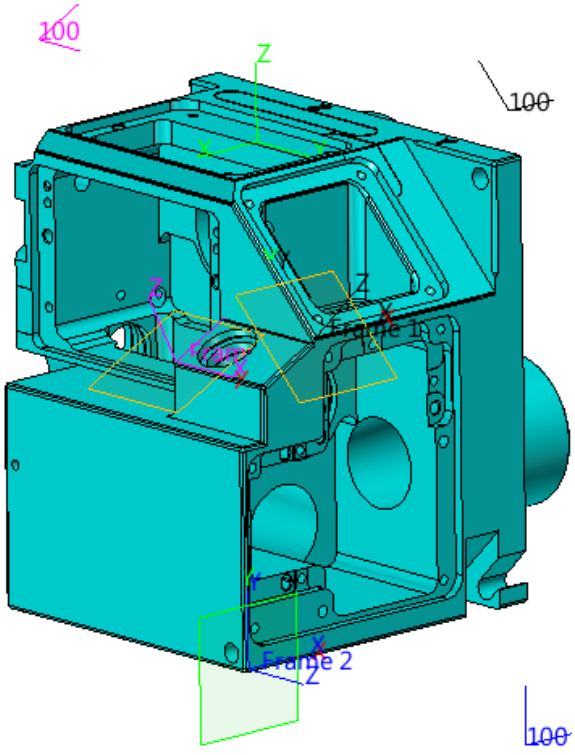


Figure41 Finished frame

1.2.3 Create Toolpath Based On Different Frame

The process of creating 2X or 3X toolpath based on different frame is the same as 4X indexing milling , so here we will skip this process of creating toolpath and just show you the result as follows:

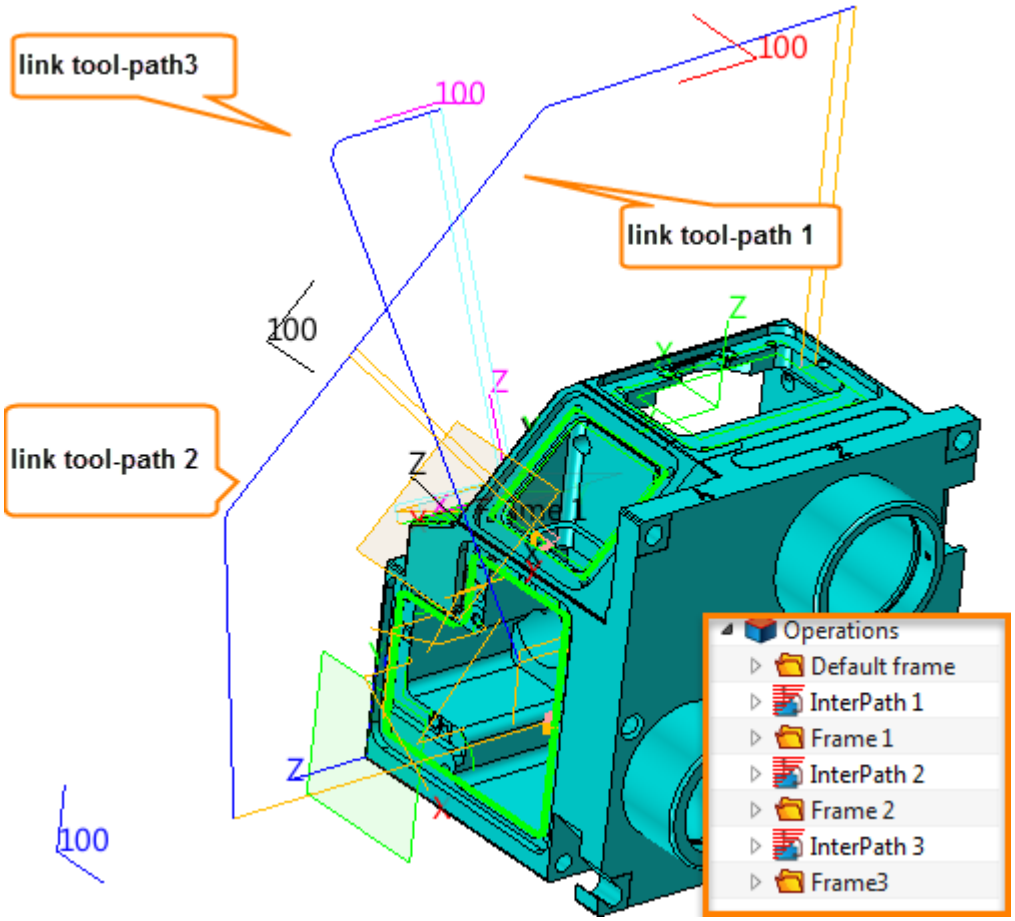


Figure42 Finished toolpath

For your reference you can open the file " **5X_indexing_With toolpath.Z3"** directly.

But you are strongly recommended to finish it from beginning again by yourself.

1.2.4 Set Up Machine and Output Space

STEP 01 Set up the machine and choose the post processor "**ZW_FUNAC_5X**" :

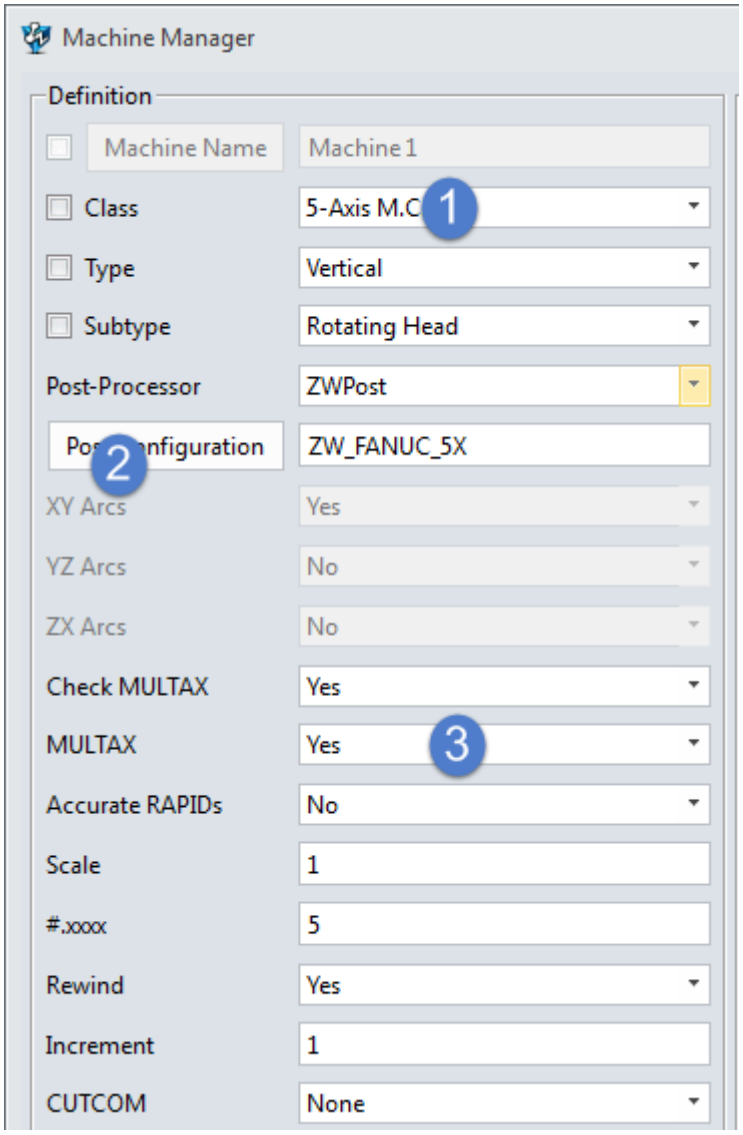


Figure43 Set up post processor

STEP 02 Set up the machine structure as AC type by the post Processor editor as follows:

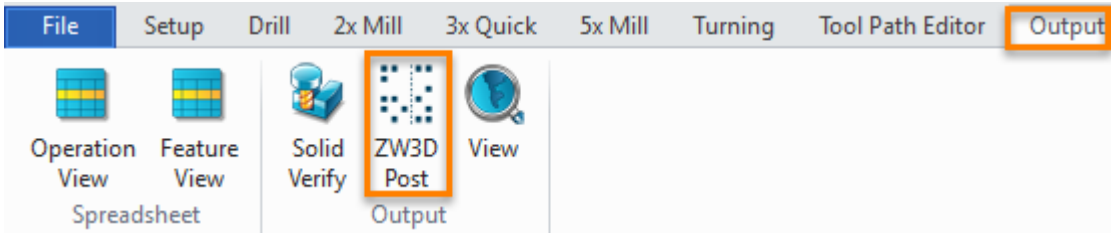


Figure44 Open ZW3D Post editor

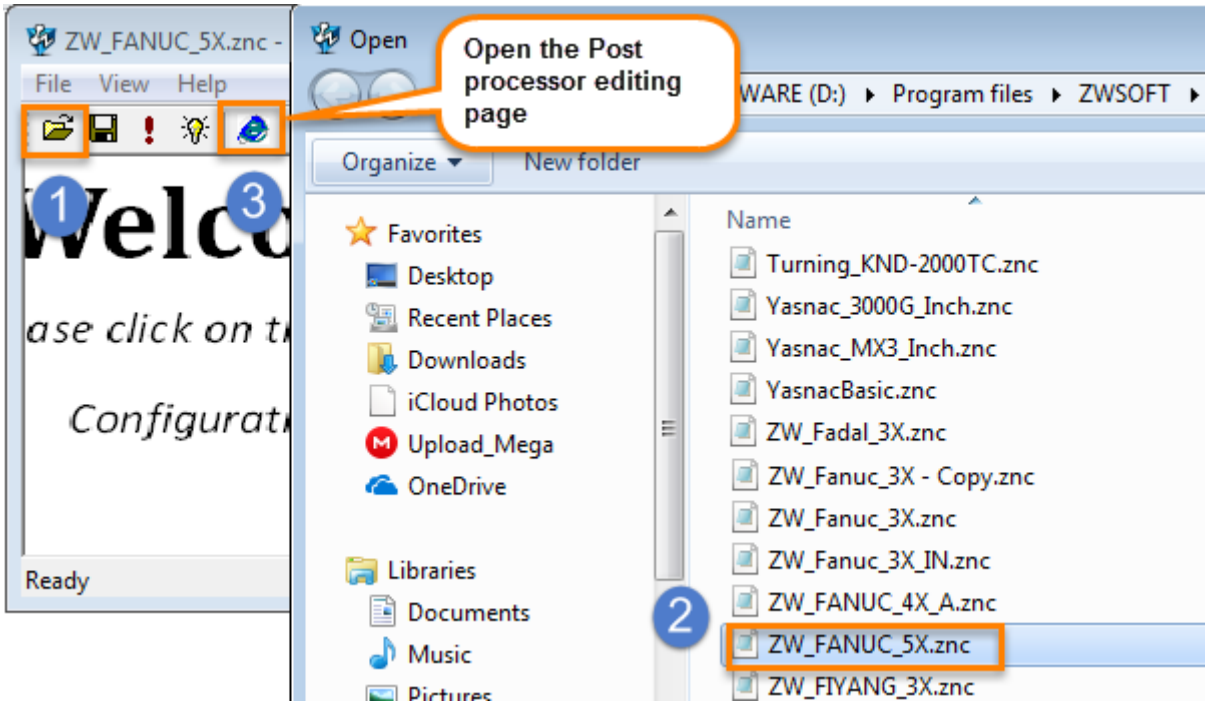


Figure45 Load the related Post Processor

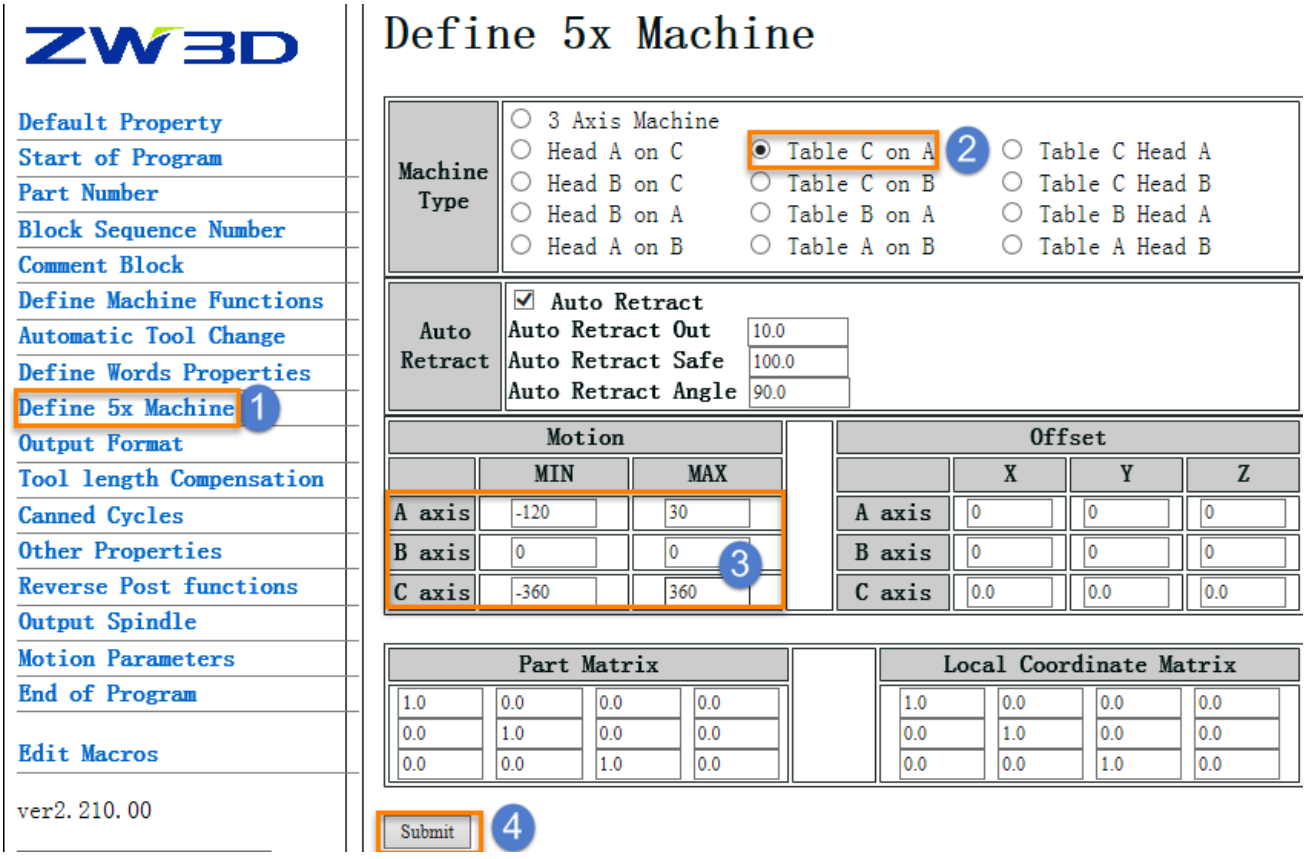


Figure46 Set up 5X machine type and rotary axis's limitation

Figure47 Save modification

STEP 03 Set up Output space as Machine as follows:

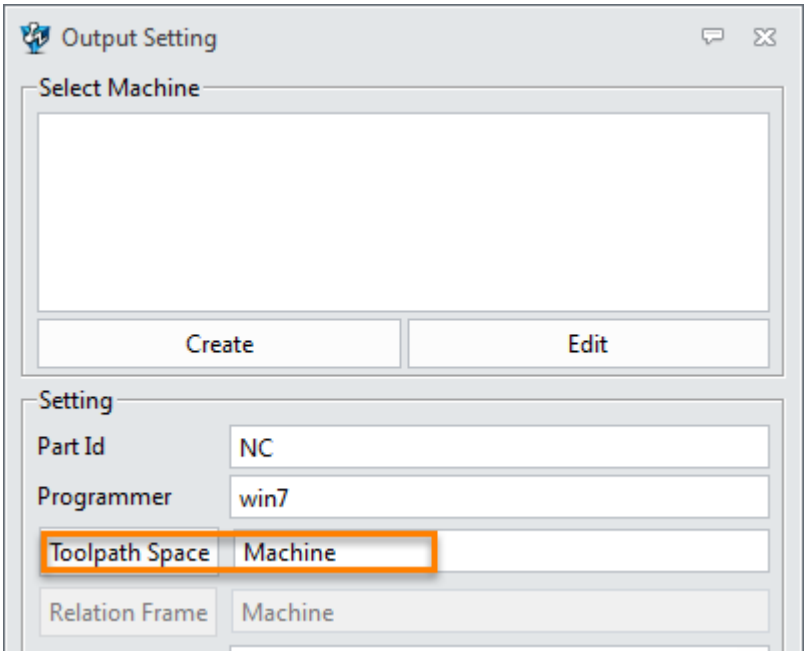


Figure48 Set up output space

1.2.5 Output NC Code

Here just choose the connected operation to output the NC code and check it as follows:

STEP 01 Choose the connected operation as output:

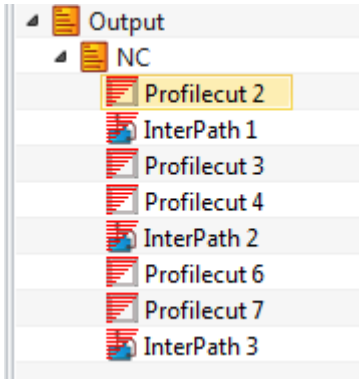


Figure49 Choose operations as output

STEP 02 Output NC code and verify

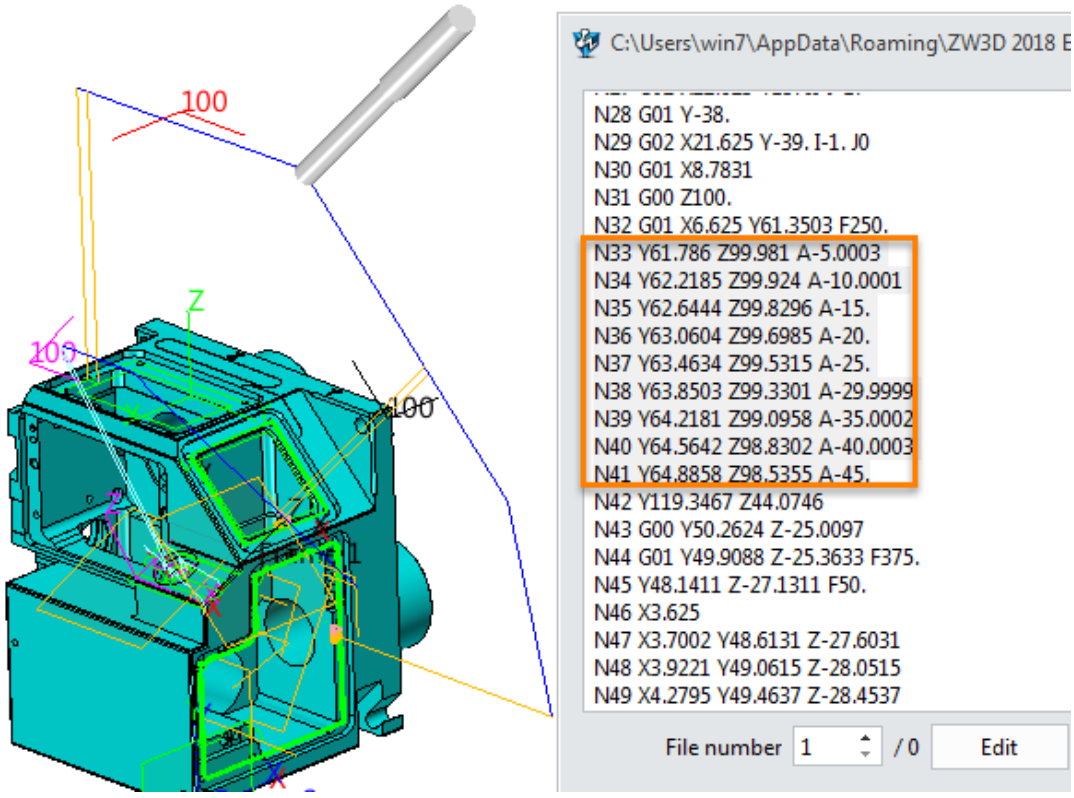


Figure50 Indexing angle code between default frame and framw1

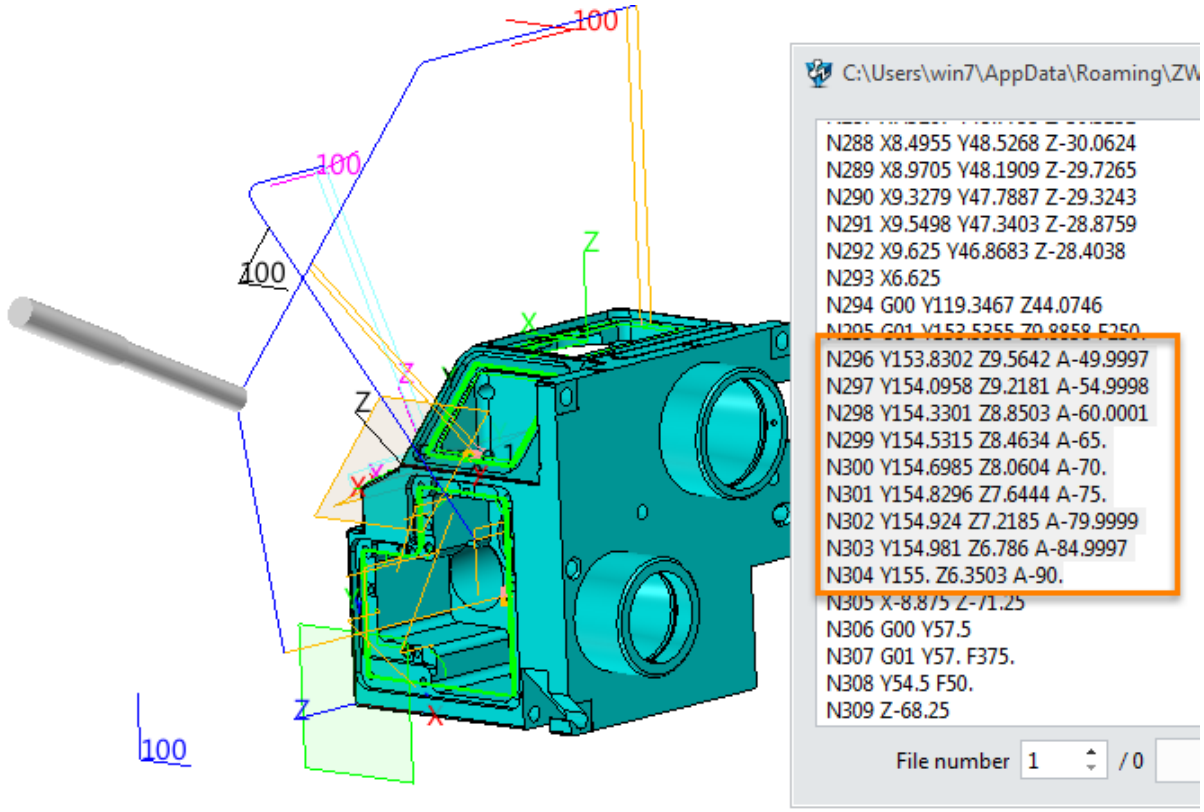


Figure51 Indexing angle code between frame 1 and frame 2

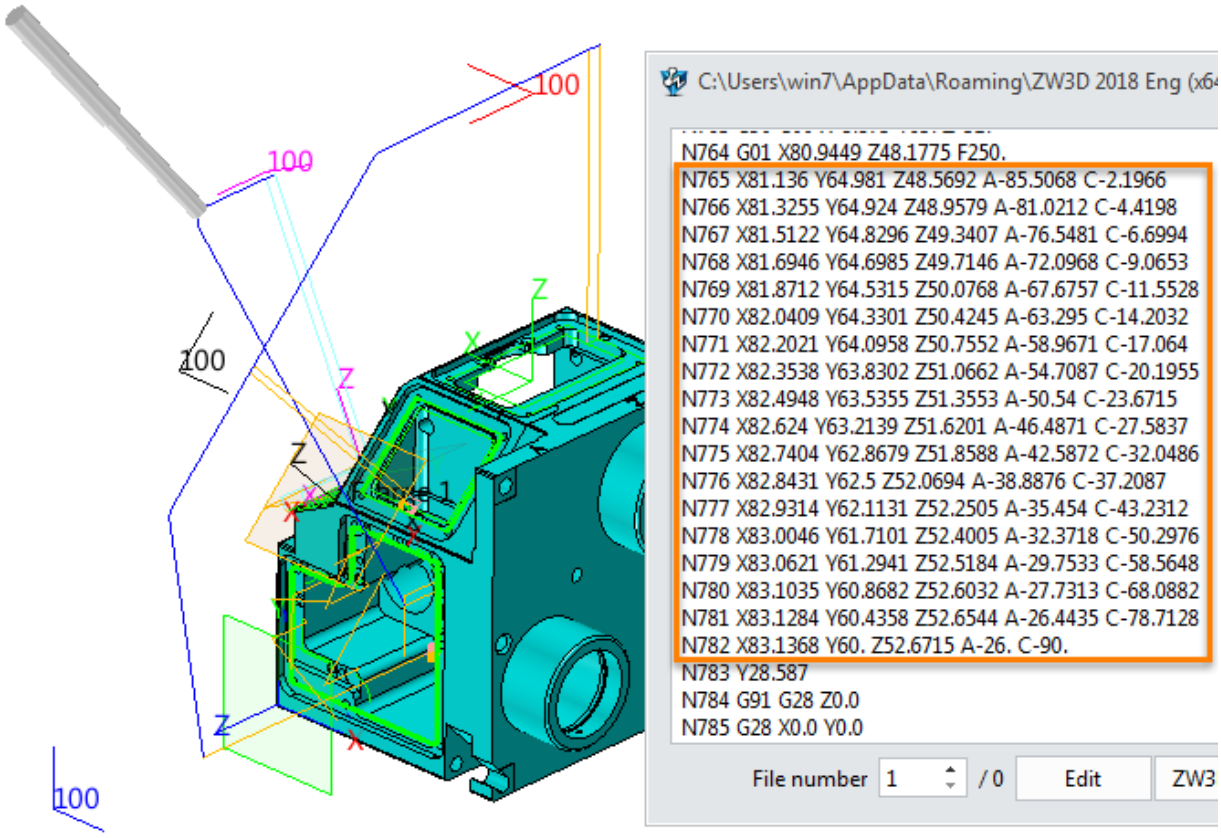


Figure52 Indexing angle code between frame2 and frame3

1.3 5X Silmutaneous Movement Operations

Next we will learn how to use the 5X operations as follows:

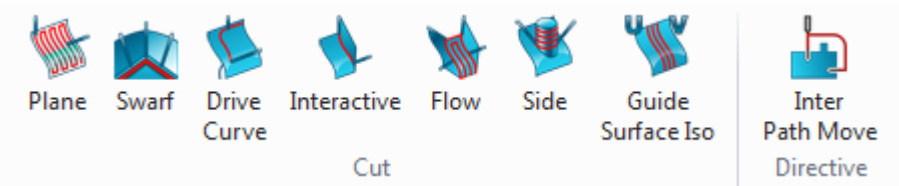


Figure53 5X Simultaneous Movement Operations

Among those operations we have introduced the InterPath Move operation, so we can skip it. As for the **Interactive** operation, since it is not used very often, it will be not introduced in this chapter, if you are interested in it then you can check the help document.

ZW3D CAM 5X simultaneous movement operations integrate both 4X and 5X functions together. Therefore, it is able to switch the 5X simultaneous operation to 4X simultaneous operation by the tool axis control parameter.

1.3.1 5X Plane Cut Operation

Philosophy: The 5 axis plane cut creates a cutting pattern based on parallel cuts at a user-specified angle with respect to the frames X axis. This cutting pattern can be used to control the tool tip or the contact location of the tool on the part. It is possible to constrain the tool axis to a plane (for 4 axis milling) or to a specific orientation (for 3 axis milling).

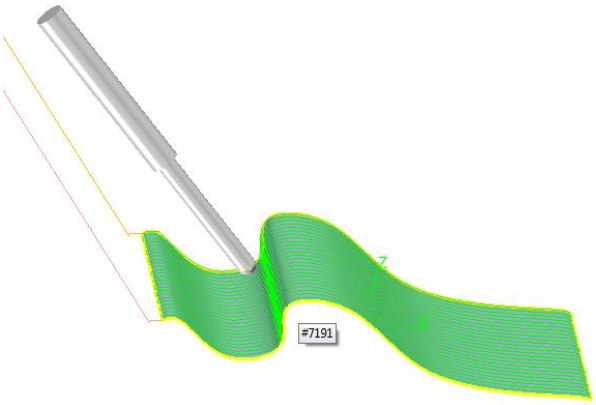


Figure54 5X Plane Cut

1. How to run 5X plane operation

5X Plane operation works on general surface, so it only requires that the general part surface can create toolpath. Next Let's open the practice file "**5X Plane Cut**.Z3" as follows:

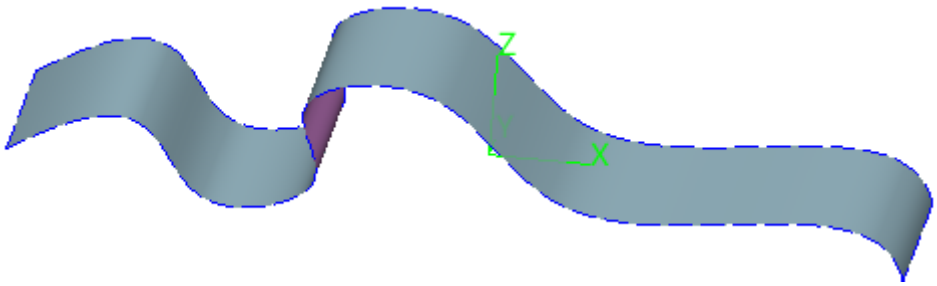


Figure55 5X Plane Cut practice part

Then we shall use this part to show you how to create 5X Plane Cut tool path on it:

I. Define a general surface as follows:

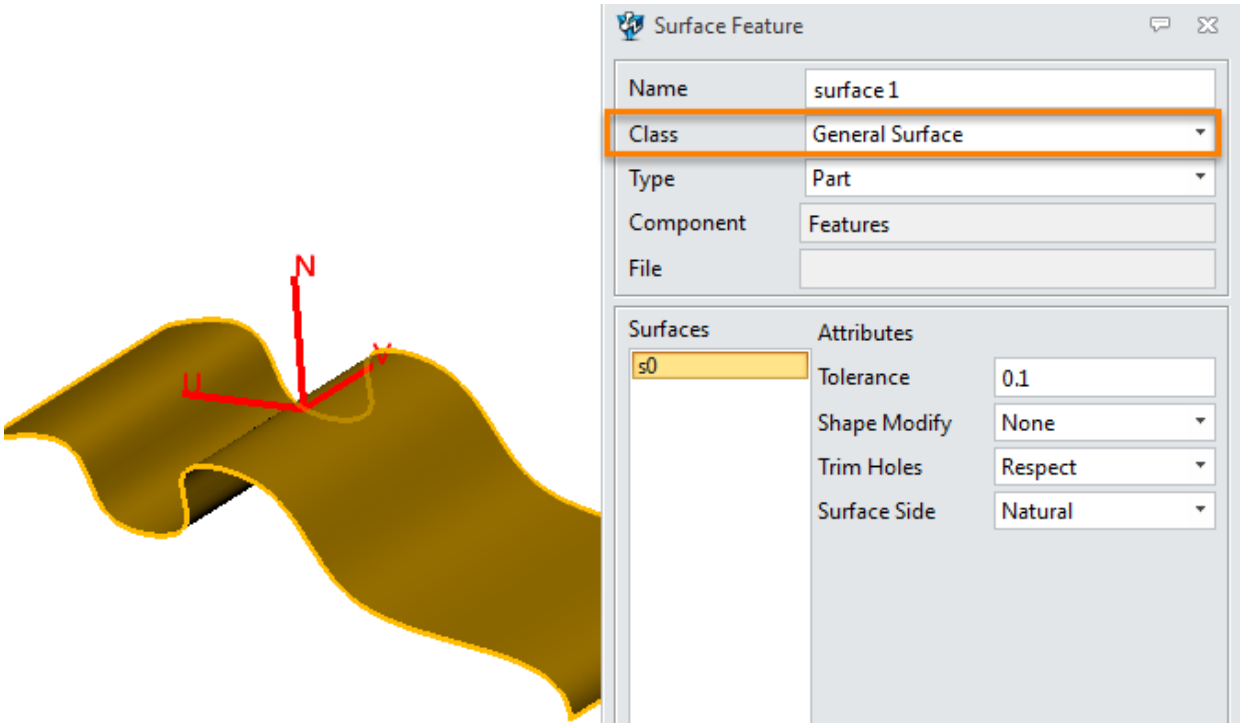


Figure56 Define general surface for 5X plane cut

II. Calculate operation by default parameter to get toolpath as follows:

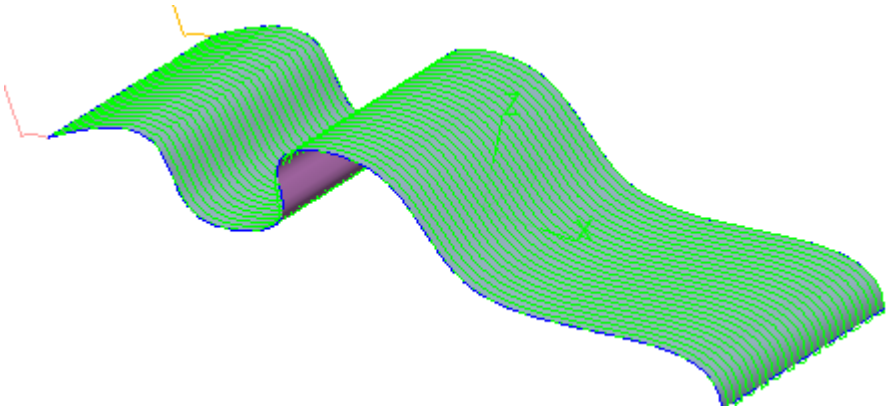


Figure57 5X Plane Cut toolpath

2. Set up 5X Plane Cut parameter:

I. Primary parameters:

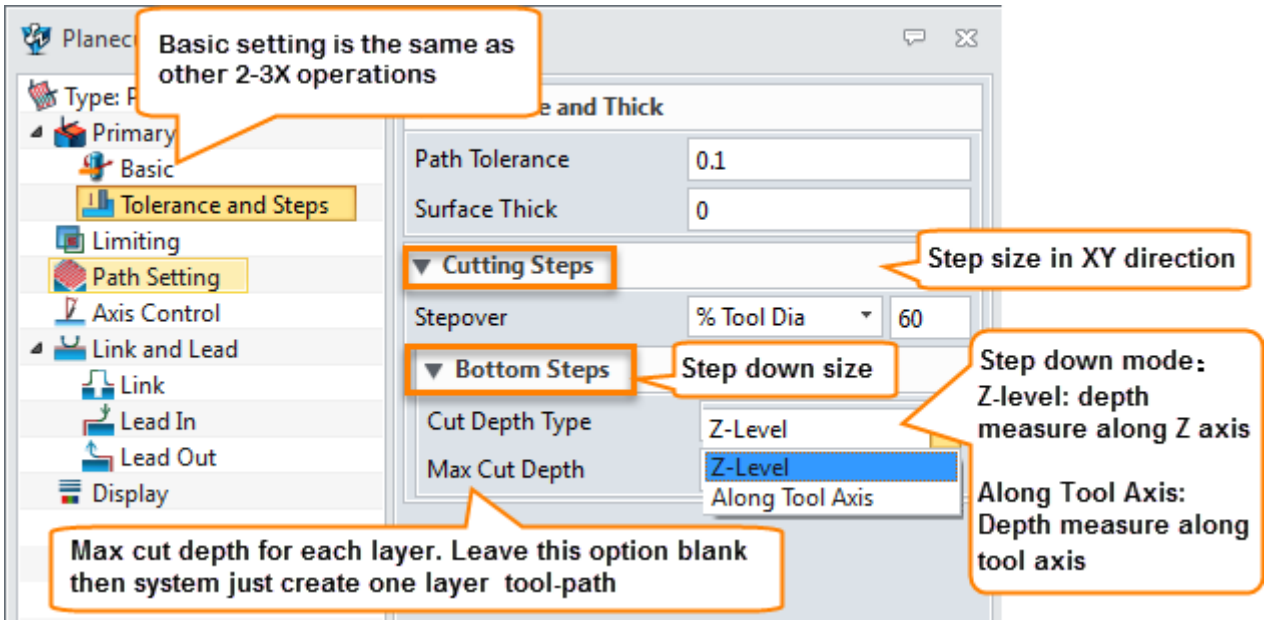


Figure58 Primary parameters

II. Path setting parameters:

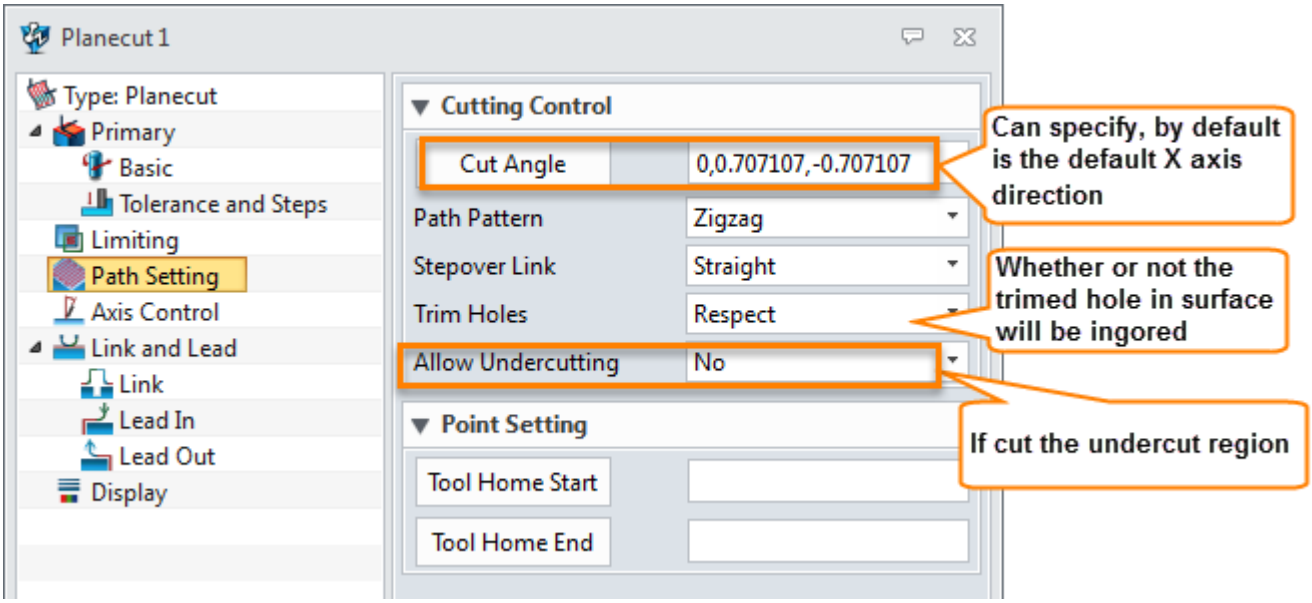


Figure59 Path setting parameters

III. Axis control parameters:

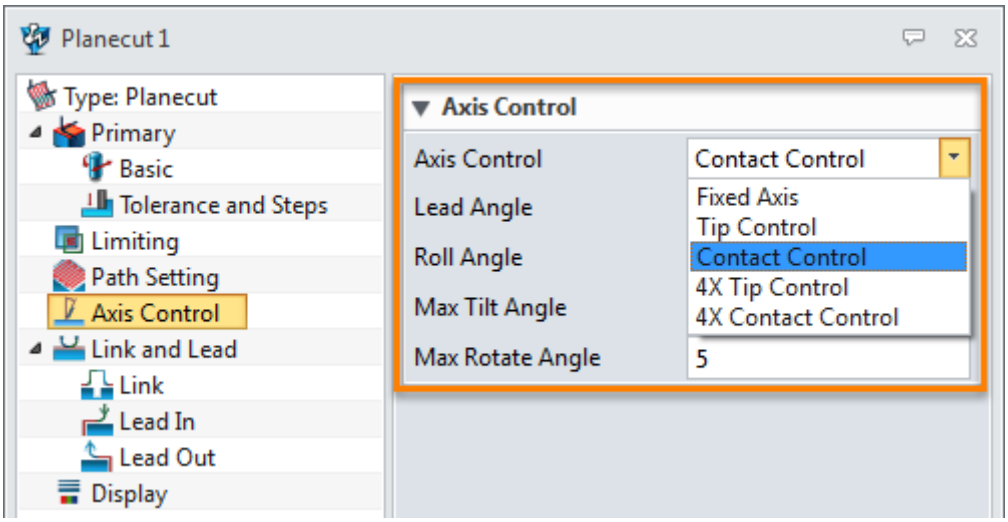


Figure60 Axis control parameters

- **Fixed Axis**: The tool axis will be determined by lead and roll angles along the cutting direction and relative to the Z axis of the frame. Actually, you can think it is the frame's Z axis, since by default the Lead and Roll angle are all 0. As follows:

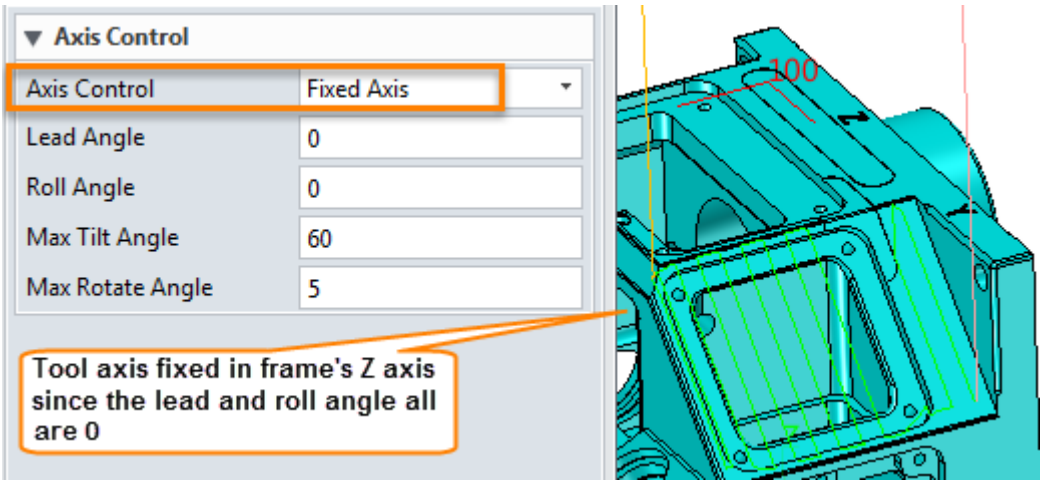


Figure61 Fixed Axis tool.

- **Tip Control**: The local contact data determines the cutter orientation while the cutter tip point is kept within the cutting plane. Actually, when the Tip of Tool is in the cutting plane then the tool axis is the surface's normal way.

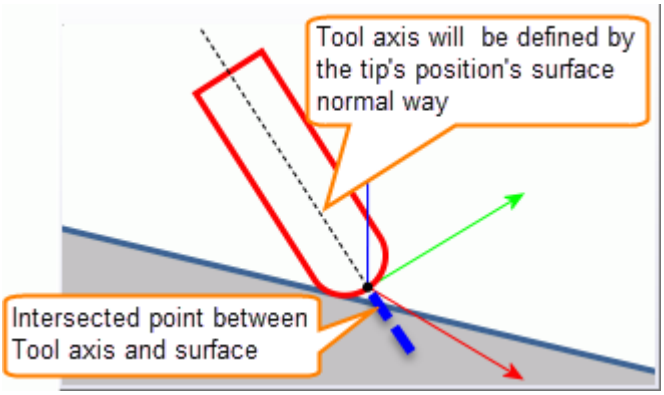


Figure62 Tool tip

- **Contact Control**: The local contact data determines both the cutter orientation and tip point. It uses the local contact point as reference and the normal way of this point together to determine the tool axis's position.

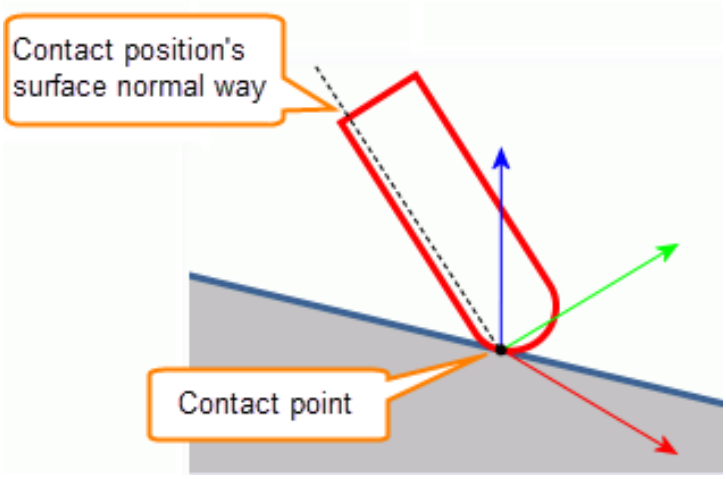


Figure63 Contact control case

- **4X Tip Control:** Tip contol option work in 4X mode, and you need to specify 4X plane as follows:

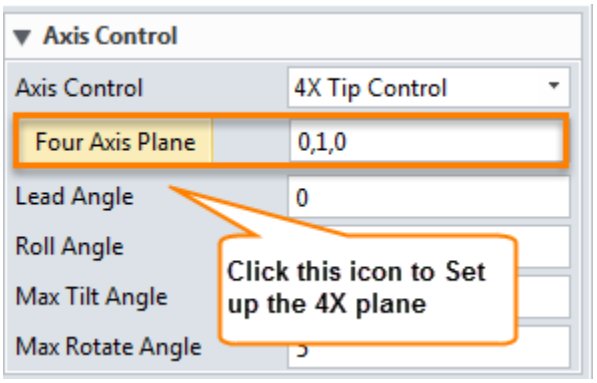


Figure64 Specify 4X plane

- **4X Contact Control**: Contact control option work in 4X mode, you also need to specify 4X plane.
- **Lead angle:** The angle of the tool axis will be tilted toward the movement direction

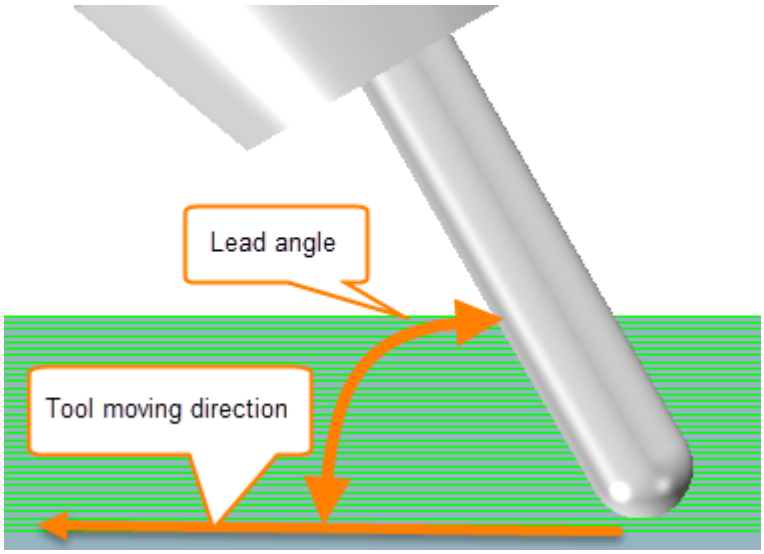


Figure65 Lead angle

- **Roll angle** : The angle of the tool axis will be tilted to the direction which is perpendicular to the forward motion direction. A positive value will tilt the tool to the right, negative to the left.

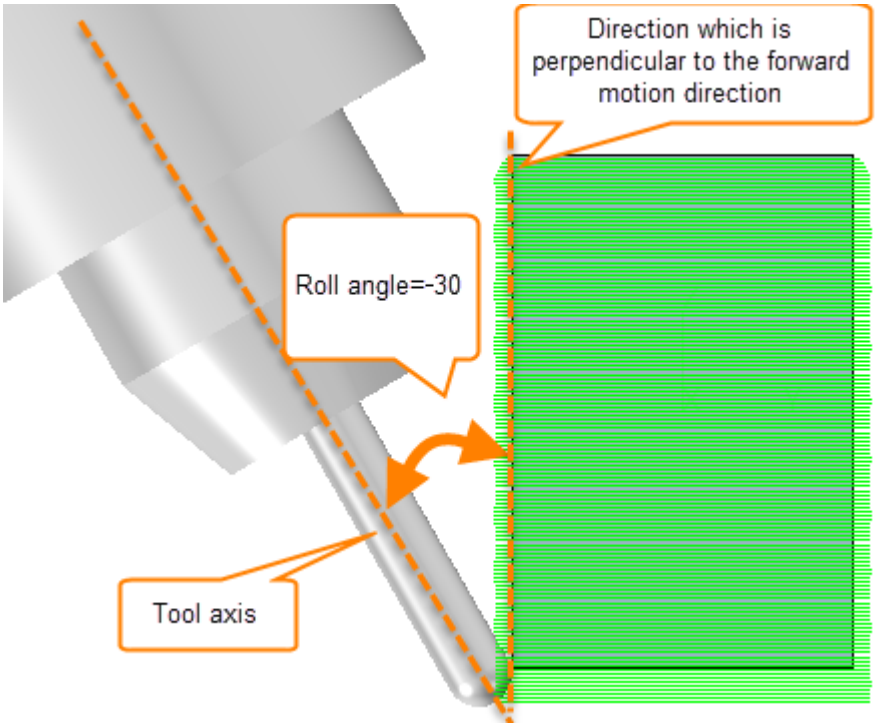


Figure66 Roll Angle

1.3.2 5X Swarf Cut

Philosophy:The 5 axis Swarf Cut operation uses **Control surfaces** to calculate the tool path. The tool axis is controlled by **Drive surfaces** with which the side of the tool maintains contact. The bottom of the tool (contact point) is controlled by **Part surfaces**.

Open the practice file " **5X_SwarfCut_Practice.Z3**" as follows:

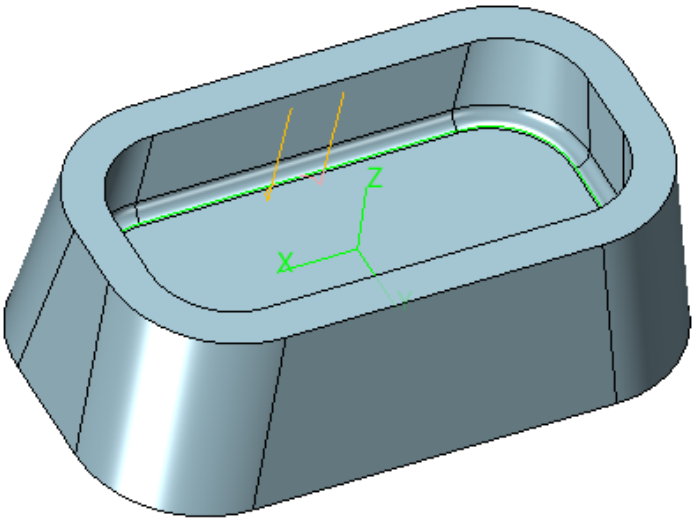


Figure67 5X Swarf Cut practice file

From the part structure we can find that it is not able to cut the side wall and radius on the bottom face by 3X tool path since the wall is titled. So here we hope the tool axis can title following side wall and keep tangent to the side wall as follows:

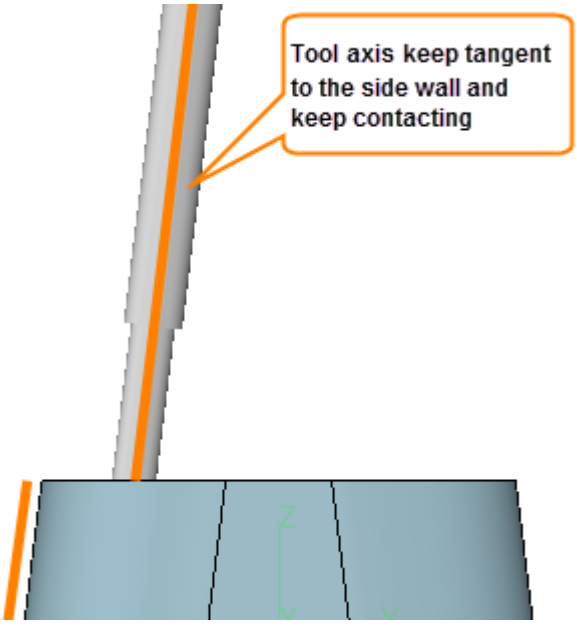


Figure68 Keep tangent to side wall face

Next let's show how to create 5X swarfCut tool path on this part as follows:

1. How to run Swarf Cut operation:

STEP 01 Define drive surface : the drive surface will be used to control the tool axis by maintaining contact with the side of tool

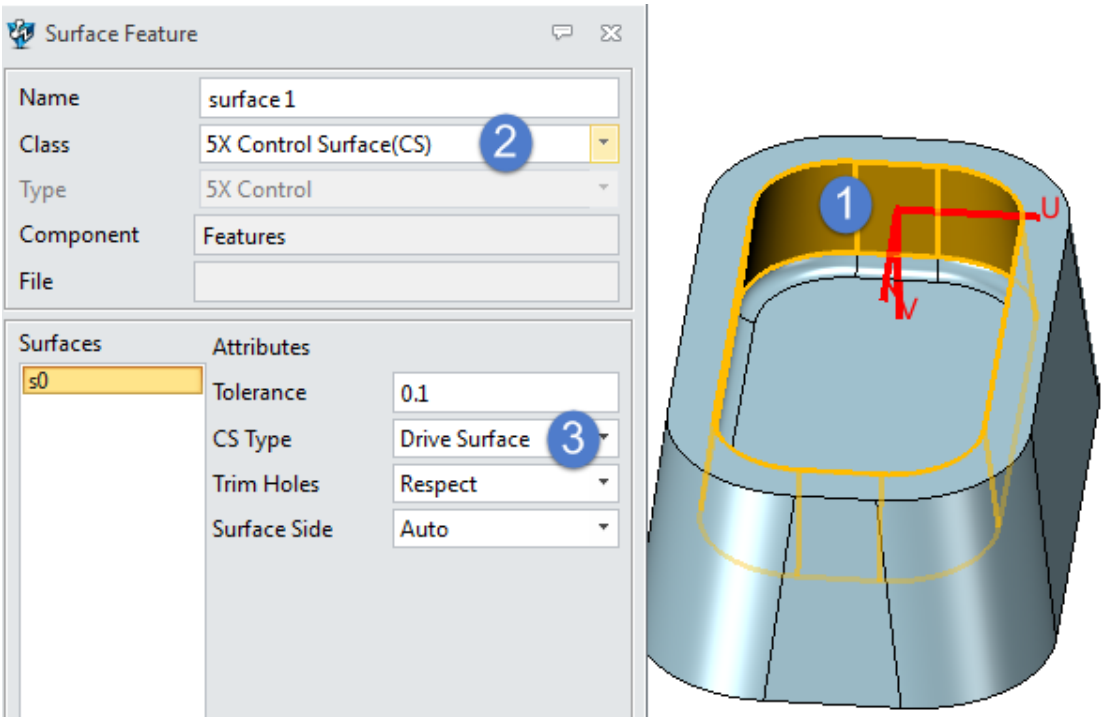


Figure69 Define drive surface

STEP 02 Define Part surface: the part surface actually is the target that will be cut as follows:

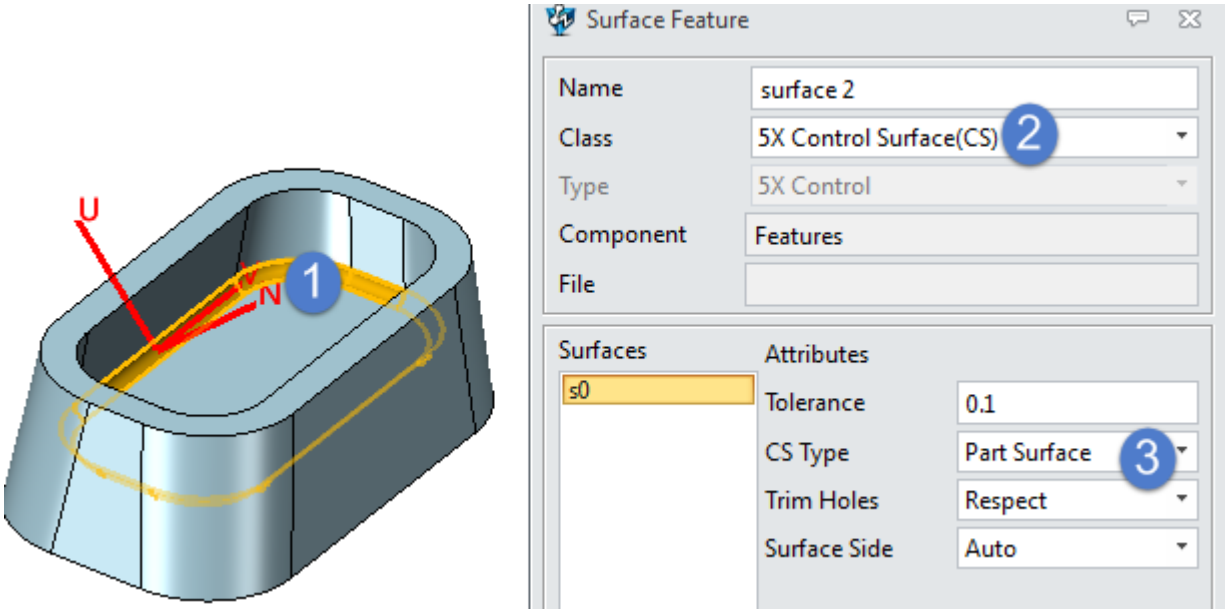


Figure70 Define Part surface

STEP 03 Choose a ball end tool with diameter of 6 mm, and calculate the operation by default parameter. Then we can get toolpath as follows：

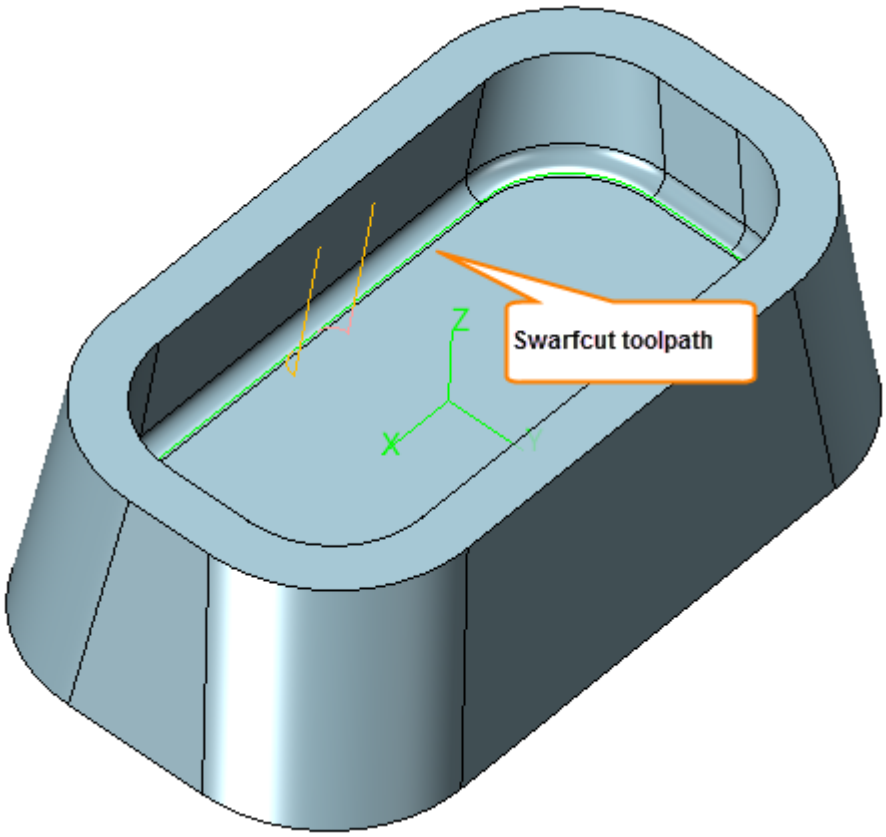


Figure71 Swarfcut toolpath

1.3.2.1 Swarf cut parameters

Next let's take a look at the main parameters of Swarfcut operation as follows:

1. Primary parameters

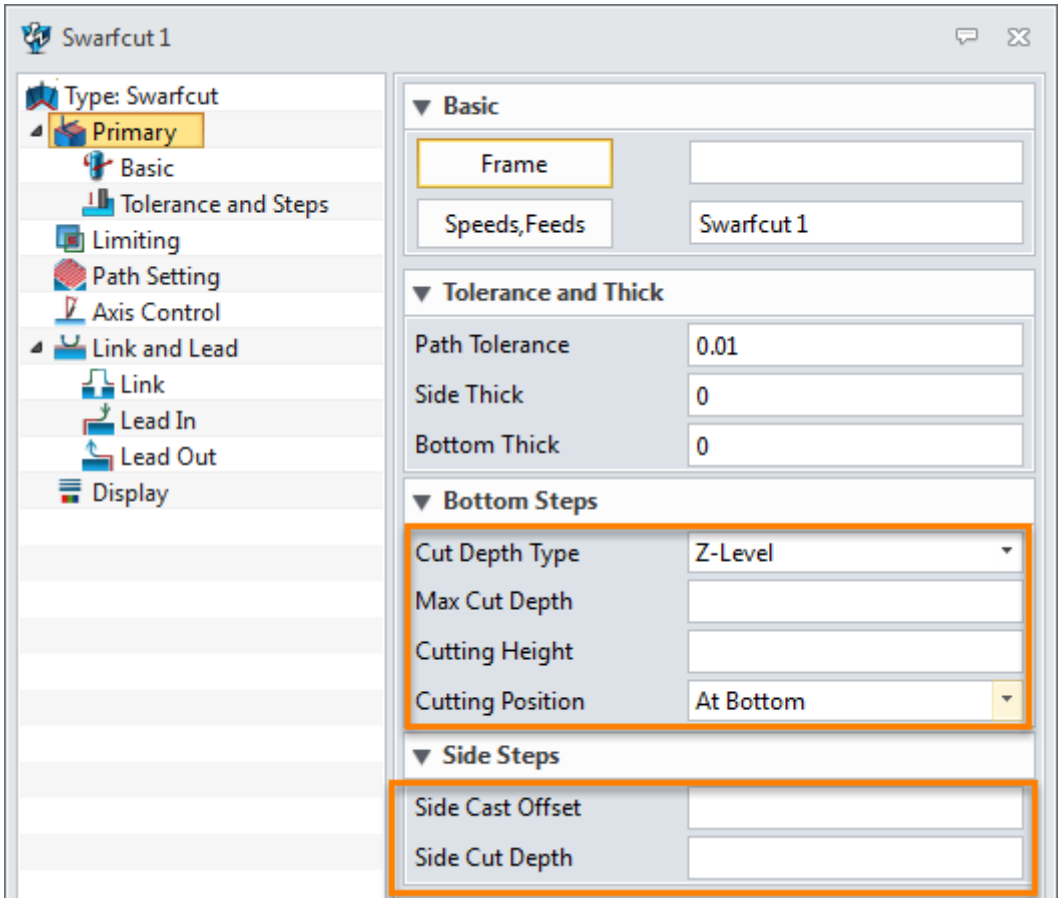


Figure72 Swarfcut primary parameters

For the Basic and Tolerance related parameter actually, every operation is the same meaning. So here let's skip it. Here are some new parameters we need to explain

- **Cut depth Type**: depth measurement direciton.
- **Z-Level**: means that depth is measured along working frame's Z axis
- **Along tool**: means that depth is measured along tool axis
- **Max Cut depth**: if we create multiple layers toolpath then this value is the maximum depth for each layer. If we leave it blank then it means only 1 layer toolpath will be created.

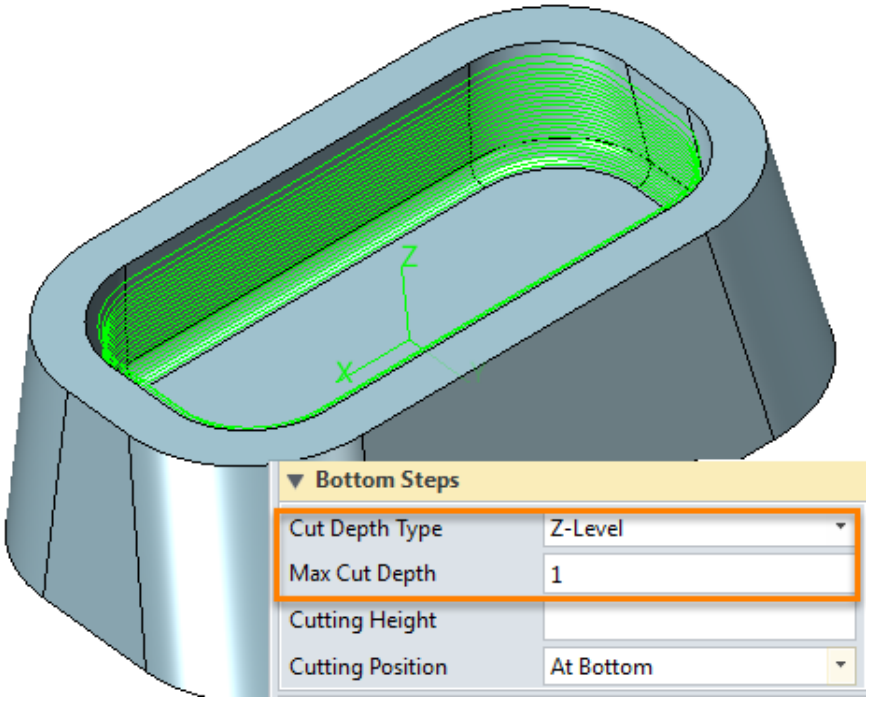


Figure73 Multiple layers swarf cut toolpath along Z level

- **Cutting height** : It is a limitation for depth and only work for **At Top** option as follows

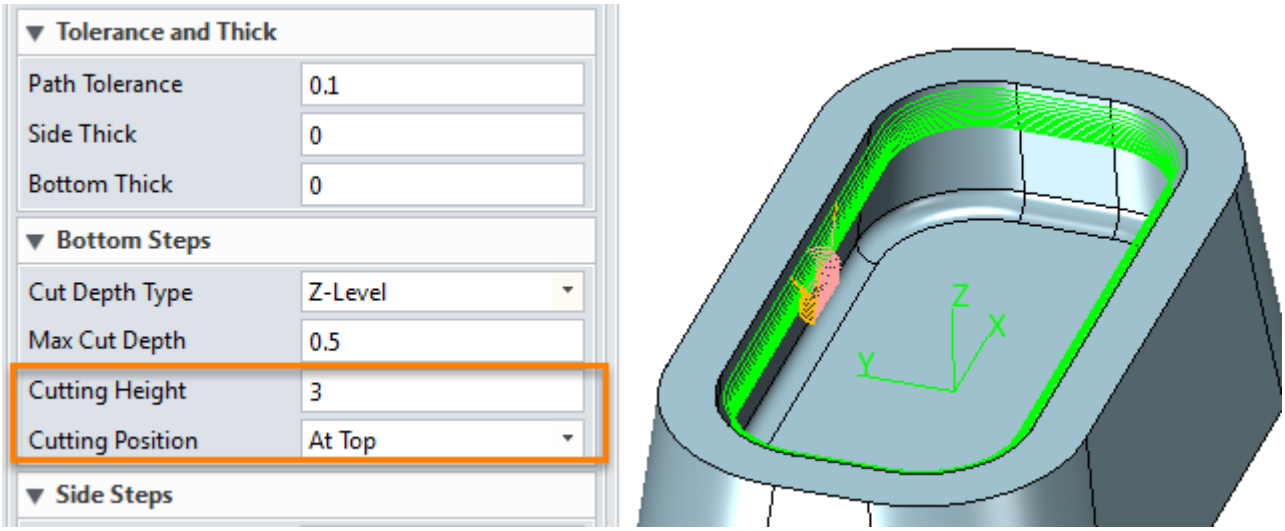


Figure74 Cutting height of swarf cut

- **Cutting position:** we can regard it as a measurement reference for depth, starting from top or from bottom. If we only create one toolpath then it is used to specify where to cut top or bottom.
- **Side cast offset**: it is used to set up the side thickness of cast part which needs to be removed.
- **Side cut depth**:something like the stepsize in XY.

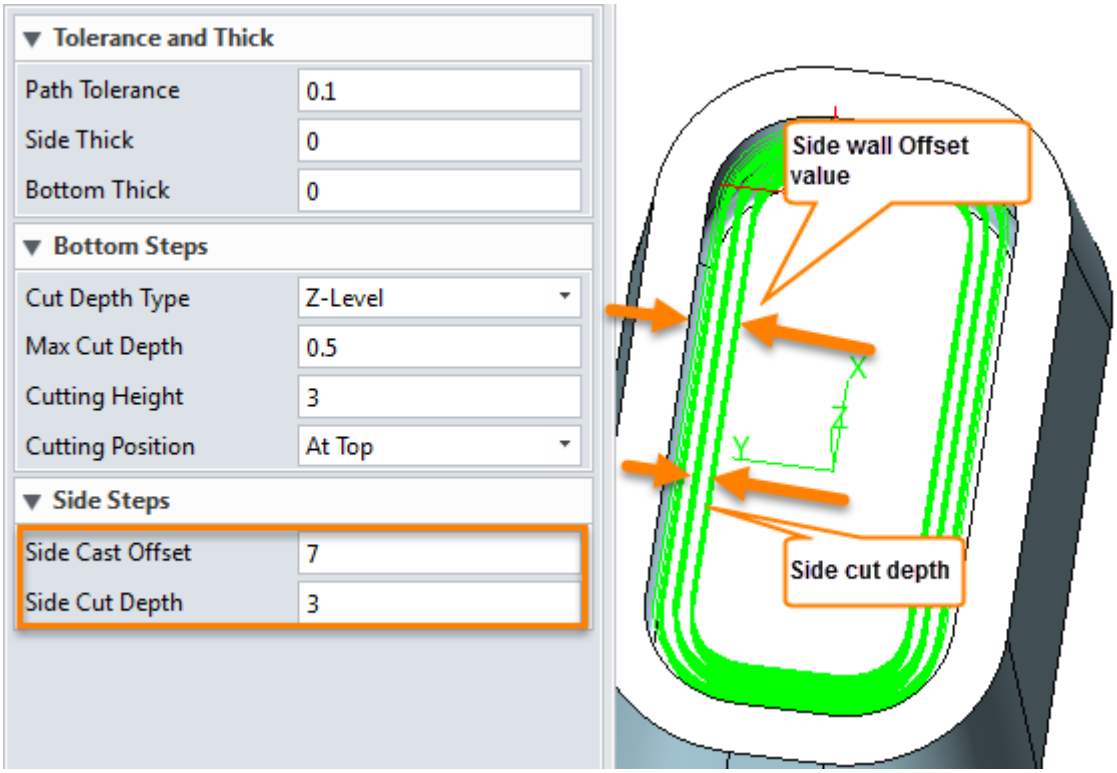


Figure75 Side steps

2. Path setting parameters

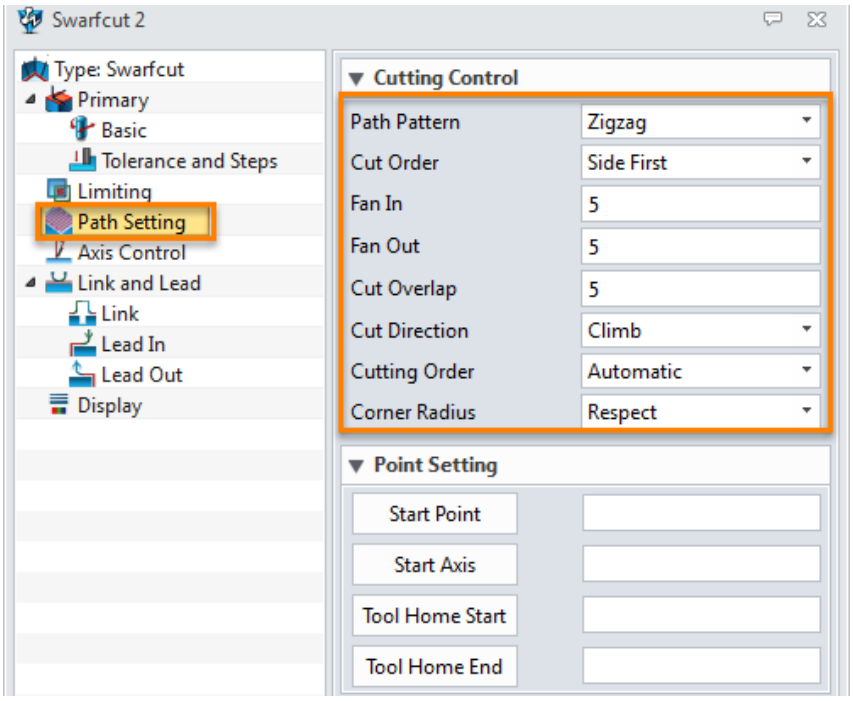


Figure76 Path setting parameters

- **Path pattern**: set up if use one way or Zigzag pattern

- **Cut order**: This determines the depth cut order. This can be applied to both Base Depths and Side Depths. (Normally used under the condition of side cast offset)
  - **Bottom First:** Cut down to base (part) surfaces first for each side cut.
  - **Side First:** Cut sides first on each level.
- **Fan In:** A distance from a corner seam (edge) at which the tool will begin to lessen the influence of the drive surfaces on the tool axis so that it can assume the optimal orientation on the corner.
- **Fan Out**: A distance the tool may traverse while transitioning from the optimal orientation in a corner to have the tool axis controlled by a drive surface.
- **Cut Overlap:** This is a re-cut distance to obtain smooth part surface when **cutting closed loops.** This distance is added at the end of the cut (retracting the beginning of the cut) at the "cut" feed rate.
- **Cut Direction:** This determines the direction of cut， which are Clime and Conventional.
- **Corner Radius:** Fillet the cut with this radius.

3. Axis control pararmeters:

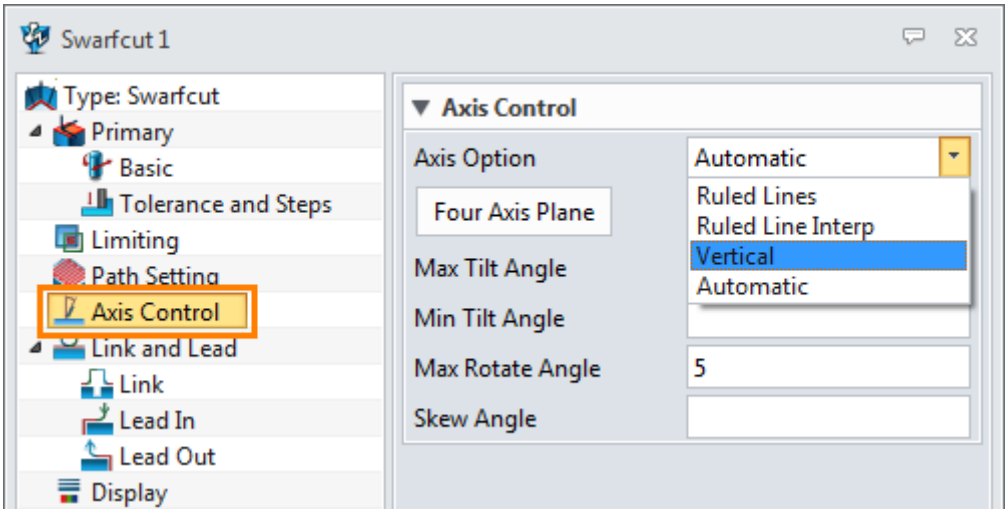


Figure77 Axis control parameters

Axis Option:

- **Ruled lines**: The tool axis always follows the ruling direction of a drive surface for ruled surfaces, it is used for the ruled drive surface.

- **Vertical**: The tool axis is both tangent to the drive surface and vertically tilted.

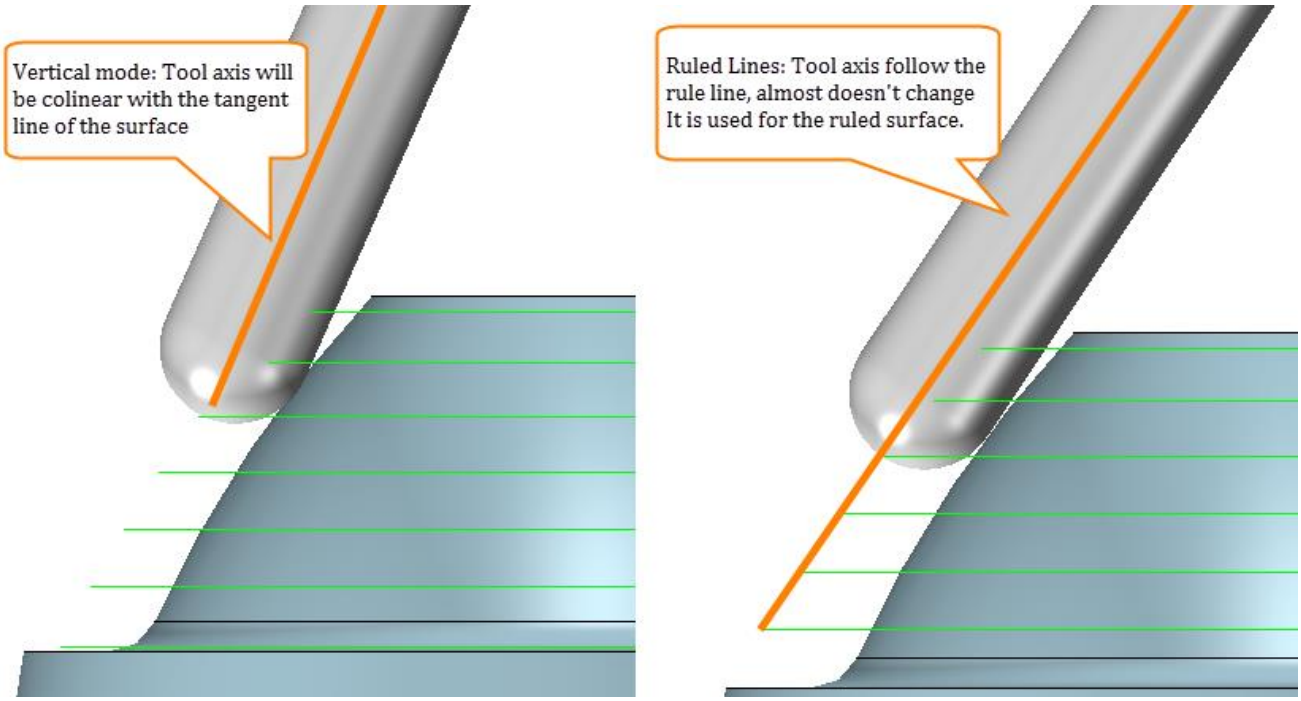


Figure78 Tool axis options

- **Automatic:** The tool axis follows "ruled lines" for curved ruled drive surfaces and will be vertical for other types of drive surfaces including flat ones.

1.3.2.2 Real case study

Next we will use 2 real cases as examples to show how to apply the swarf cut operation to real work.

1. Case 1: "5X_Impeller.Z3"

Let's open the real case " **5X_impeller.Z3** " file as follow:

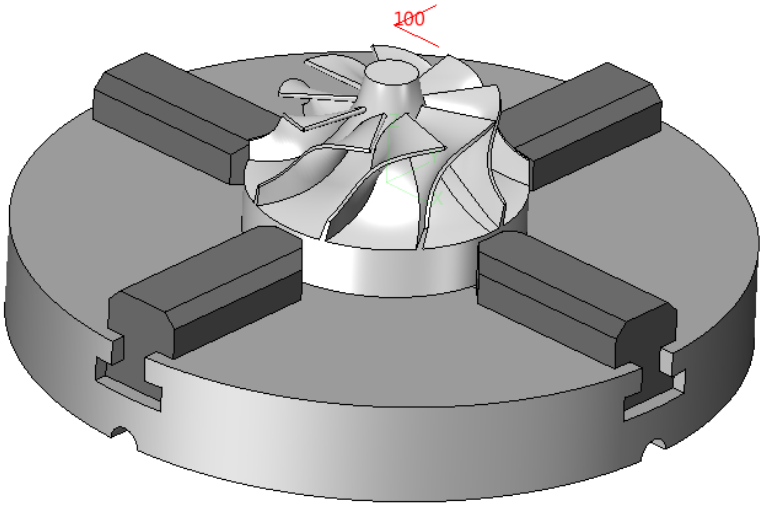


Figure79 5X_Impeller file

Then we will create swarf cut toolpath to cut the blade as follow:

STEP 01 Define the 5X control surface as follow:

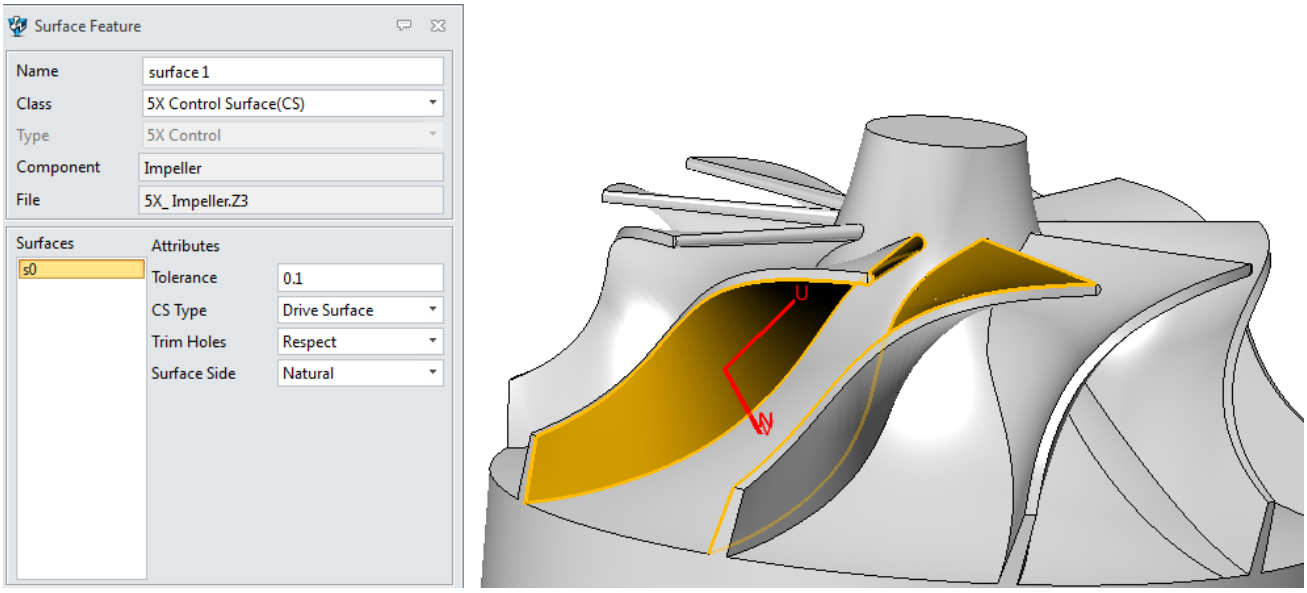


Figure80 Define 5X Drive Surface

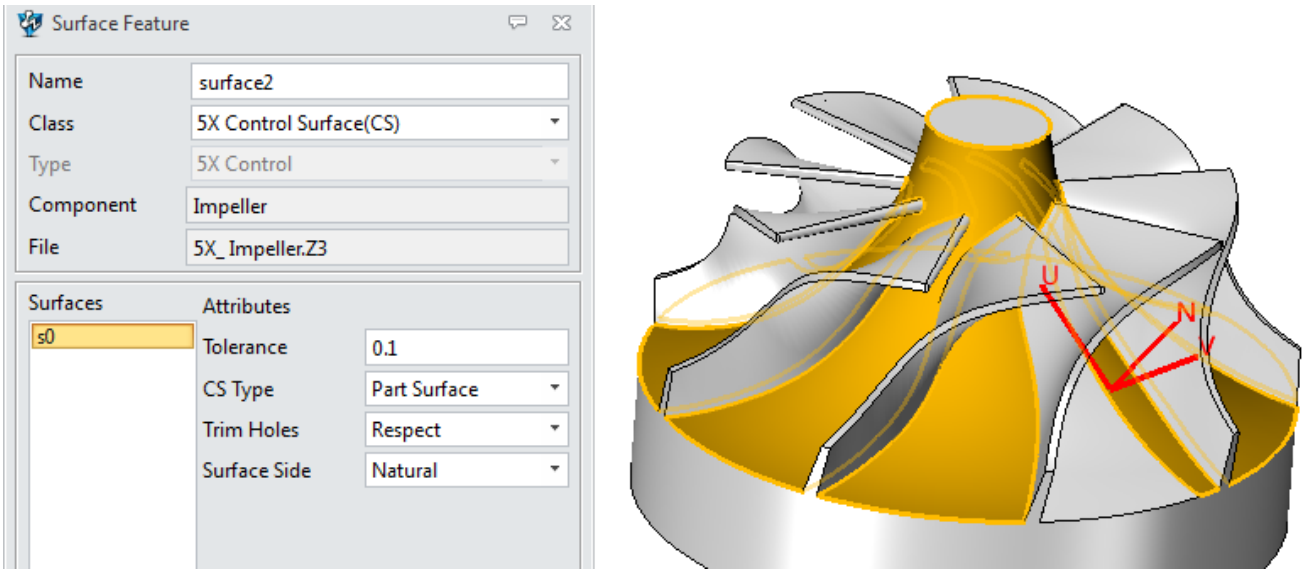


Figure81 Define 5X Part Surface

STEP 02 Set up tool size and operation parameters

Here we assume that the corner radius is 2 mm so that we can choose the ball end mill with the radius of 2 mm. Then set up operation parameters as follows:

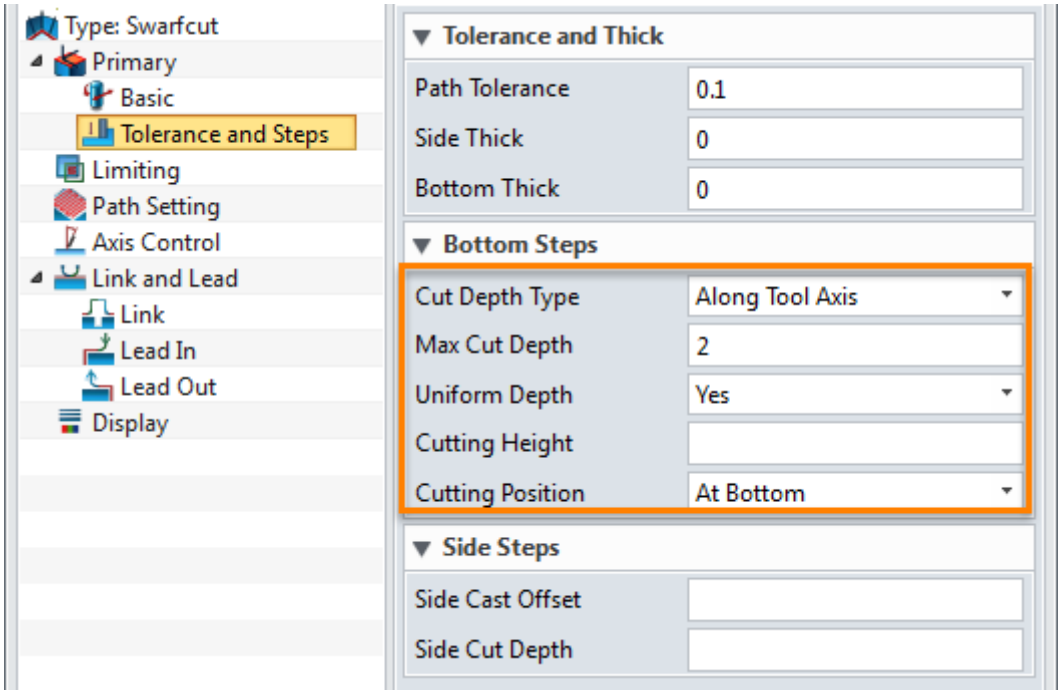


Figure82 Define Primary parameters

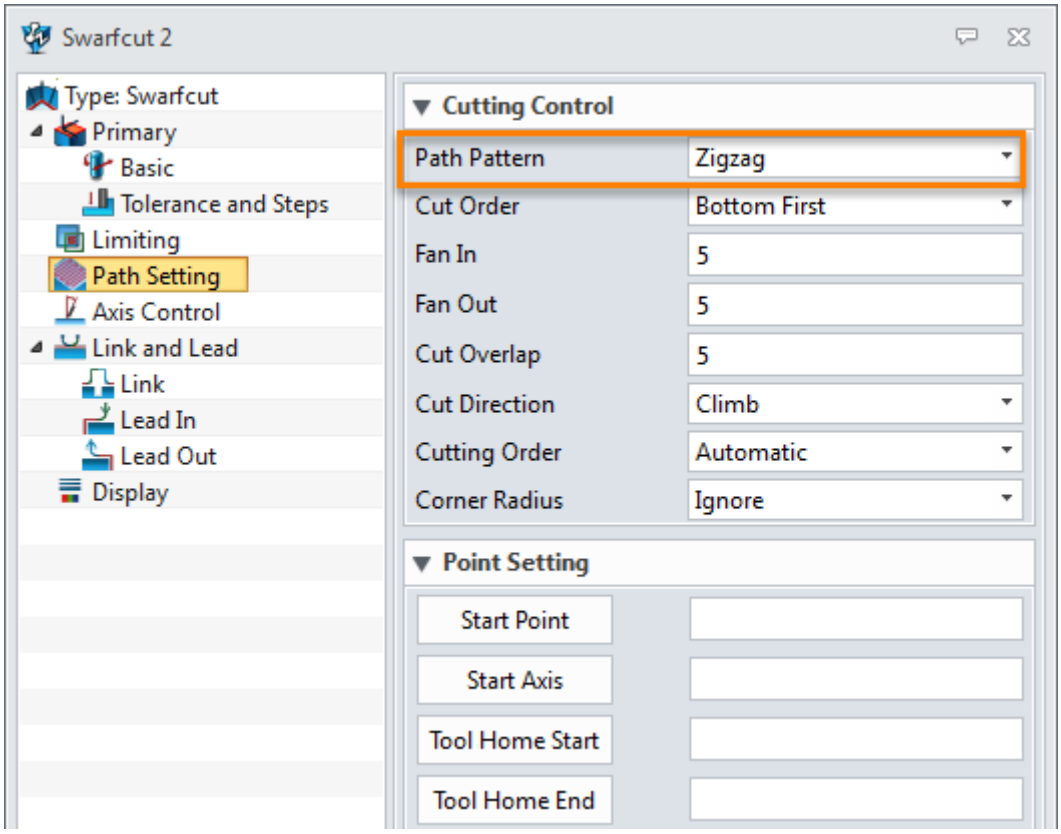


Figure83 Define Path setting parameters

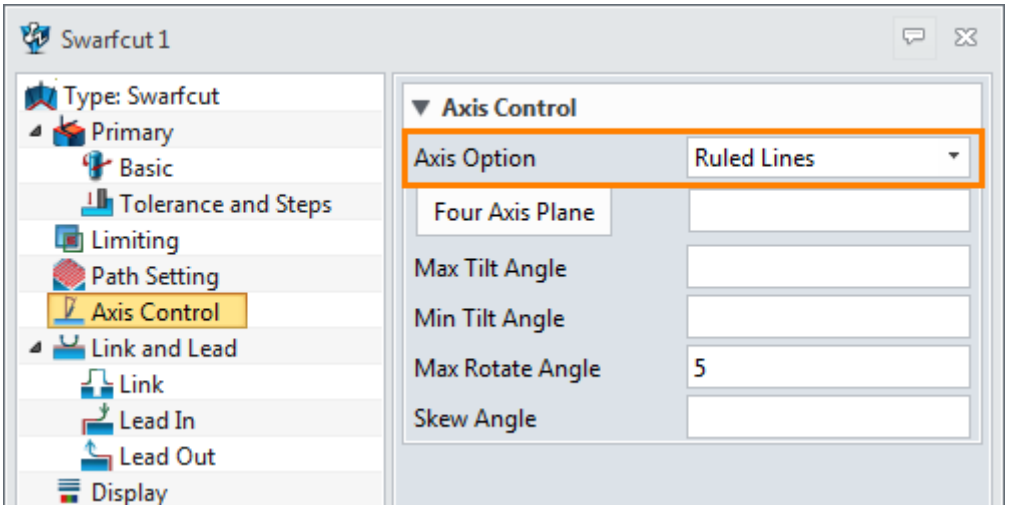


Figure84 Define Axis Control

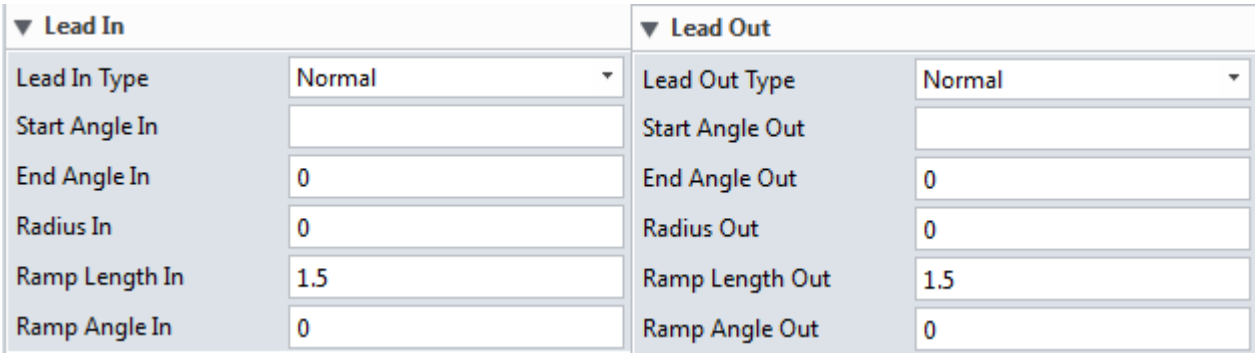


Figure85 Lead in and lead out

For the rest parameters keep their default setting.

STEP 03 Calculate toolpath as follows:

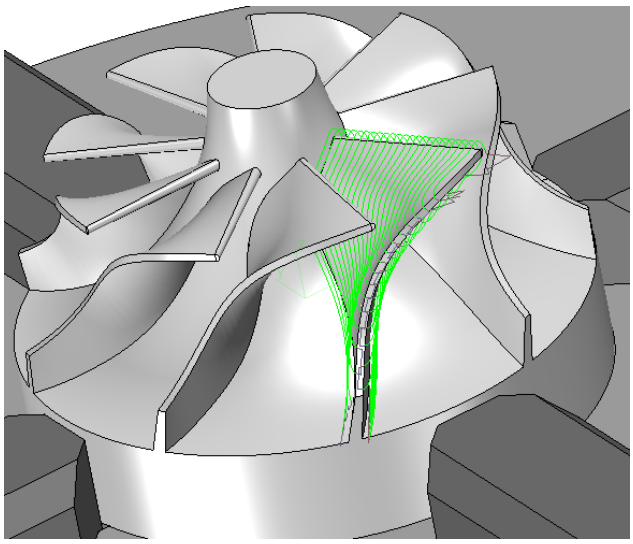


Figure86 Swarf cut toolpath

STEP 04 Pattern the toolpath by transforming function and setting up the pattern parameters as follows：

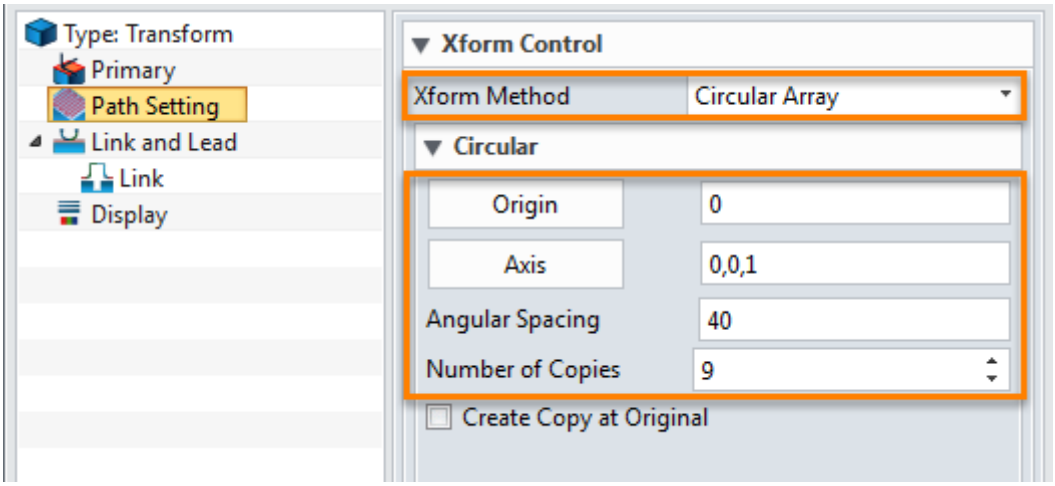


Figure87 Pattern the swarf cut toolpath around Z axis

The finished result is as follows:

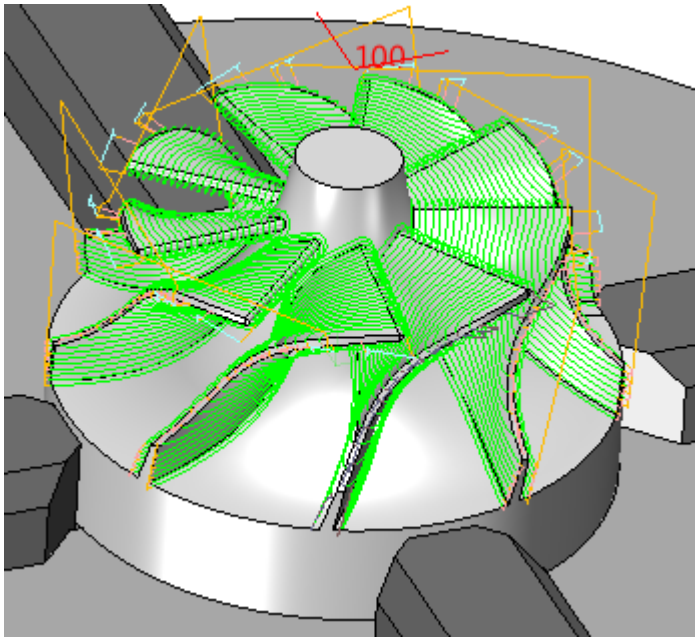


Figure88 Finished swarf cut for impeller blade

Verify the toolpath movement by verifying function as follows:

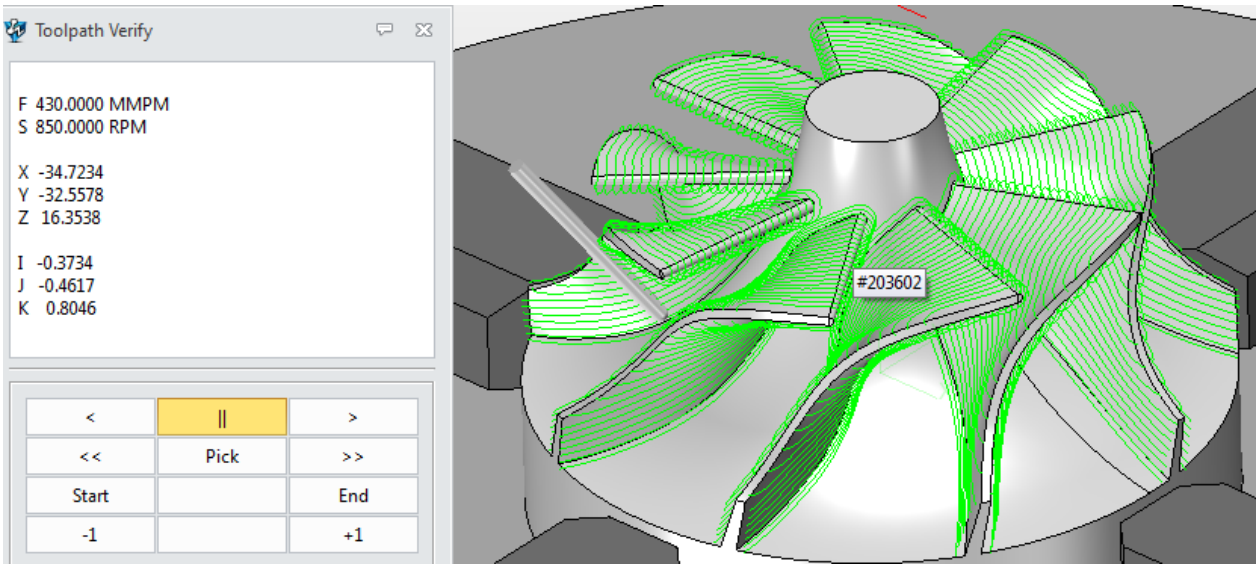


Figure89 Verify swarf cut toolpath

After saving the file. From the result we can find that actually the blade part has been finished. We will keep on creating toolpath on this impeller file by other operations and finally when we finished the tutorial the impeller model also will be finished.

2. Case 2: "SwarfCut_A.Z3"

Please open the case file "**Swarfcut_A.Z3**" as follows:

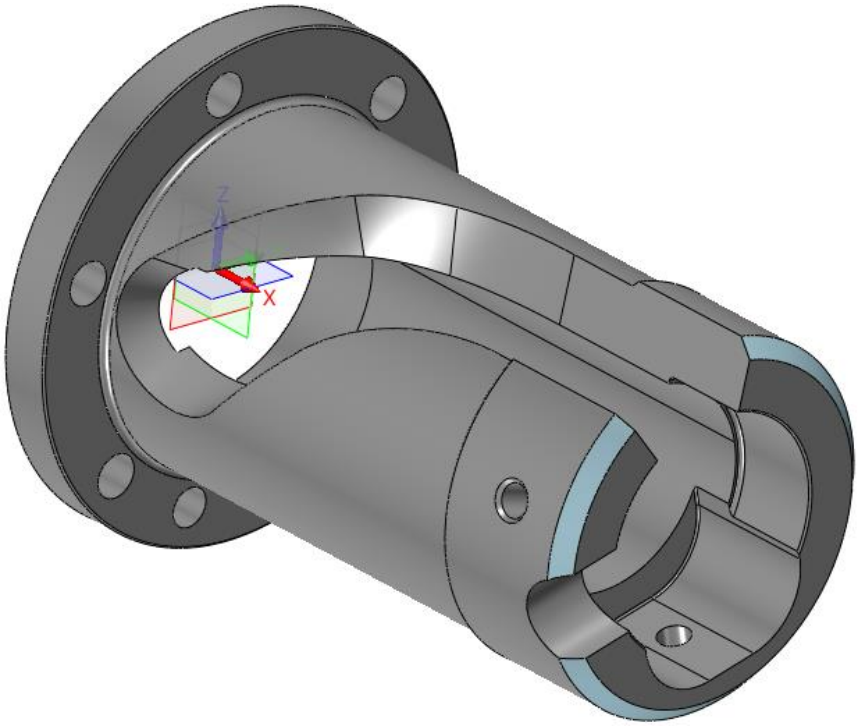


Figure90 Swarfcut_A file

Task: finish the highlighted surface as follows:

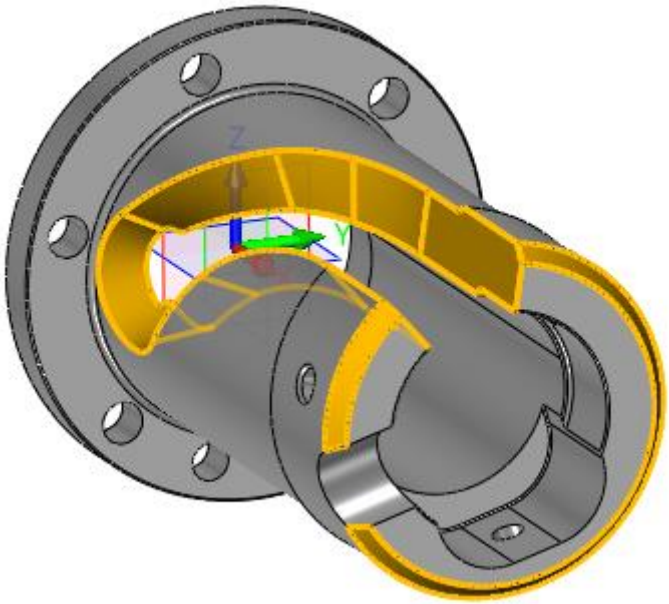


Figure91 cutting required faces

STEP 01 Heal the highlighted surface： Since this file is an imported file and the swarfcut operation has strictly requirements for geometry quality, so it is better to heal the file first before programming. Details are as follows

I. Delete loops

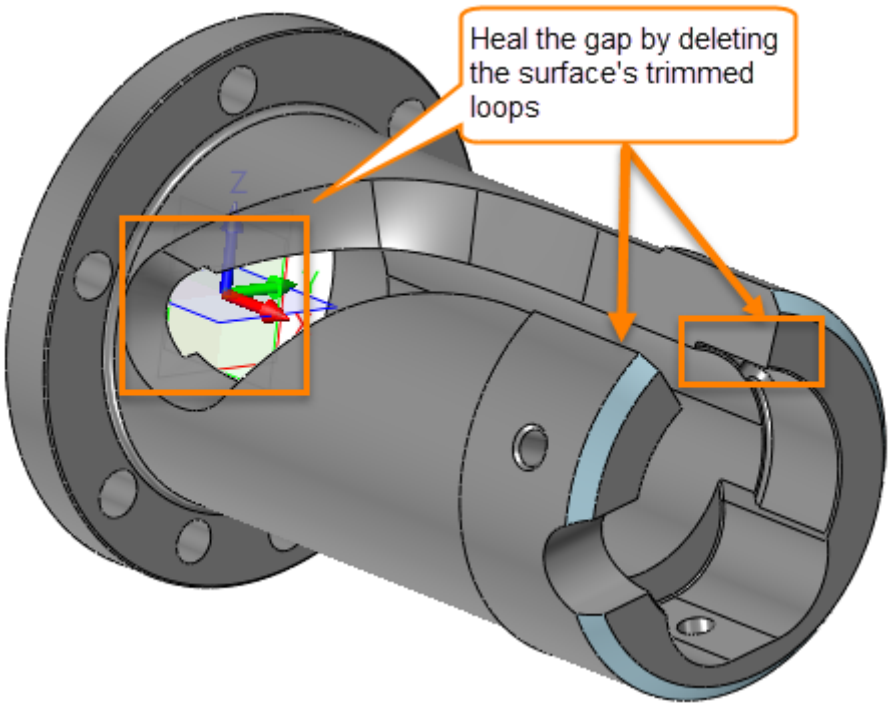


Figure92 Delete surface trimed loops

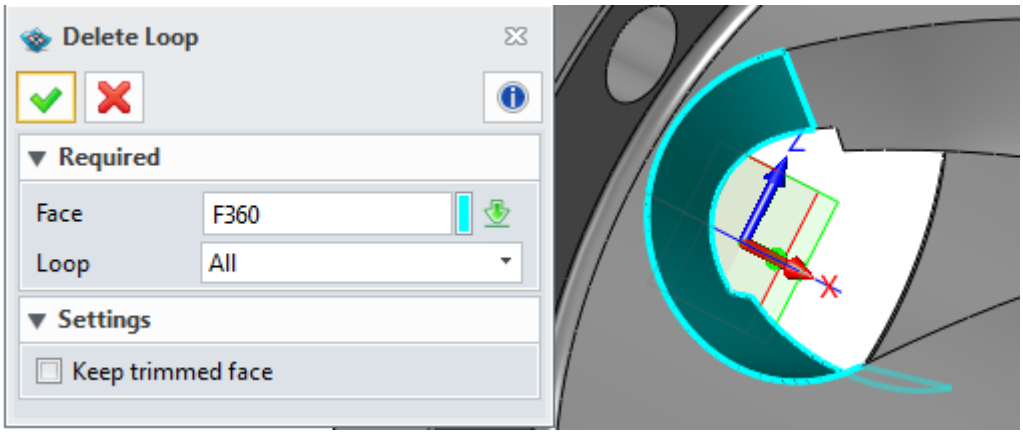


Figure93 Delete loops operation 1

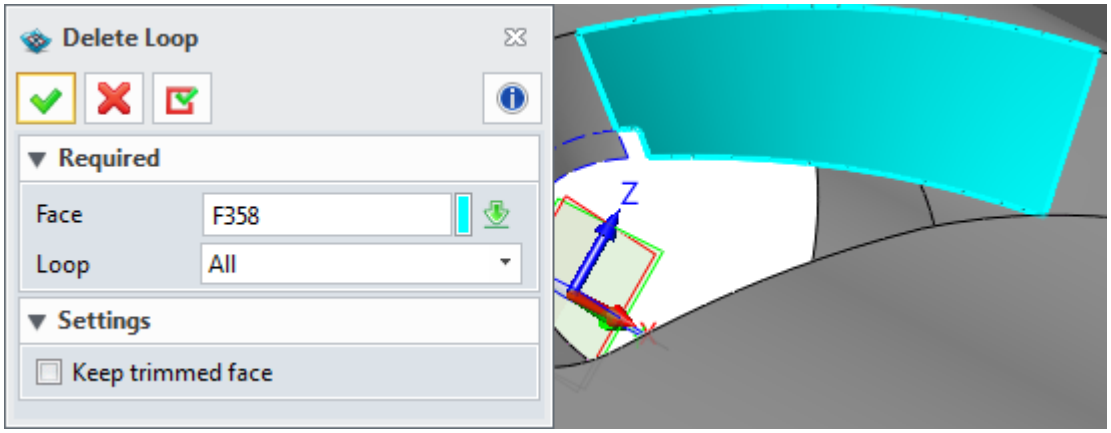


Figure94 Delete loops operation2

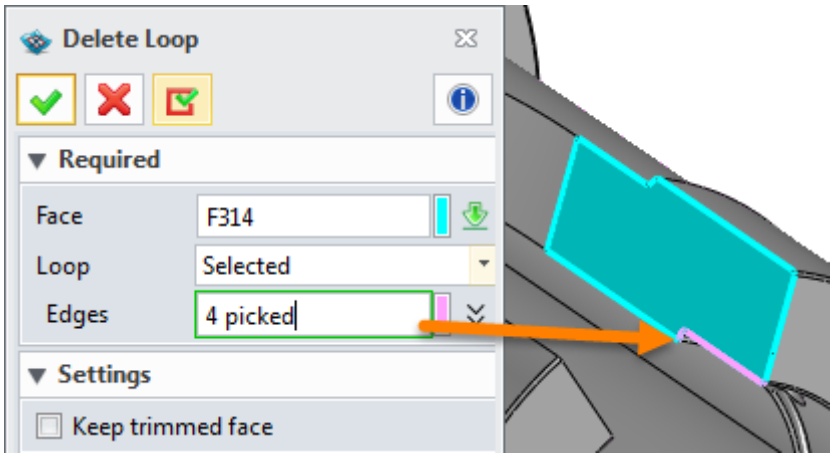


Figure95 Delete loops operation 3

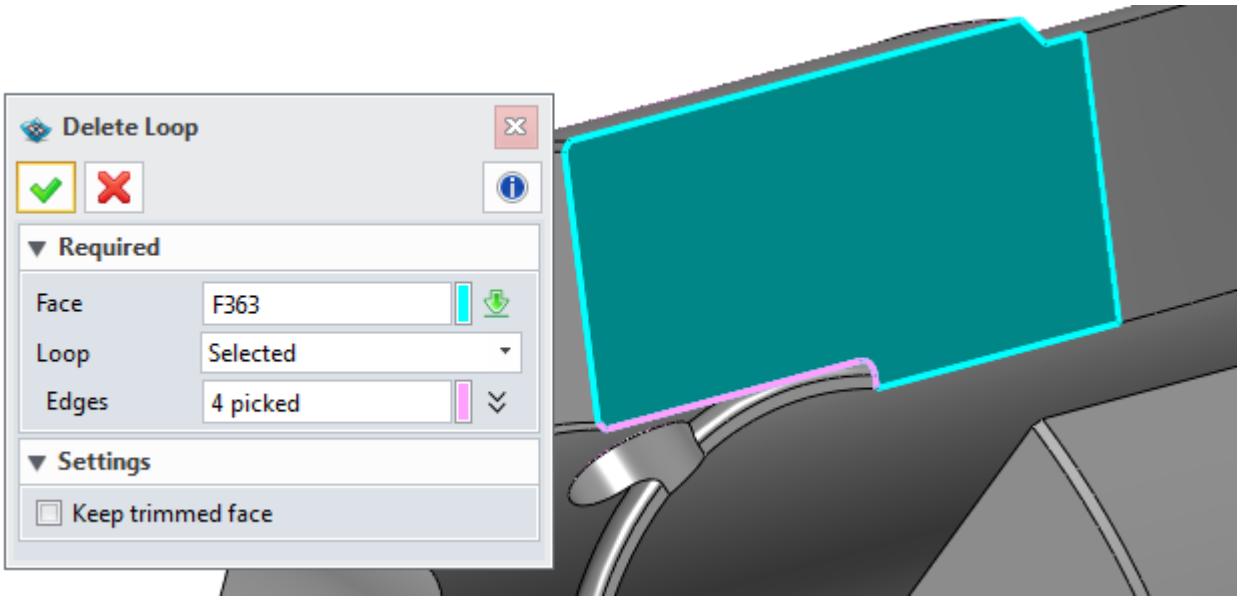


Figure96 Delete loops operation 4

II. After deleting the loops, we can get the following result:

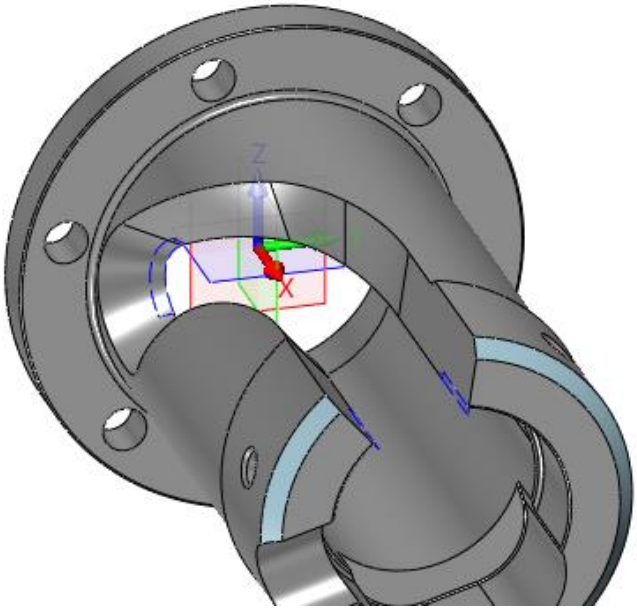


Figure97 Result after deleted loops

STEP 02 Modify surface: Trim the surface to curve as follows:

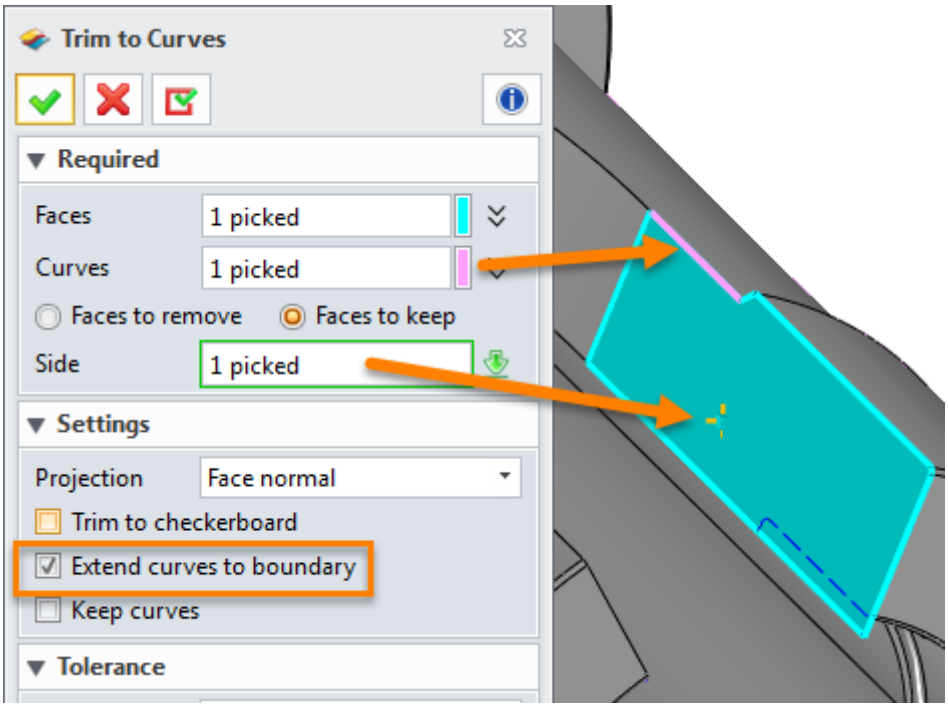


Figure98 Trim surface_right side

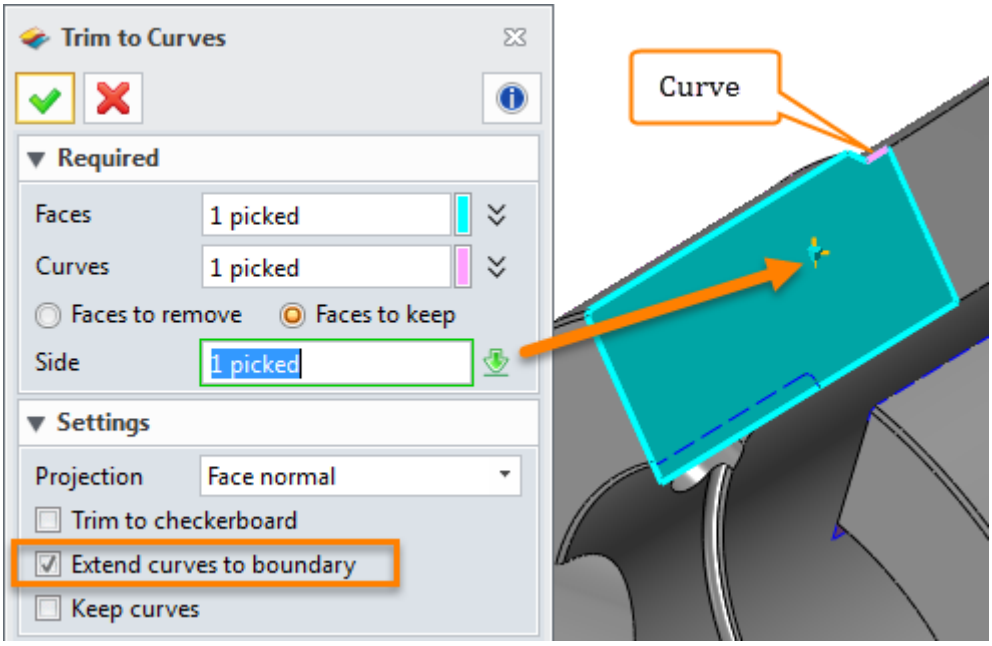


Figure99 Trim surface_left side

Finally we can get the result as follows:

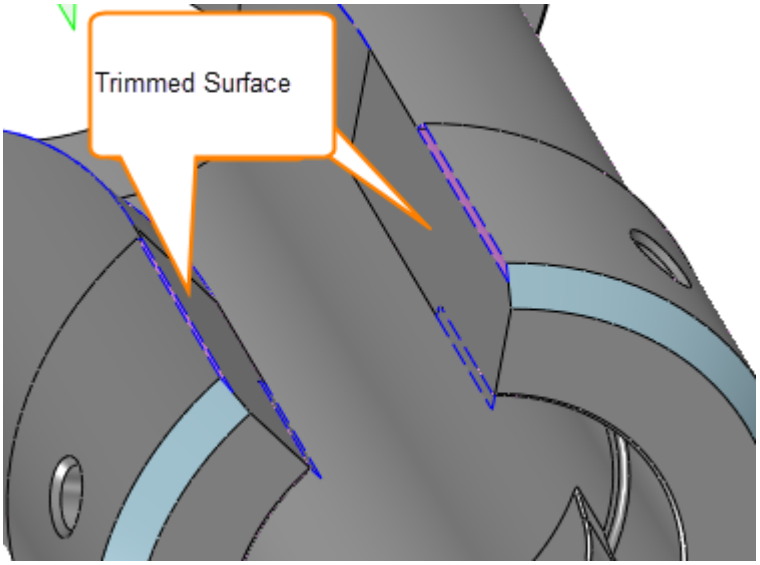


Figure100 Trimmed surface

STEP 03 Ensure that the trimmed surface can be tangent to the adjacent surface by **Match Tangency** Tool as follows:

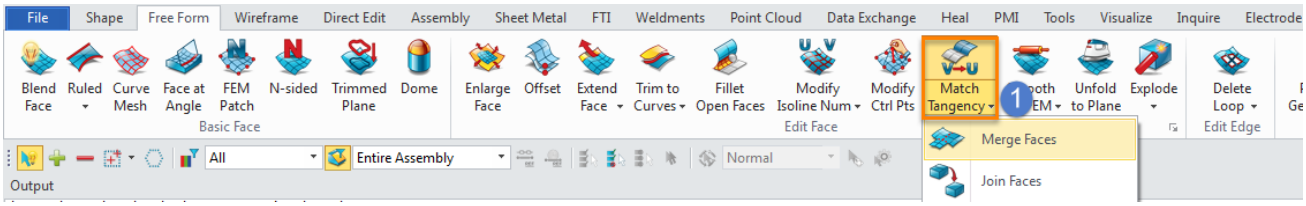


Figure101 Match Tangency command

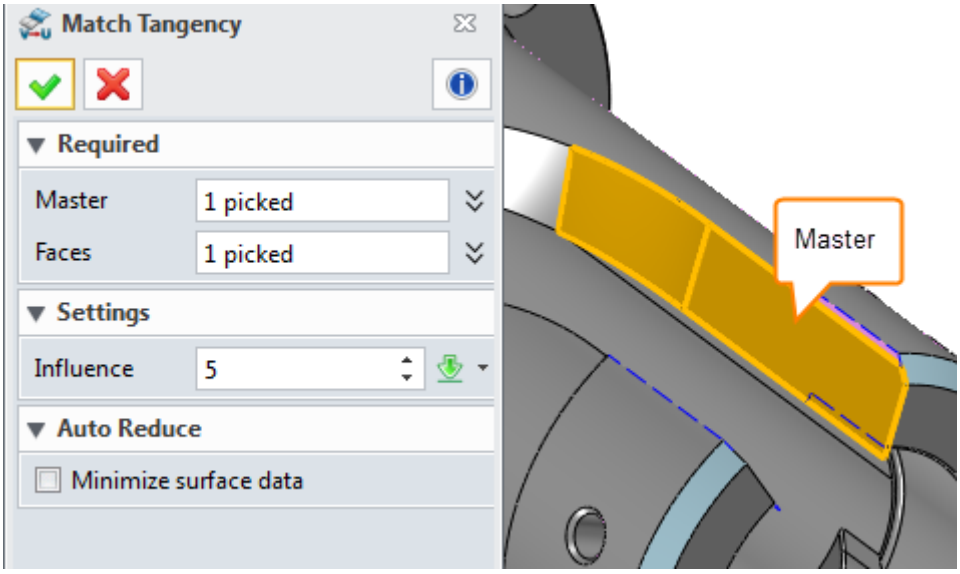


Figure102 Match trimed surface tangency to adjacent surface

Please use the same way to match the trimmed surface tangency on the both right and left side

STEP 04 After healing the surface we can go back to CAM and create swarfcut toolpath for the target surface as follows:

I. Define the drive surface:

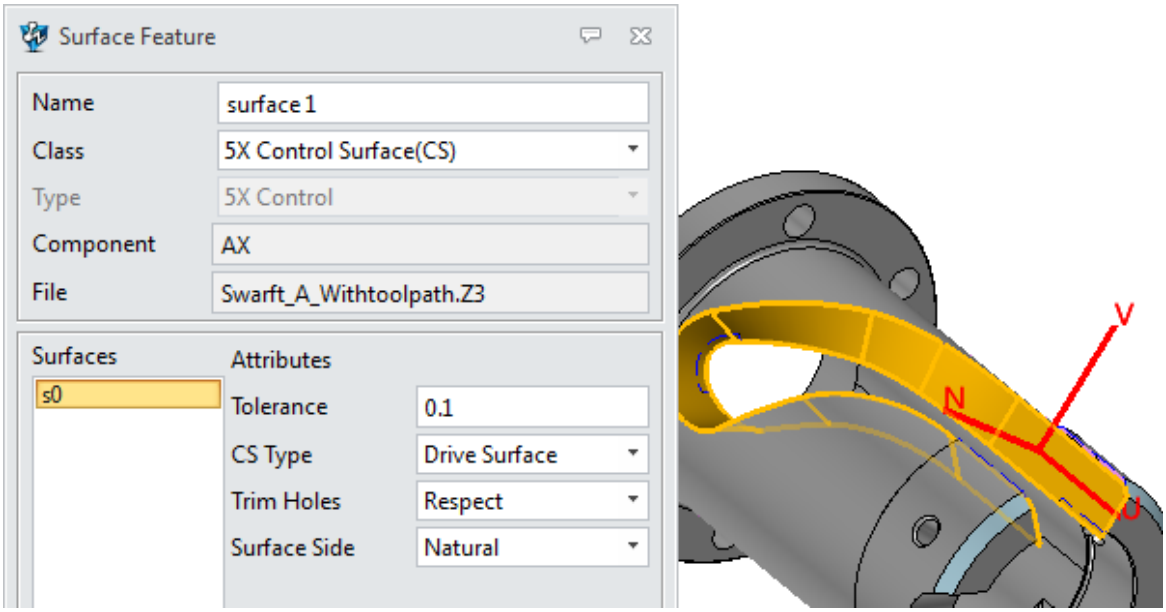


Figure103 Define drive surface

Here we will cut the drive surface to define a drive surface as feature input.

II. Set up tool size : Use 10 mm flat end mill.

III. Set up parameter as follows:

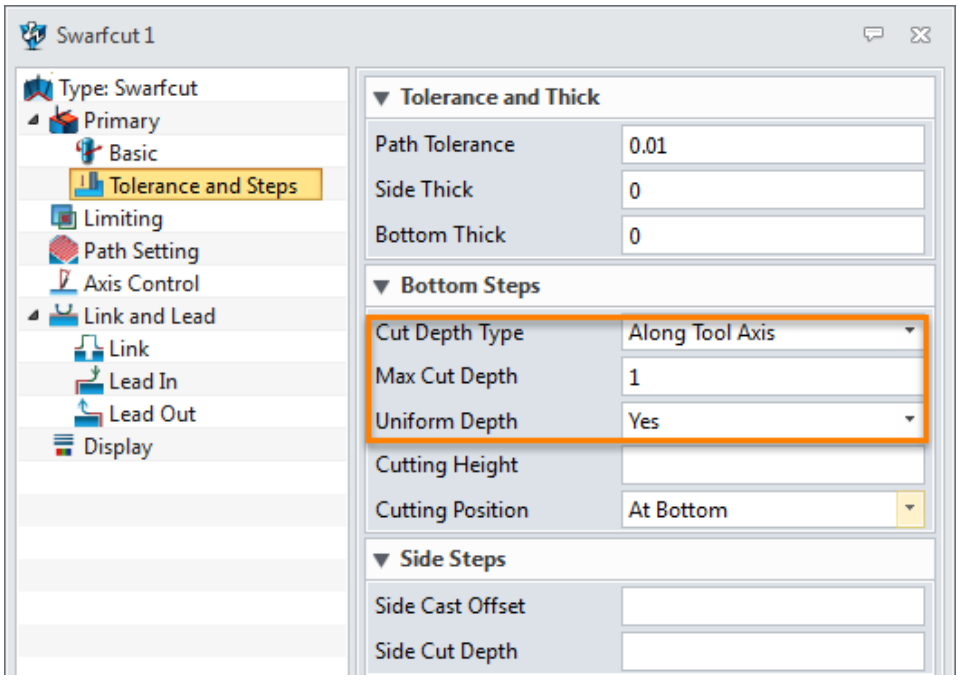


Figure104 Primary parameters

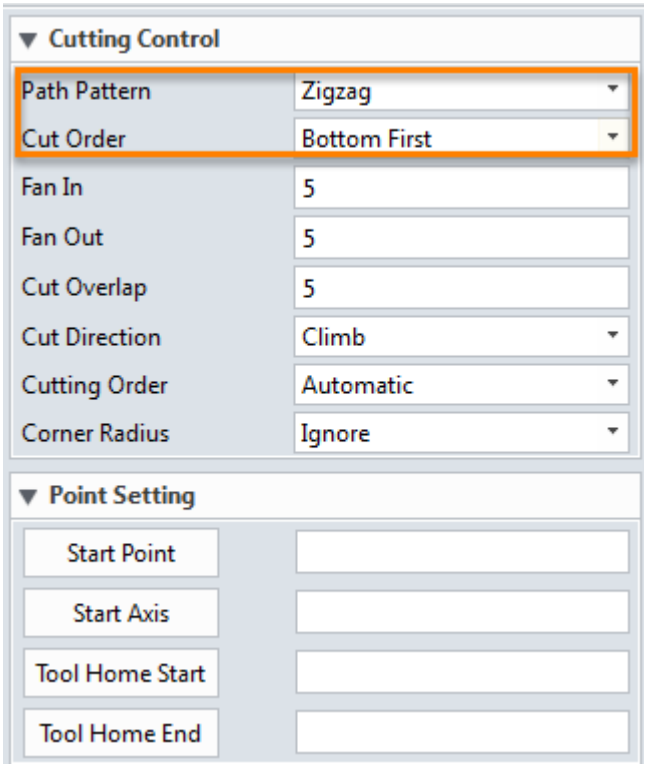


Figure105 Path setting parameters

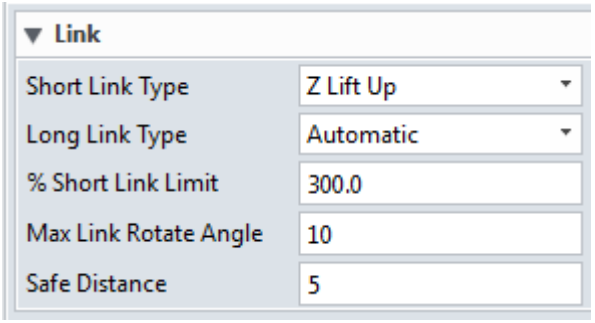


Figure106 link

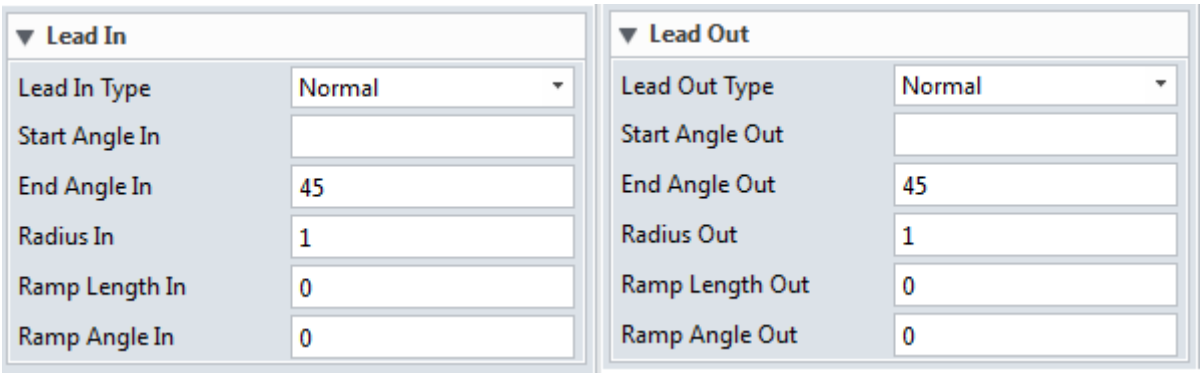


Figure107 Lead in and out

IV. After calculating the toolpath we can get the following result:

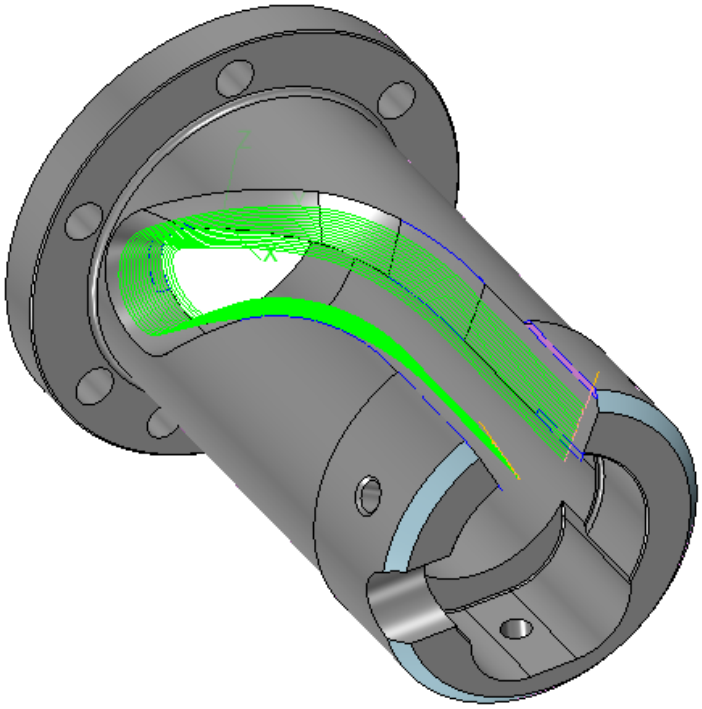


Figure108 Finished Swarfcut toolpath for pocket

Now we can check the toolpath by verifying as follows：

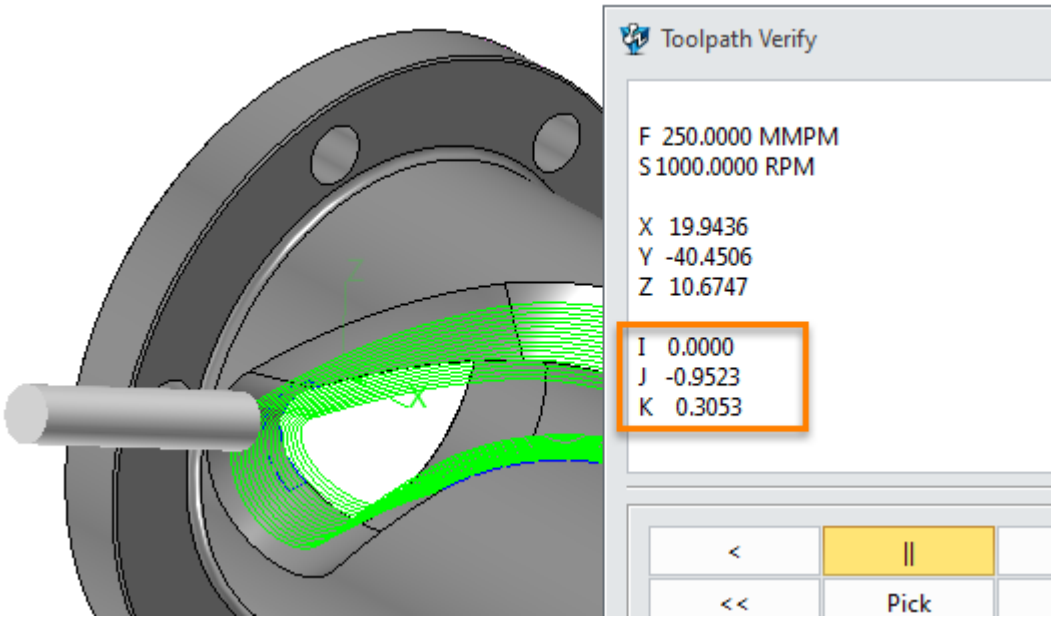


Figure109 Verify toolpath

we can find that actually this is 4X milling simultaneous movement toolpath. So let's look at the Four Axis Plane option as follows:

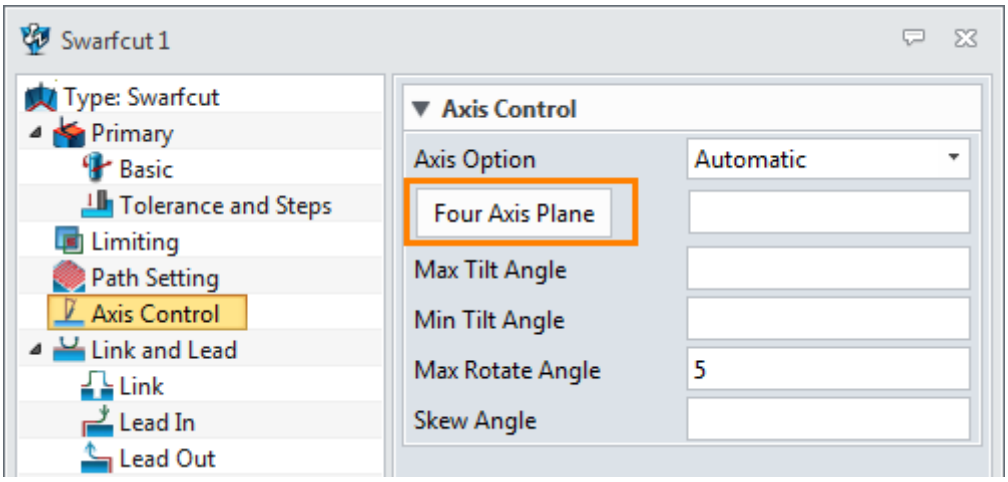


Figure110 Define 4X plane for swarfcut operation

4X Plane option actually is used to force the toolpath to a 4X Tool path. By default the system will automatically create the toolpath according to the surface's situation. Here the 4X tool path is enough to finish the pocket so even without setting up the Four Axis Plane system can automatically create the 4X toolpath for it. But the end chamfer surface will be different. In order to make a better toolpath here we can modify the part again to the following result:

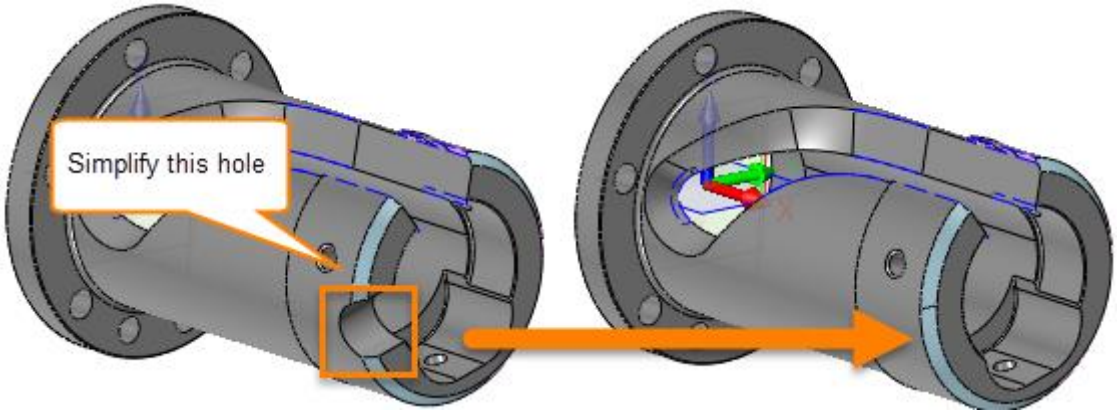


Figure111 Simplify hole in end

Next we will go to create toolpath on the chamfer face by the four axis plane option or without it seperately.

STEP 01 Define the chamfer face as drive surface:

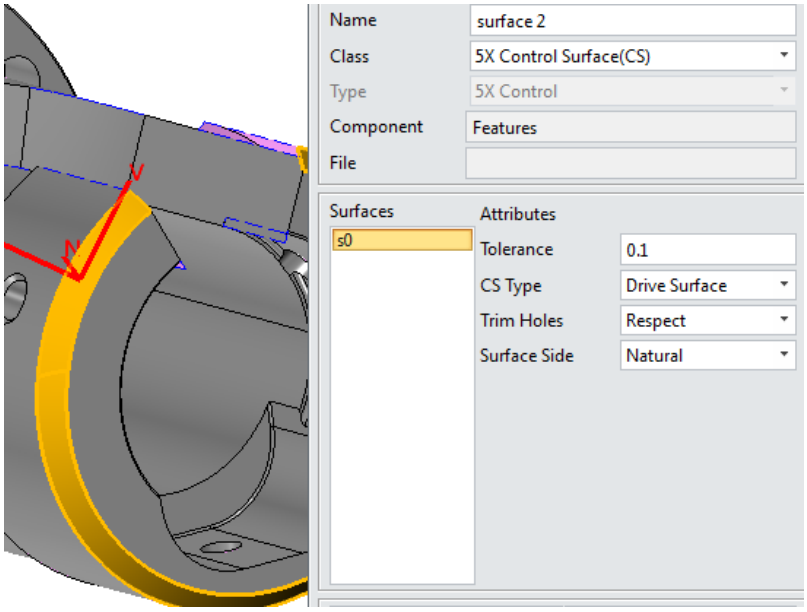


Figure112 Define Define chamfer face as drive surface

STEP 02 Set up parameters and choose the same tool (D10) as follows:

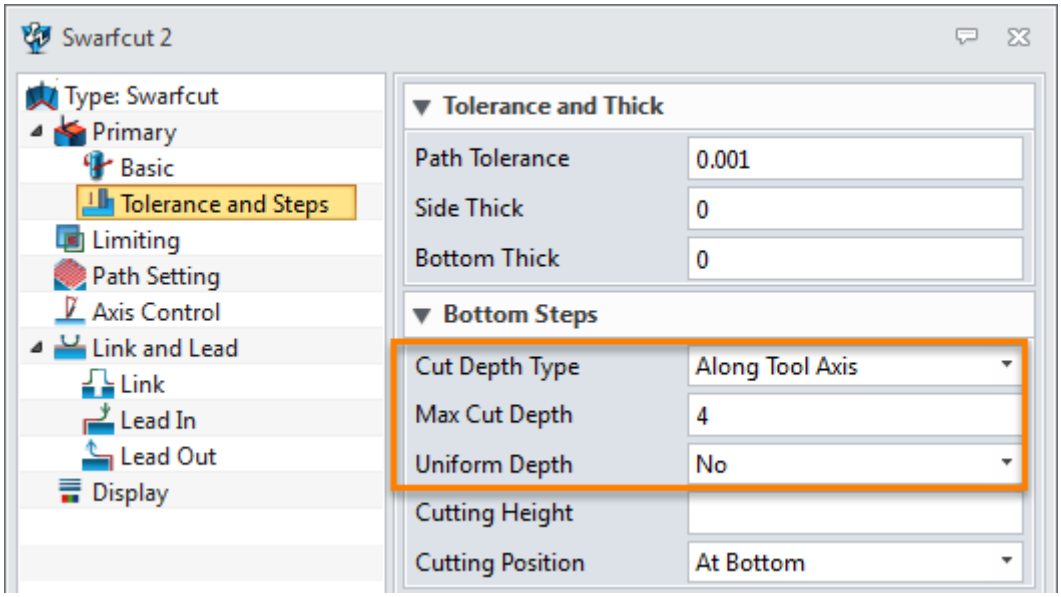


Figure113 Primary cutting parameter for cutting chamfer face

STEP 03 Create toolpath without 4X plane option:

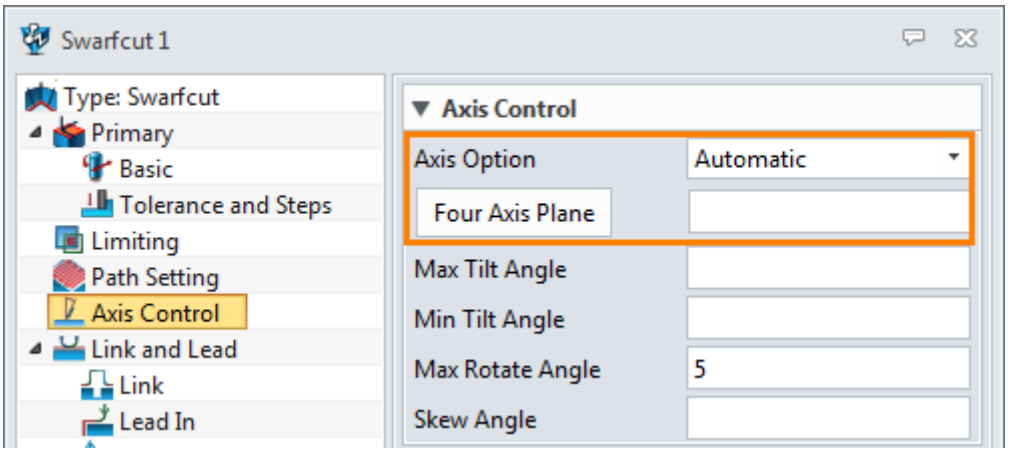


Figure114 Axis control setting

The toolpath is as follows:

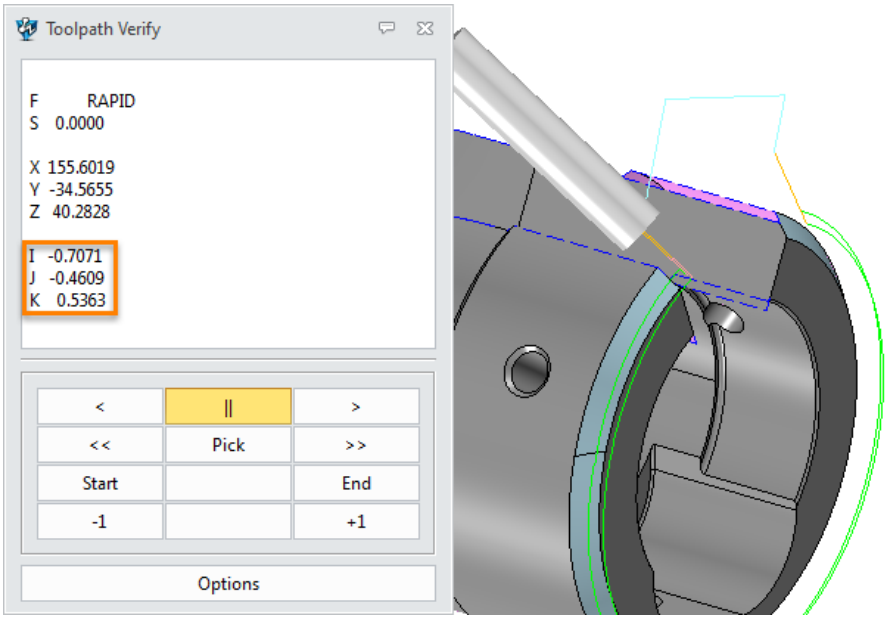


Figure115 Swarf cut toolpath without 4X plane option

From the result you can find that now actually it is a 5X toolpath not 4X simultaneous toolpath. So it will conflict with your machine. How to solve it? By 4X plane option as follows:

STEP 04 Set up the 4X plane according to real condition (here let's choose A)

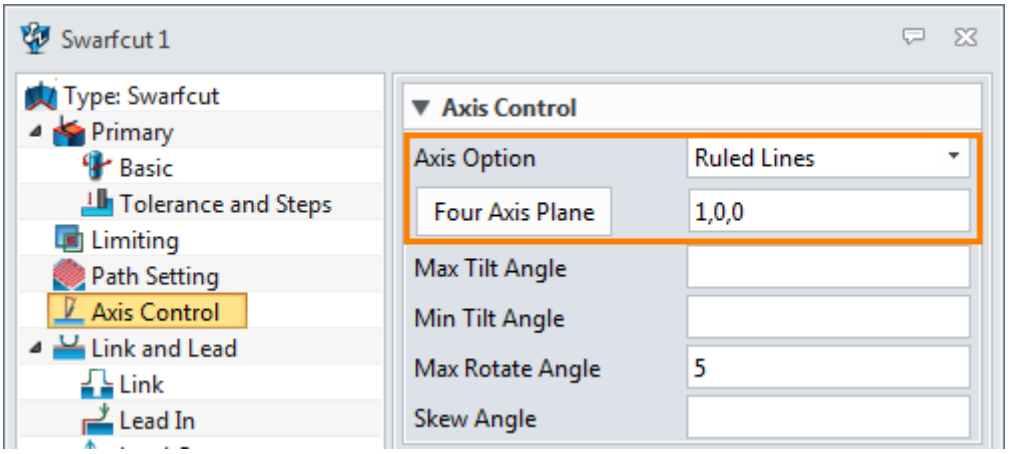


Figure116 Set up 4X plane

STEP 05 After calculation we can get another toolpath as follows:

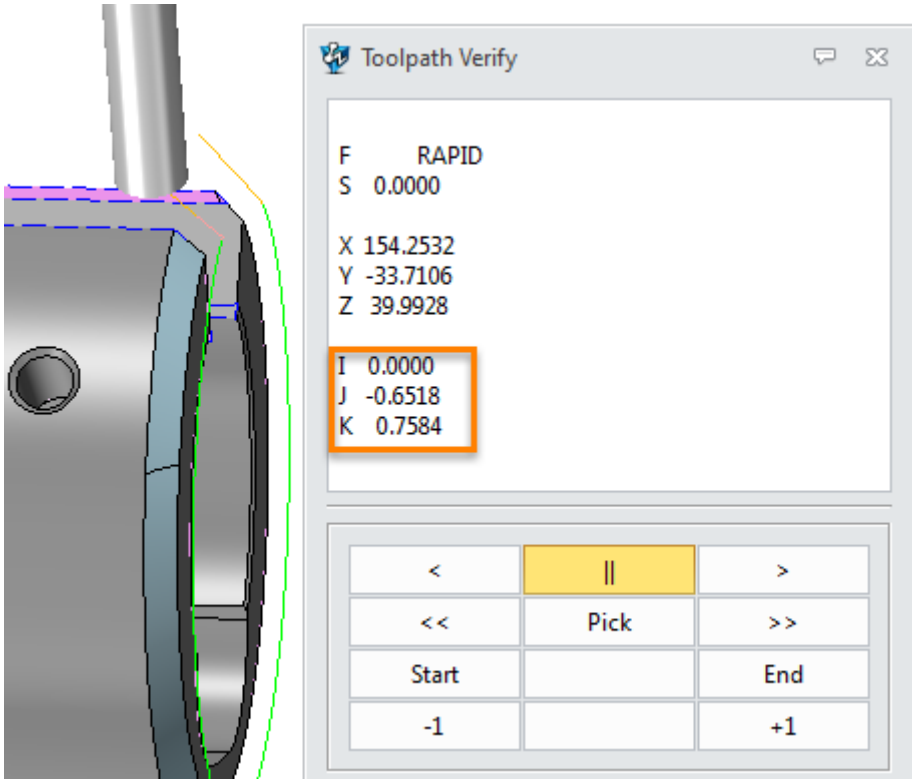


Figure117 4X swarf cut with Flat end mill tool

If we use ball end mill tool and create more toolpath then we can get a better 4X toolpath for finishing this chamfer face as follows:

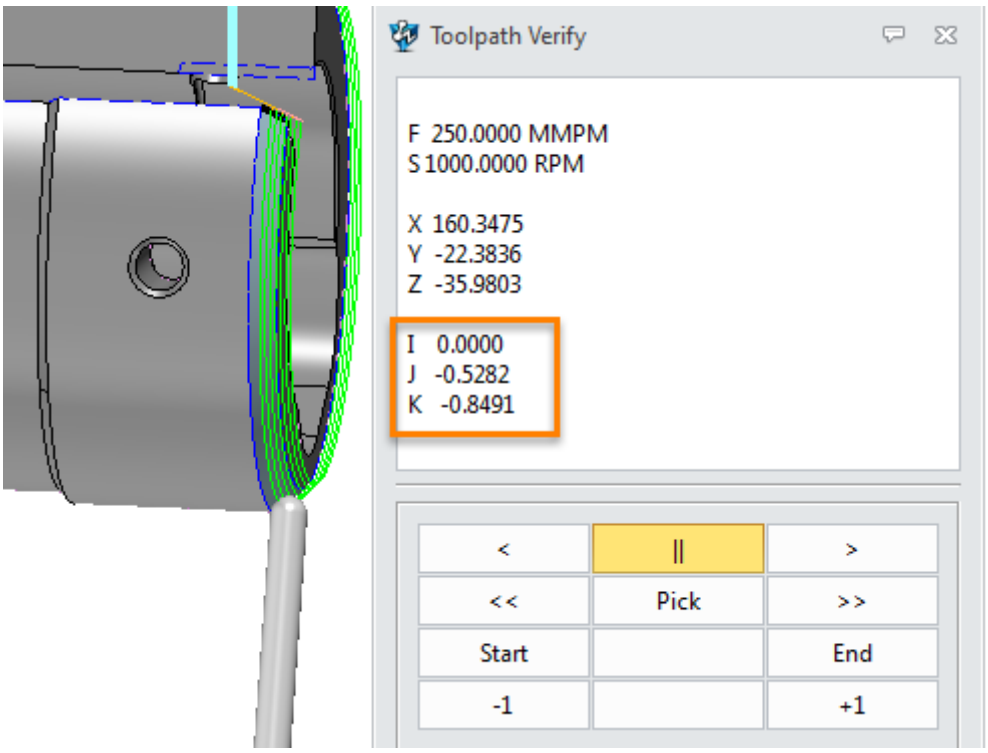


Figure118 4X Swarf cut toolpath with ball end mill tool

1.3.3 5X Drive Curve Cut

Philosophy: The 5 axis **Drive Curve Cut** uses 3D driving curves to calculate the tool path. The cutter is driven along these curves and respects the surface geometry to be cut. This operation shares the same Axis Control capability as the 5 Axis Plane Cut operation. The remaining parameters are also similar.

Open the Z3 file "**DriveCut_ISOCut_HelicalGear.Z3**" as follows:

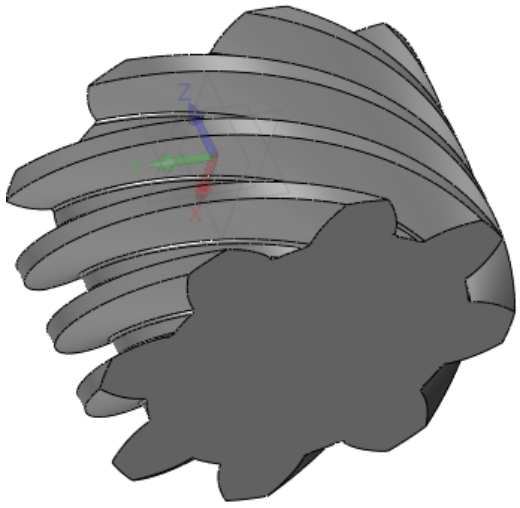


Figure119 DriveCut_ISOCut_HelicalGear file

Next, we will use this file to introduce how to run the **Drive Curve** cut operation and Guide Surface ISO operation. First, let's look at drive curve cut operation.

1. How to run Drive Curve Cut operation

STEP 01 Create the drive curve:

I. Choose the 3D medial command from the wireframe module as follows:

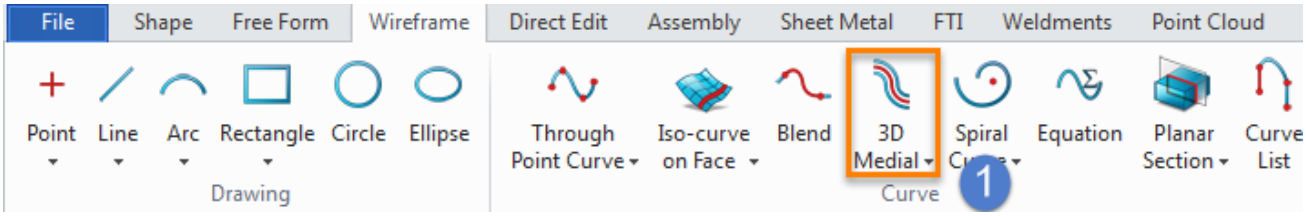


Figure120 3D Medial command

II. Pick the edges on bottom surface of any tooth socket as input as follows:

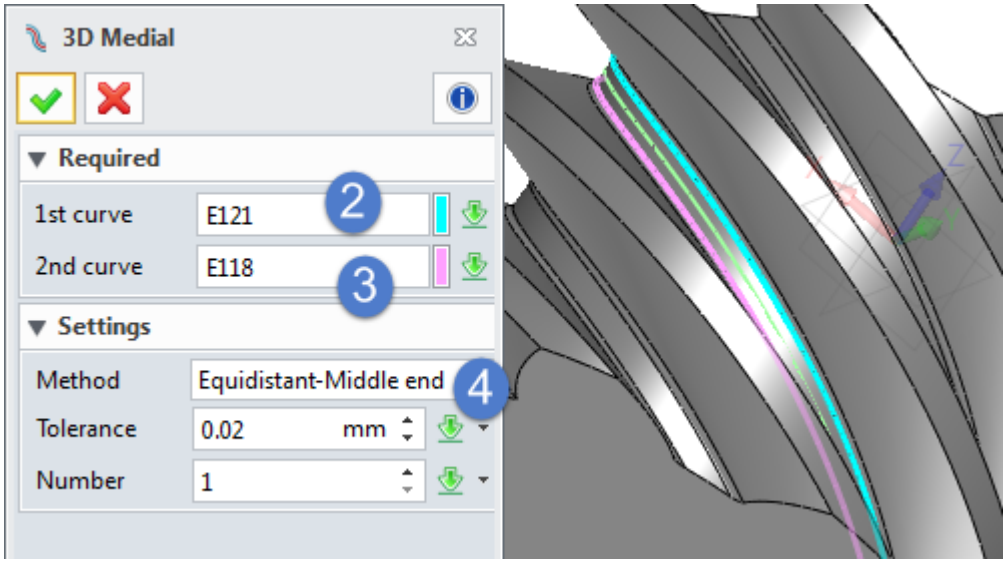


Figure121 Input curves for creating 3D medial curve

After creating the 3D medial curve then enter into CAM let's create Drive curve cut toolpath.

STEP 02 Define dirve curve and part surface

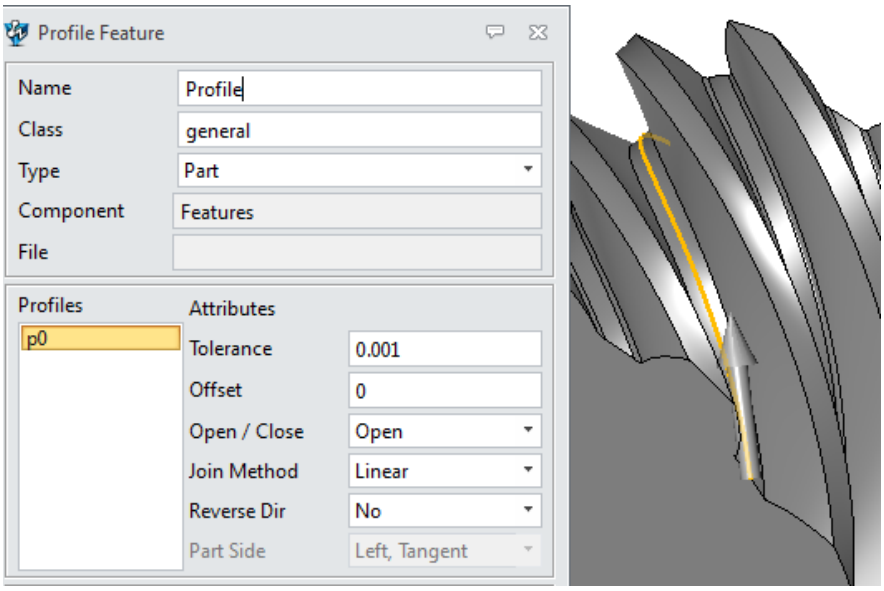


Figure122 Define profile for Drive curve cut

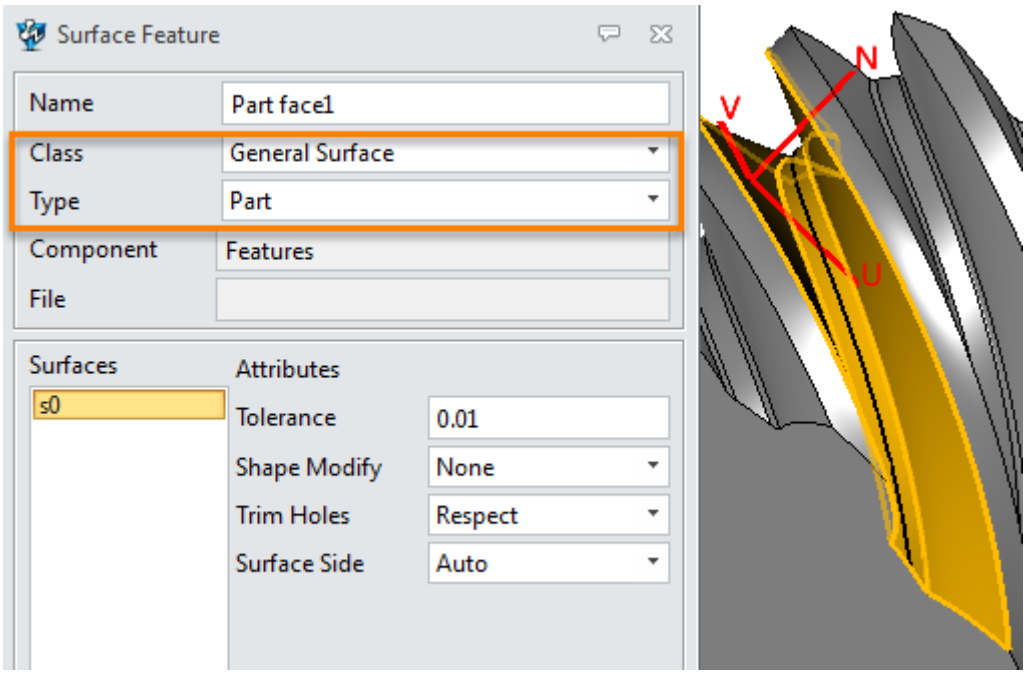


Figure123 Define part surface

STEP 03 Customize a taper tool

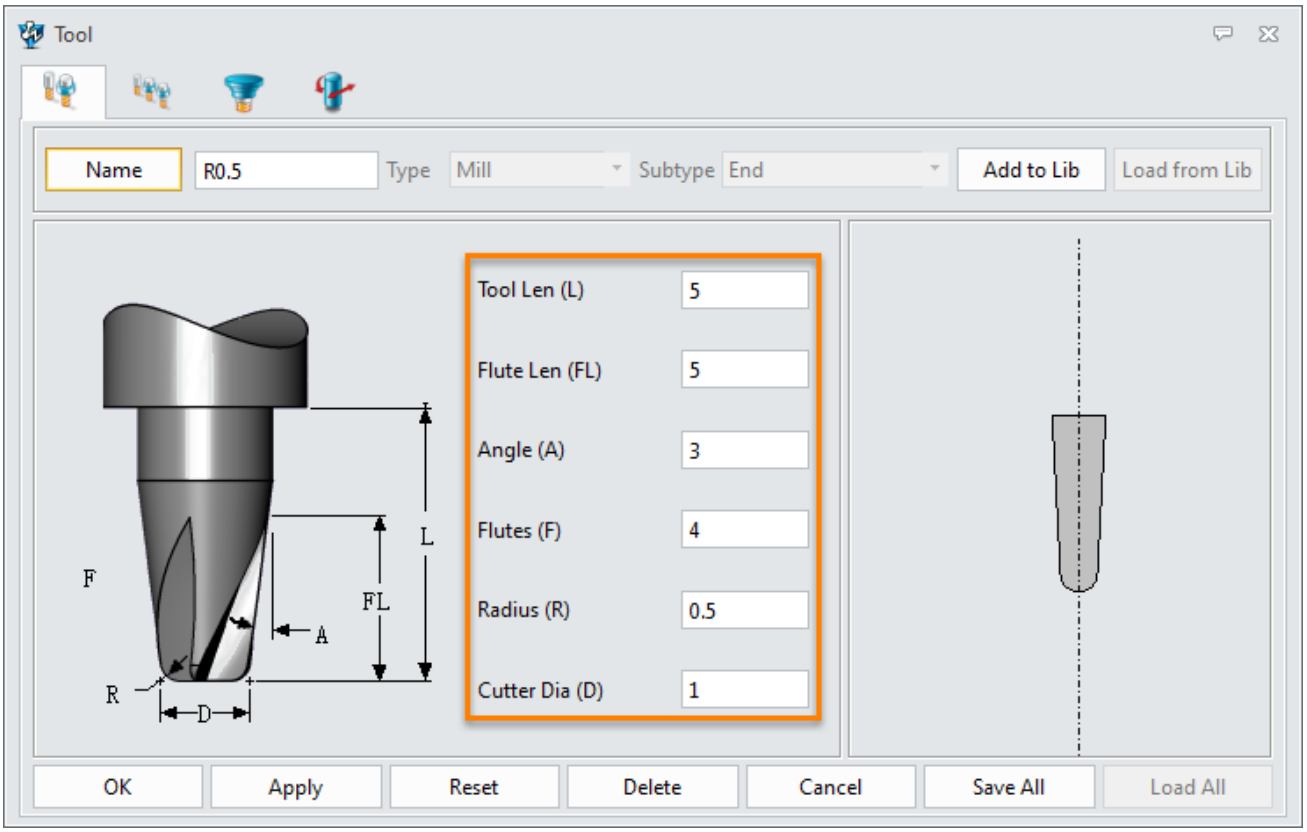


Figure124 Cutomize taper tool

STEP 04 Set up parameters:

Since most of the parameters are similar to the previous introduced PlaneCut and SwarfCut operations so here we just take some special and different parameter to explain the meaning. For others we just show the setting as follows:

I. Primary parameters:

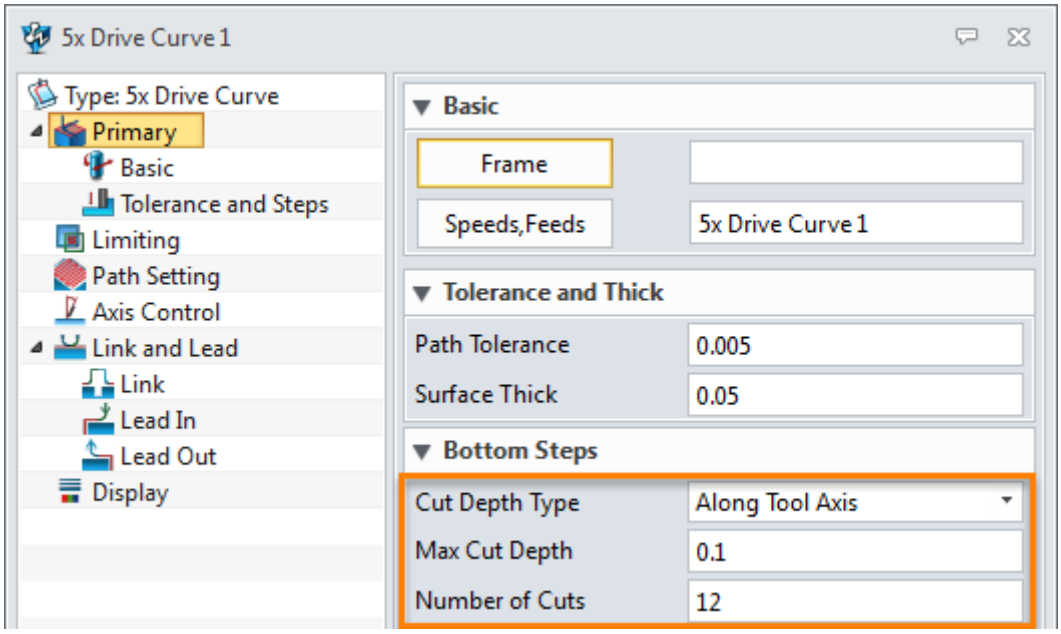


Figure125 Primary parameter of 5X Drive Curve

- **Number of Cuts**: means how many cutting layers in depth way.

II. Path setting parameters:

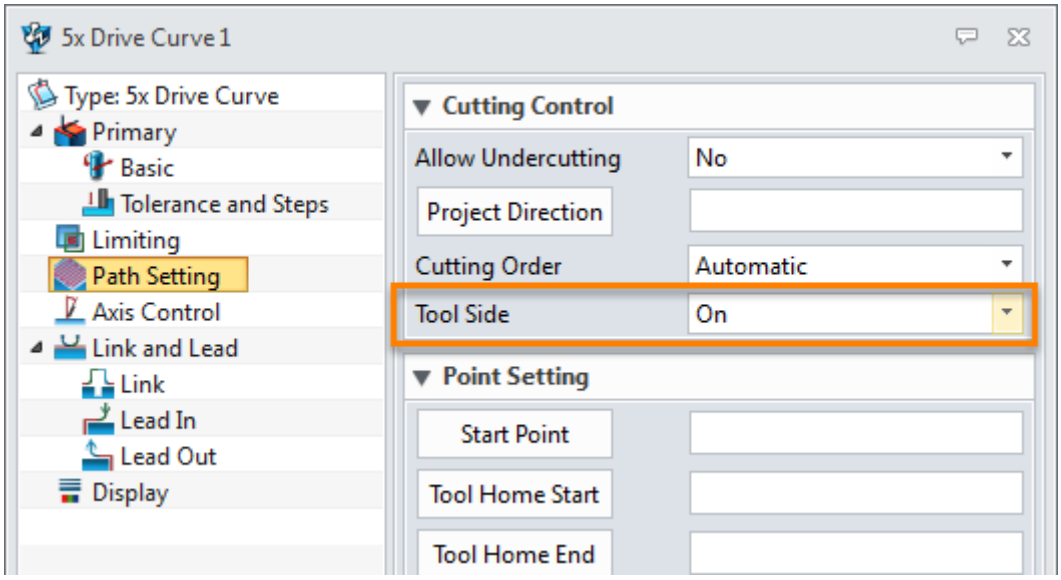


Figure126 Path setting parameters of 5X Drive Curve

- **Tool side**: includes ON, Left/Right/Center of ball as follows:

•On: The curve offset defined in any profile feature will be ignored.

•Left/Right: The cutting tool follows the corresponding left (or right) side of each driving curve when looking down from the z-axis. The left or right offset equals the sum of the curve offset of the profile feature and the tool radius.

•Center of ball : means the center of ball end mill will locate on the drive curve, it is the same function as the On if tool is not ball end.

III. Axis control parameters:

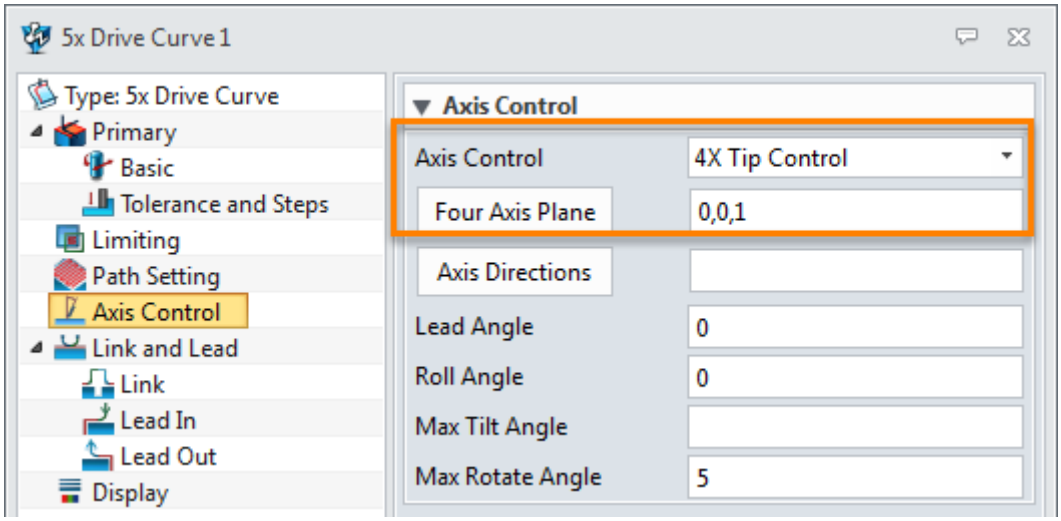


Figure127 Axis control parameters of Drive Curve

IV. Link, lead in and lead out parameters:

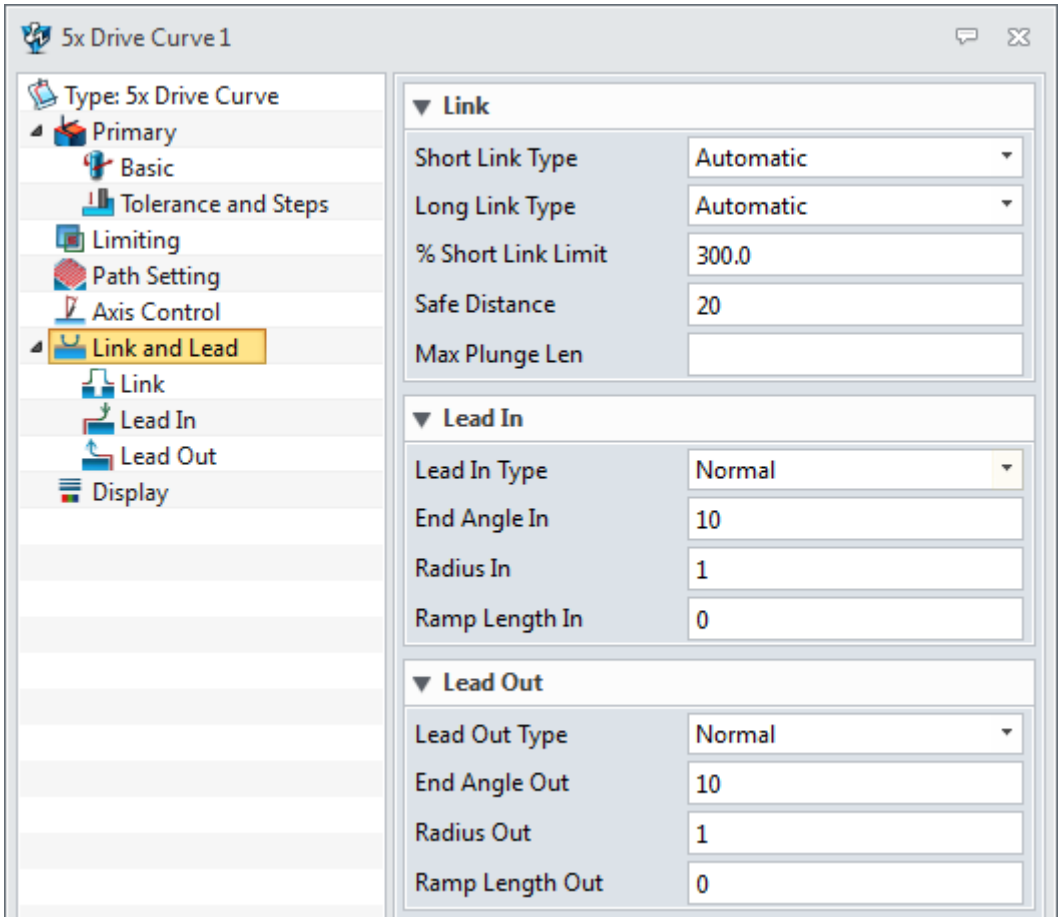


Figure128 Link,lead in and lead out parameters of 5X Drive Curve

STEP 04 Calcualte toolpath as follows:

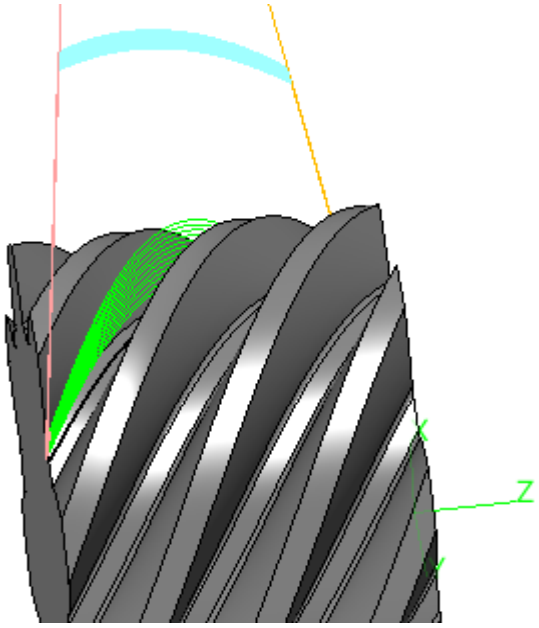


Figure129 5X Drive Curve toolpath

So far we finished the 5X drive curve toolpath for the tooth socket. But as we can see that we had left some material over, so it is necessary to use another operation to finish the whole tooth socket. Then let's take a look at 5X GuideSurface ISO Cut operation as follows:

1.3.4 5X Guide Surface ISO Cut

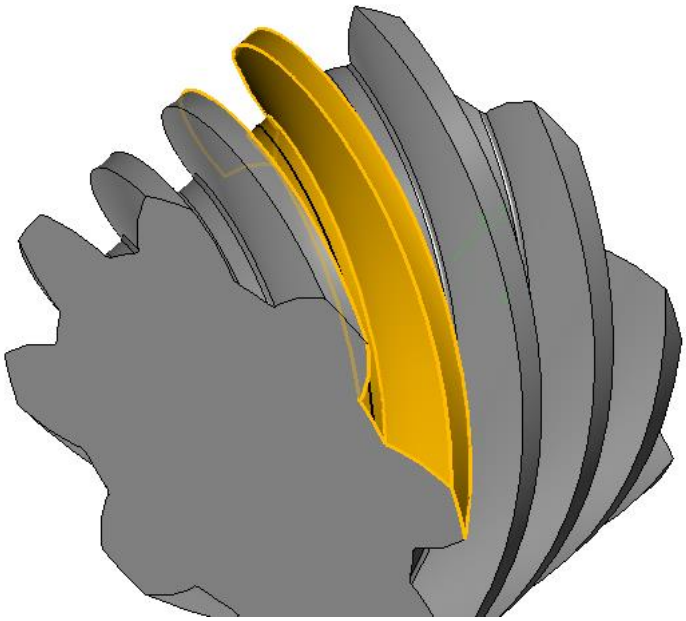


Figure130 Finish required region for the tooth socket

Phylosophy :

1) Each 5 axis **Guide Surface Iso Cut** operation must have **drive surface(s)** selected in its feature list with general surfaces input as the cutting target.

2) The **guide surface** defined by a drive surface **forces tool axis to follow its normals along the isolines.**

3) If the field "**Cut Drive Surface**" is toggled as "Yes", the drive surfaces will serve as the cutting target in addition to other general surfaces in the feature list. If "No", the drive surfaces will be ignored. The iso direction will be either U-isolines or V-isolines,, conventional or climbing.

Therefore it is easy to know if we want the tool axis to follow a normal way along a special pattern we still need a drive surface as follows:

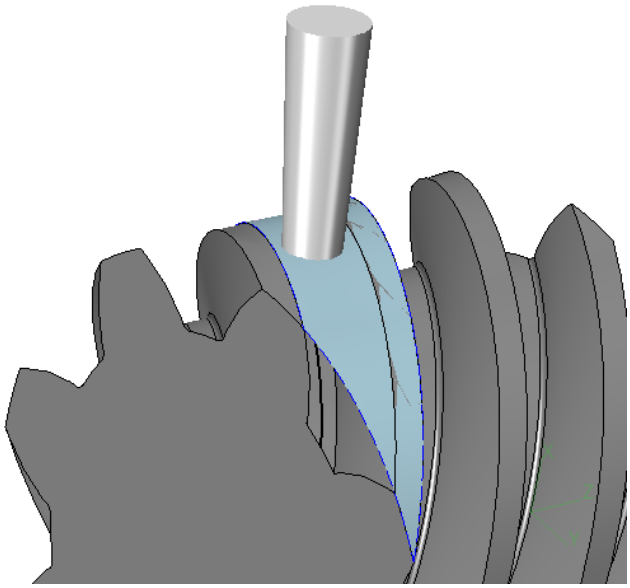


Figure131 Tool axis normal to drive surface and move along the isoline

Next let's create the toolpath as follows:

STEP 01 Create drive surface

I. Copy the origial part file and rename it as auxilairy face as follows:

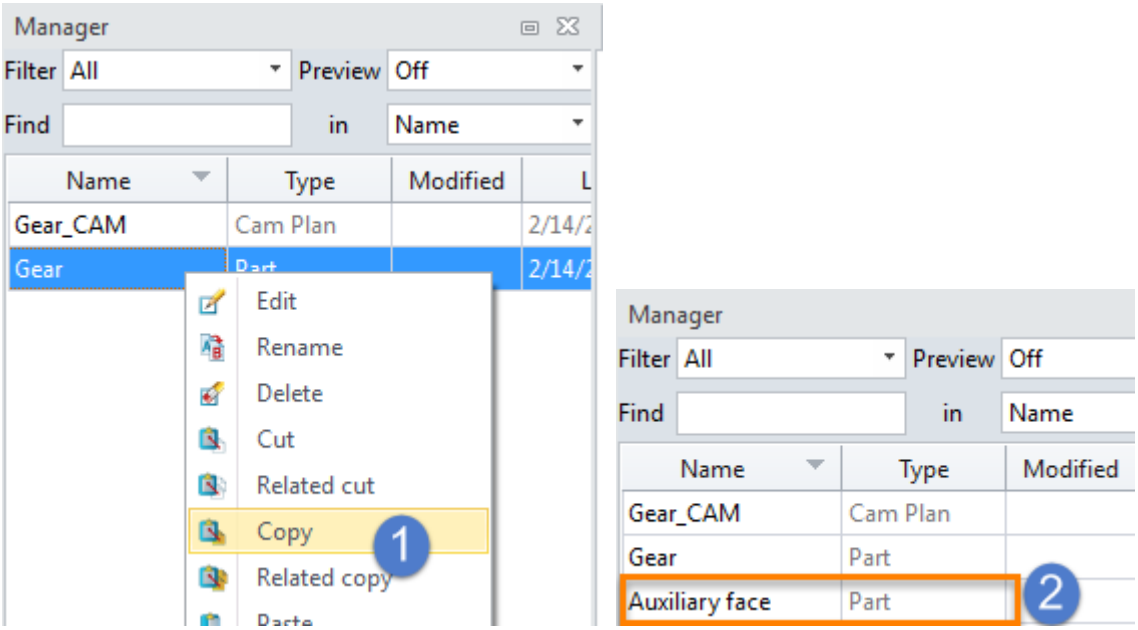


Figure132 Copy original gear part

II. Enter into the new Auxiliary face part file and create surface as follows:

1) Create a blend curve at one end as follows:

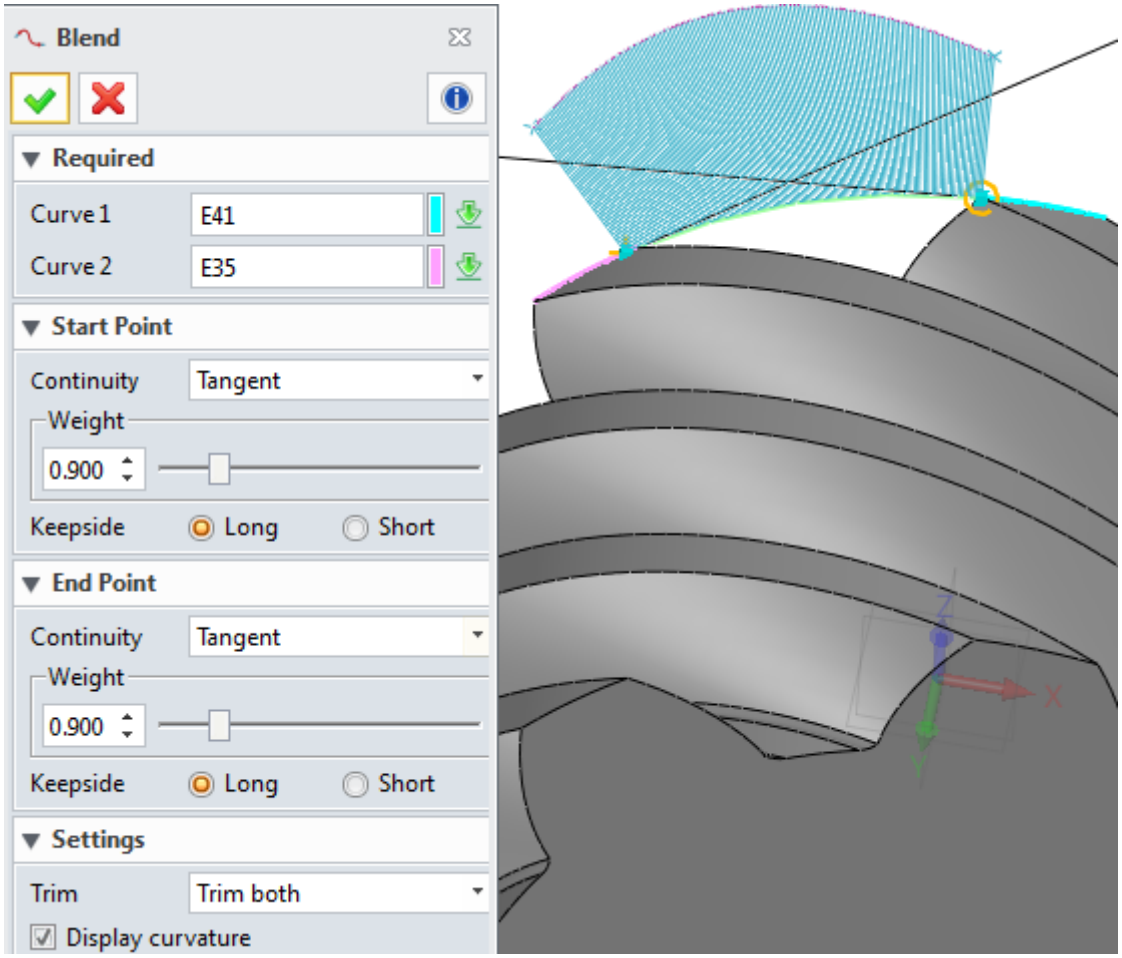


Figure133 Create a blend curve between 2 edges

2) Then make use of the curve to create a surface by "**Bi Rail Loft**" command as follows:

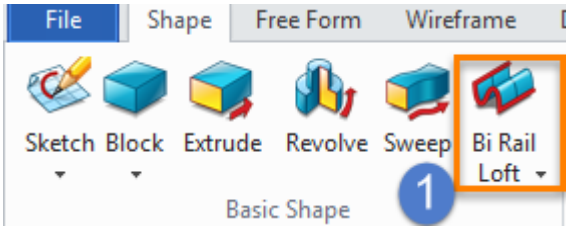


Figure134 Bi Rail Loft

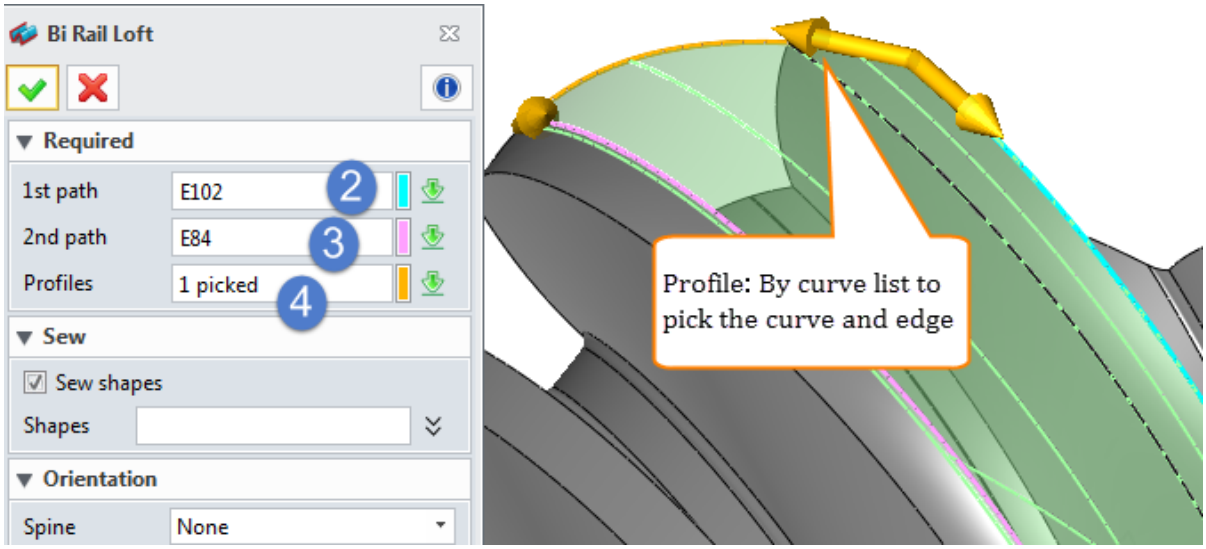


Figure135 Create Surface by Bi Rail Loft command

3) Then delete the gear shape and just leave the surface alone as follows:

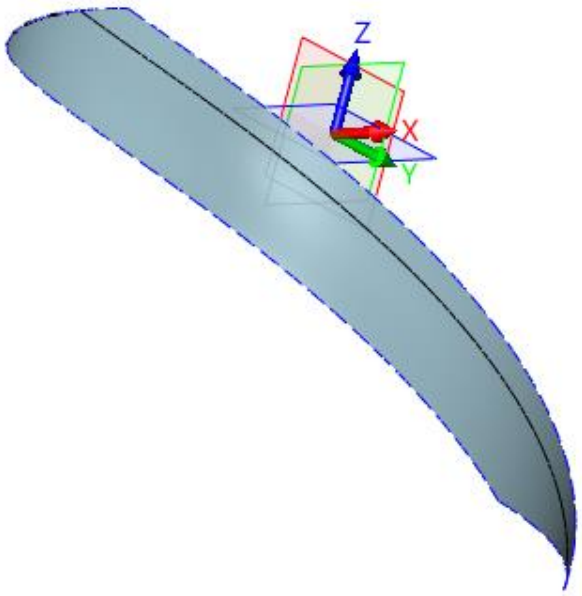


Figure136 Leave the created surface only

4) Merge the surfaces together as follows:

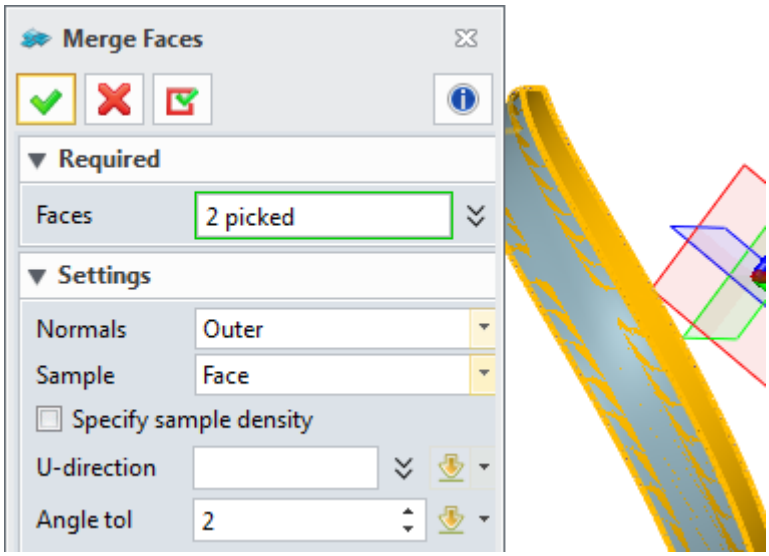


Figure137 Merge the surface together

STEP 02 After finishing the drive surface, go back to CAM and add the auxiliary face into as follows:

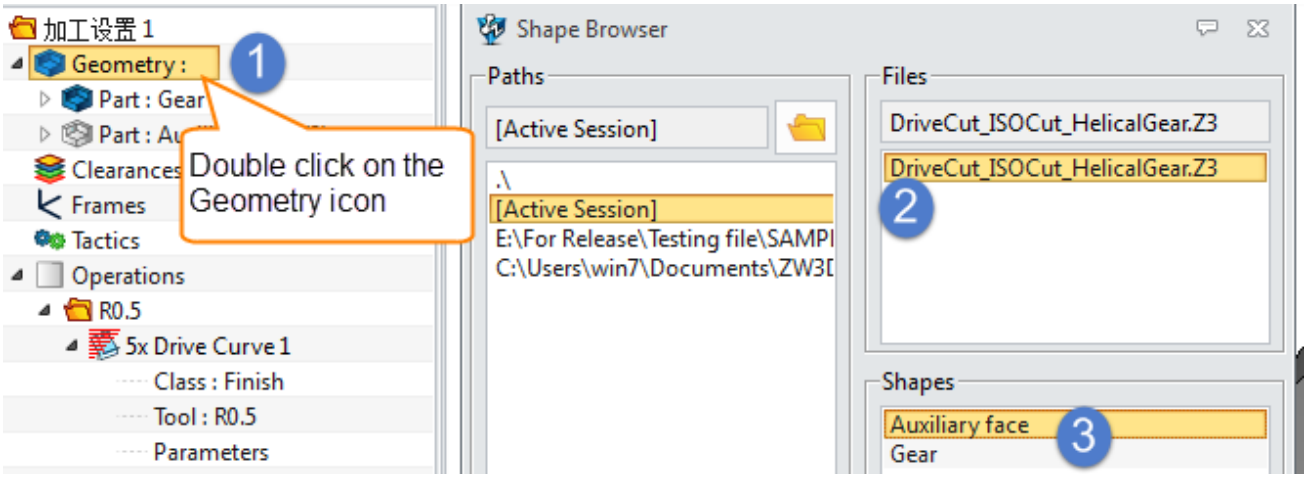


Figure138 Add auxiliary face into CAM

STEP 03 After defining the auxiliary face as drive surface, define the cutting region as part surface as follows:

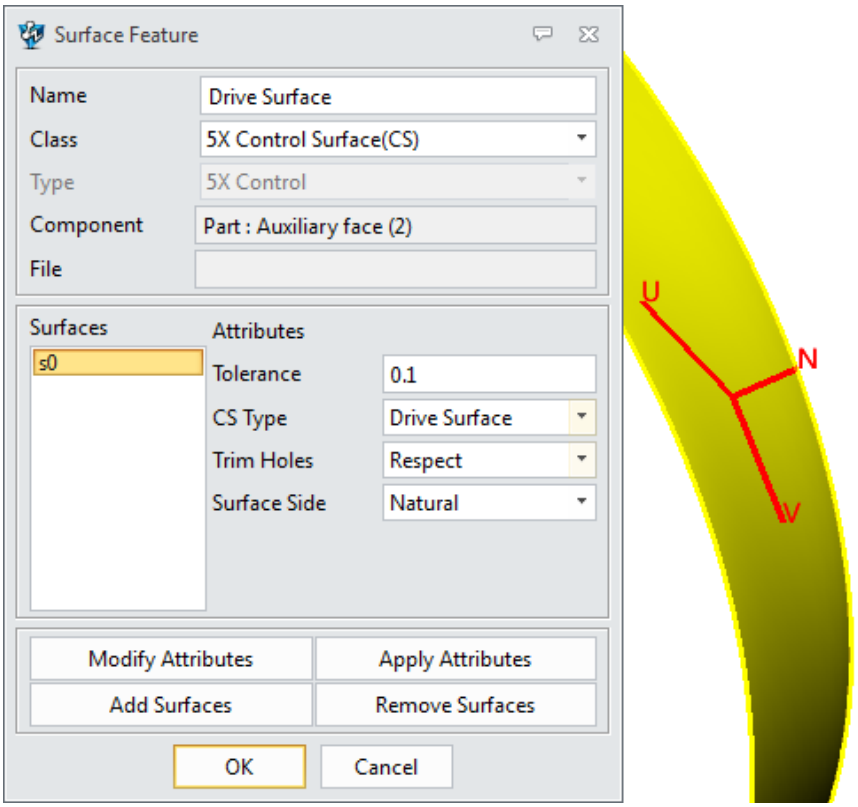


Figure139 Define drive surface

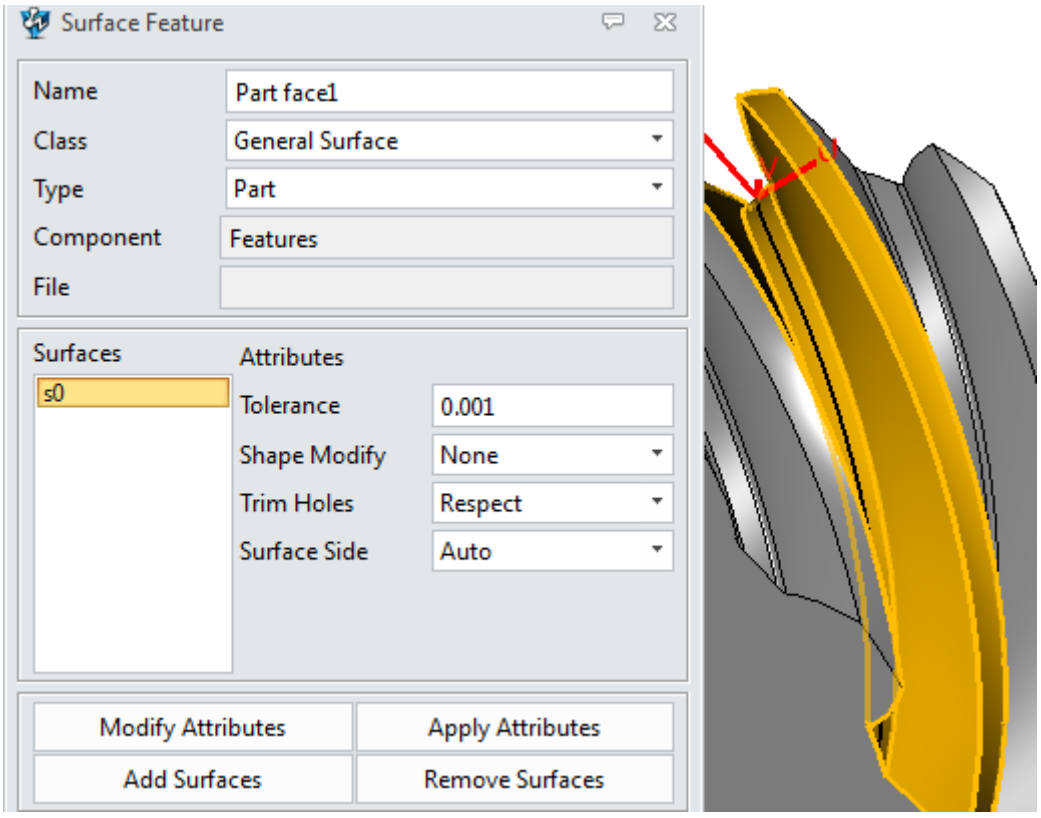


Figure140 Define part surface for 5X Guide Surface ISO Cut

STEP 04 Choose the same tool used in drive curve operation and then set up the parameters as follows:

I. Primary parameters:

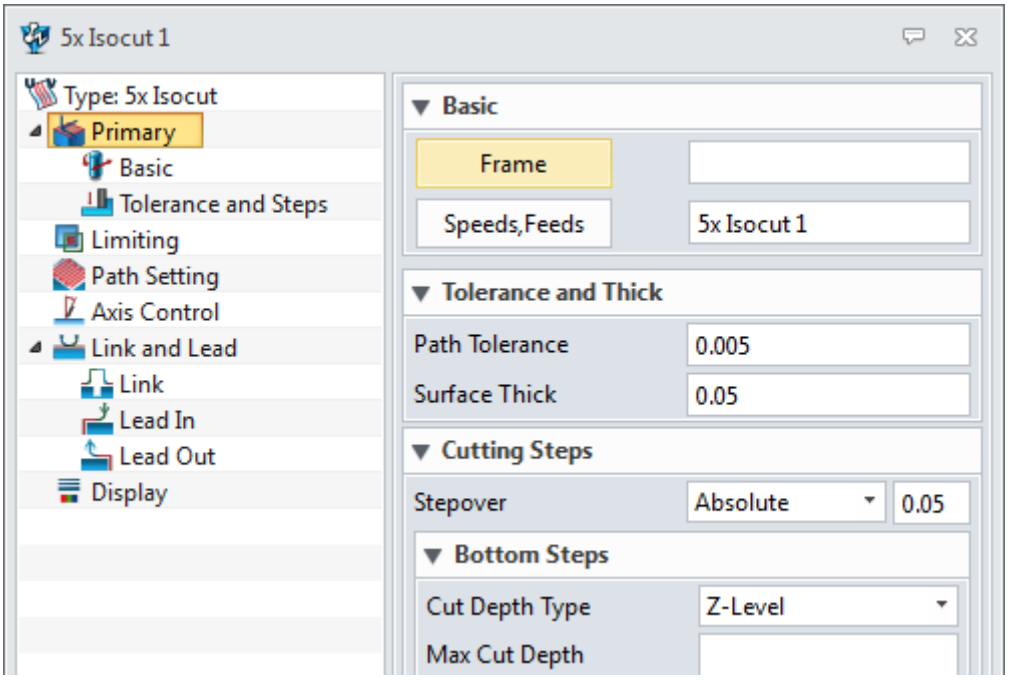


Figure141 Primary parameters of 5X IsoCut

- **Stepover**: can choose differnet type,including Absolute, %Tool Dia, Scallop, Num of Cuts etc.It is the same menaing for the Absolute, %Tool Dia, Scallop,options with what in 3X milling operations.
- **Num of Cuts** means how many toolpath layer you can determine to cretae, which will fill the whole part face.

II. Path setting parameters:

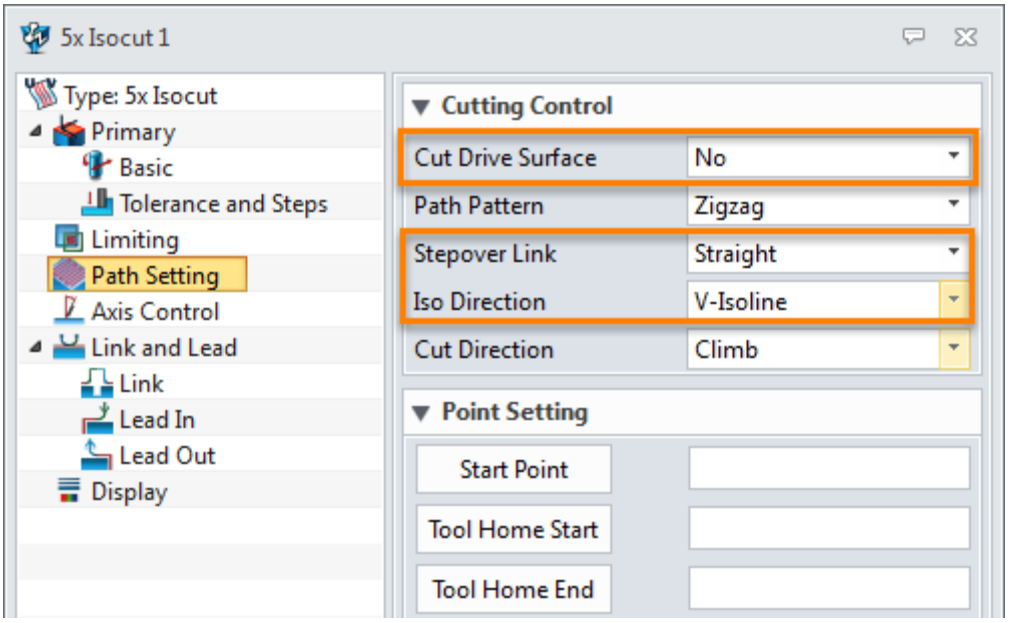


Figure142 Path setting parameters of 5X IsoCut

- **Cut Drive Surface**: if we choose yes then the drive surface will serve as milling target at the same time. If No, then it will ignore it, and only use the drive surface to guide the tool axis.
- **StepoverLink:** Straight or Round
- **ISO direction**: if along the U direction or V direction of drive surface

III. Axis Control parameters

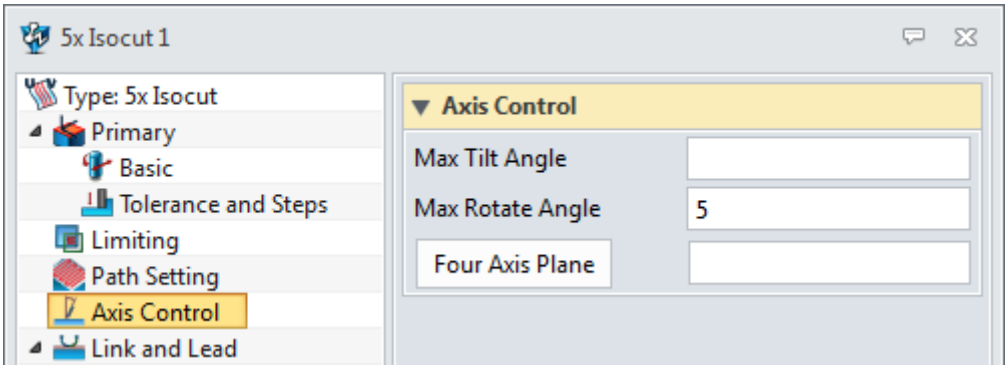


Figure143 Axis Control parameters of 5X IsoCut

Here the four axis plane will affect your output, you can set up according to the real machine's structure.

IV. Leak and lead parameters:

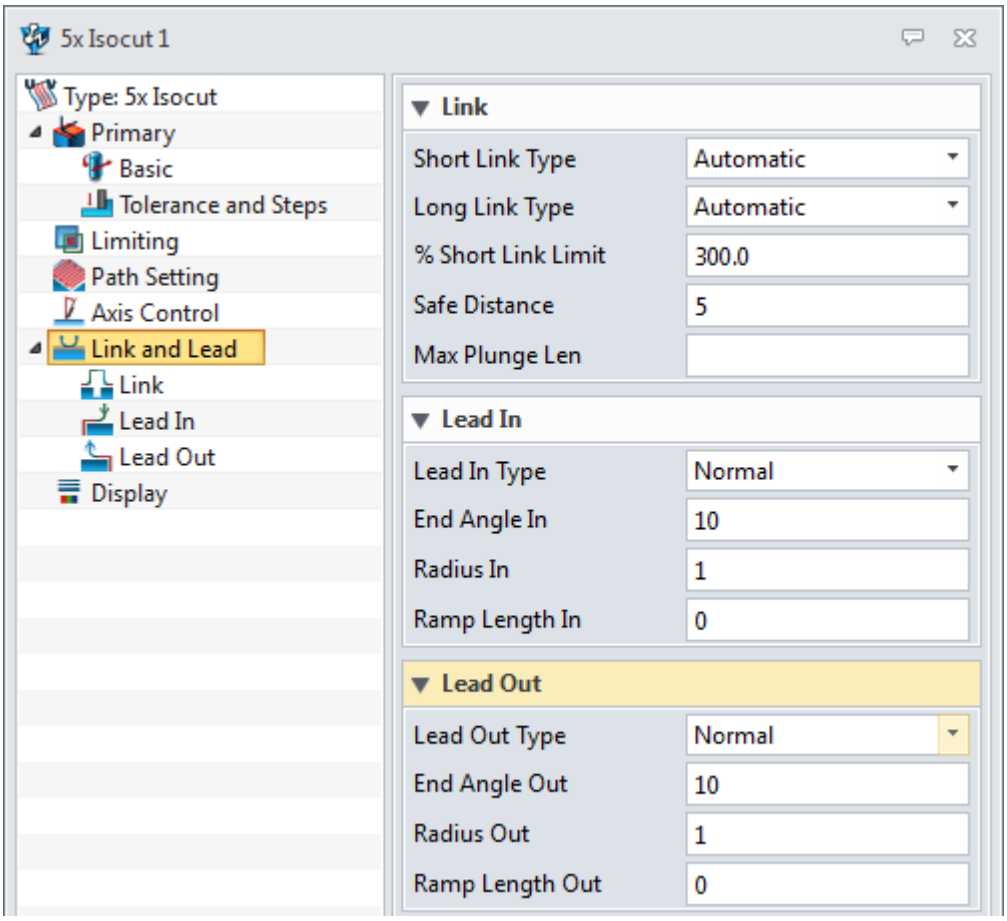


Figure144 Link and lead parameters

STEP 05 Calculate the toolpath as follows:

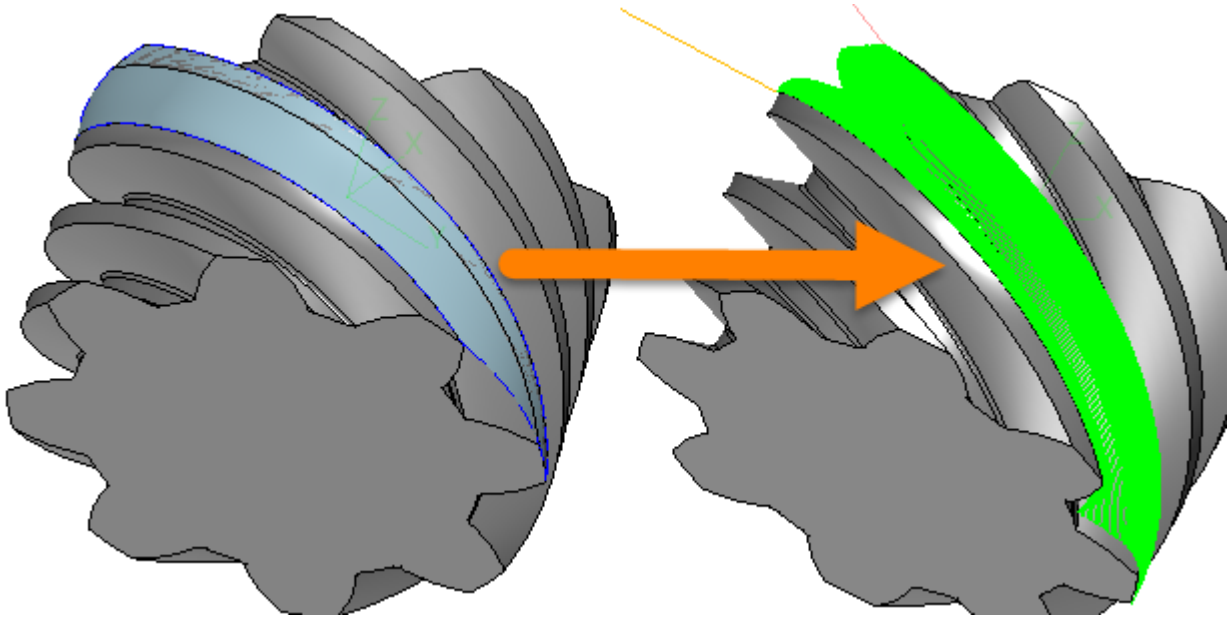


Figure145 5X Guide Surface ISO Cut toolpath

1.3.5 5X Flow Cut

Philosophy: The 5 axis Flow Cut operation requires either a 5 axis Swarfcut or a 5 axis Drive Curve Cut as a reference operation that contains two separated cuts. These two cuts will be used as flowing curves. The Swarf cut or Drive Curve Cut can also have multiple depths. ZW3D CAM will select the two bottom cuts as flowing curves. It is very useful for machining areas between two tilted walls (turbine blades for example).

So let's open the file "**5X_Impeller.Z3**" again, when introducing the swarf cut operation we have created the swarf cut toolpath to finish the blade as follows:

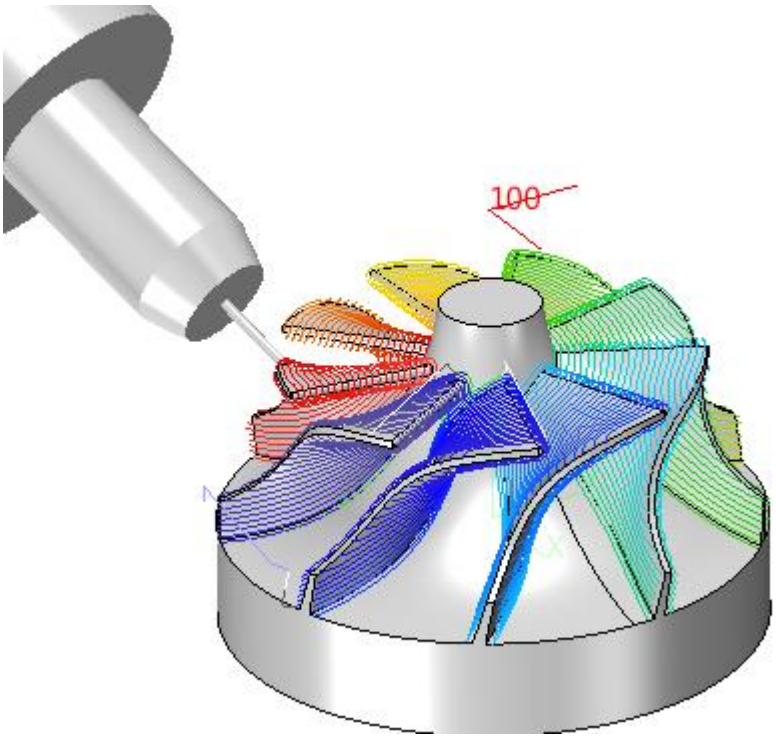


Figure146 5X impeller with Swarf cut toolpath

Here we will make use of the part to introduce 5X Flow Cut operation. Now we need to create toolpath for the areas between blades and the out surface of the blade as follows:

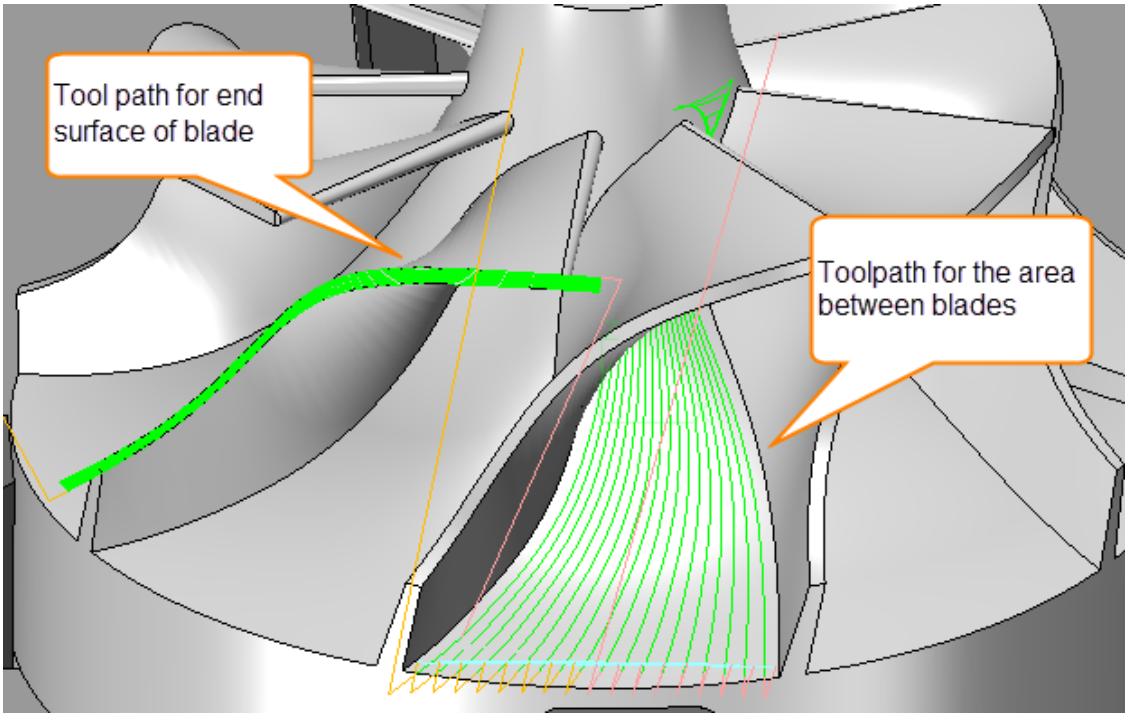


Figure147 Tool path for impeller

Next we will use both introduced swarf cut operation and drive curve operation as reference separately to create the 5X flow toolpath on the desired areas.

1. Reference to Swarf Cut operation

STEP 01 Create another swarf cut toolpath on the root of blade as follows:

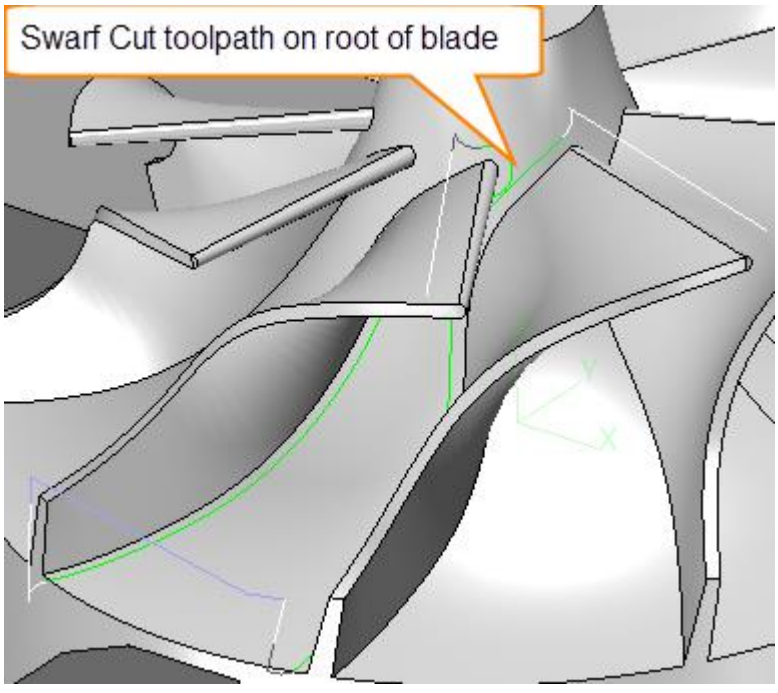


Figure148 Swarf cut toolpath on root of blade

STEP 02 Create 5X flow cut reference to the swarf cut as follows:

I. Define part surface for 5X flow cut as follows:

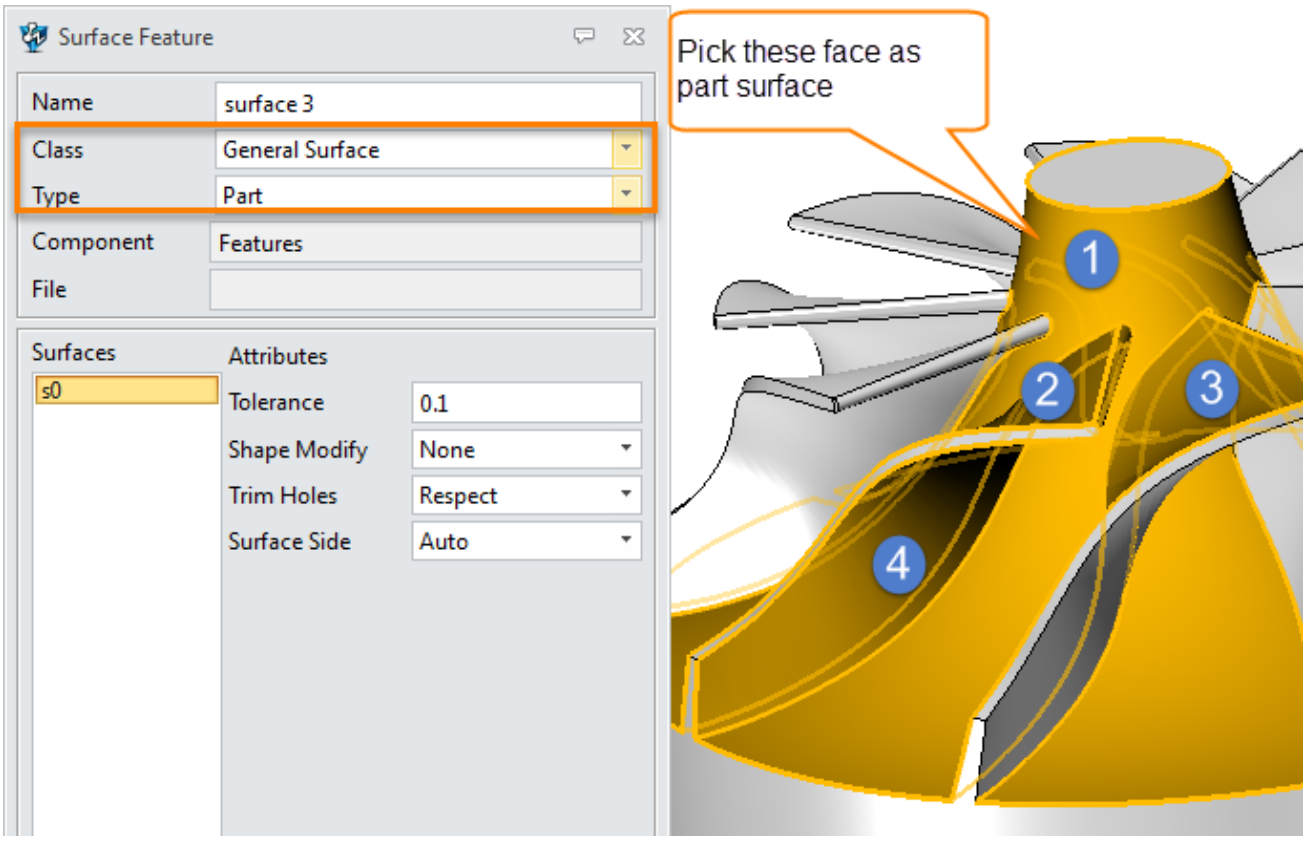


Figure149 Define part surface for 5X flow cut

II. Choose swarf cut as reference operation as follows:

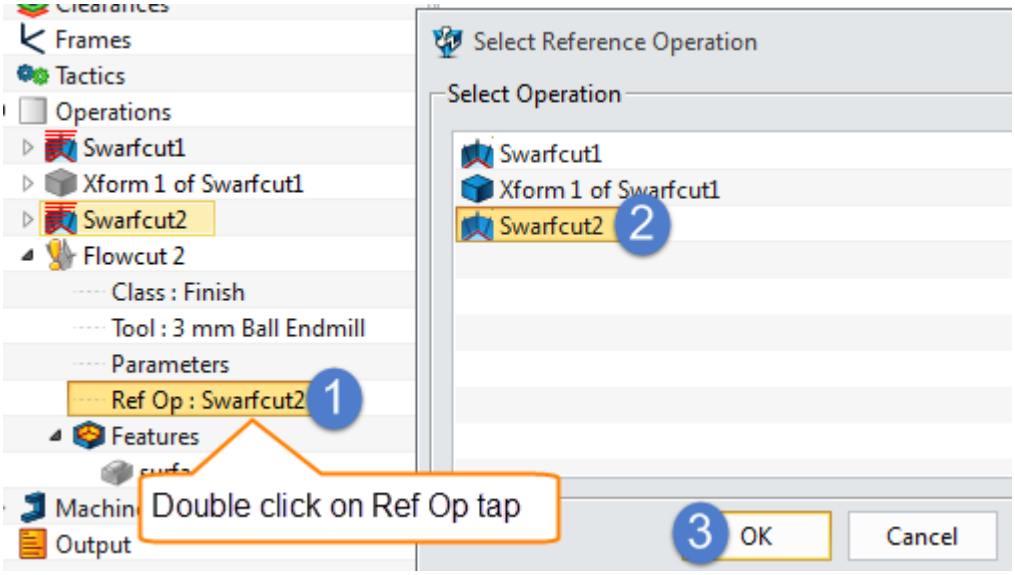


Figure150 Pick swarfcut2 as reference

III. Choose Tool : 3 mm Ball End Mill

IV. Set up pararmeters

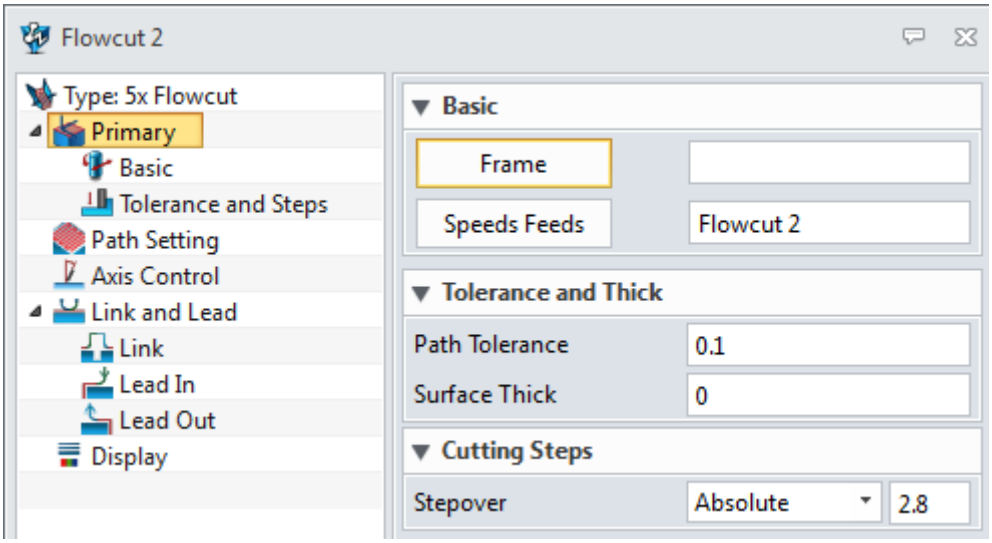


Figure151 Primary parameters

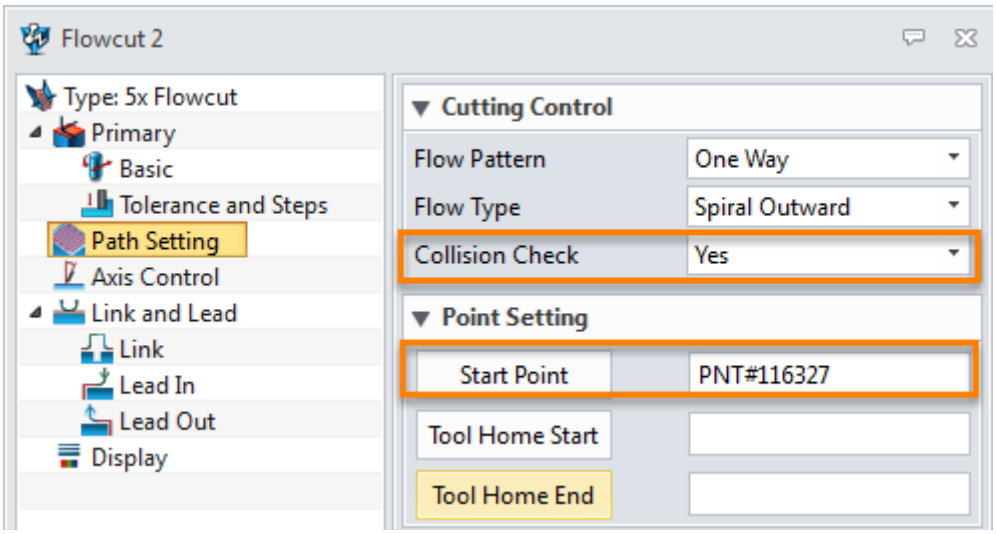


Figure152 Path setting parameters

- **Collision check**: choose yes to check if the toolpath will collide with stock
- **Start Point**: it is allowed to set up the start point as follows:

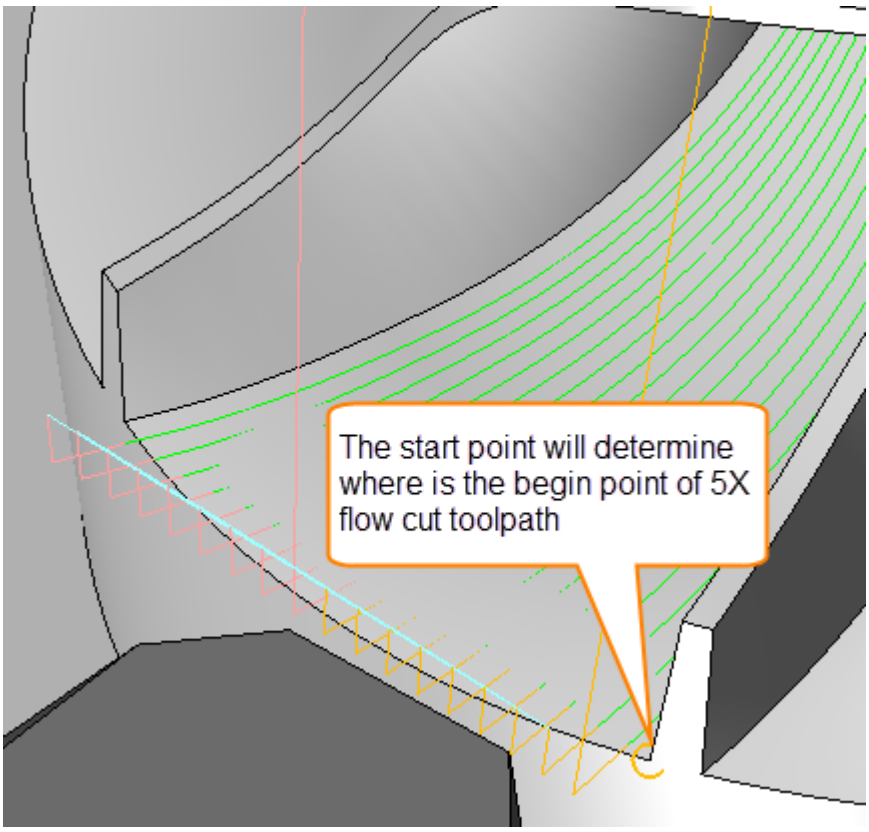


Figure153 Strat point for 5X flow cut

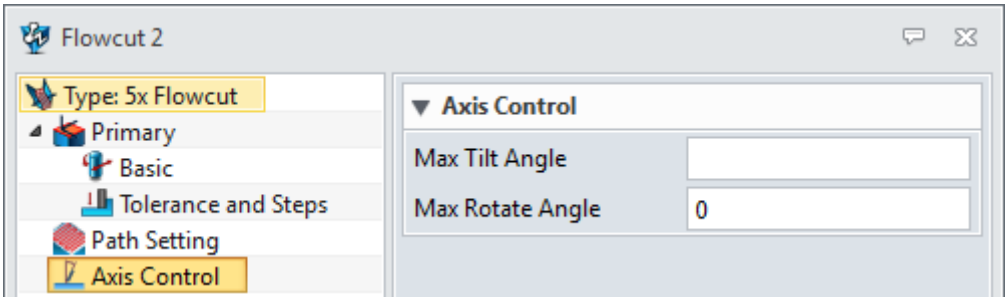


Figure154 Axis control parameters

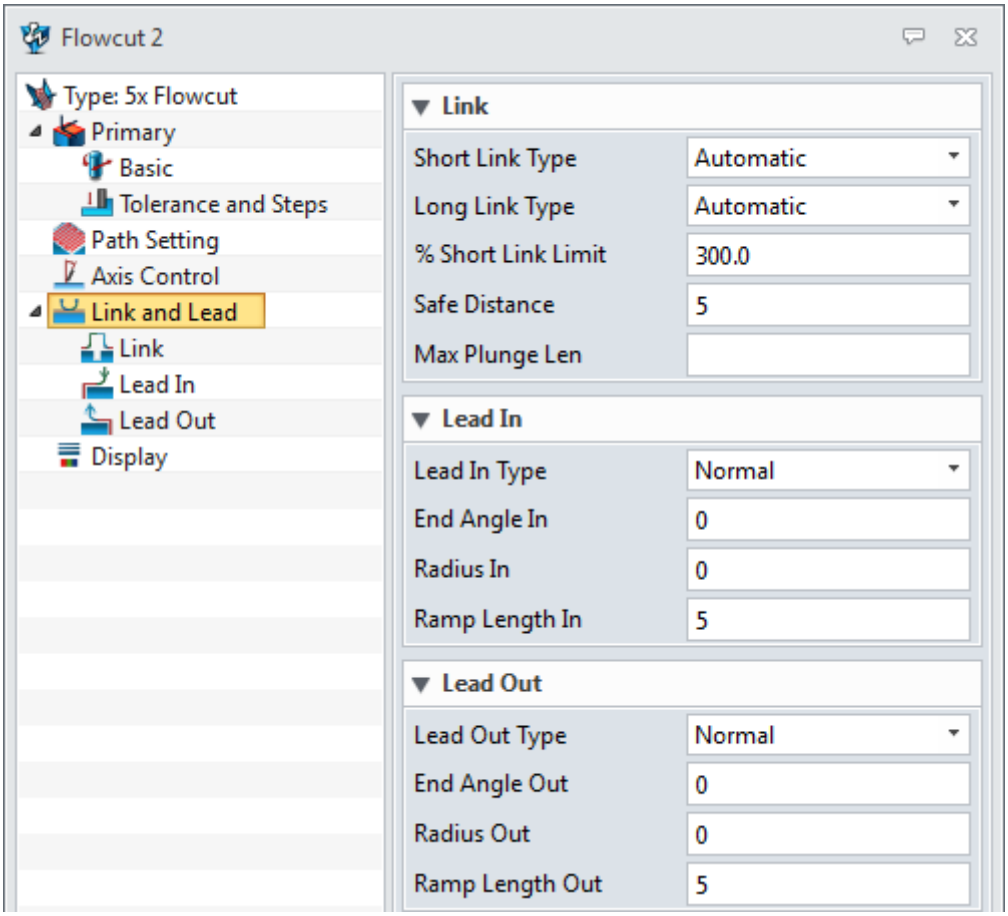


Figure155 Link and lead

STEP 03 Calculate the toolpath and we will get the following result:

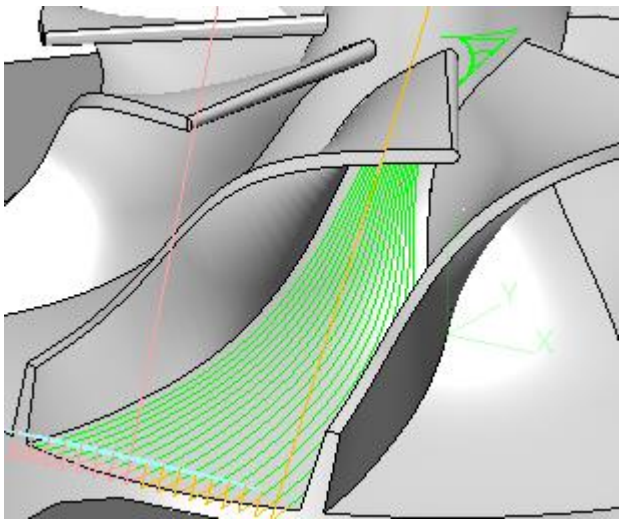


Figure156 5X Flow Cut toolpath

2. Reference to 5X Drive Curve operation

Here we will create 5X flow cut toolpath on the end face of blade as follows:

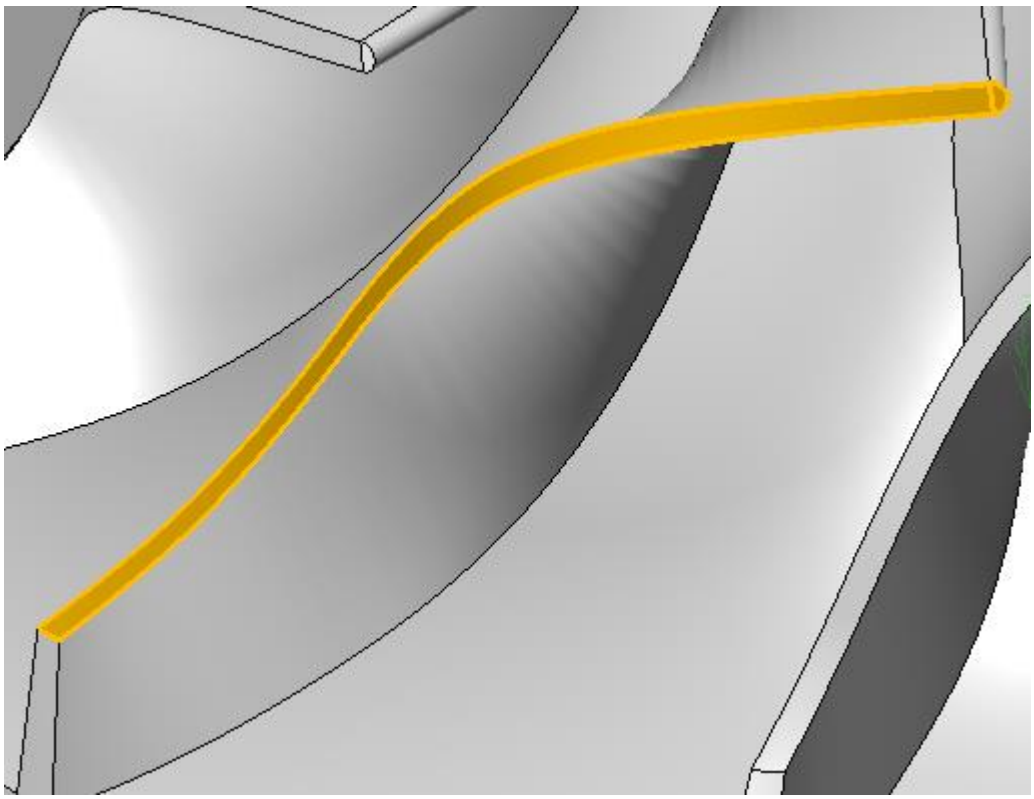


Figure157 End face of blade

In order to make the toolpath that can fully cover the whole end surface, it is necessary to create an auxiliary surface to help create the toolpath. As follows:

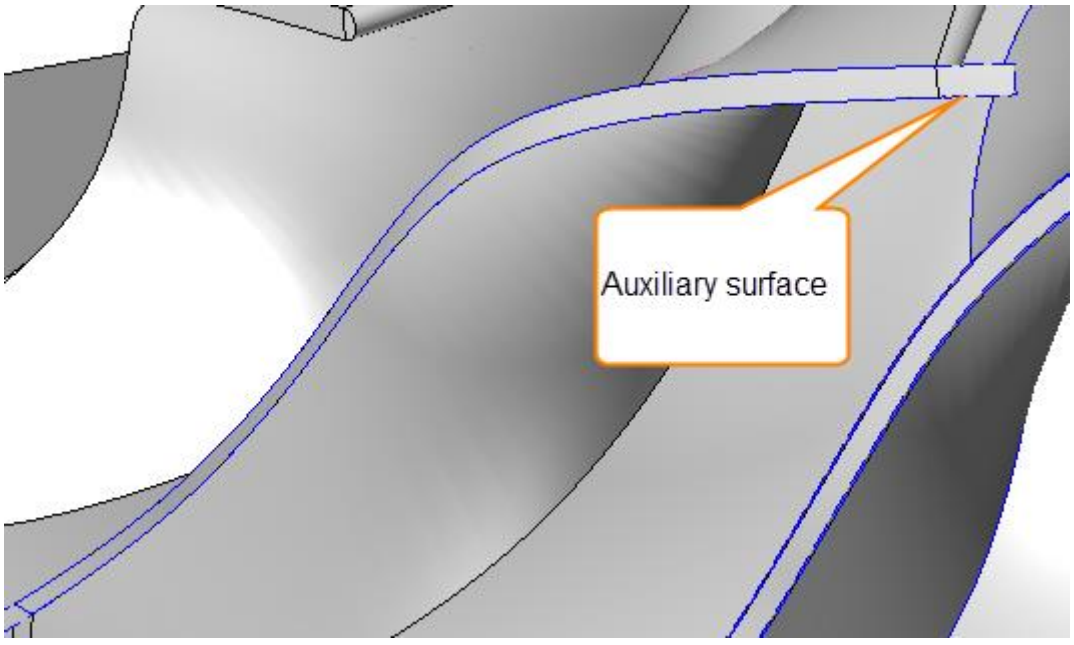


Figure158 Auxiliary surface for 5X flow cut

Here we will not introduce how to create the auxiliary surface again, you can find it in the file of "5X_impeller" as follows:

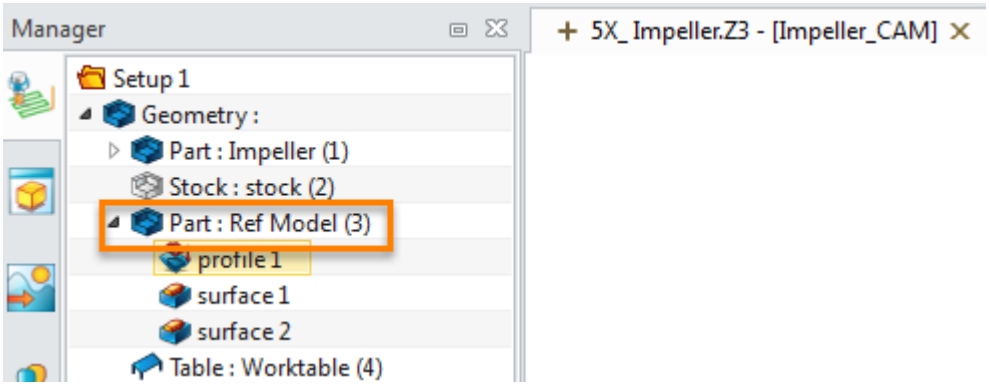


Figure159 Auxiliary surface mode for 5X Impeller

Next let's make use of the existing auxiliary surface to create the toolpath

STEP 01 Define the surface as part surface and choose the profile as drive curve as follows :

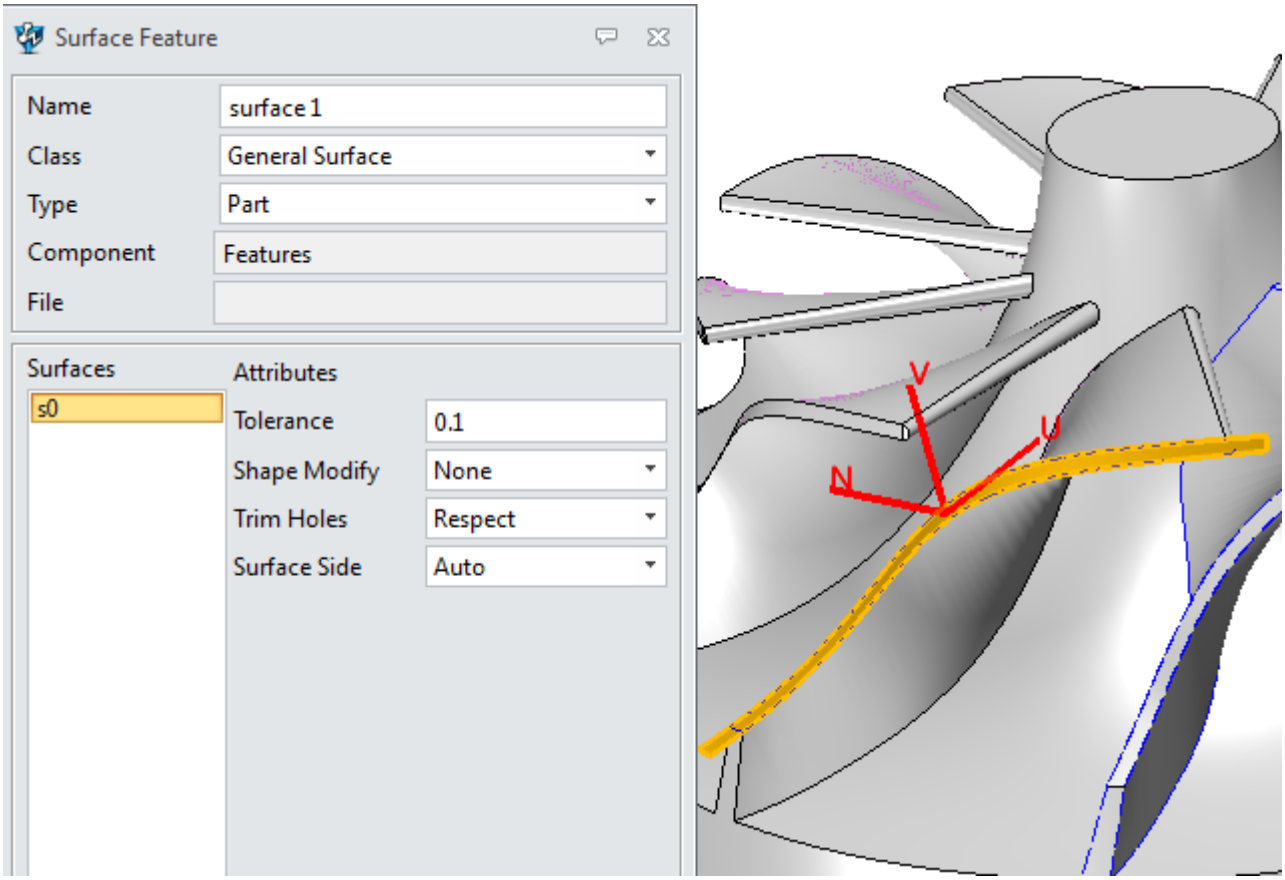


Figure160 Define the auxiliary surface as part surface for drive curve operation

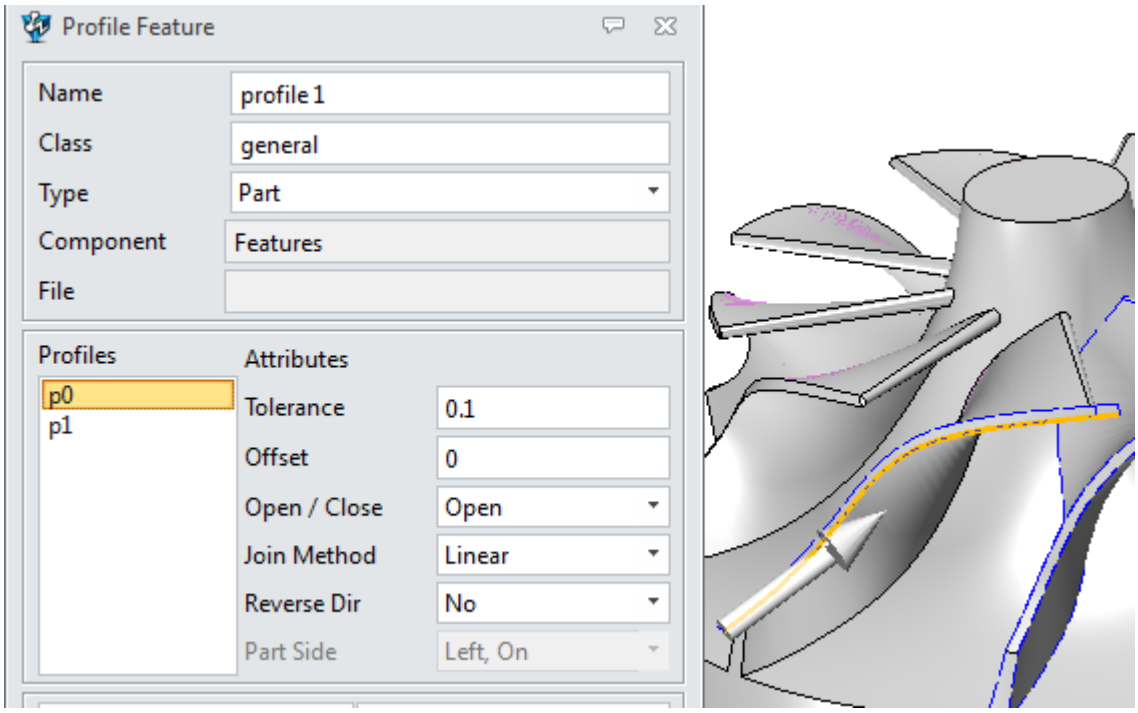


Figure161 Define profile for drive curve

STEP 02 Choose ball end mill of 3 mm and set up parameters to create toolpath as follows :

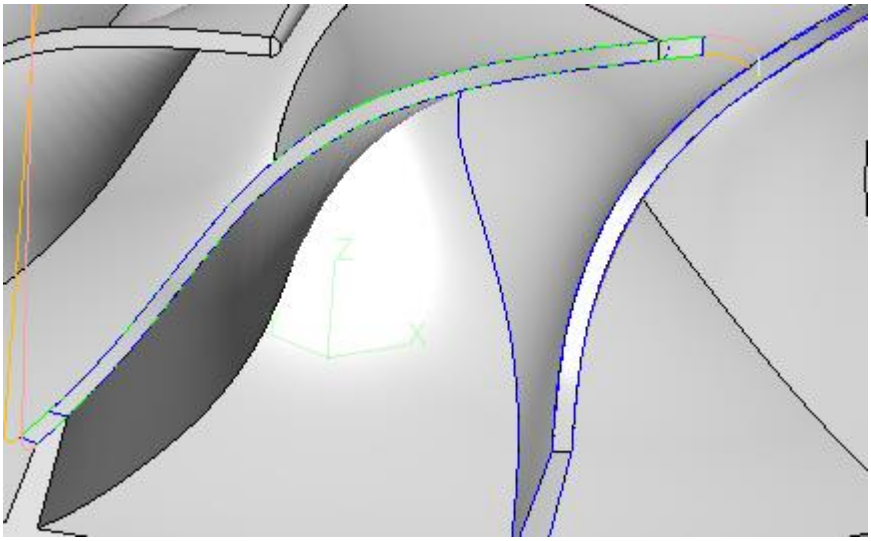


Figure162 5X Drive curve toolpath

STEP 03 Create 5X flow cut as follows:

I. Set up the feature and reference operation

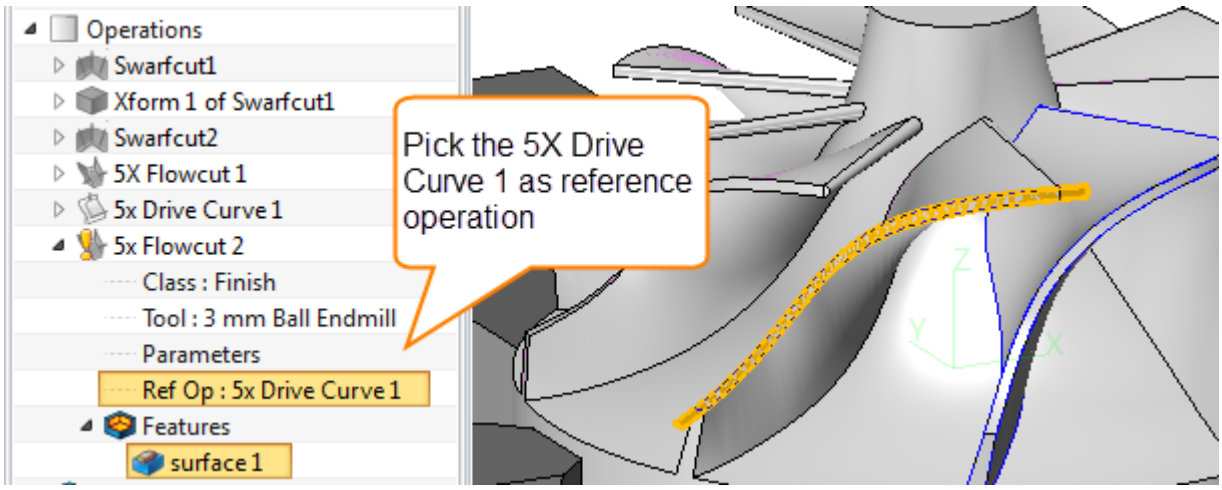


Figure163 Reference to 5X Drive Curve to create Flow cut toolpath

II. Set up parameters

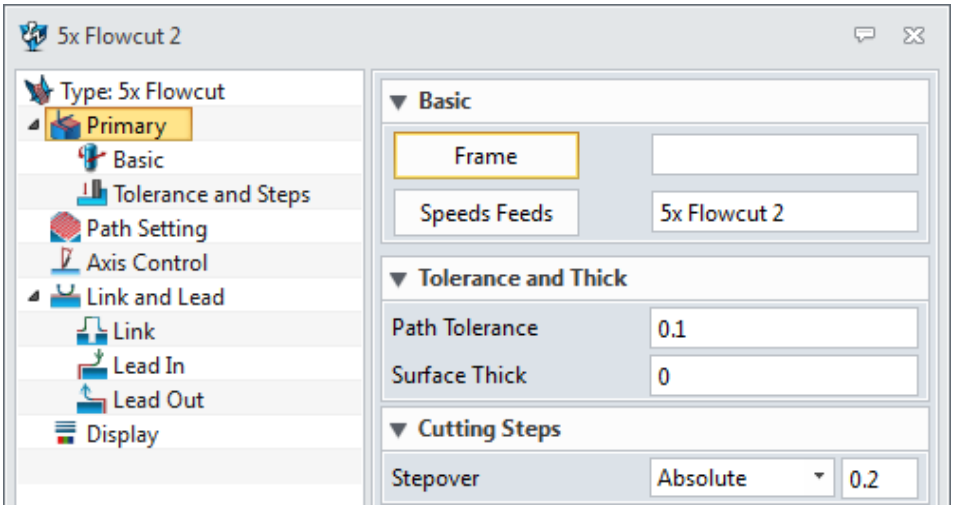


Figure164 Primary parameters

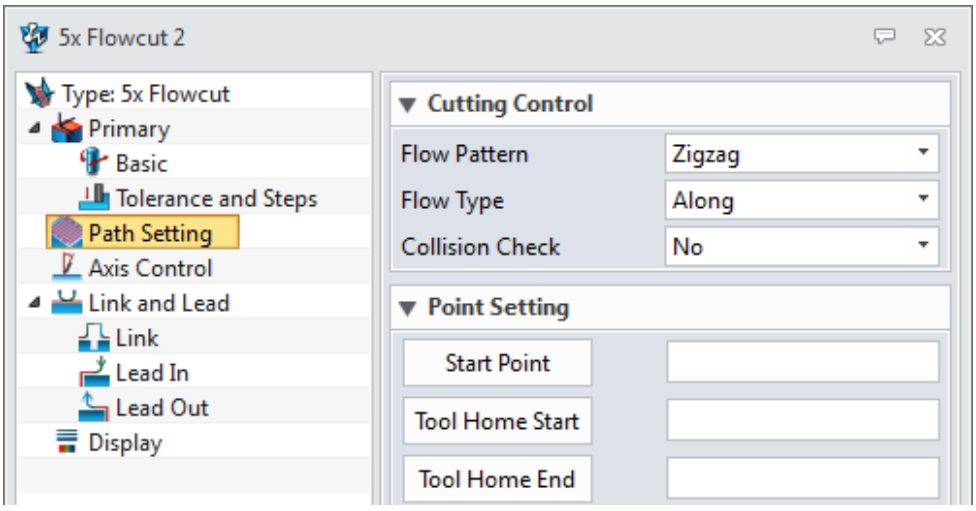


Figure165 Path Setting parameters

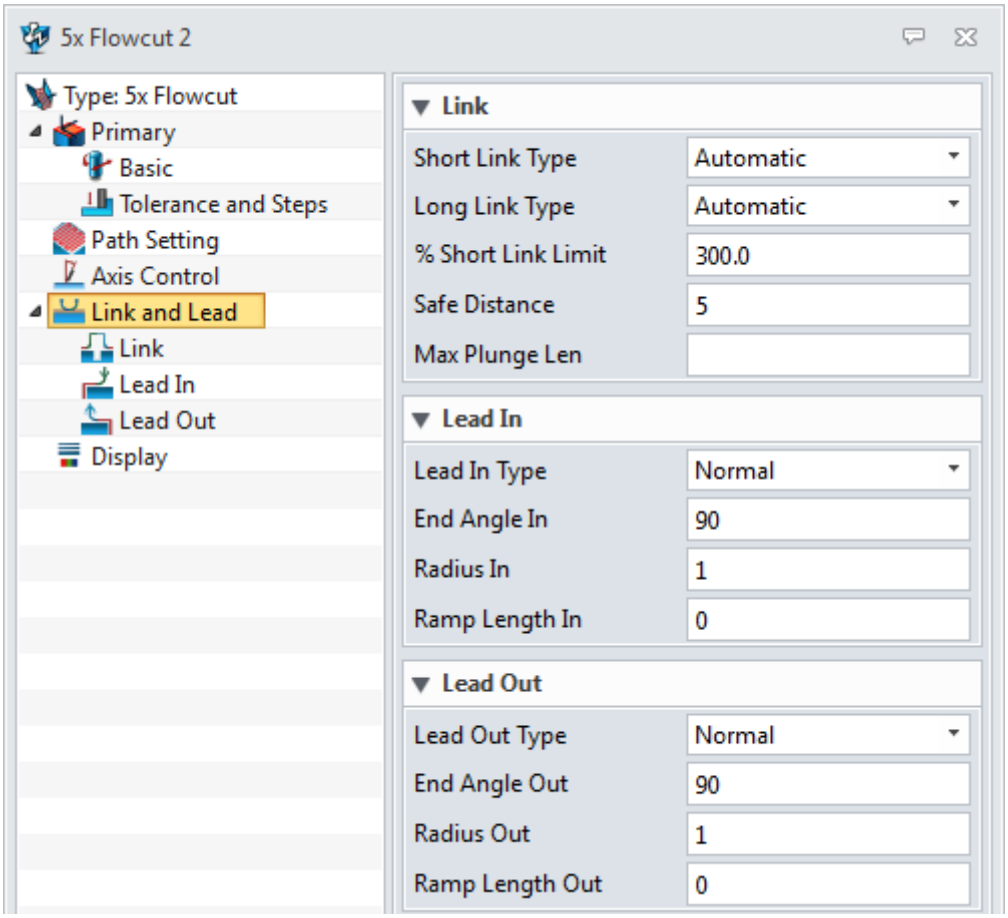


Figure166 Link and Lead parameters

III. Calculate and get the result as follows:

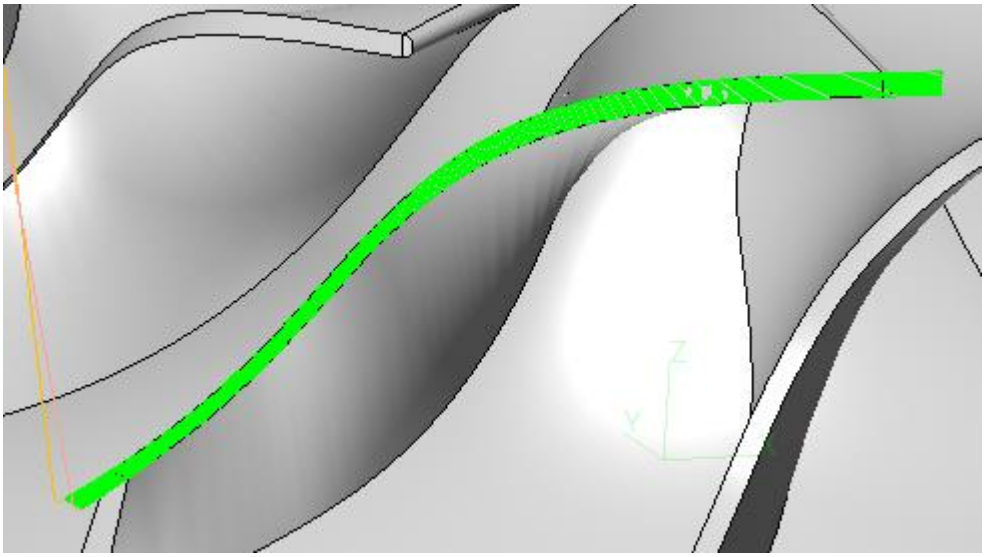


Figure167 5X Flow Cut Toolpath on end face of blade

Then we can use the transform function to pattern the toolpath, finally we can get the following result :

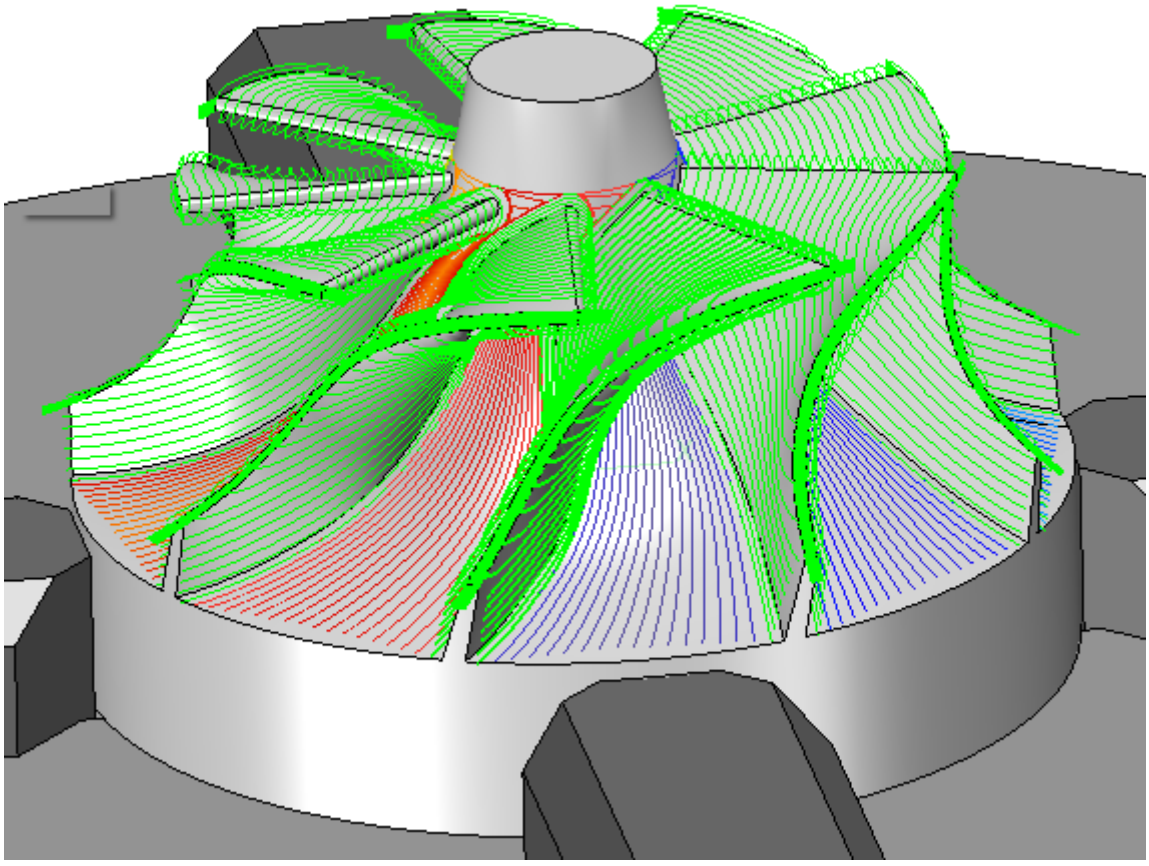


Figure168 Toolpath for the impelle

After finishing these steps please hide the auxiliary part and then save the file. Next let's take a look at the last operation of 5X, side cut.

1.3.6 5X Side Cut

Philosophy: The **5 axis Side Cut** operation accepts parts or general surface features as geometric inputs. Based upon different axis control options, it allows you to position the cutter in various orientations including normal or side tangent to the part with lead, roll and skew angles. This operation is a good choice for turbine top machining or complex pocket finishing with point control.

First we can take the "**5X_Impeller.Z3**" file as an example again to show how to use the side cut operation. Now we need to cut the top region, the final result will be as follows:

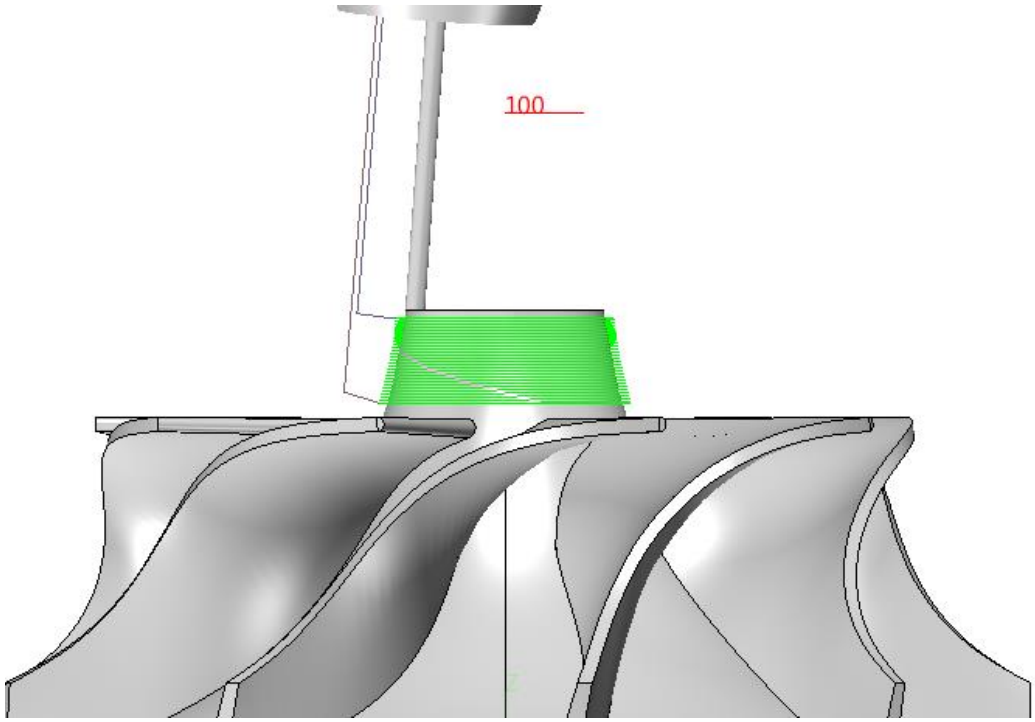


Figure169 5X side cut for the top region of impeller

Next let's create the toolpath step by step as follows:

STEP 01 Choose 5X Side Cut operation and define the general surface

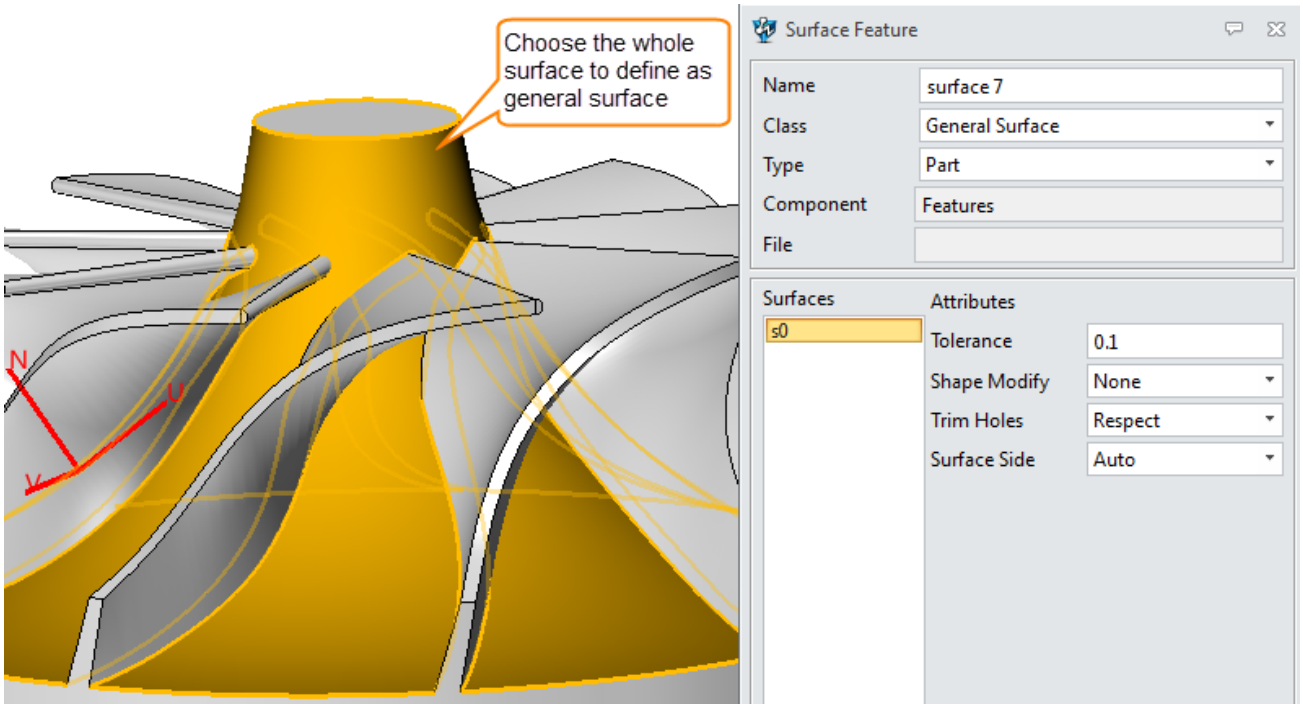


Figure170 Define general surface for 5X side cut

Normally a general surface is enough to create the toolpath of 5X side cut.

STEP 02 Next let's set up the parameters:

I. Primary parameters:

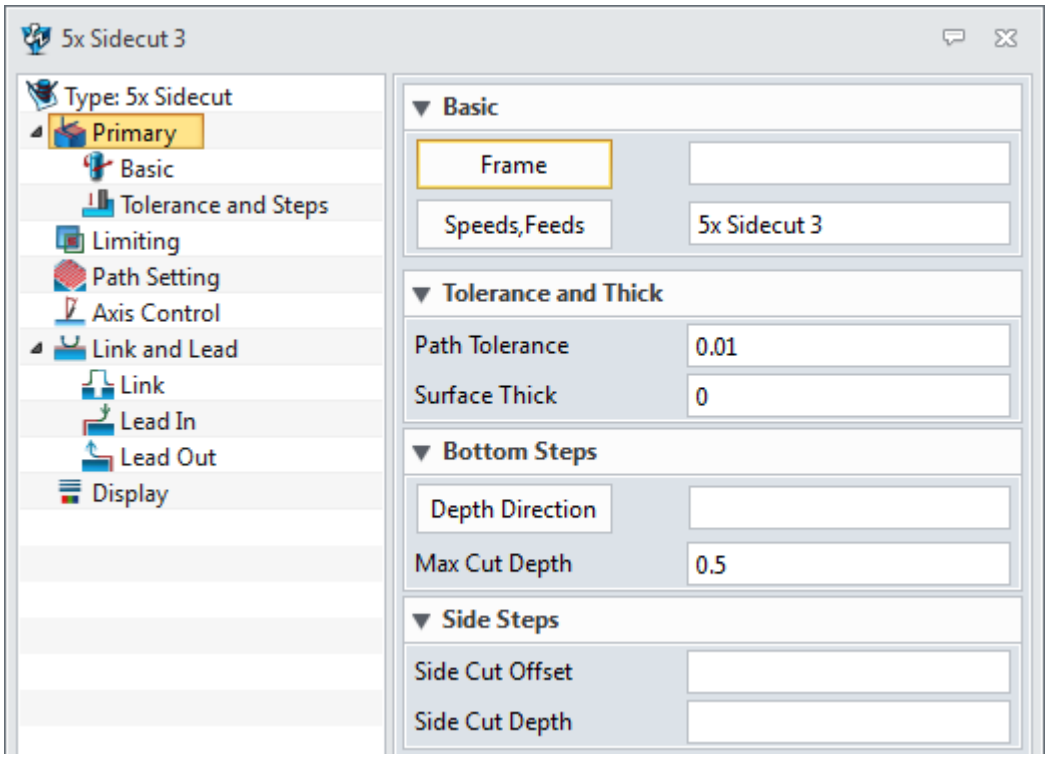


Figure171 Primary parameters

II. Limit parameters:

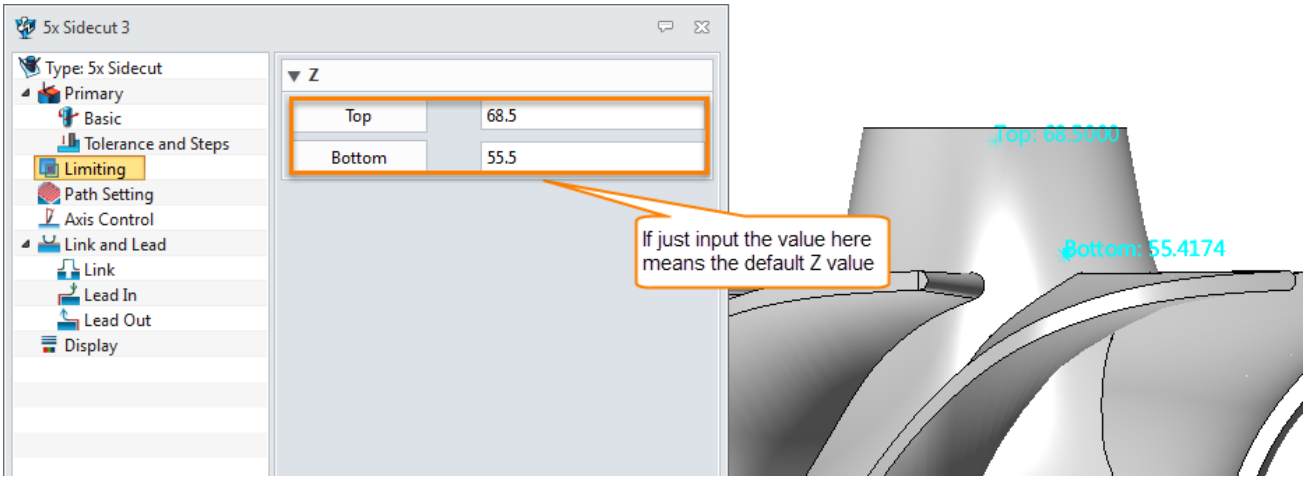


Figure172 Set up the top and bottom point

III. Path setting parameters:

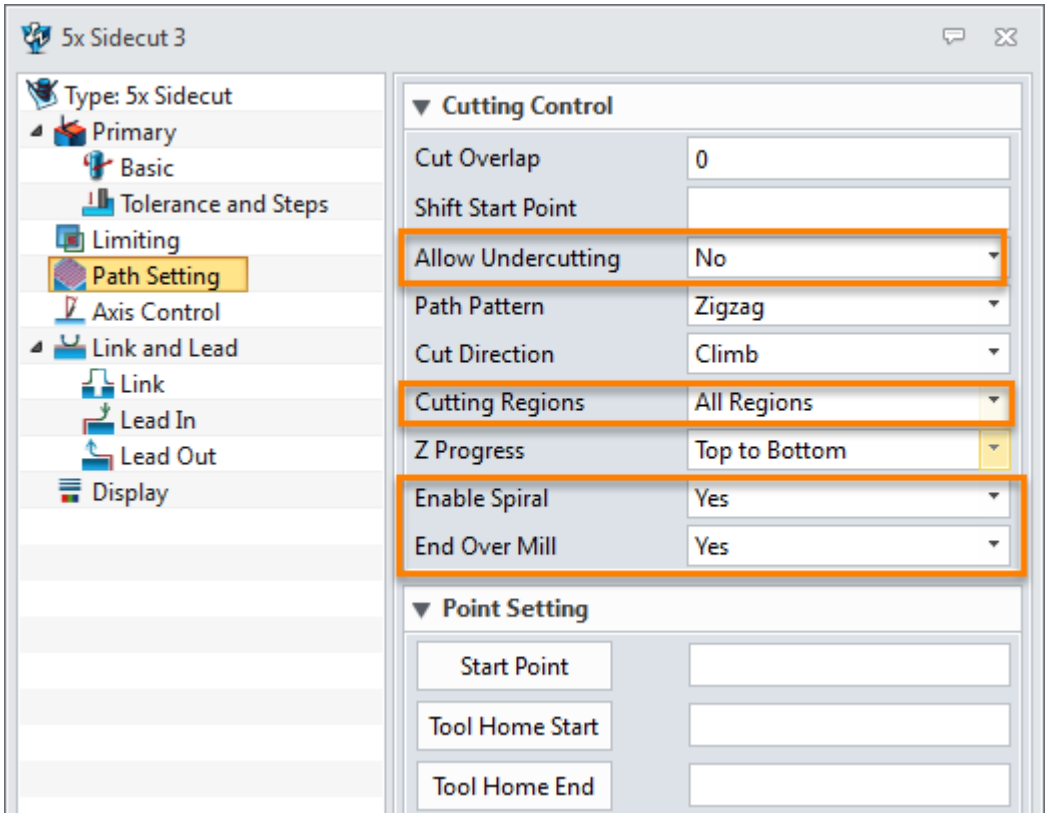


Figure173 Path setting parameters

- **Allow Undercutting:** If there is undercut need to cut, choose this option to allow tool axis to rotate
- **Cutting Regions:** Includes All Regions, Pockets Only, Outside Only,

- **All regions**: means all of the region in the target surface or part will be taken into account
- **Pockets Only**: Just cut the pocket, need to combinate with the option of "**Control Point**" in Axis control.
- **Outside Only**: just cut the outside of surface or part.
- **Enable Spiral**: If we allow the toolpath move as spiral pattern or not.

IV. Axis control parameters:

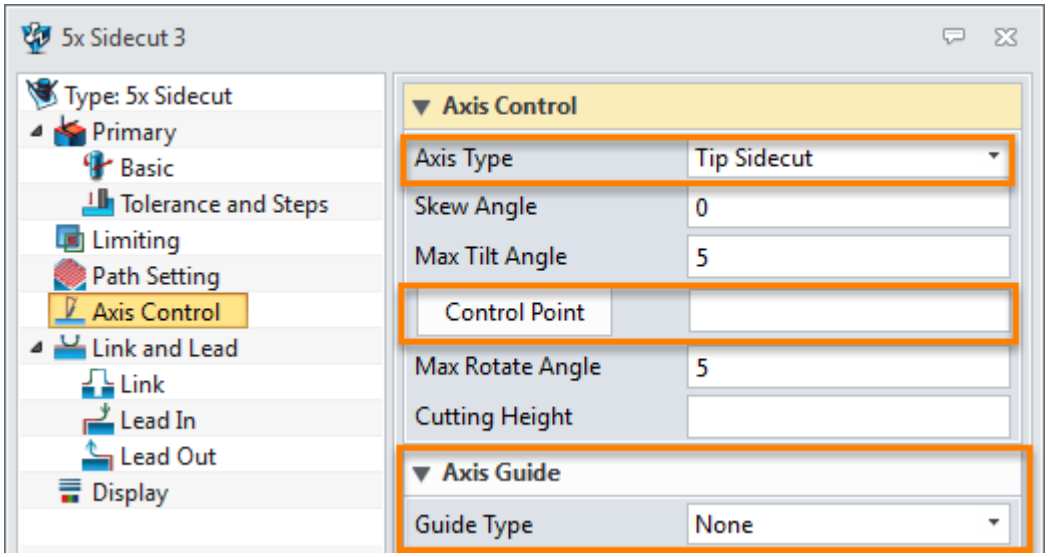


Figure174 Axis control parameters

- **Axis Type**: besides the introduced types in this operation there are 2 more different types :
  - **Tip SideCut**: Tool Tip determine the tool axis's orientation and then cutter side will be tangent to the part tangent plane.
  - **Contact SideCut**: Tool's contact point determine the tool axis's orientation and then cutter side will be tangent to the part tangent plane
- **Control Point**
  This parameter is used to **machine pockets**. If defined, the control point overwrites the Axis Control parameter so that the tool axis passes through the control point.
- **Axis Guide**

  In 5x sidecut, Axis Guide tries to define a few types of primitive surfaces as the guide surface by using the same concept as the guide surface does in 5x isocut. It is used mainly to machine pockets or porting (like driving curve).

For more details about the axis guide function we will use some cases to explain it in details. Here let's skip it first, and choose none.

STEP 03 For the rest parameters please refer to the previous introduction to finish by yourself and then calculate the toolpath as follows:

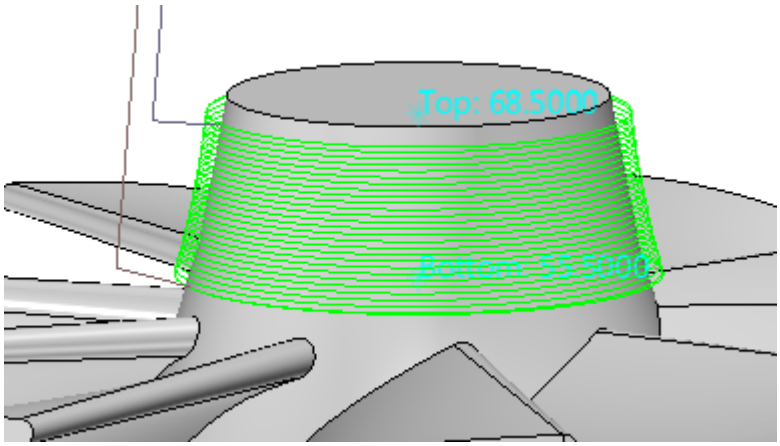


Figure175 5X side Cut toolpath on the top region of impeller

After saving the file. So far actually we have finished the toolpath for the impeller.

Next, we will take another example to show how to use the axis guide function to solve some problems:

1. **Statue Case**: Open the Statue Cut case file "**Statue_SideCut.Z3**" as follows:

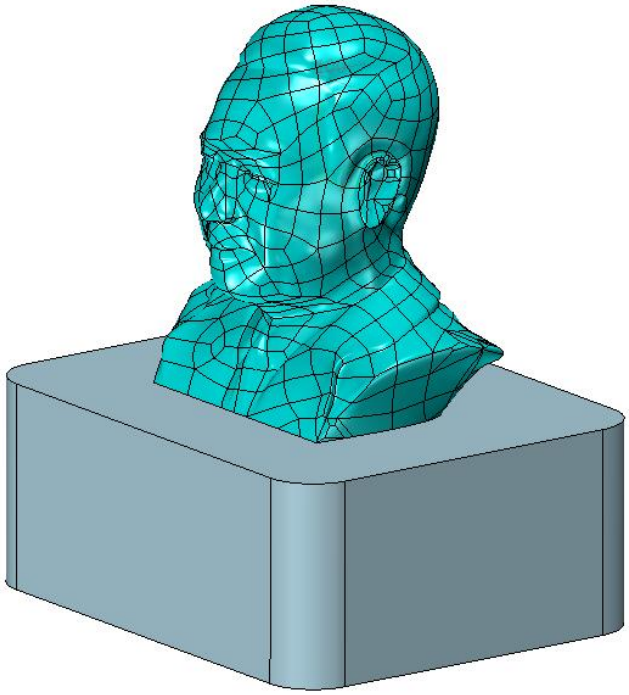


Figure176 Statue_SideCut case

STEP 01 Enter into CAM and create 5X Side Cut operation and then add the general surface as follows:

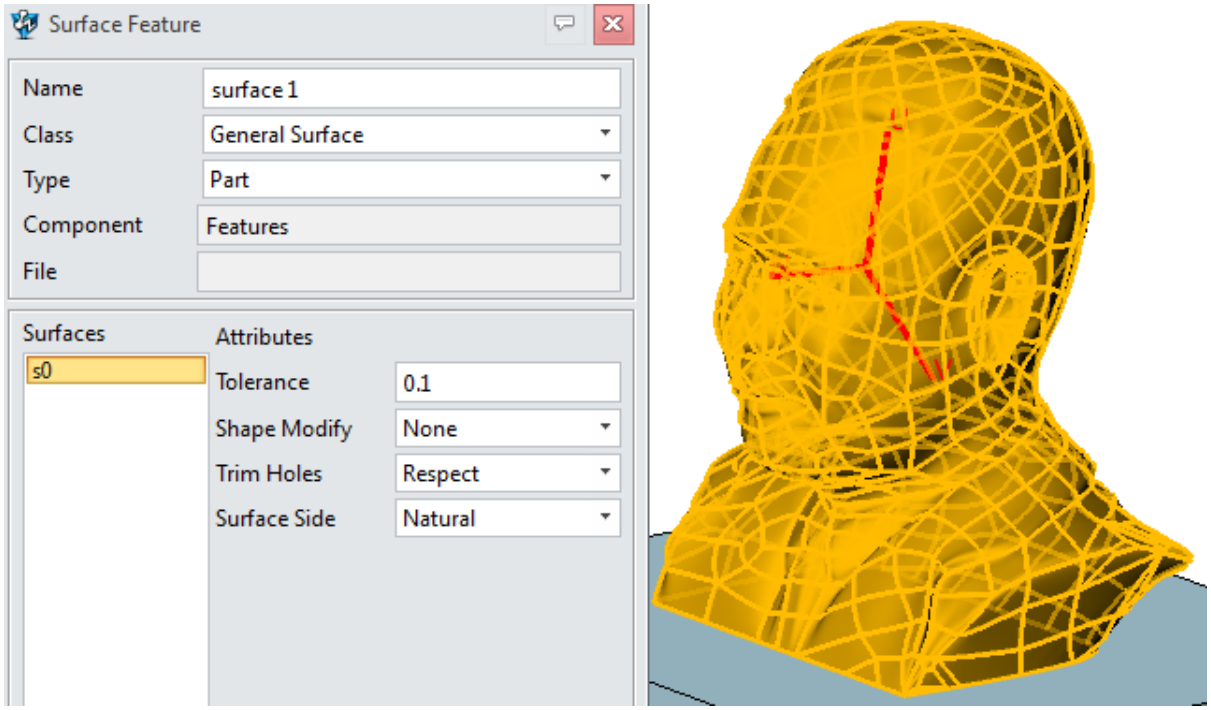


Figure177 define the statue part as general surface

In oder to let the tool stop at the bottom of statue we can define the bottom face as Start check surface as follows:

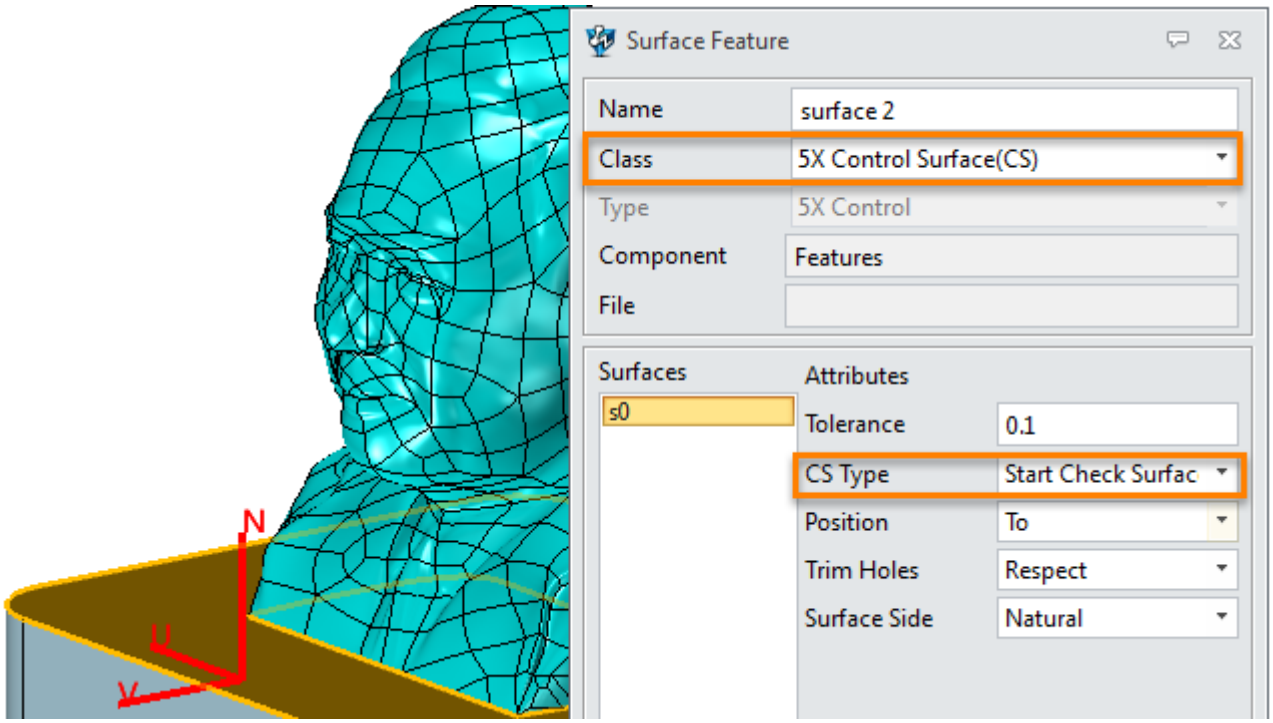


Figure178 Define Start Check Surfac

STEP 02 Set up the parameter according to the above introduction as follows:

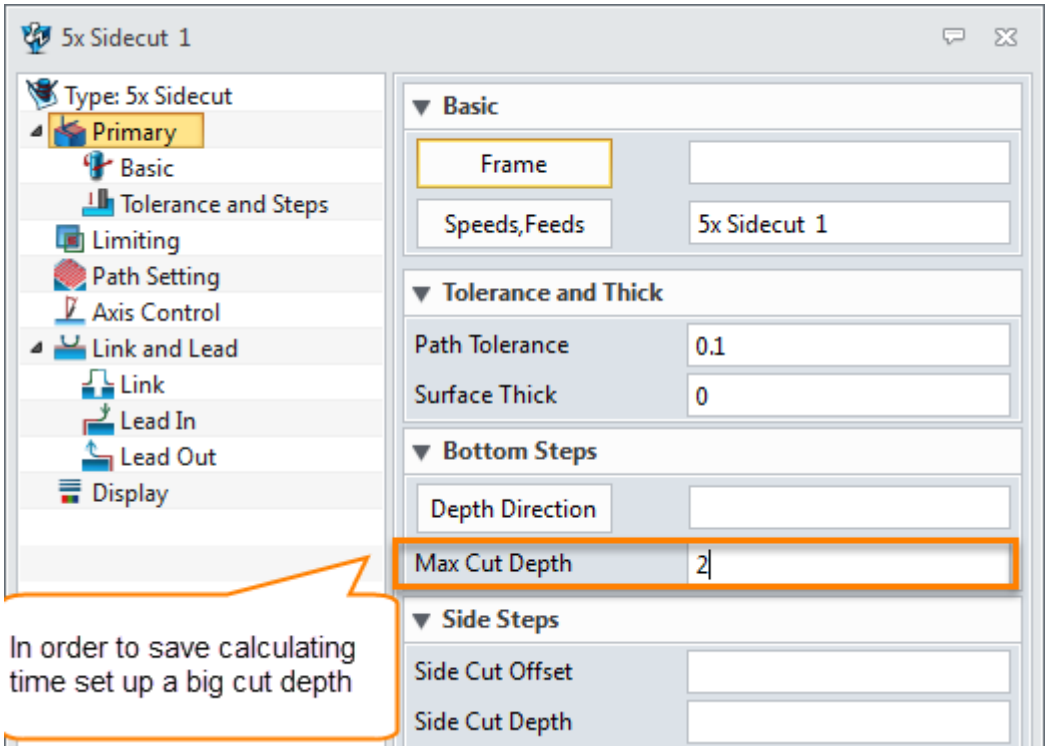


Figure179 Primary parameters for statue cut

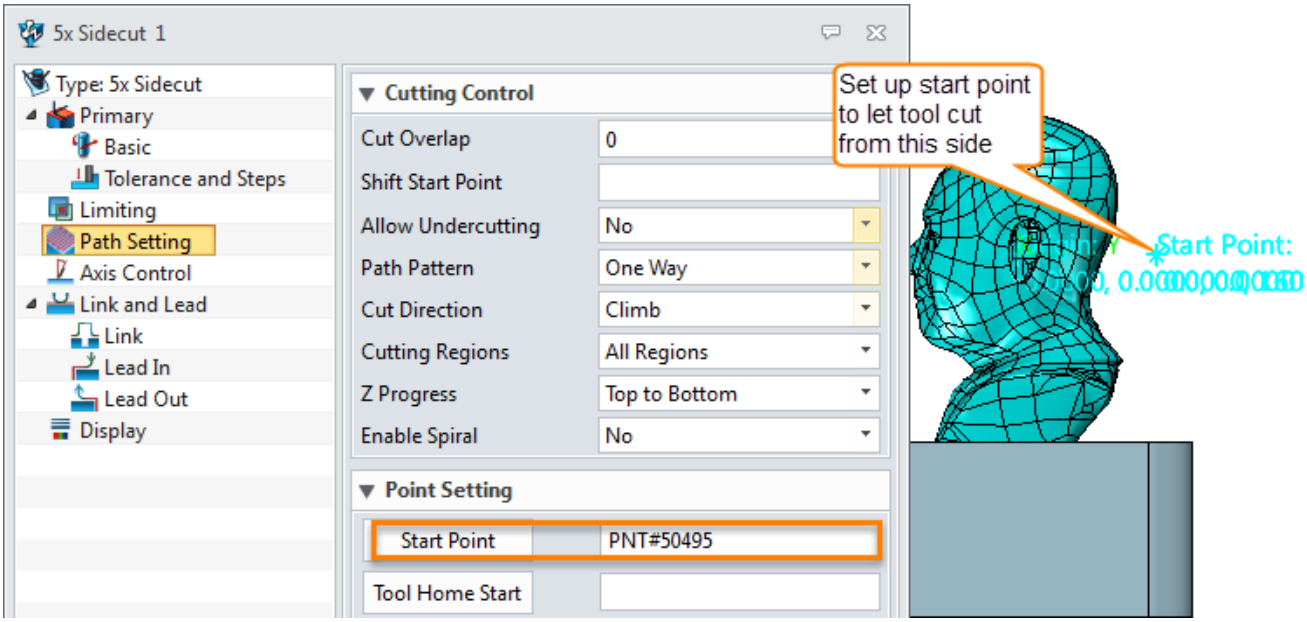


Figure180 Path setting parameters

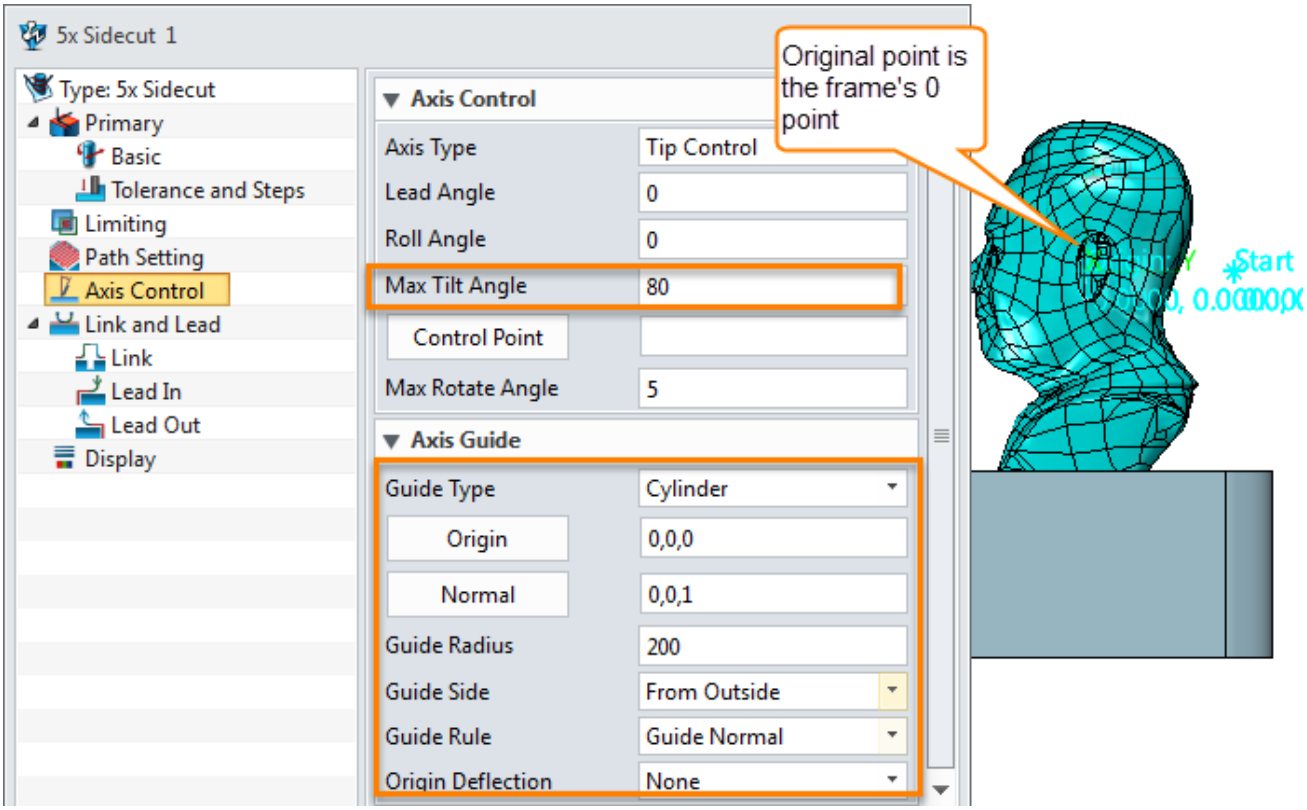


Figure181 Axis control parameters

Now let's stop here and calculate the toolpath to check what we will get. Following is the result:

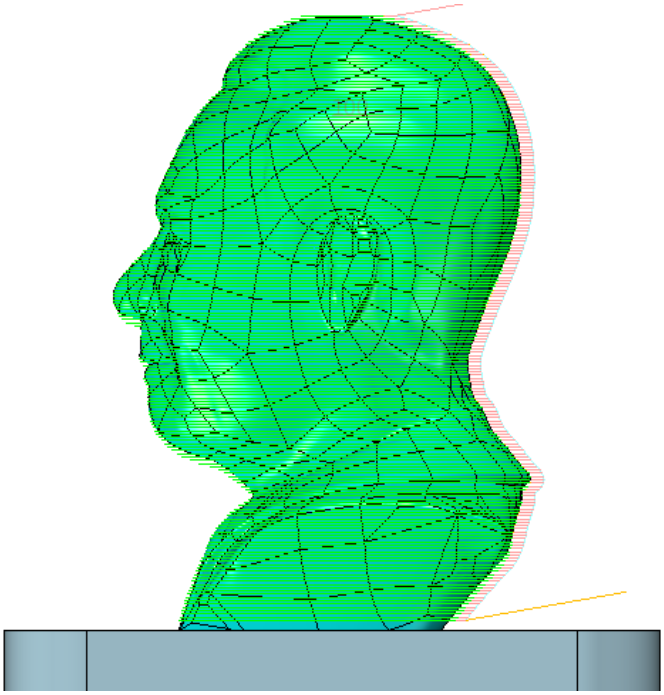


Figure182 5X Side Cut toolpath for the statue

Next let's make use of the created toolpath to explain axis guide parameters, as follows:

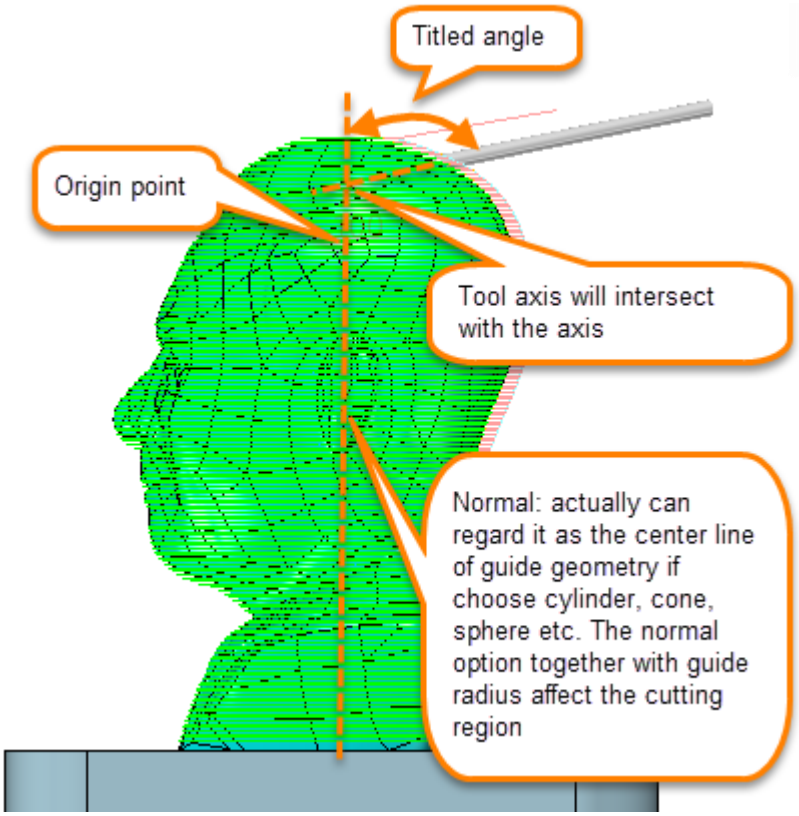


Figure183 Explanation for axis guide parameters

- **Guide type**: normally use the cylinder, cone, sphere and drive curve. For drive curve we will use another case to show how to use it.
- **Origin:** Can be regarded as the center point of section profile in XY as follows, the picked normal way will pass this origin.

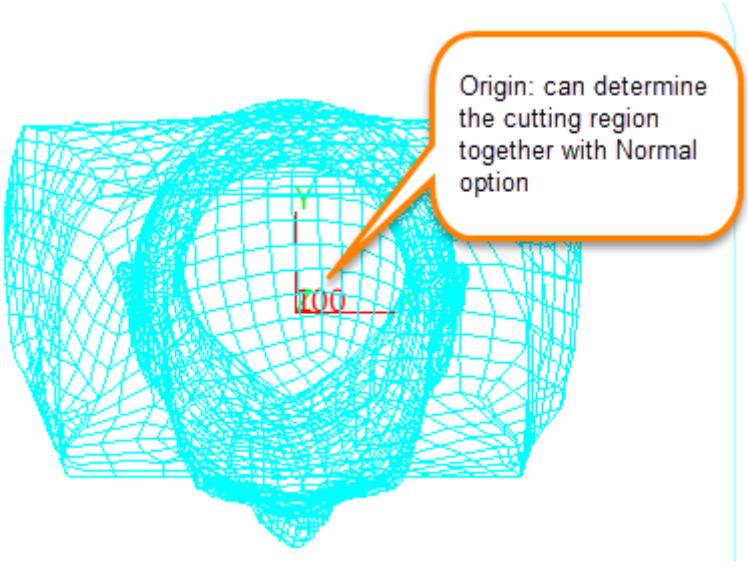


Figure184 Origin option

- **Normal**: if choose Cylinde, Cone, Sphere etc type, regard it as the center line of the guide geometry. which will affect the cutting region together with guide radius.
- **Guide Radius**: It is the section profile's radius of guide geometry which depends on the guide type.
- **Guide Side**: If cut the outside or Pocket.
- **Guide Rule**: if the tool axis guide by guide geometry normal or cutting part's normal.
- **Origin Deflection**: if deflect the origin to allow the tool to engage into the small corner.

Now that most of the pameters are illustrated, let's use another case to explain how to use the drive curve type guide and control point together to do the porting cut.

2. **Porting cut Case**: Open the file "**5X PortingCut.Z3**" as follows:

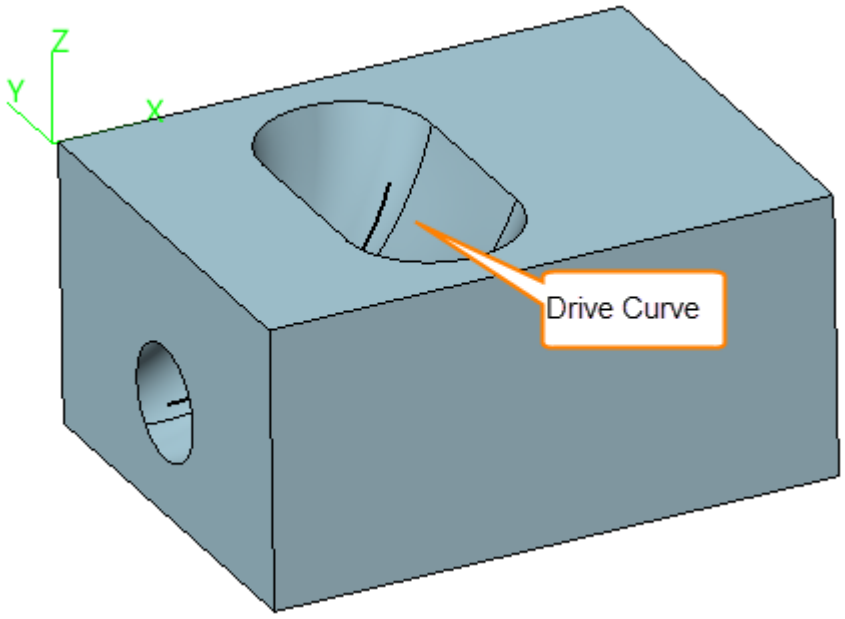


Figure185 5X porting cut case

In this file a drive curve had been created for reference. So in this case we will skip the jobs on how to create the drive curve. Next let's create toolpath for the port shape step by step:

STEP 01 Define profile for drive curve:

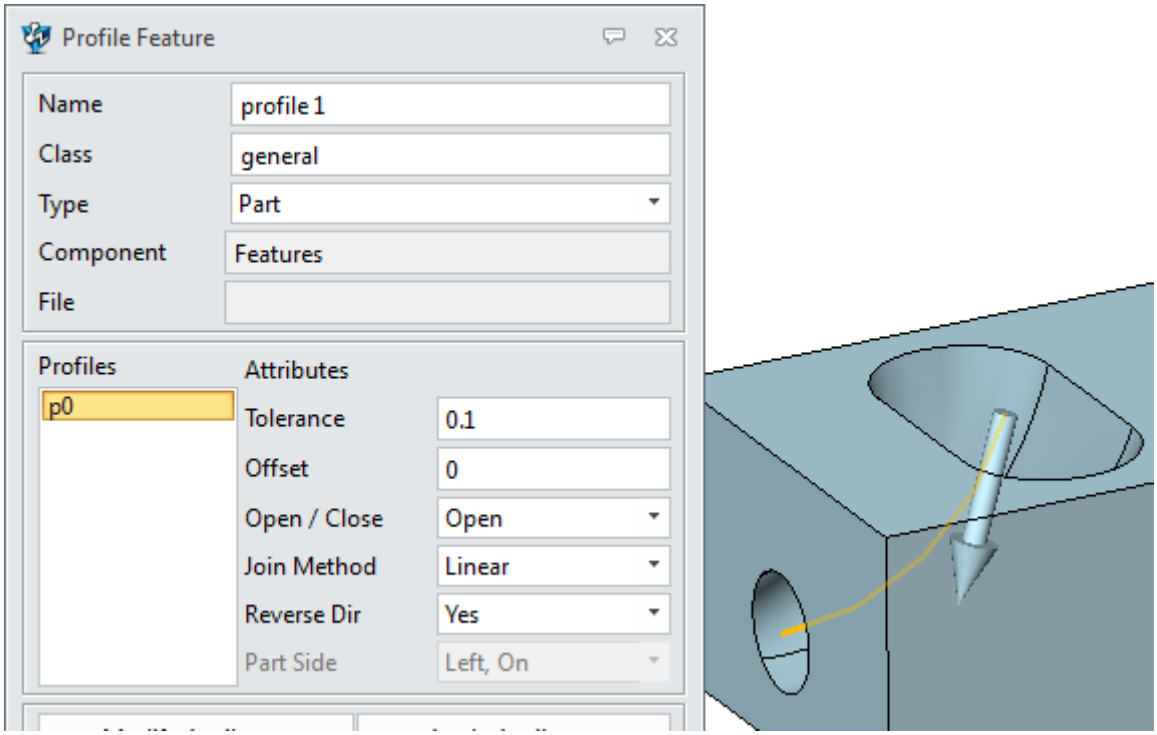


Figure186 Define drive curve by profile

STEP 02 Define cutting surface as general surface:

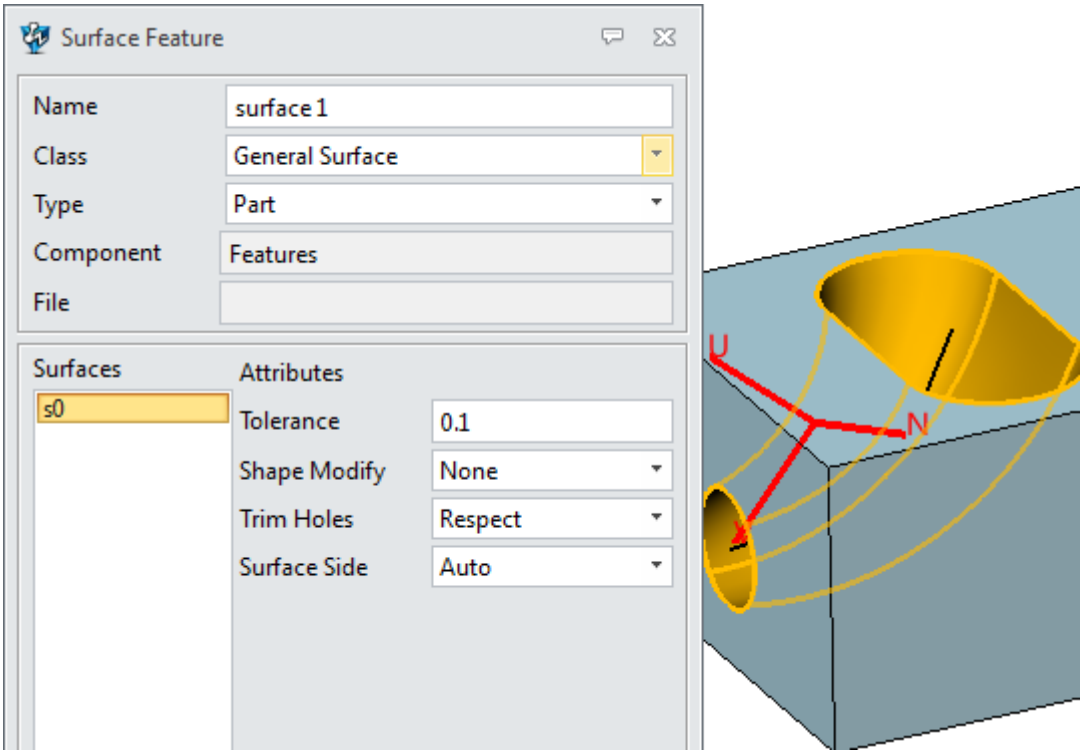


Figure187 Define general surface

Note: it is important to make sure the surface's direction is positive.

STEP 03 Define a lollipop tool as follows:

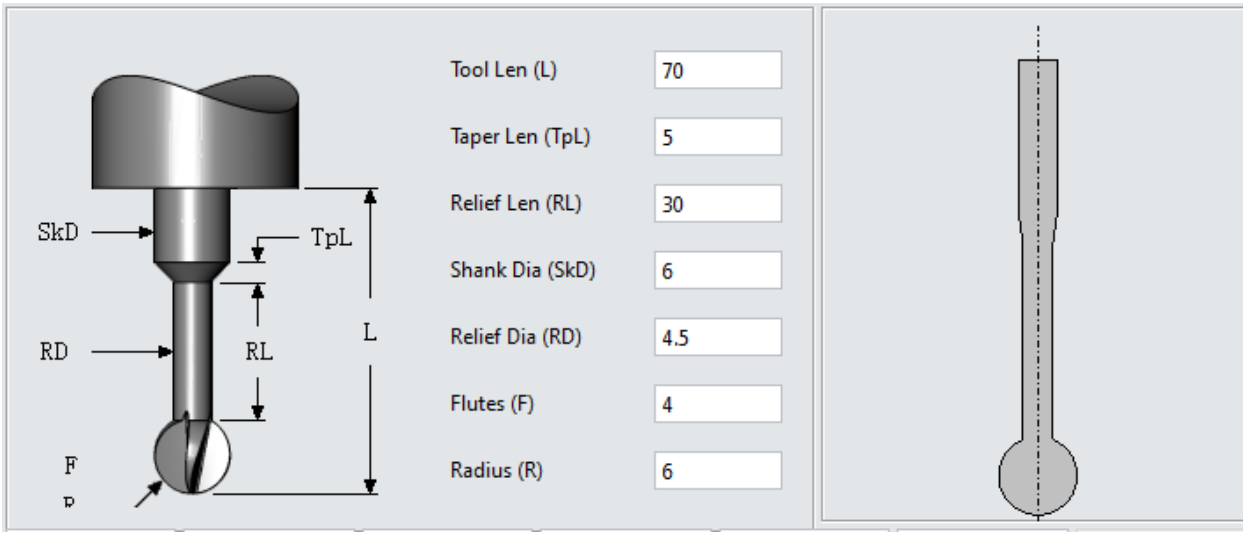


Figure188 Define lollipop tool

STEP 04 Set up parameters:

Here we will skip the Primary parameters and Path setting parameter and just focus on axis control parameter as follows:

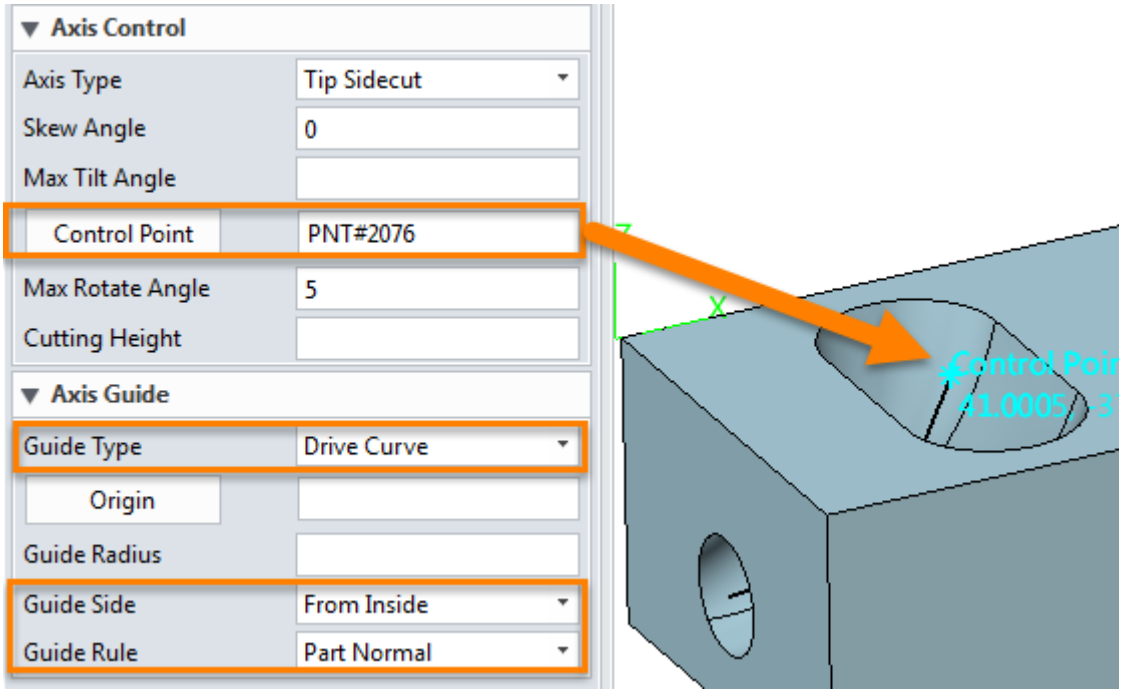


Figure189 Set up axis control parameters

Because here used the drive curve as the guide geometry: the tool axis will be guided by the drive curve's tangent way, so the guide radius makes no sense for this situation, just leave it blank.

STEP 05 After finishing setting up parameter, calculate the operation Toolpath as follows.

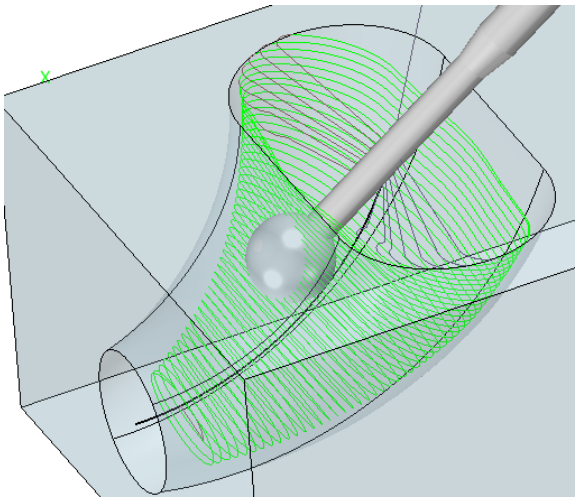


Figure190 Porting cut toolpath by 5X Side Cut operation

Summary:

We have introduced how to achieve the 5X indexing milling by sub frame and Interpath function. And with the practical examples, we have introduced how to run 5X simultaneous movement operations and apply these operations to the real case.

Notice:

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

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

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

ZW3D Support Team: zw3d@zwsoft.com