ZW3D from Entry to Master Tutorial

2X Machining

ZW3D™ V2023 CAM from Entry to Master 2X 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 CAM From Entry to Master 2X Machining***, a master tutorial.

Thanks for being our user!

The ZW3D Team

Contents

1 Introduction

Key Points:

- ✧ General Workflow to Create 2X Toolpaths
- ✧ CAM UI and Configuration

1.1 Getting Started

ZW3D is an integrated CAD/CAM solution, which provides various tools covering the whole process of product development. This enables users a more productive workflow from digital threads to shop floor operations.

***Notes**: This tutorial is based on version ZW3D 2020, some functions or icons may not match the current version.*

1.1.1 General Steps to Create 2X Machining Toolpaths

The general workflow to create 2X toolpaths is as follows:

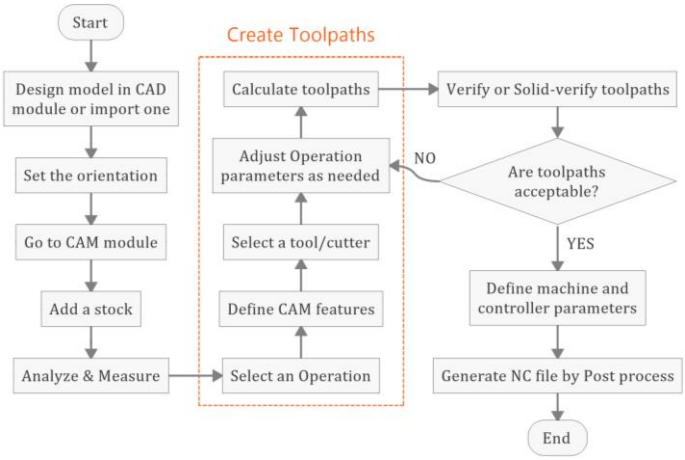


Figure 1 General Steps to Create 2X Machining Toolpaths

STEP 01 Design model in CAD module or import an external geometry.

STEP 02 Align the model to a proper direction for machining.

STEP 03 Go to CAM module.

STEP 04 Add a stock.

STEP 05 Analyze and measure the key features of the model to help choose a correct strategy.

STEP 06 Select an operation, define CAM features, pick a tool and then calculate toolpaths with suitable parameters.

STEP 07 Define machine and controller parameters.

STEP 08 Post process NC code to file.

1.1.2 Open the Part File

A part used in CAM module can be a solid or surface or both mixed together. Users can create a part with CAD module or import one from another 3D modeling system like NX, SolidWorks, CATIA, etc.

In this chapter, we'll take an existing ZW3D part as an example, which is located in the ZW3D installation folder. Click ***Open*** button or use ***Ctrl+O*** to open the file ZW3D 2X Machining.Z3, as shown in Figure 2.

The typical location is: C:\Program Files\ZWSOFT\ZW3D 201X Eng (x64)\training\2X machining model\ZW3D 2X Machining.Z3.

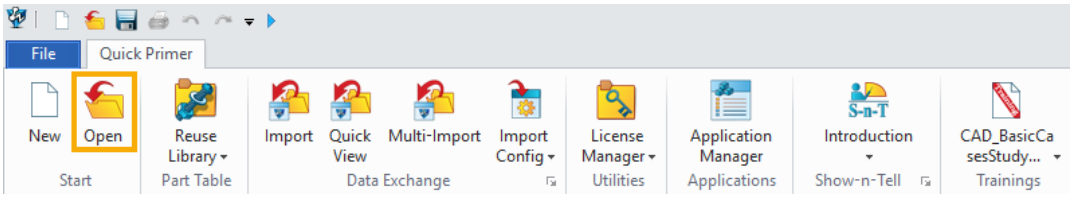


Figure 2 Open a Part

1.1.3 Access to CAM Module

STEP 01 Click ***CAM Plan*** button in Document Aware Toolbar. Or right-click on the blank area of Graphic Window and choose ***CAM Plan***.

STEP 02 Select a template, and press ***OK***.

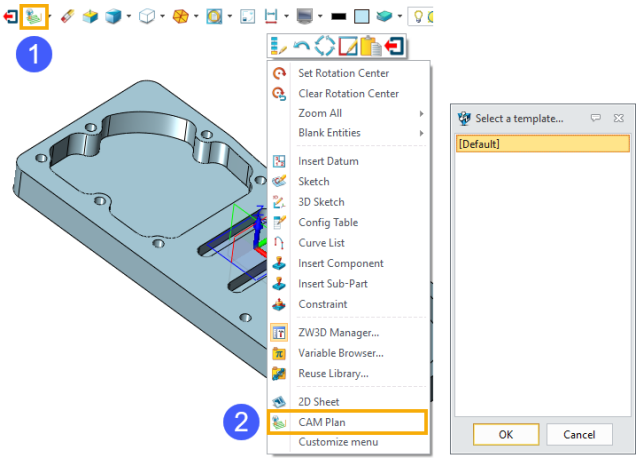


Figure 3 Access to CAM Module

1.1.4 Back to CAD Level

Click the ***Exit*** button in Document Aware Toolbar, or right-click on the blank area of Graphic Window and choose ***Exit*** to get back to CAD level

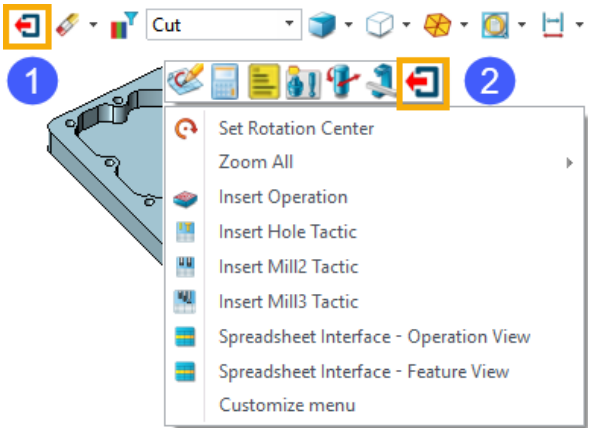


Figure 4 Back to CAD Level

1.2 CAM Interface

1.2.1 Layout

The following figure shows the typical layout of ZW3D CAM window when a file is opened.

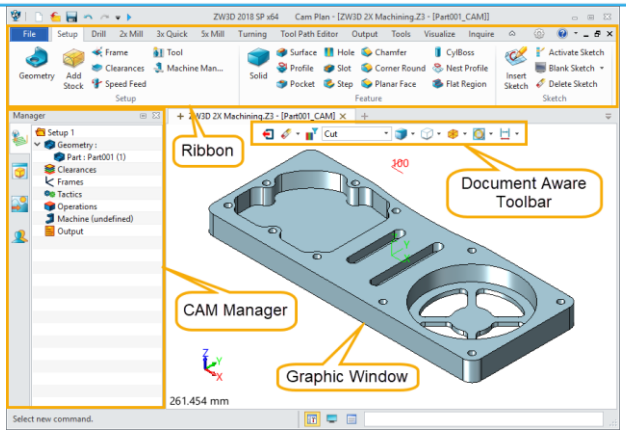


Figure 5 Interface Layout

Ribbon: All the functions in need to create toolpath are included in Ribbon Bar. Users can infer the function of every button from its name. And a detail introduction of Ribbons used in 2X machining will be provided later.

CAM Manager: Users can control and review the whole machining workflow here. It will be shown in detail in the next chapter.

Document Aware Toolbar: There are many useful tools in this area, like ***Exit***, picking filter, display settings, etc.

Graphic Window: This is the place where the geometry and toolpaths are shown.

1.2.2 Customize Ribbon

Users can also right-click on the blank area of Ribbon to customize the most interface. As the figure shows below, users can rearrange the Ribbon locations, modify ***Hotkey*** settings, or redefine mouse actions.

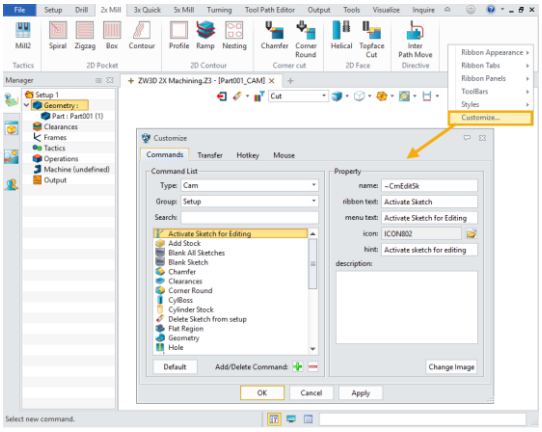
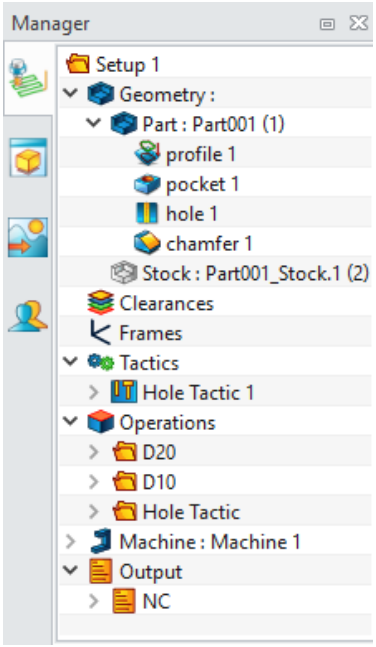


Figure 6 Customize Ribbon

1.2.3 CAM Manager

CAM Manager is the most useful place when you are trying to generate toolpaths for a part. As a beginner, you'd better follow the steps list in CAM Manager from top to down. The figure below shows the main structure of the CAM Manager.



Geometry: All the geometry and CAM features are listed here, including part, stock and other objects. Users can select, add/delete, show/hide, or set attributes and features to them.

Clearances: Used to define tool clearance distance.

Frames: Insert and manage local Frames, which can be used as the programming coordinate system.

Tactics: You can add hole tactics or milling tactics by right-click here.

Operations: You can manage all Operations in this area, including setting parameters, select tools, calculate toolpaths, and verify the results.

Machine: Define post Configuration

Output: NC Code output control

Figure 7 CAM Manager

1.2.4 Customize CAM Manager

Users can right-click on the icon ***Setup*** or any text in CAM Manager to customize it.

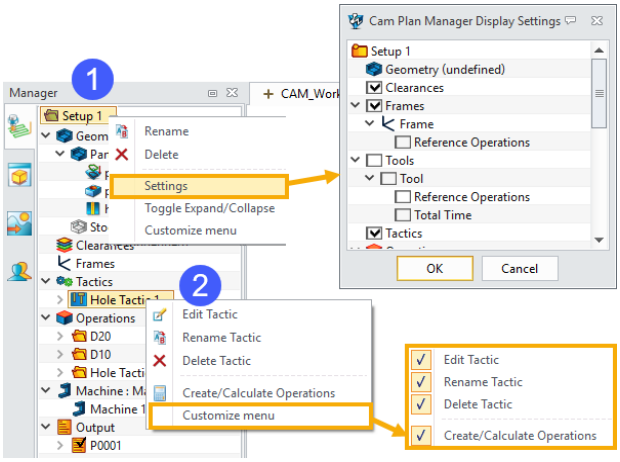


Figure 8 Customize Manager

Double-click on the right place

Double-click on the text of the ***Manager Tree*** or the small icon in front of it is different in results. Double-click on the icon will hide or display the toolpath, but on the text will bring you to Parameters window. Right-click on the text will open the context menu.

1.2.5 Configuration

Click the gear button on the right corner of top to find ***Configuration*** dialog box. These options customize how ZW3D works and interacts to the specifications you set.

You can control the file management strategies, back-up interval, memory size for Undo/Redo. You can set the default way to display geometry. You can also define CAM options (including Tool Library, default floors, output folder etc.)

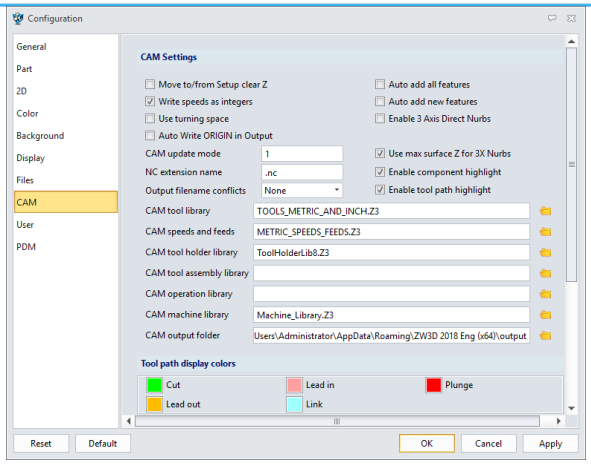


Figure 9 Configuration Dialog

General: mainly used for file management and some general options, including single/multiple object file, back-up, session management etc.

Part: set options for CAD module

2D: Set options for 2D Drawings.

Color: Most colors are set here.

Background: Set background style.

Display: Customize numeric settings and toggle settings, and default display style

Files: Set all kinds of file folder locations.

CAM: Define some unique options, strategies and colors used in CAM

User: Define some simple user information

PDM: Set PDM options.

What do these options mean in CAM Configuration dialog?

Here is a list of common 2X settings in CAM Configuration that you'd better know. Please press F1 to find more information.

Move to/from setup clear z	When checked, each operation will start and end at the setup's ***Clear Z*** position. OR each operation will start and end at the ***Frame*** or local ***Clear Z*** position.
Write speeds as integers	When checked, spindle speed values are integers in the CL file. Otherwise, they'll have numbers after a decimal place.
Using turning space	When checked, the coordinate space in CAM changes to Turning (XZ coordinate).
Auto add all features	When checked, ZW3D will automatically add part and stock features to newly created operations.
Auto add new features	When checked, ZW3D will automatically add newly created CAM features to all existing toolpath operations

Auto write ORIGIN in output	When checked, ZW3D will automatically write the origin coordinate value (in global CS) of local Frame which is used to generate toolpaths into the output file.

2 2X Milling

Key Points:

- ✧ Prepare Part and Stock
- ✧ Create 2X Milling Operations
- ✧ Generate & Verify Toolpaths
- ✧ Output NC Files

2.1 Getting Started

You need a part, a stock and a rough analysis to help you get a preliminary machining plan before you start to create toolpaths.

This chapter uses the same model as previous.

2.1.1 Get Part Ready

STEP 01 Open the part file in CAD Module.

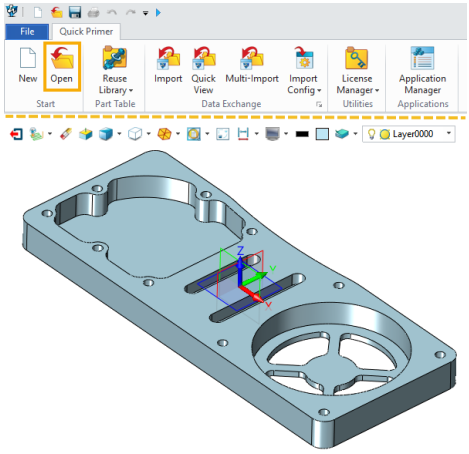


Figure 10 Open a Part in CAD Module

STEP 02 Align the model with a proper coordinate system according to your machine. Normally the origin locates on one of the corners or the graphic center of the model. Users can do this with the help of the ***Move*** command. In this case, the coordinate is already set to right place.

2.1.2 Add a Stock

After the part is ready, the first thing after going into the CAM module is to add a stock. ZW3D provides 2 methods to create a stock.

Method1: Click on Ribbon ***Add a stock*** to create a regular stock according to the geometry size of the part, as the figure shows below.

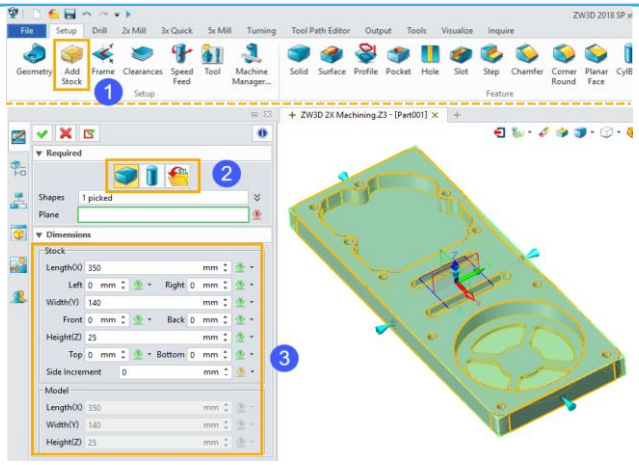


Figure 11 Create a Stock with Ribbon

STEP 01 Click on ***Add a Stock***.

STEP 02 Select a form for the stock. It can be a block, cylinder or external STL shape.

Shapes: Select the geometry to enclose.

Plane: Select a reference plane to orient the stock. If blank, the reference plane is parallel to XY.

STEP 03 Define the dimensions of the stock.

Method2: Insert a geometry as a stock. The geometry may come from another object in the same Z3 file or an external file.

STEP 01 Click on Ribbon ***Geometry*** to call out ***Shape Brower*** window, then select the geometry you want to insert as a stock. Or you can also right-click on ***Geometry*** text in CAM Manager tree to get access to the same place.

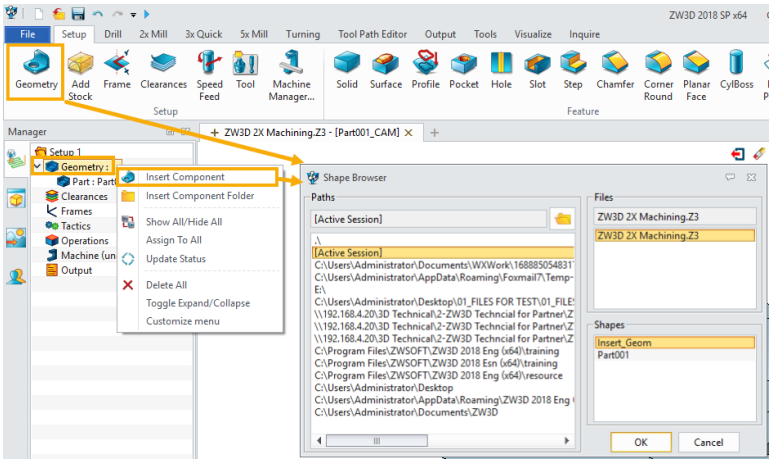


Figure 12 Create a Stock by Inserting a Geometry

STEP 02 Set the inserted geometry as a stock.

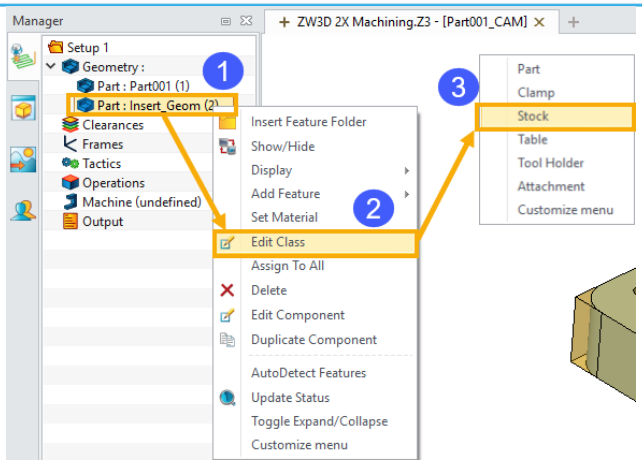


Figure 13 Set the Inserted Geometry as A Stock

2.1.3 Analyze and Measure

It is important to users to do a preliminary analysis and measurement on the part, including part material, the corner radius, hole diameters, key features and other parameters which may help decide the cutter or operation plan. The tools you may refer to are shown below, you can also do this step in CAD Level.

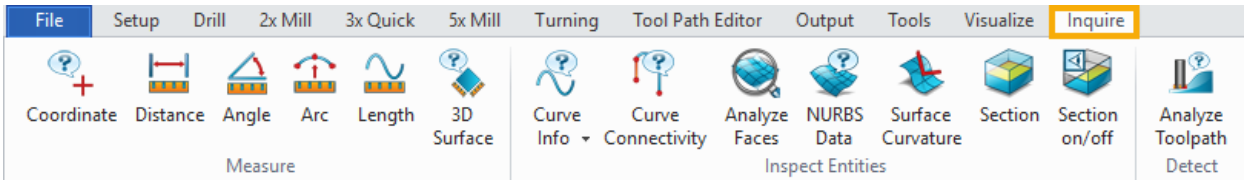


Figure 14 Useful Tools for Analysis and Measurement

Here is a list to help you understand how to use the tools in the ***Inquire*** tab.

Coordinate: Inquire the coordinate value of the picked point

Distance/Angle/Arc/Length: Inquire the Distance/Angle/Arc/Length information

3D Surface: Inquire the area of an enclosed planar 3D or 2D region

Curve Info/Curve Connectivity: Inquire curve information

Analyze Faces: Analyze surface imperfections, draft angles, curvature, and other surface characteristics

NURBS Data: Inquire the NURBS data of a face or free surface

Surface Curvature: Inquire the property of a surface, including the area, boundary length and maximum curvature of the surface.

Section: Create a 3D section view

Section on/off: Turn on/off section view

Analyze Toolpath: Check if any collision or other error occurs in toolpath

2.2 Create Toolpaths

2.2.1 Overview of 2X Milling Operations

ZW3D provides various types of operations, this tutorial will take a brief overview of all the options one by one.

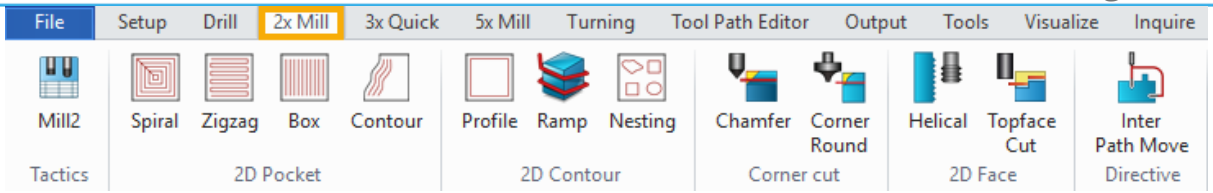


Figure 15 Operations for 2X Milling

Mill2: With this command, users can create a series of “intelligent” rule-based 2X operations for the whole part by one click. ZW3D names such function as “Tactics”. Similar ones can also be found in 3X, 5X and Drill module.

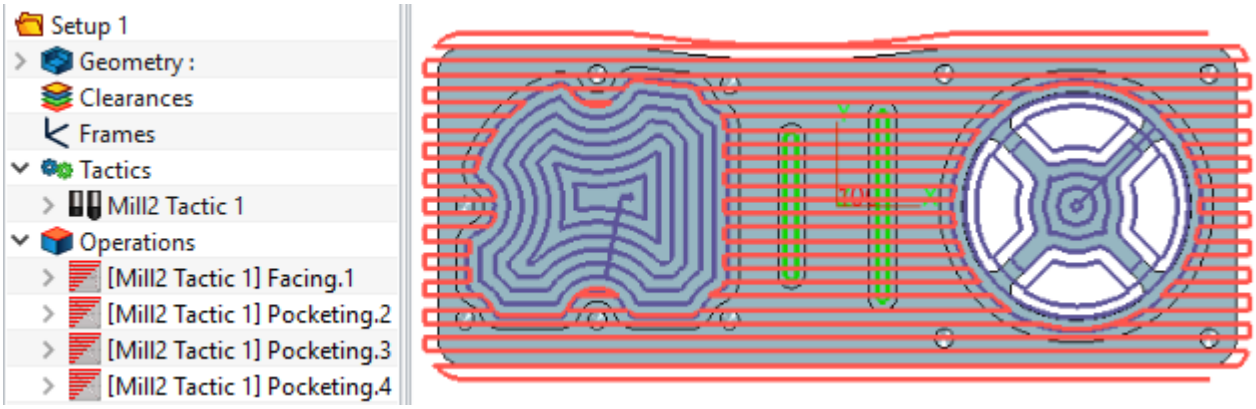


Figure 16 Mill2 Tactic Toolpaths

Spiral: This operation is a facing or pocketing technique, which advances the tool at each depth by proceeding toward or away from the part boundaries.

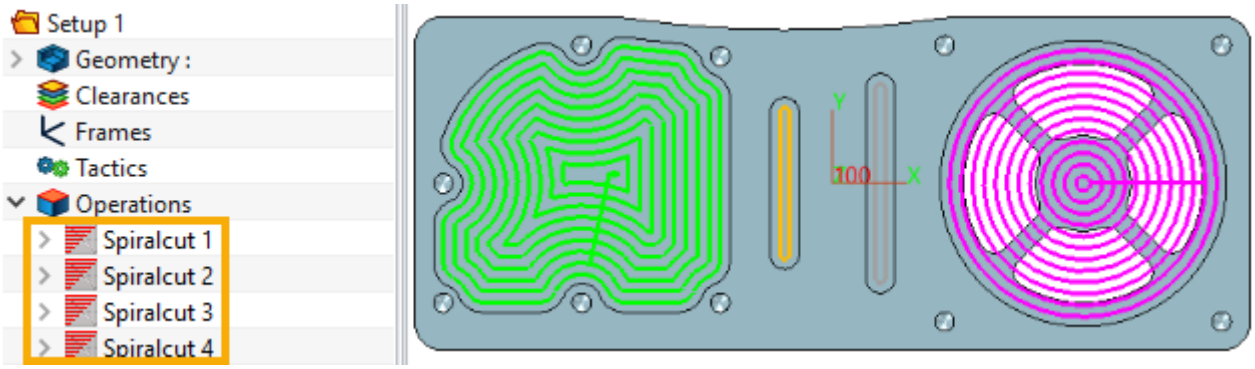


Figure 17 Spiral Operation Toolpaths

Zigzag: This operation is a facing or pocketing technique, which advances the tool at each depth through a sequence of straight parallel cuts, reversing direction at the end of each cut.

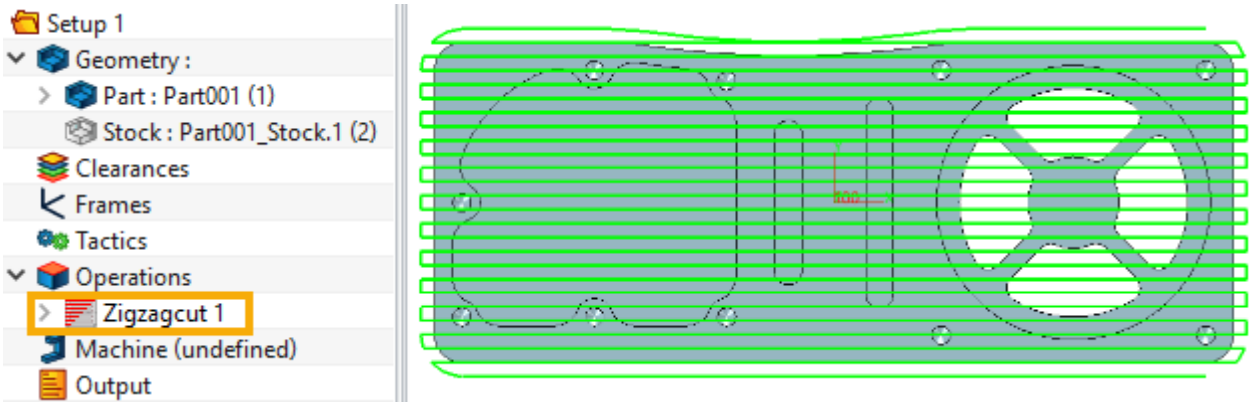


Figure 18 Zigzag Operation Toolpaths

Box: This operation is a facing or pocketing technique, similar to the zigzag cut except the cuts are all in the same direction. The tool is lifted between each cut.

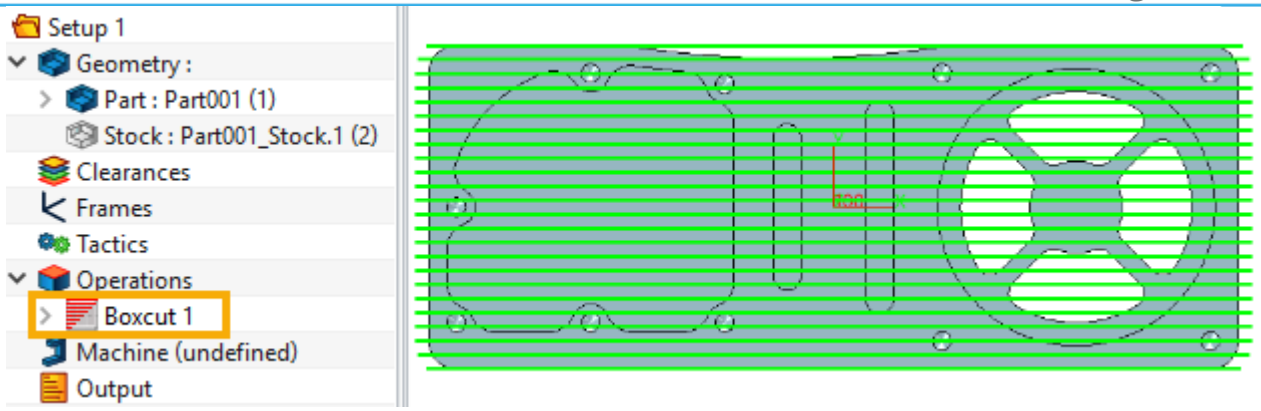


Figure 19 Box Operation Toolpaths

Countour: This operation will generate a toolpath parallel or perpendicular to the reference curve (medial or spine) for each cutting zone.

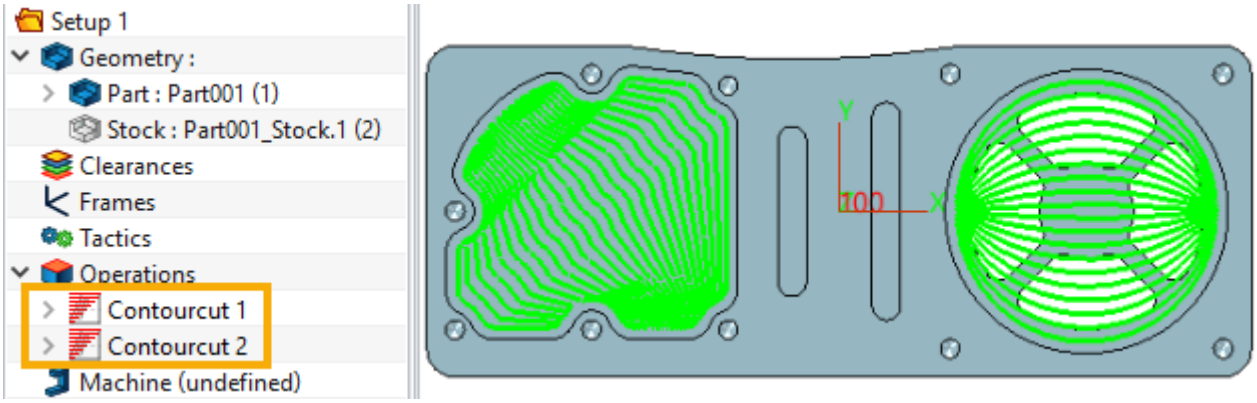


Figure 20 Contour Operation Toolpaths

Profile: This operation cuts any number of open or closed curve boundaries (CAM Profile features) or CAM Components containing geometry profiles.

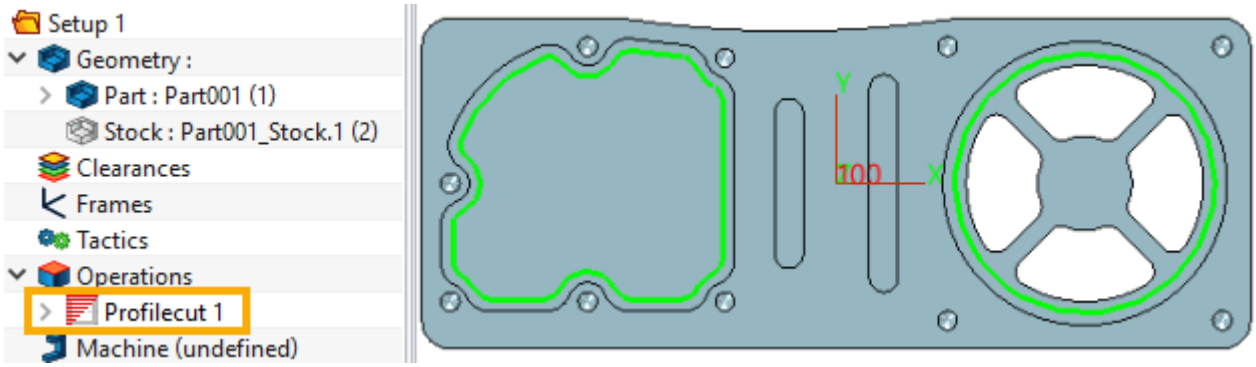


Figure 21 Profile Operation Toolpaths

Ramp: The toolpath created by the Ramp Cut operation is similar to the toolpath created by ***Helical Cut*** operation for a hole feature.

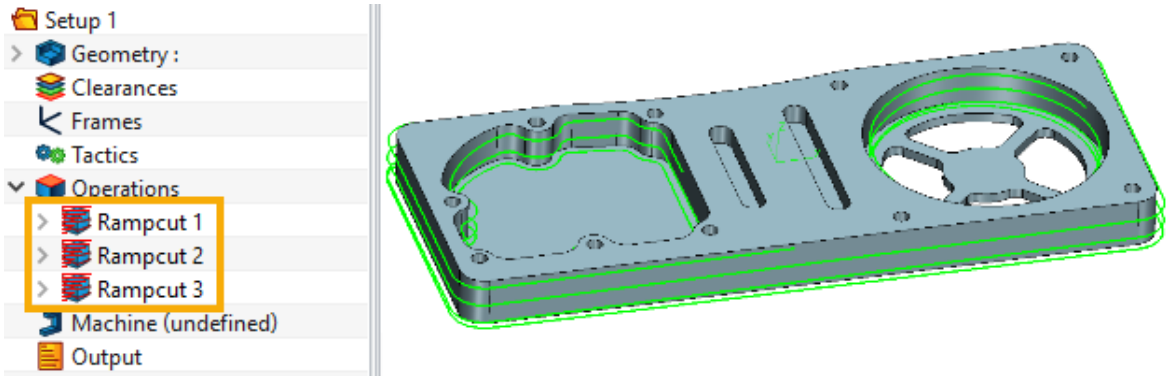


Figure 22 Ramp Operation Toolpaths

Nesting: This operation is used to arrange parts on a plate of material to maximize the number of parts that will fit.

Chamfer/Corner Round: This operation is used to cut chamfer or fillet.

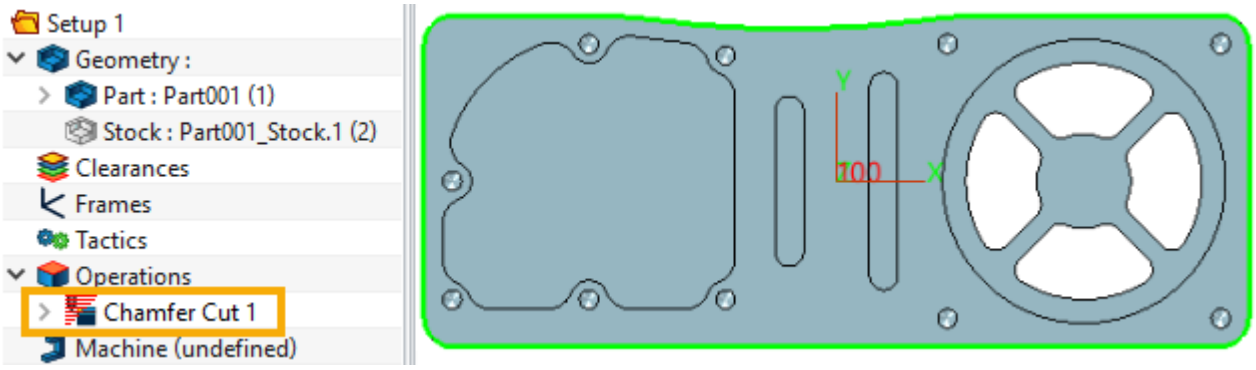


Figure 23 Chamfer Operation Toolpaths (Corner Round is similar)

Helical: This operation is used to machine male and female threads.

Topface Cut: This operation is used to get a planar face on the top of the whole part or some planar faces of the part.

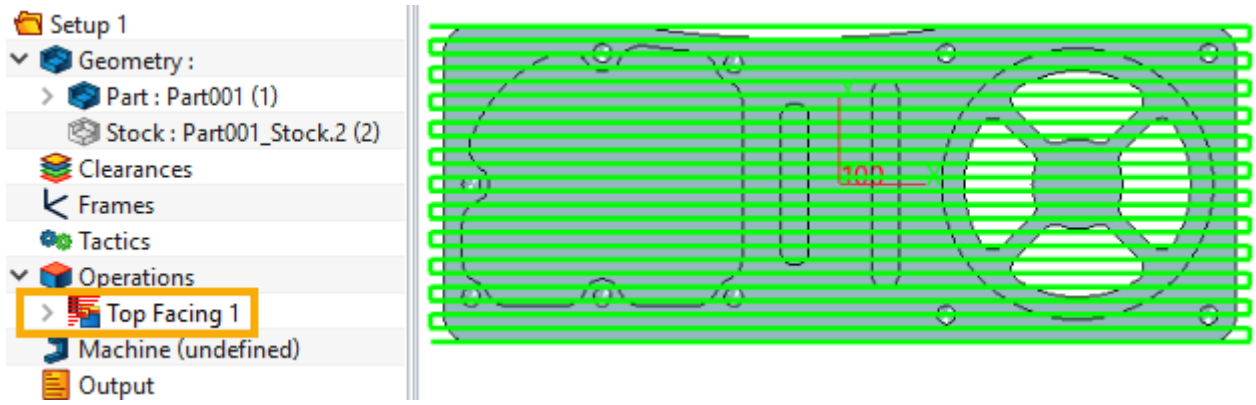


Figure 24 Topface Operation Toolpaths (Corner Round is similar)

Inter Path Move: This operation is to create safe links between tool paths.

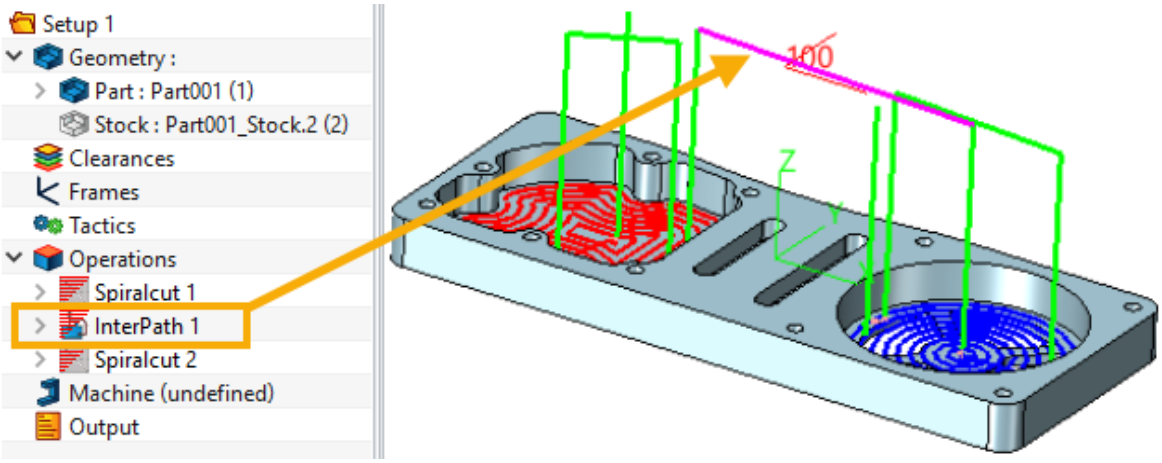


Figure 25 Inter Path Move Operation Toolpaths

2.2.2 Mill2 Tactic

Mill2 Tactic is used to create a series of “intelligent” rule-based 2X operations for the whole part by one

click. This function will help users to save a lot of time when they are creating 2X toolpaths, especially on simple parts and the ones which don't require a very fine finishing.

Mill2 Tactic will automatically detect the CAM features of the part, and then choose a suitable operation (as well as the parameters and cutter) for each of them based on the defined rules. The tools are chosen from Library, users can use the default Library or build a new one.

Here are the steps to create a Mill2 Tactic.

STEP 01 Click on Ribbon ***Mill2*** to add a Mill2 Tactic in the Manager.

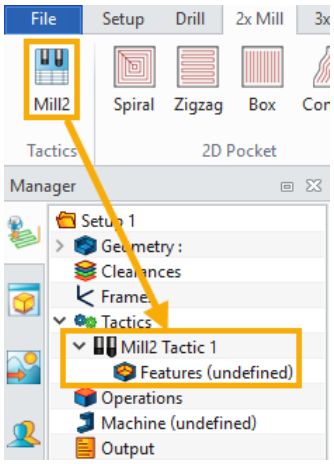


Figure 26 Create a Mill2 Tactic

STEP 02 Right-click or double-click on the text ***Features***, to add the part in.

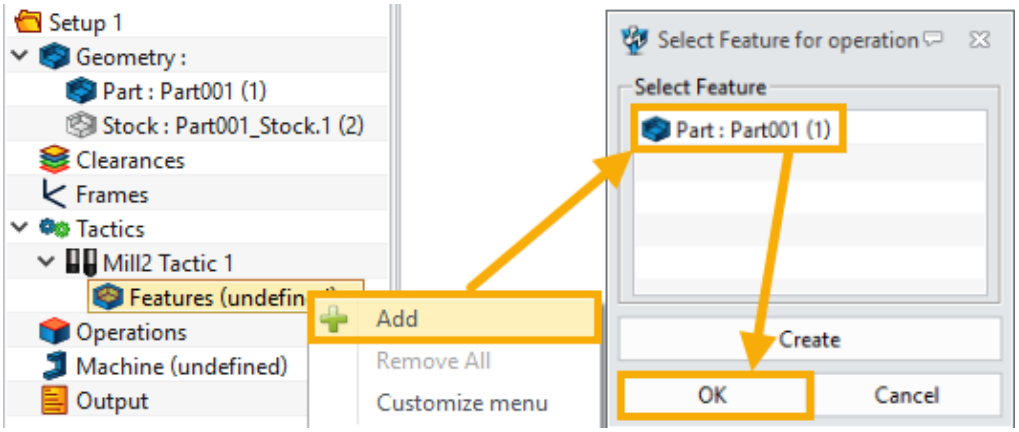


Figure 27 Add a part in the Features

STEP 03 Right-click on the text ***Mill2 Tactic 1***, then select ***Create/Calculate Operations*** to generate toolpaths.

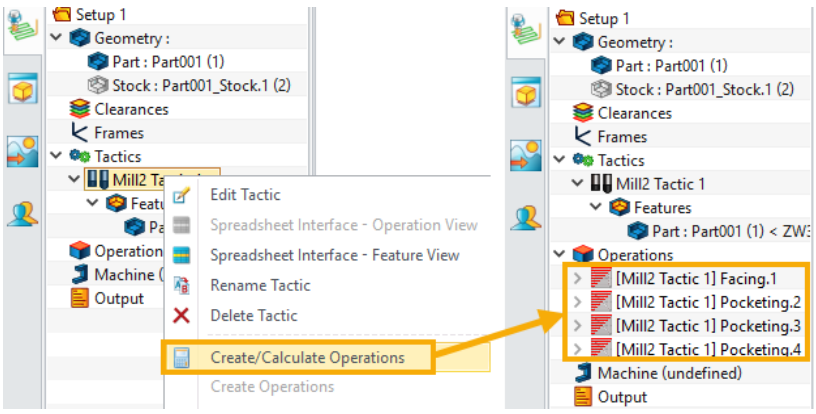


Figure 28 Generate Mill2 Tactic Toolpaths

How to edit rules of Tactics?

Users can right-click on the text ***Mill2 Tactic 1*** to view the settings (-

Figure 28 shows) or get access to editing the rules. Click on ***Edit Tactic*** will call out Tactic Manager as shown below. Users can define the name, material type, tool library and rules here.

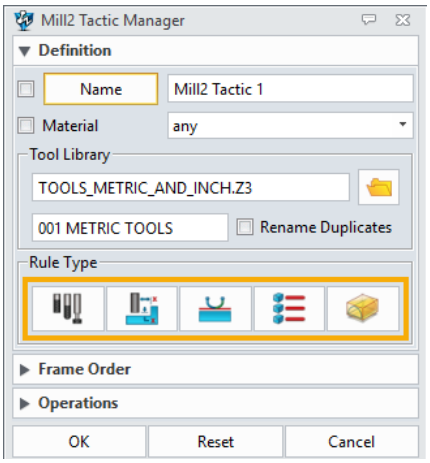


Figure 29 Mill2 Tactic Manager

2.2.3 CAM Features

The figure below shows all the types of CAM Features in ZW3D. This section will take ***Profile*** as an example to show the general steps on how to create a CAM Feature.

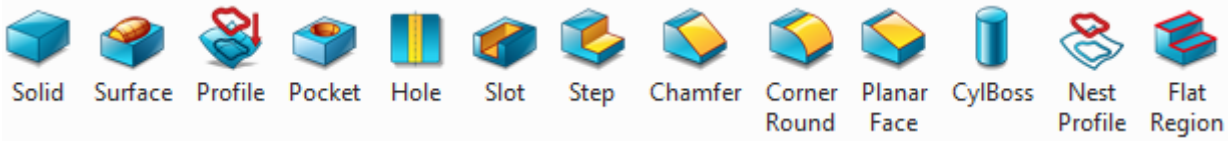


Figure 30 CAM Features in ZW3D

STEP 01 Click on the Ribbon ***Profile*** to call out the select window.

STEP 02 Select the edges or curves to make up a ***Profile***.

STEP 03 Set the attributes and press ***OK***, then the feature will show in the Manager.

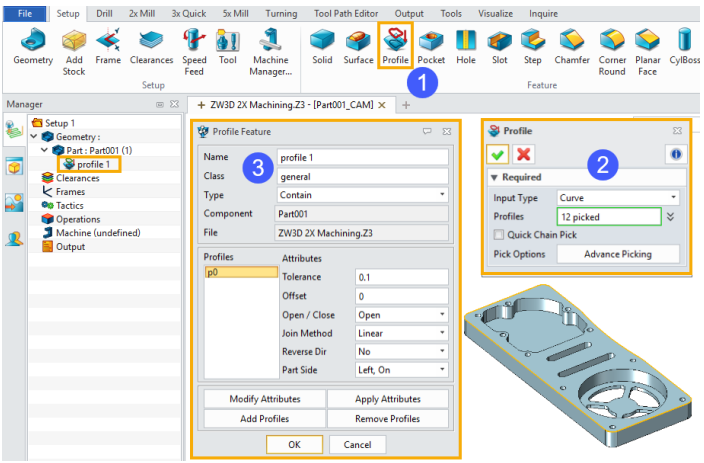


Figure 31 Create a Profile Feature

There are another 2 ways to call out feature select window. One is to right-click on the part in CAM Manager; the other is to right-click or double-click on the text ***Feature***.

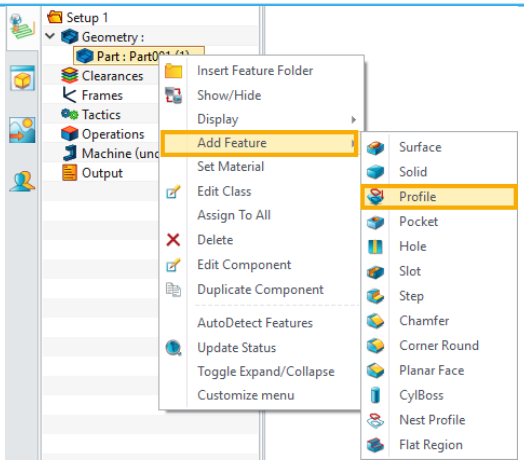


Figure 32 Another 2 Ways to Call Out Feature Select Window (1)

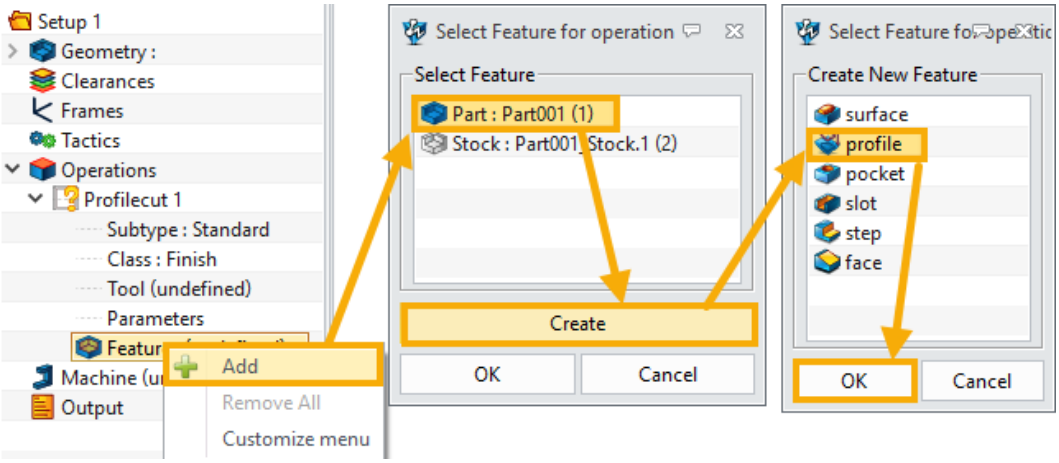


Figure 33 Another 2 Ways to Call Out Feature Select Window (2)

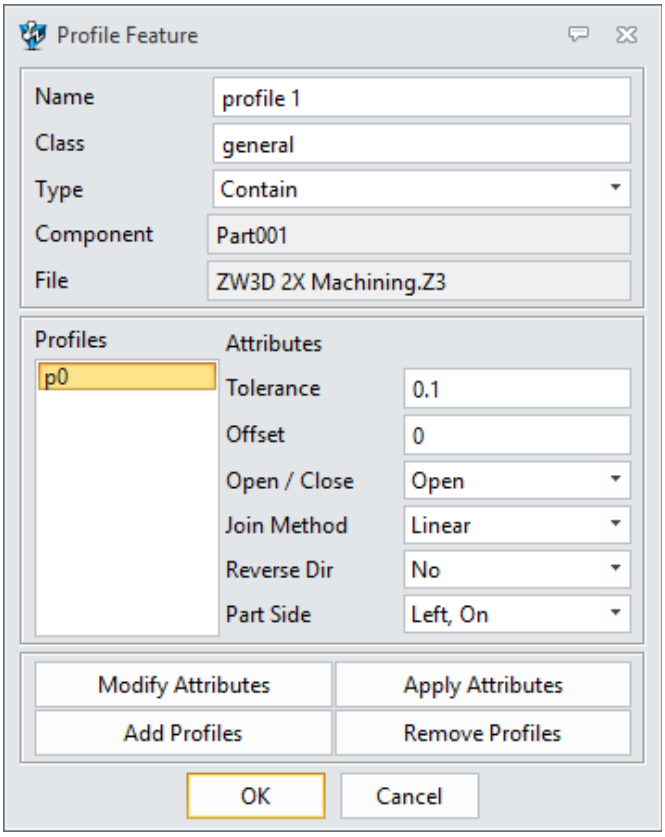


Figure 34 Profile Feature Attributes

Class: Define the class of the Feature as "general" except for 5X Swarf Operation.

Type: Define the type of the Feature. "Part" is used to derive toolpaths while "Contain" to contain or limit toolpath movement.

Tolerance: Define the tolerance used to determine if the profile is closed.

Offset: Define the offset value of selected curves, which is intended to remain on the part after the CAM operations are completed.

Open/Close: Define whether the CAM profile will form an open or closed boundary

Join Method: Define how gaps in the Feature are to be closed

Reverse Dir: Define the direction of the Profile feature - in effect, it changes the part side from right to left or vice versa.

Part Side: Define which side of the profile is the part side

2.2.4 Tool Manager

Users can customize ***Tools*** or ***Tool Library*** with Tool Manager. You can click the Ribbon ***Tool*** or right-click on the text ***Tool*** in CAM Manager Tree to get access to ***Tool Library*** as the below figure shows. Users can either input the parameters to create a tool or direct load one from Library.

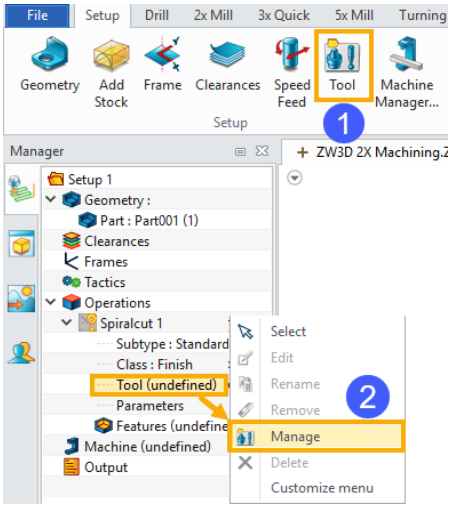


Figure 35 Access to Tool Manager

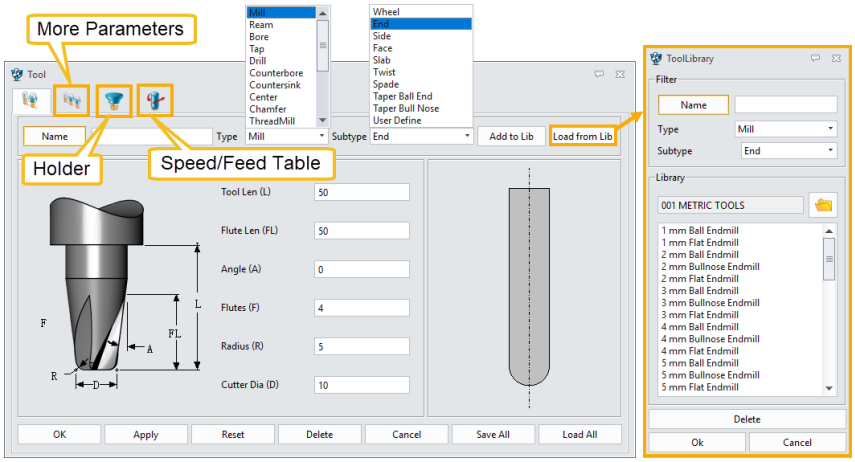


Figure 36 Tool Manager

2.2.5 Tool Library

Here is an example showing the steps to create a new Tool Library. We'll use this in Chapter 4.

STEP 01 Create a new CAM Plan file named ***Tool Library*** or any other one you prefer.

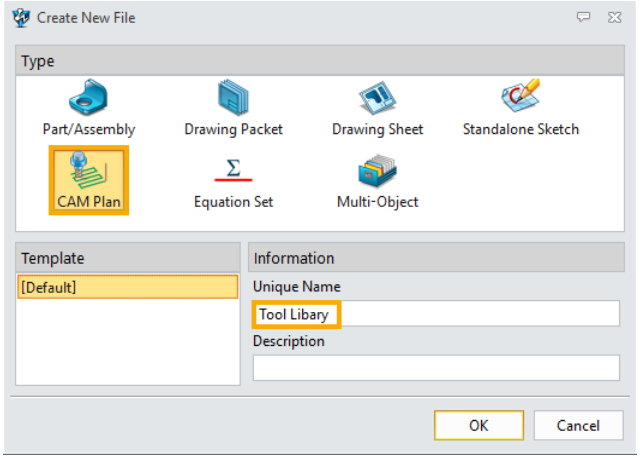


Figure 37 Create a New CAM Plan File

STEP 02 Save the file.

STEP 03 Open Tool Manager and define tool ***CenterDrill 5mm*** as below.

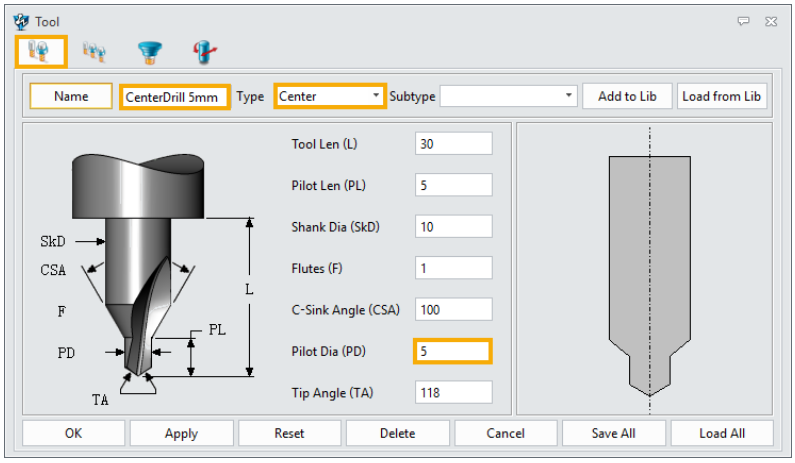


Figure 38 Tool “CenterDrill 5mm” Settings (1)

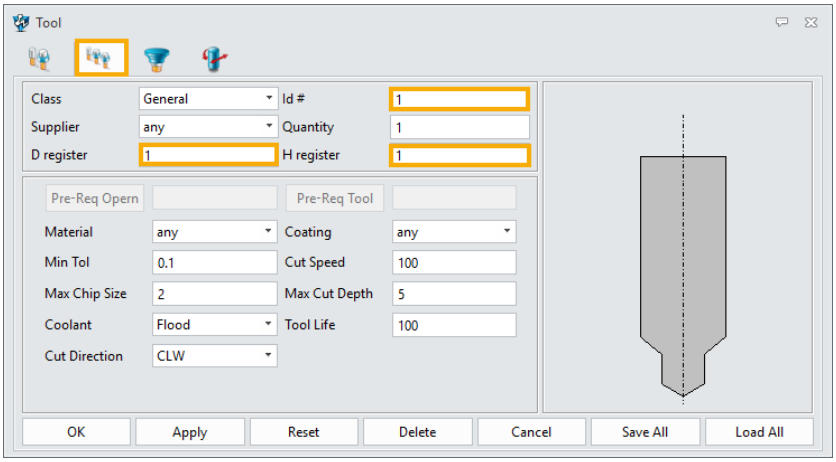


Figure 39 Tool “CenterDrill 5mm” Settings (2)

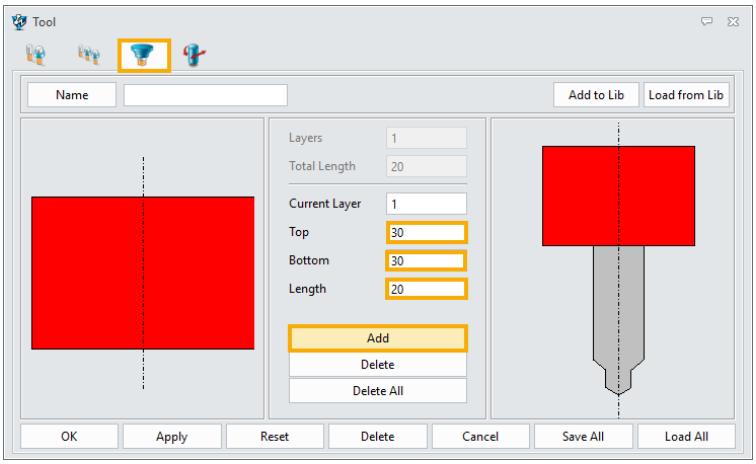


Figure 40 Tool “CenterDrill 5mm” Settings (3)

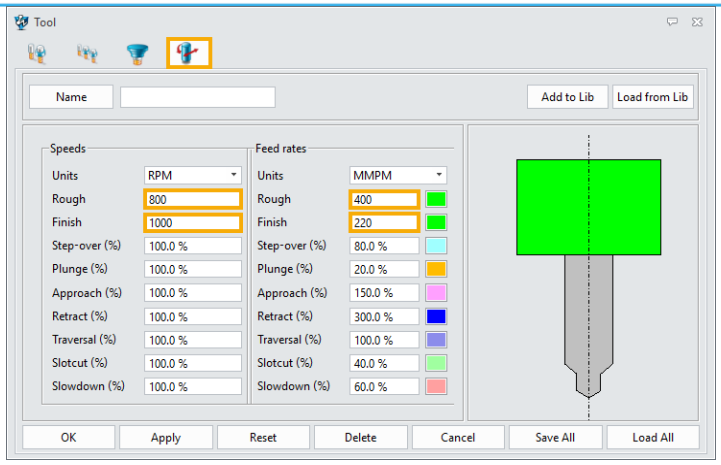


Figure 41 Tool “CenterDrill 5mm” Settings (4)

STEP 04 Click ***Save All*** button on the Tool Manager after all the parameters of ***CenterDrill 5mm*** are defined. Then select the correct path to save all tool data in the ***Tool Library*** file.

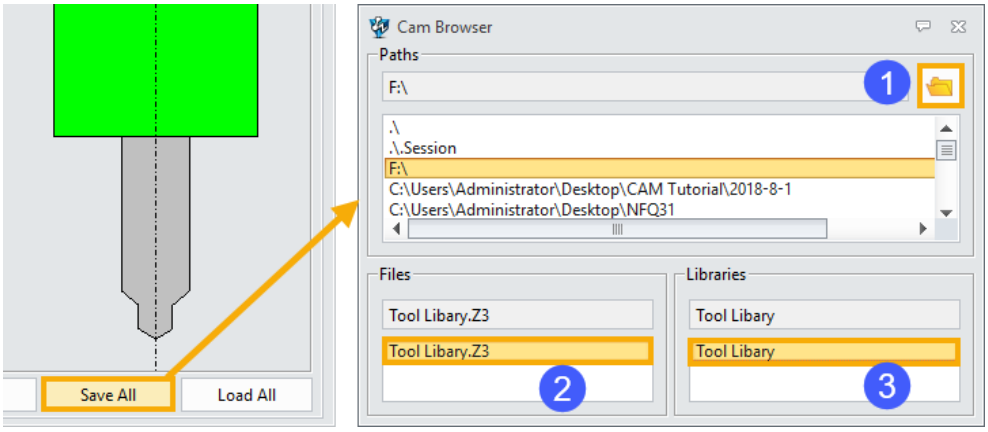


Figure 42 Save the Tool “CenterDrill 5mm”

STEP 05 Define the tool ***Drill 8mm*** as below, and then click ***Save All*** as previous.

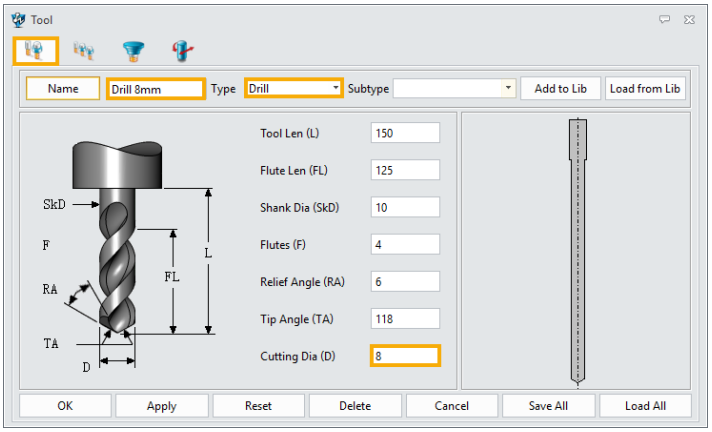


Figure 43 Tool “Drill 8mm” Settings (1)

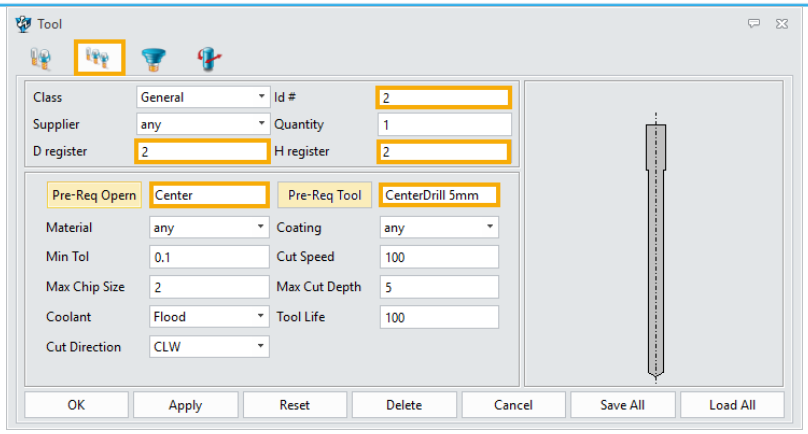


Figure 44 Tool “Drill 8mm” Settings (2)

STEP 06 Define the tool ***Drill 10mm*** as below, and then click ***Save All*** as previous.

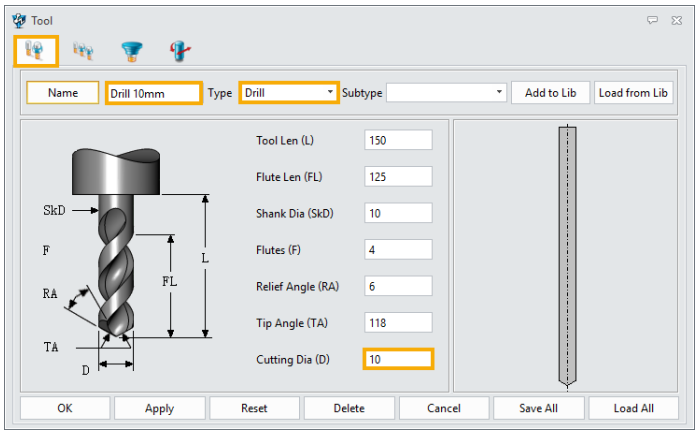


Figure 45 Tool “Drill 10mm” Settings (1)

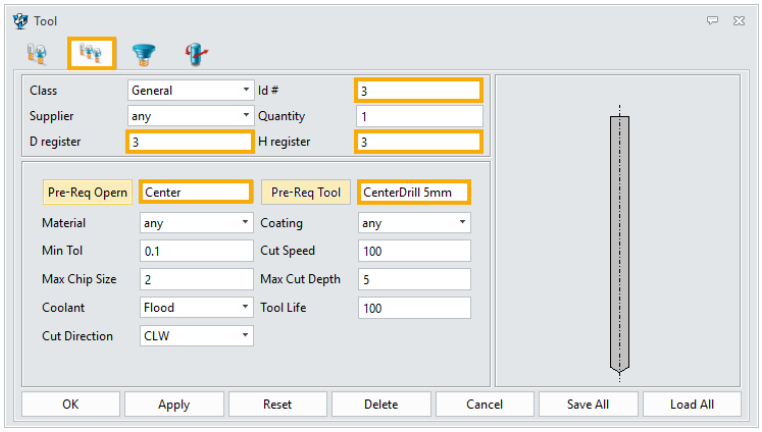


Figure 46 Tool “Drill 10mm” Settings (2)

STEP 07 Define the tool ***M10X1.25*** as below, and then click ***Save All*** as previous.

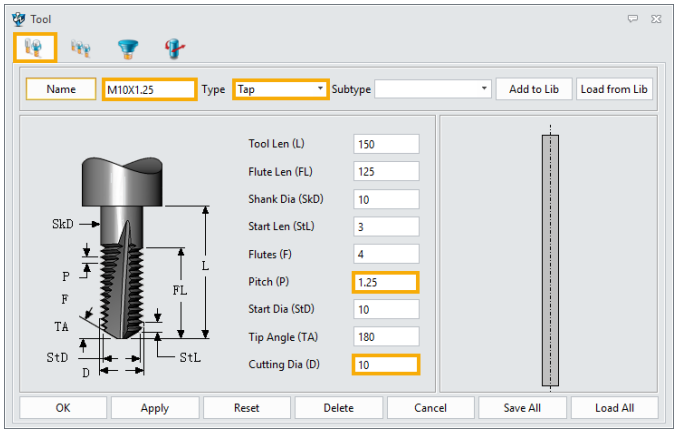


Figure 47 Tool “M10X1.25” Settings (1)

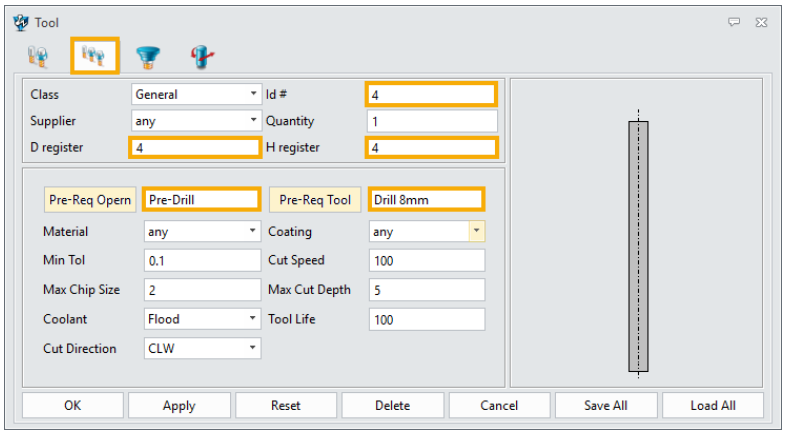


Figure 48 Tool “M10X1.25” Settings (2)

STEP 08 Save the file again and exit. The next time you want to use a cutter from the ***Tool Library*** we have just created, you can open the ***Tool Manager*** window and click ***Load from Lib*** or ***Load All*** button.

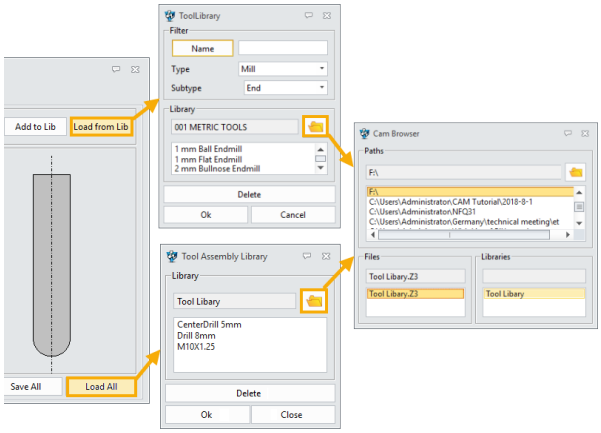


Figure 49 Load Tool Library

2.2.6 Typical Parameters of 2X Operation

It is very important to set suitable parameters of an Operation, or the calculation won’t develop a good result. This section will give a detail introduction of typical 2X parameters. Some unique type of parameters will be further discussed in a case study later.

Select an Operation (Spiral for example), then double-click or right-click on the text ***Spiralcut1*** or ***Parameters*** to call out parameter settings.

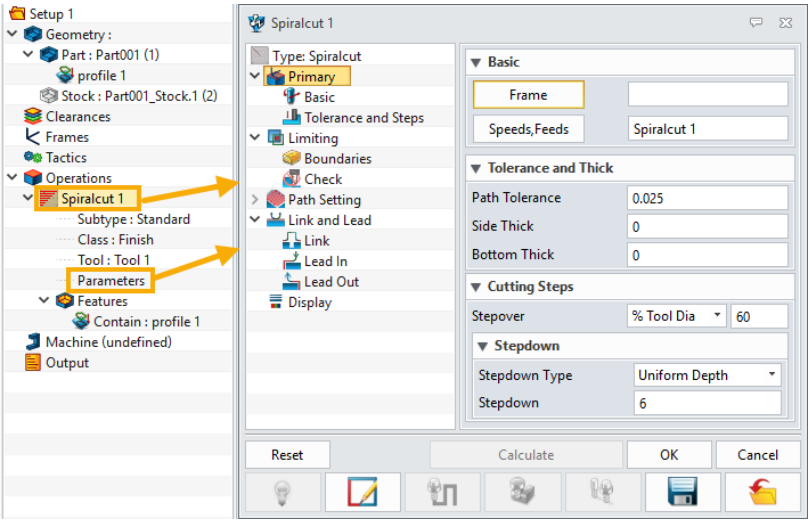


Figure 50 Parameter settings (Primary)

Primary -Frame: Define the coordinate system used to calculate toolpaths.

-Speeds, Feeds: Define Speed/Feed Table used in this Operation.

-Path Tolerance: Define toolpath tolerance.

-Side Thick: Define the thickness of material left on boundary sides after the operation is done.

-Bottom Thick: Define the thickness of material left above the bottom of the region after the operation is done.

-Stepover: Define the spacing of adjacent cuts between cuts.

-Stepdown Type: Define how tool moves downwards.

-Stepdown: Define the height between cuts.

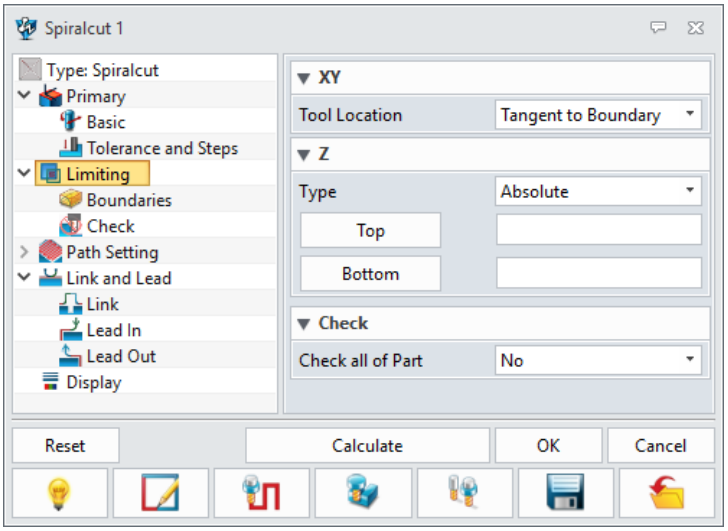


Figure 51 Parameter Settings (Limiting)

Limiting -Tool Location: Define whether the tool can move past boundary.

-Type: Define the type of the input value in ***Top*** and ***Bottom*** below.

-Top/Bottom: Define the top/ bottom of the operation.

-Check all of Part: Define whether the Part feature will be considered by the operation, no matter if it has been added in.

Tips

Hang the mouse over any input field for a second, ZW3D will pop up a picture to illustrate it.

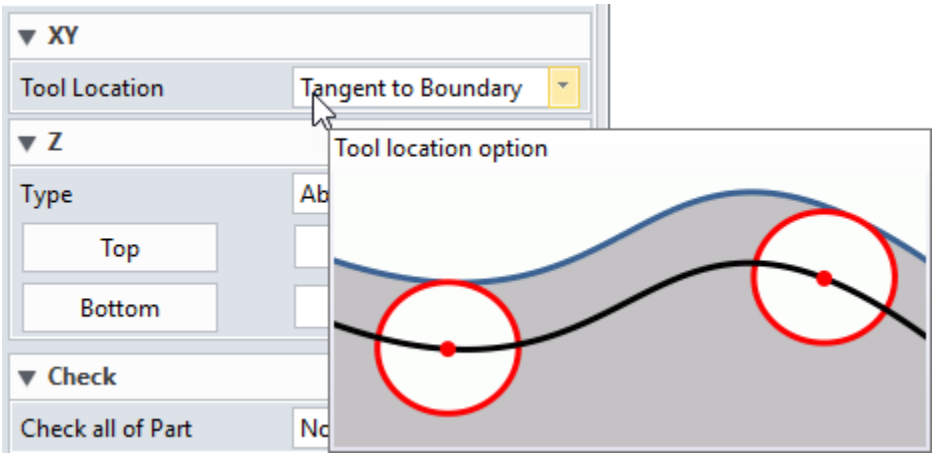


Figure 52 CAM Input Field Tips

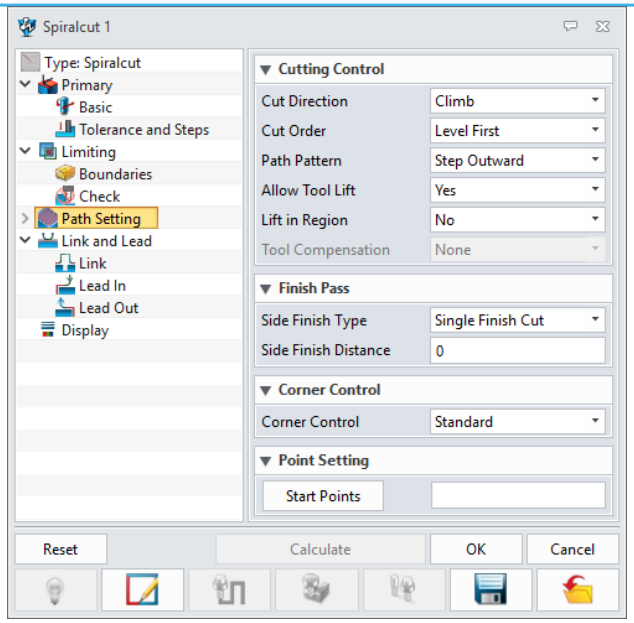


Figure 53 Parameter settings (Path Setting)

Path Setting -Cut Direction: Define the move direction of the tool.

-Cut Order: Define cut order.

-Path Pattern: Define the moving pattern of toolpaths.

-Allow Tool Lift: Define whether to allow lifting the tool.

-Lift in Region: Define whether to allow lifting the tool when cutting one region.

-Tool compensation: Define whether to output Cutter Compensation statements when generating the active tool path.

-Side Finish Type: Define the type of final clean up made on the part boundaries.

-Side Finish Distance: Define the material thickness left after finishing.

-Corner Control: Define how the tool behaves when it changes direction.

-Start Points: Define preferred regions on the boundaries to begin cutting.

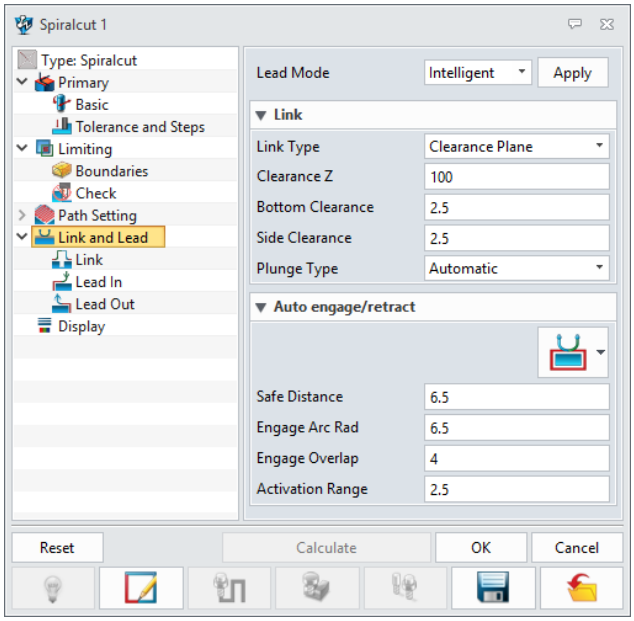


Figure 54 Parameter settings (Link and Lead)

Link and Lead-Lead Mode: Define the lead mode.

-Link Type: Define the link type when the tool moves to next starting point.

-Clearance Z: Define the Z value of the Clearance plane if the Link Type is ***Clearance Plane***.

-Bottom Clearance: Define the safety lift distance (above the ***Top Point*** set in Limiting Tab) for inter-cut moves.

-Side Clearance: Define the gap between part boundaries and inter-cut moves.

-Plunge Type: Define the type of engage plunge motion.

-Safe Distance: Define the distance between the tool and part surface when engage or retract.

-Engage Arc Rad: Define the engage arc radius.

-Engage Overlap: Define the re-cut distance to obtain a smooth part surface.

2.2.7 Verify Toolpaths

ZW3D provides 2 methods for verifying toolpaths. One is named ***Verify***, which is used to check how the tool moves along cuts. The other one is a simulation process called ***Solid Verify***, which shows how a stock becomes a part.

Method 1: Verify

You can click on the button ***Verify*** on the bottom of the parameter setting window, or right- click on the operation name and select ***Verify*** to start the process. In the popup window, you can see the ***Feed/Speed*** and location of the tool. Besides, you can also move the tool forward, backward or to a location by ***Pick*** button.

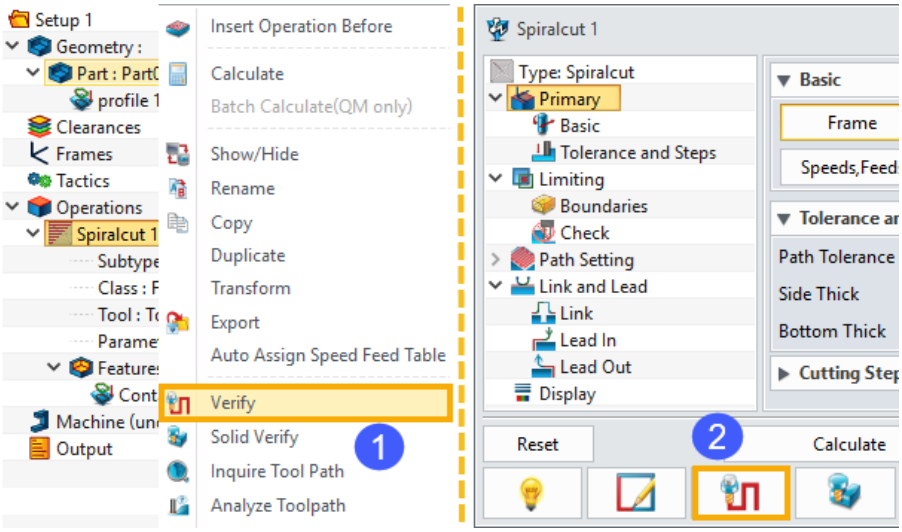


Figure 55 Access to “Verify”

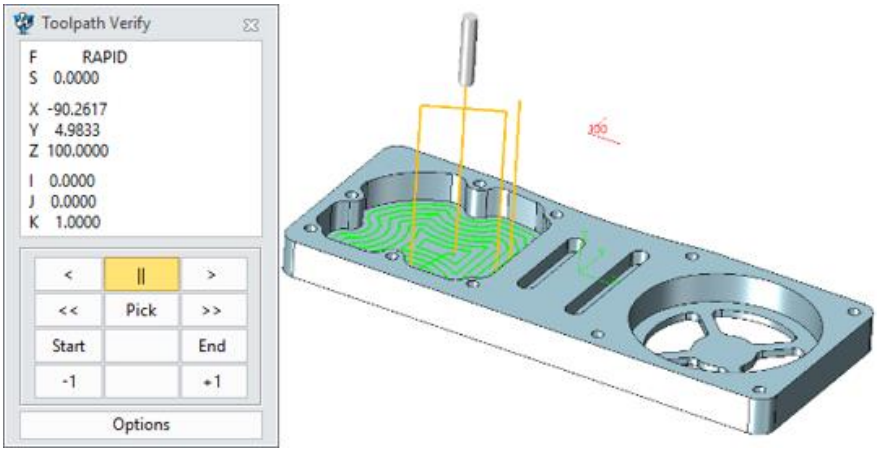


Figure 56 “Verify” Form

Method 2: Solid Verify

You can click on the button ***Solid Verify*** on the bottom of the parameter setting window, or right-click on the operation name, and select ***Solid Verify*** to go to the simulation space.

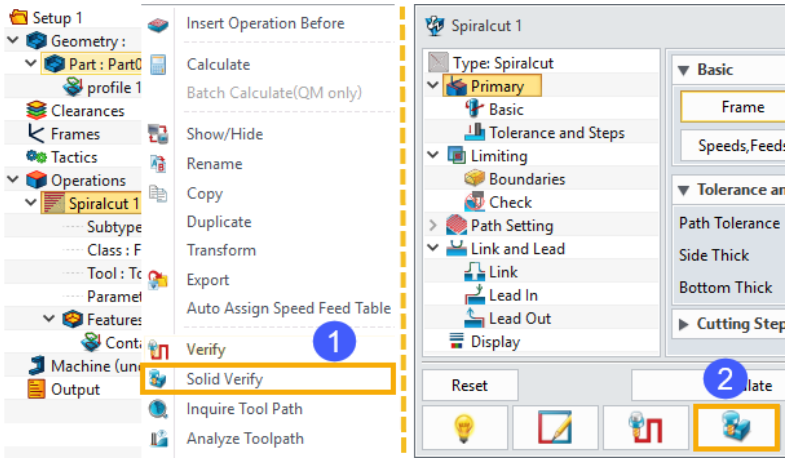


Figure 57 Access to “Solid Verify”

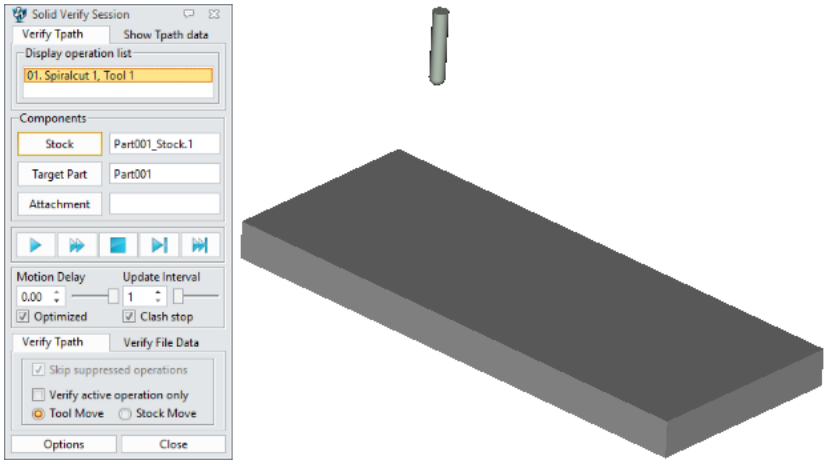


Figure 58 “Solid Verify” Environment

2.2.8 Define Machine and Controller Parameters

If toolpaths are acceptable, the next step should be done is to define machine and controller parameters. Users can click on ***Machine Manager*** in Ribbon or ***Machine*** in CAM Manager Tree to go to the setting window.

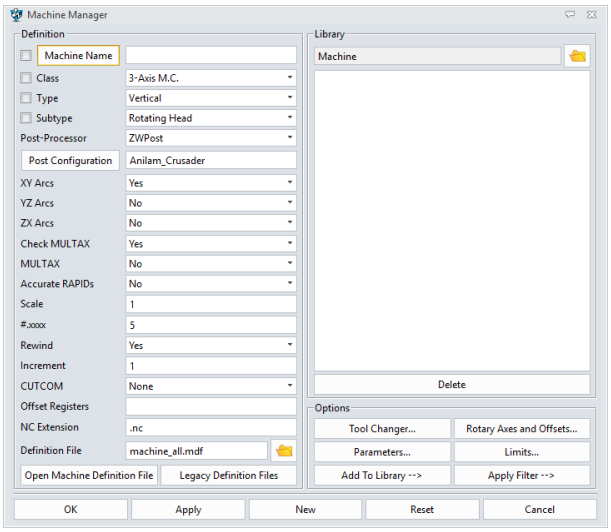


Figure 59 Machine Manager

Machine Name: Define the name of your machine.

Class: Define the classification of your machine.

Type: Define the tool/table movement type of your machine.

Subtype: Define subtype according to ***Type*** above.

Post-Processor: Define the post-processor to be used.

Post Configuration: Define the post configuration.

XY Arcs, YZ Arcs, ZX Arcs: Define whether to support circular interpolation in the XY, YZ, or ZX planes.

Check MULTAX: Define the name of your machine.

MULTAX: Define whether to output tool axis vectors when creating NC codes.

Accurate RAPIDs: Define whether the machine is capable of accurately moving from point to point as defined in the CL Output file during RAPID Traversal moves.

Scale: Define the scaling factor to apply to motion in NC codes.

#.xxxx: Define the number of decimal digits in NC codes.

Rewind: Define whether to rewind the tape at the end of the NC code.

Increment: Define the tape block numbering increment.

CUTCOM: Define whether to output cutter compensation statements in NC codes.

Offset Registers: Define Specify the range and format for Work Offset Registers.

NC Extension: Define the extension of NC file.

Definition File: Define machine definition file.

2.2.9 Generate NC files

There are 2 methods to generate NC files.

Method 1: Create NC files from ***Operations***

STEP 01 Right-click on ***Operations*** in CAM Manager.

STEP 02 Select the way in which you want to output.

Create Output for All: Create one output file containing all operations.

Create Output for Individual: One output file only contains one operation.

Method 2: Create NC files from ***Output***

STEP 01 Right-click on ***Output*** in CAM Manager and select ***Insert NC***. Or simply double-click on the text ***Output***.

STEP 02 Right-click on the newly created output line ***NC*** and select ***Add Operations*** to involve the operations in.

STEP 03 Right-click on the newly created output line ***NC*** and select ***Output CL*** or ***Output NC*** to generate ***CL/NC*** files. You can also select ***Settings*** or double-click on ***NC*** to customize the output settings.

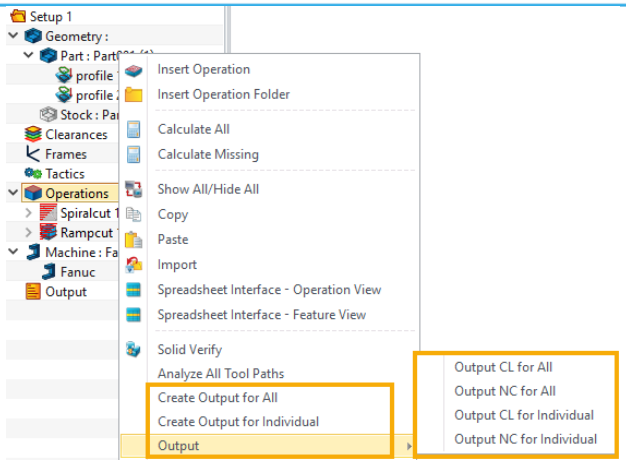


Figure 60 Create NC files from “Operations”

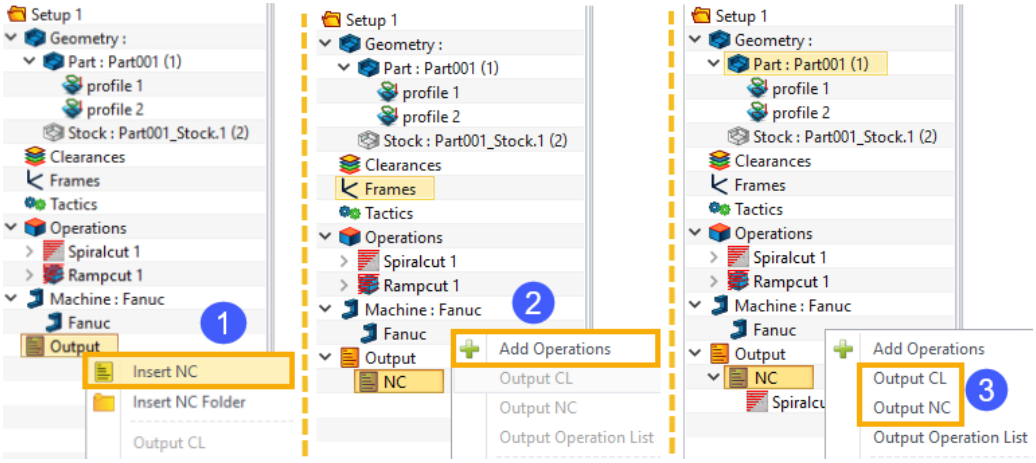


Figure 61 Create NC files from “Output”

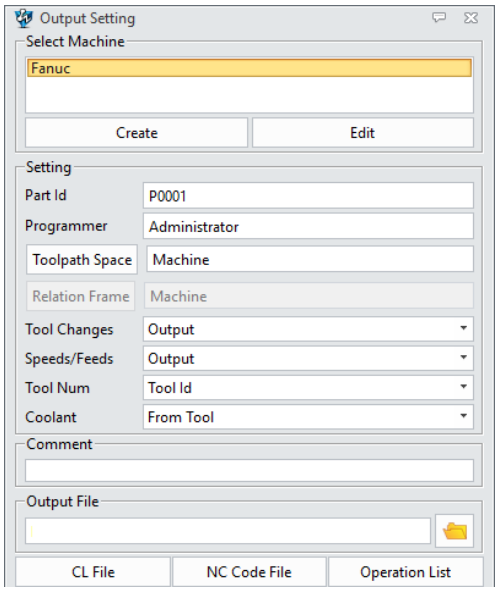


Figure 62 “Output” settings

Select Machine: Select a machine defined earlier from the list.

Part Id: Define the ID number of the part.

Programmer: Define the name of the programmer.

Toolpath Space: Define Coordinate system used to generate NC file. Two options are available, ***Machine*** (global) and ***Local***.

Relation Frame: Define local frame used to create NC file.

Tool Changes: Define whether to enable tool changes.

Speeds/Feeds: Define if spindle speed and feed rates should be specified in the output files.

Tool Num: Define ID number of the tools used in the output file.

Coolant: Use this option to over-ride the coolant mode (specified in the Tool Properties Form) at the time of output.

Comment: Define a comment to be output at the start of the output program.

Output File: Define the output file folder location.

3 Drilling

Key Points:

- ✧ Hole Feature & Hole Filter
- ✧ Drilling Operations & Hole Tactic

3.1 Hole Feature

This chapter uses the same model as previous.

Click ***Hole*** in Ribbon or right-click on the part in CAM Manager to create hole features.

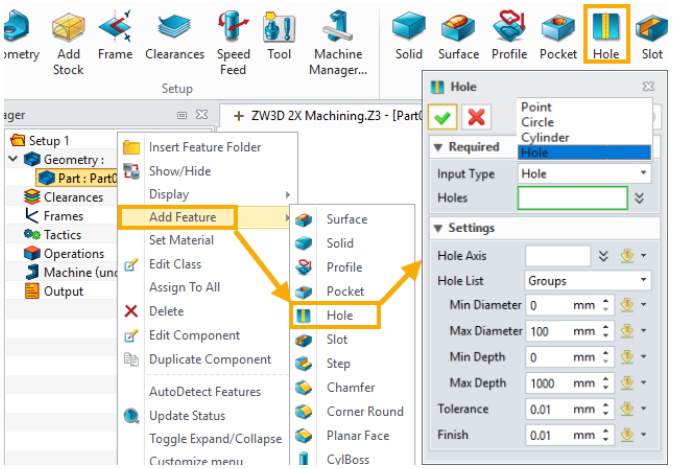


Figure 63 How to Create Hole Features

Input type: Select the geometry type used as the pick filter.

Holes: The number of holes.

Hole Axis: If an axis is specified, only those holes with the same axis will be selected.

Hole List: Define how the selected holes will be grouped.

Min/Max Diameter, Min/Max Depth: Define the range of diameter and depth, only holes within the range will be selected.

Tolerance: Define hole diameter tolerance. If the tool's diameter differs from the hole's by an amount greater than this tolerance, ZW3D will give a warning message.

Finish: Define hole surface finish thickness. For reference only and is not used when calculating toolpaths.

Users can customize the attributes of hole features after picking.

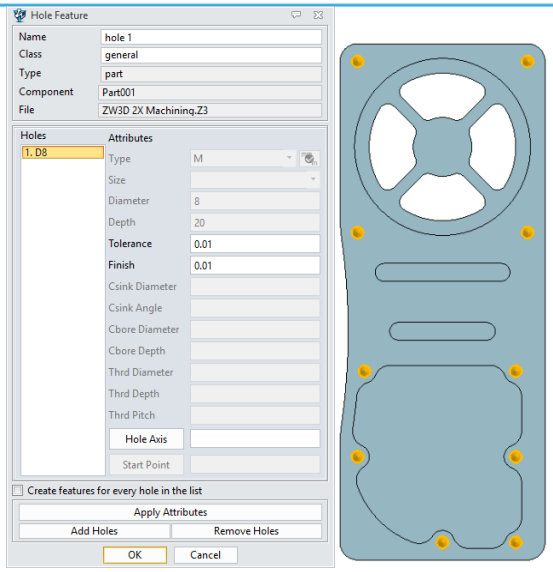


Figure 64 Hole Feature Attibutes

Type, Size: Define the thread type and size of the holes.

Diameter, Depth: Show the values of diameter and depth.

Csink Diameter/Angle: Define the countersink diameter/Angle.

Cbore Diameter/Depth: Define the counterbore diameter/depth.

Thrd Depth/Pitch: Define thread depth/pitch.

Hole Axis: Define an alternate hole axis. Otherwise, ZW3D will calculate it for you.

Start Point: Define the start point of holes defined by Point, Circle and Cylinder.

3.2 Drilling Operations

3.2.1 Overview of Drilling Operations

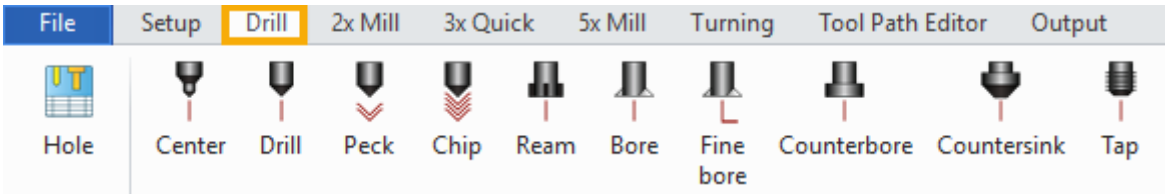


Figure 65 Drilling Operations

Hole Tactic: Like Mill2 Tactic of 2X Milling. With this command, users can create a series of “intelligent” rule-based drilling operations for the whole part by one click.

Center: This operation is used to provide starting holes for other drilling operations.

Drill: This operation is used to drill common holes.

Peck: This operation is a deep drilling method. The drill withdraws to the approach plane after advancing a specified distance into the hole, facilitating chip removal.

Chip: This operation will make the tool to withdraw a specified distance after advancing into the hole to prevent the binding of chips.

Ream: This operation is used for finishing by removing a small amount of material from a pre-drilled hole.

Bore: This operation is used to enlarge, finish and accurately locate holes.

Fine Bore: This operation is used to create a fine hole surface by avoiding drag tools along hole sides from the bottom upward.

Counterbore: This operation is used to spot-face, making room for the heads of bolts or screws, or to enlarge holes.

Countersink: This operation is used to machine a cone-shaped enlargement at the top of a hole.

Tap: This operation is used to create threads in a hole.

3.2.2 Hole Tactic

Steps to create hole tactic are same as Mill2 Tactic.

STEP 01 Click on Ribbon ***Hole*** to add a ***Hole Tactic*** in the Manager.

STEP 02 Right-click or double-click on the text ***Features***, to add the part in.

STEP 03 Right-click on the text ***Hole Tactic 1***, then select ***Create/Calculate Operations*** to generate toolpaths.

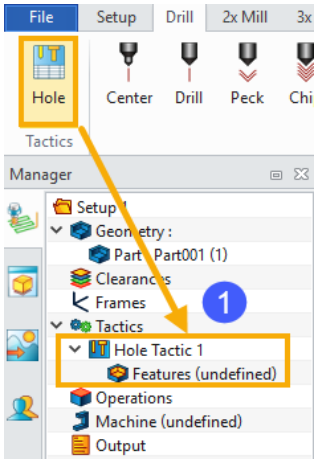


Figure 66 Create a Hole Tactic

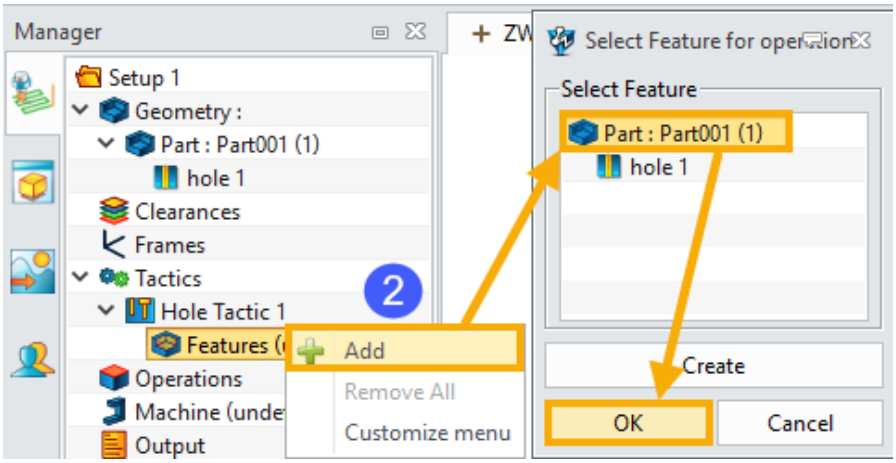


Figure 67 Add a Part in the Features

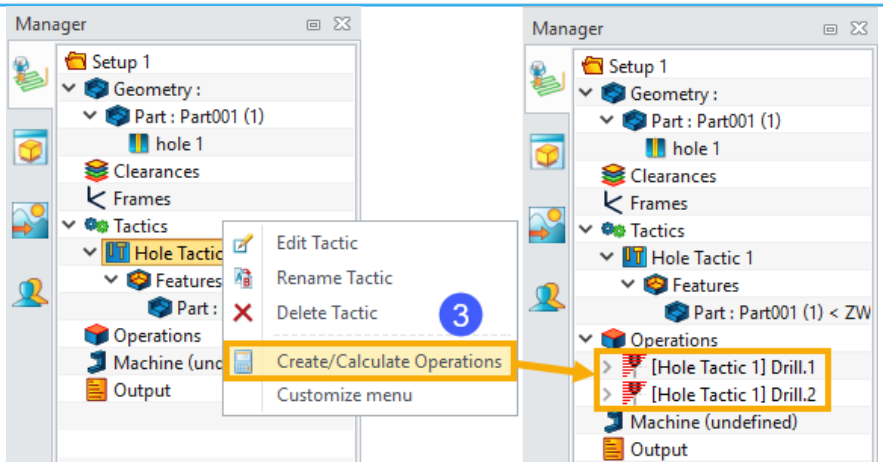


Figure 68 Generate Hole Tactic Toolpaths

4 2X Milling Case Study

Key Points:

- ✧ Calculate Toolpaths with Clamps on the Part
- ✧ Draft Milling
- ✧ Corner Control & Rest Mill
- ✧ Tool Compensation

4.1 Case 1

In this chapter, we'll take a 2D drawing as an example, which is located in ZW3D installation folder. The typical location is "C:\Program Files\ZWSOFT\ZW3D 201X Eng (x64)\training\ 2X machining model\2X_Case_01.Z3".

The part is shown below. The stock and clamps have already been created in the same file.

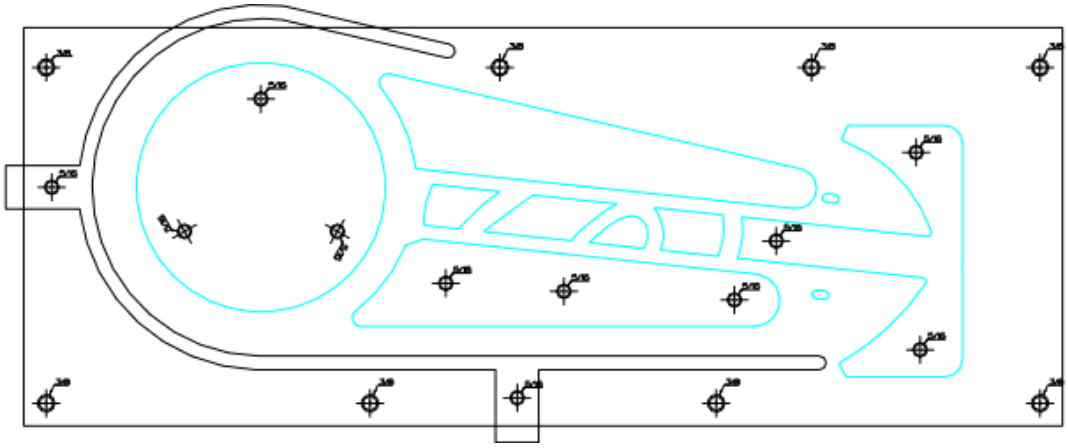


Figure 69 Part in File *2X_Case_01*

4.1.1 Sequence of Machining

1) Mill the top face of the part (Z=0), taking clamps into consideration.
2) Drill holes.
3) Cut pockets.

4.1.2 Mill the Top Face

STEP 01 Click on Ribbon ***Geometry*** or right-click on the text ***Geometry*** in CAM Manager to insert the clamps and stock.

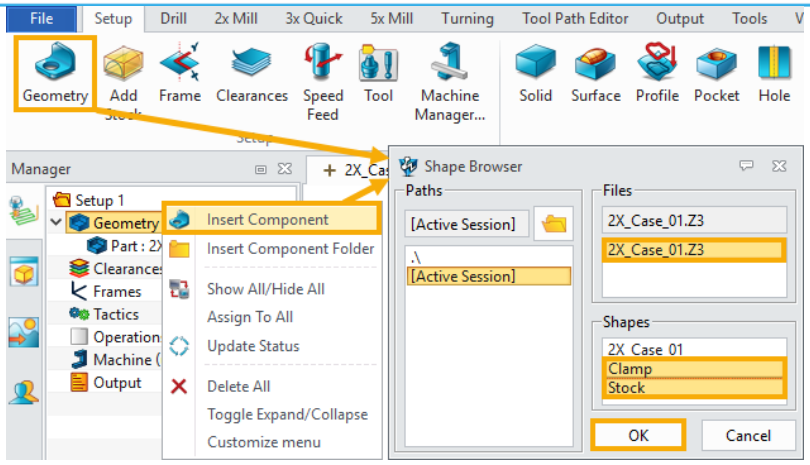


Figure 70 Insert Clamps and Stock

STEP 02 All the inserted geometry is ***Part*** Class by default. Right-click on the clamps and stock and select ***Edit Class*** to set the right class.

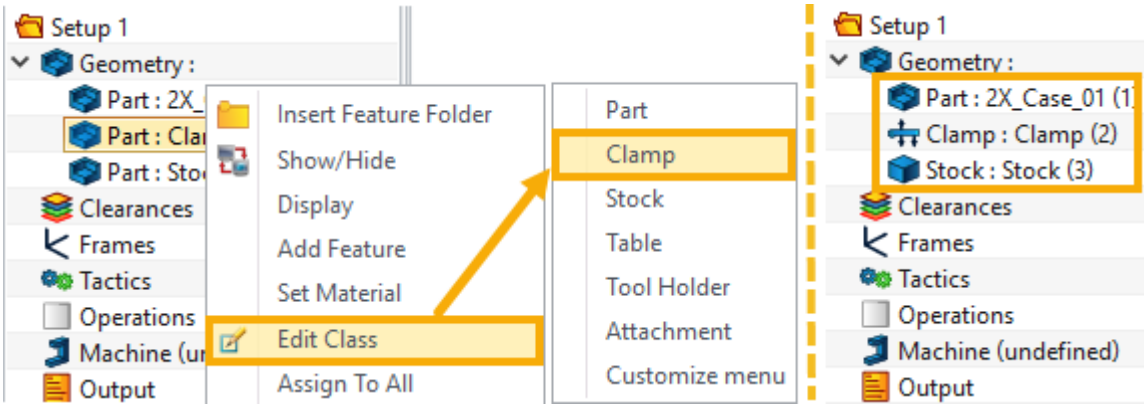


Figure 71 Set Correct Class

STEP 03 Create a Profile based on the outer rectangle.

STEP 04 Create a Zigzag operation take the ***Profile***, ***Clamps***, and ***Stock*** as machining feature. The tool is defined as below figure shows.

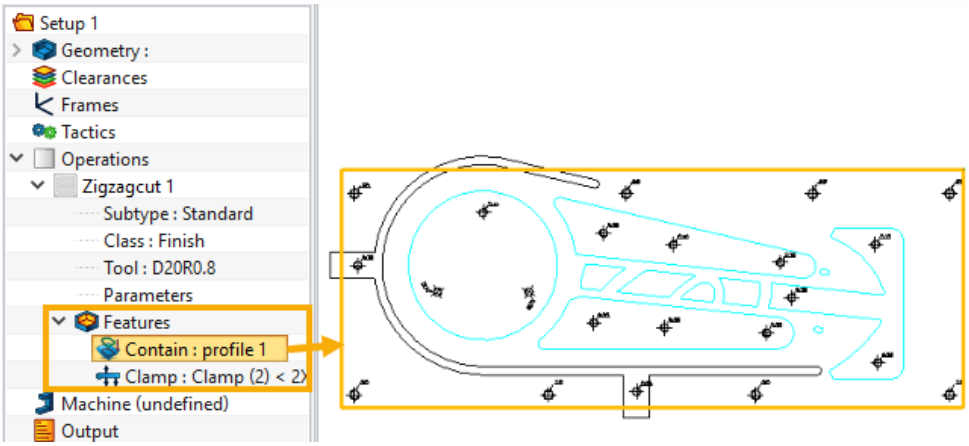


Figure 72 CAM Feature

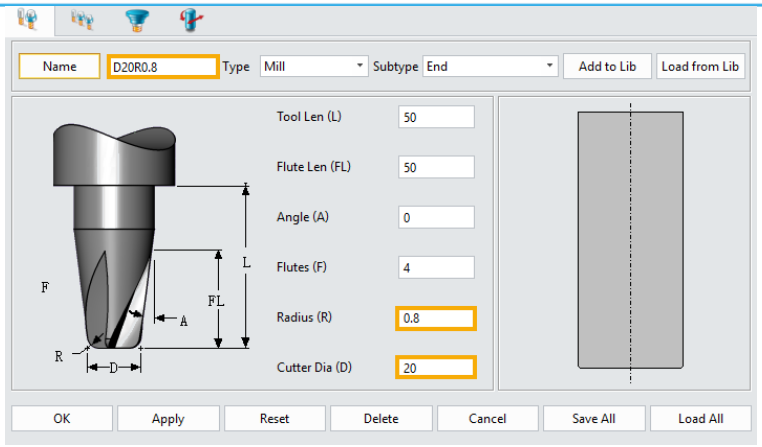
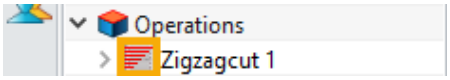


Figure 73 Tool "D20R0.8"

How to quickly hide or show?



Double-click on the small icons in front of a geometry or operation means to show or hide them.

STEP 05 The parameters are set as below figures show. These settings will drive the tool to mill a face 2mm into the stock, and avoid the areas where clamps are located.

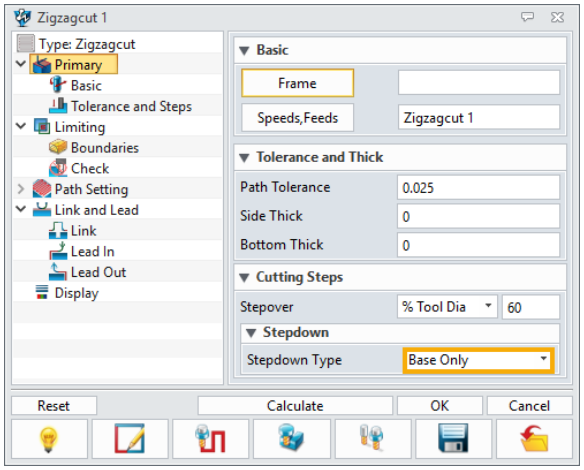


Figure 74 Zigzagcut 1 Parameters (1)

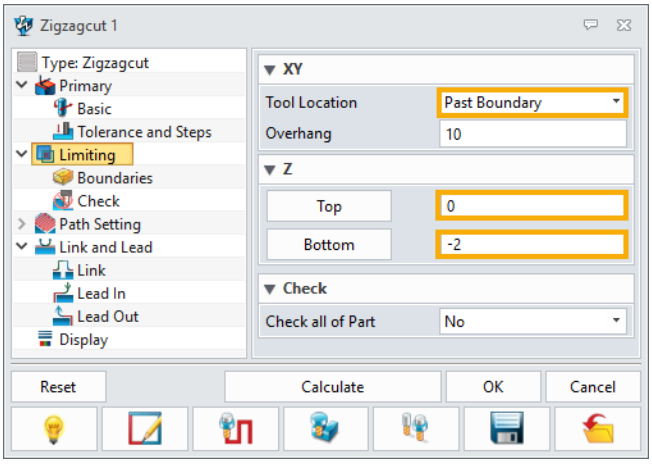


Figure 75 Zigzagcut 1 Parameters (2)

STEP 05 Calculate the toolpaths. The Zigzagcut 1 toolpaths are shown as below.

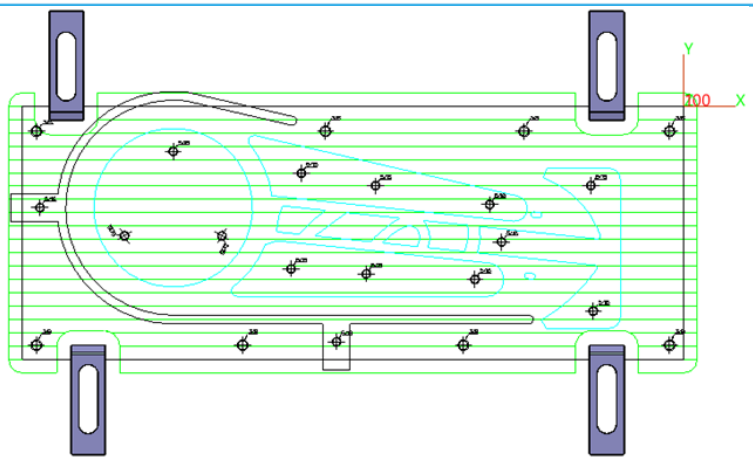


Figure 76 Zigzagcut 1 Toolpaths

Then how can we deal with the remained material that is covered by clamps? ZW3D provides a solution called ***Clamp Cleanup***.

STEP 01 Right-click on operation ***Zigzagcut 1*** and select ***Duplicate*** to make a copy of it.

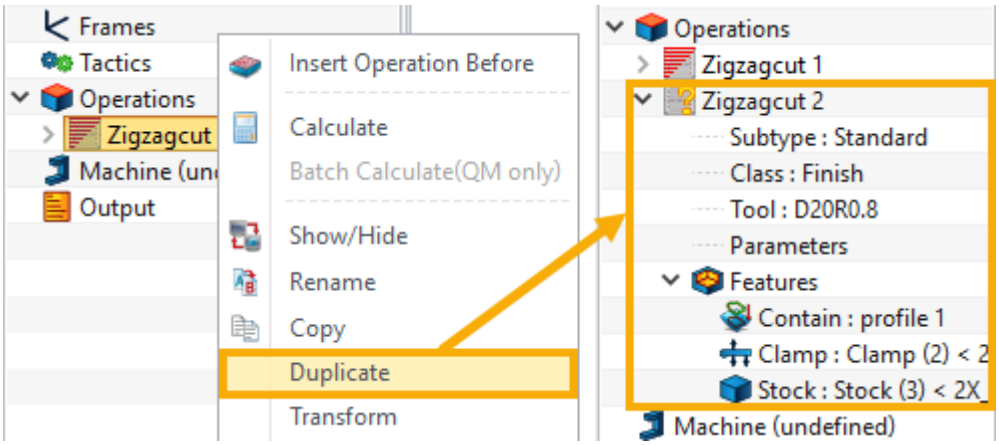


Figure 77 Duplicate Zigzagcut 1

STEP 02 Double-click or right-click on the text ***Subtype*** under operation ***Zigzagcut 2*** to change it to ***Clamp Cleanup***.

STEP 03 Double-click or right-click on the text ***Ref Op*** under operation ***Zigzagcut 2*** to change it to ***Zigzagcut 1***.

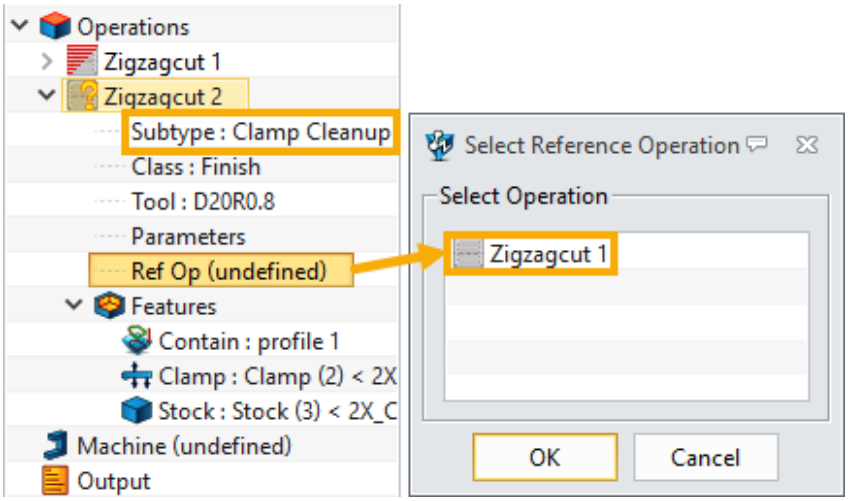


Figure 78 Set Subtype and Reference Operation of Zigzagcut 2

STEP 04 Calculate the toolpaths. See the figure below.

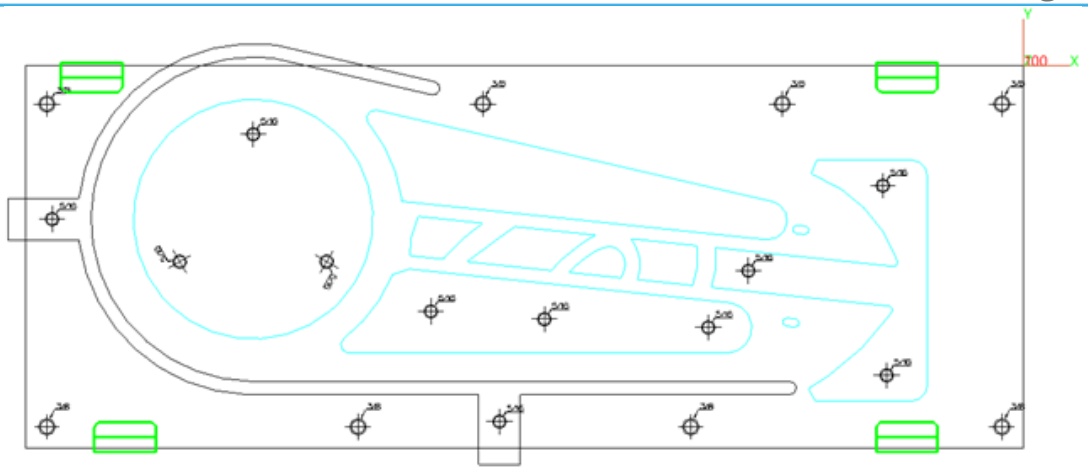


Figure 79 Zigzagcut 2 Toolpaths

4.1.3 Drill Holes

There are two types of holes in the part. One has a diameter of 7mm, marked as ***5/16***, and the other has a diameter of 8.2mm, marked as ***3/8***.

In this case, we'll take holes of diameter 7mm as an example.

STEP 01 Create Hole feature. When selecting holes with a diameter of 7mm, set the ***Input Type*** to ***Circle***, and both the ***Min/Max Diameter*** to 7mm. Then window-pick the whole part or use ***Pick All*** to pick all the arcs in part.

After click ***OK*** ✔, ZW3D will select all the hole we want from the whole part.

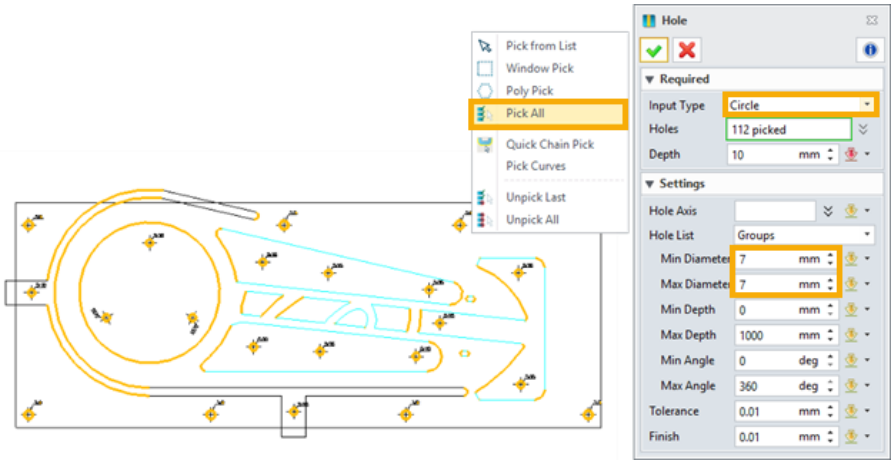


Figure 80 Hole Filter

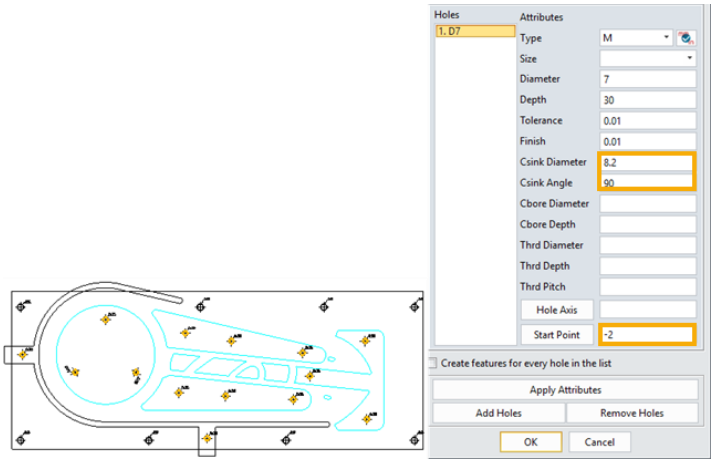


Figure 81 Hole Feature

STEP 02 Create a Drill operation and define the tool as ***Drill 7mm***. The parameters are shown as below.

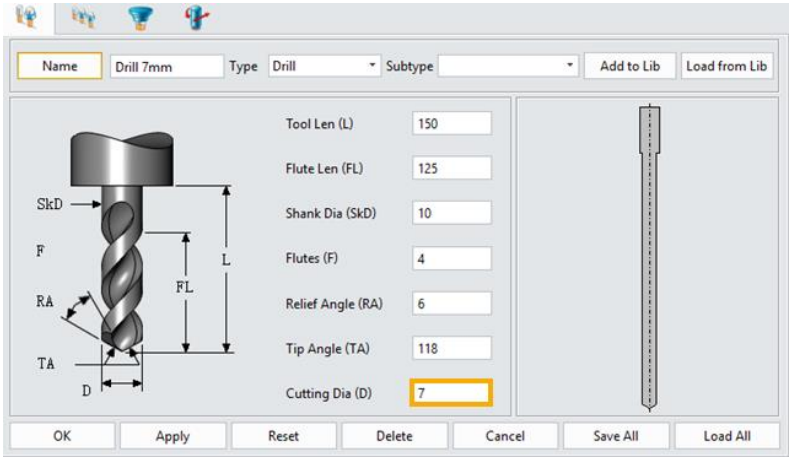


Figure 82 Drill Tool

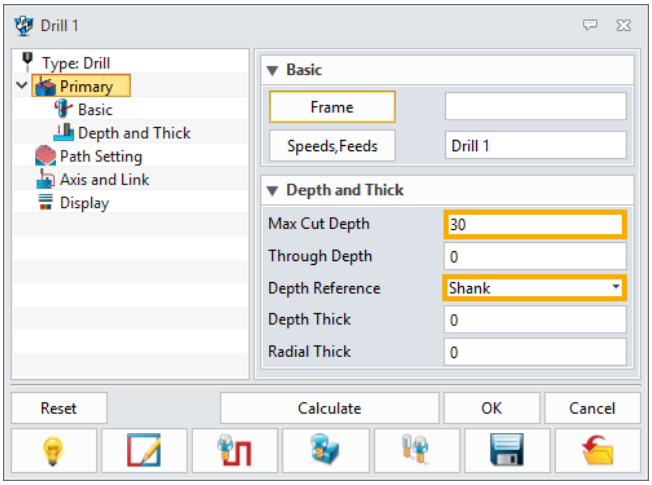


Figure 83 Drilling Parameters

STEP 03 Create a Countersink operation and define the tool as ***Csink***. Leave all the parameters by default.

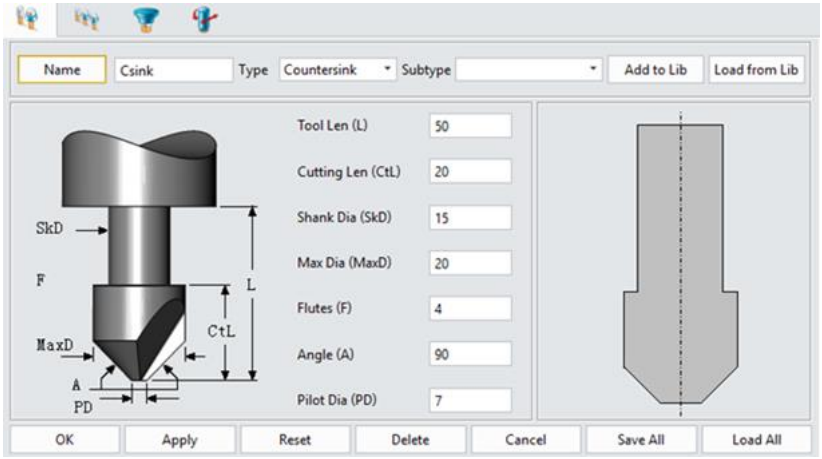


Figure 84 Countersink Tool

4.1.4 Draft Mill

STEP 01 Create a ***Pocket Feature*** based on the big circle on the left.

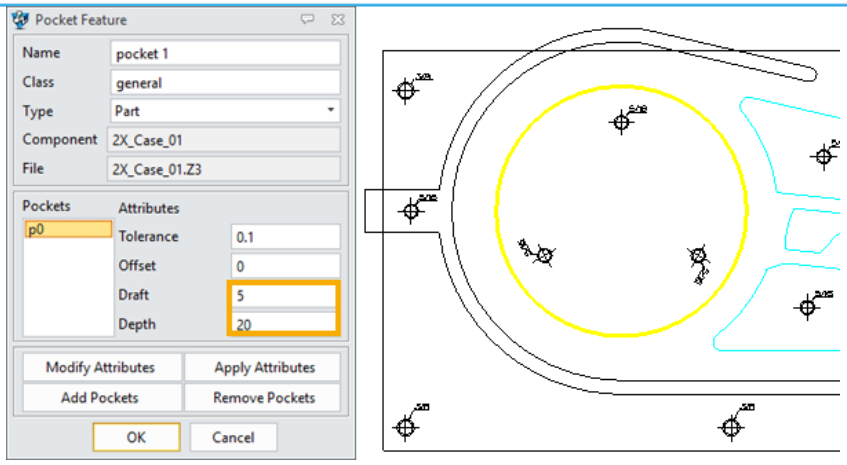


Figure 85 Pocket Feature

STEP 02 Create a ***Spiral*** operation, taking the ***Pocket*** and stock as machine feature, using the same tool as Zigzag. The parameters are set as below.

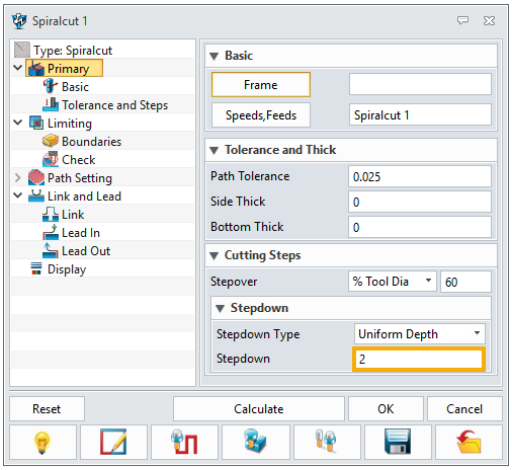


Figure 86 Sprial Operation Parameters (1)

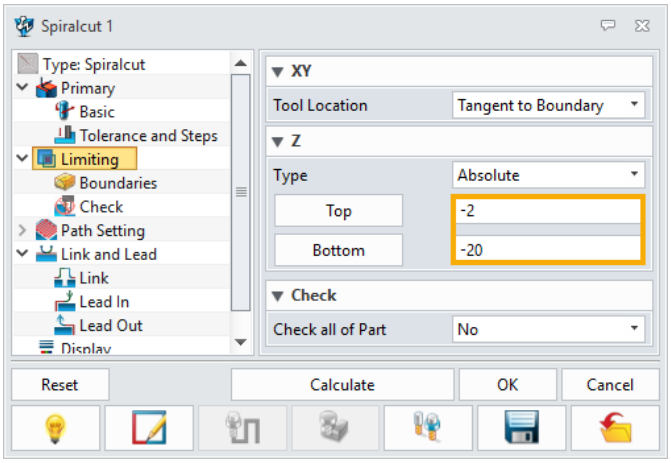


Figure 87 Sprial Operation Parameters (2)

STEP 03 Calculate the toolpaths.

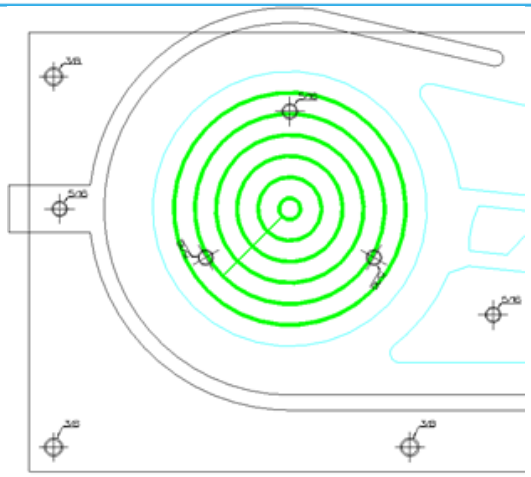


Figure 88 Sprial Operation Toolpaths (Top view)

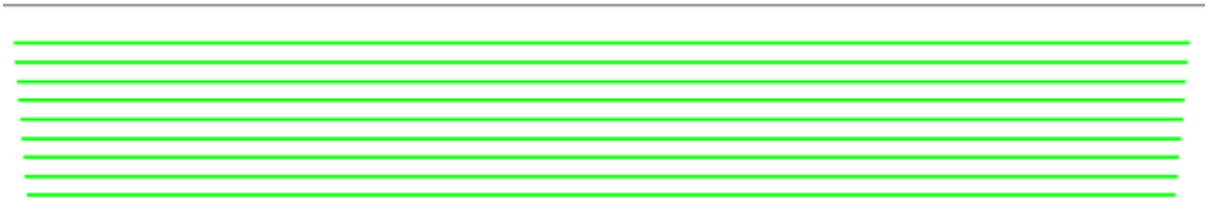


Figure 89 Sprial Operation Toolpaths (Front view)

4.1.5 Rest Mill

STEP 01 Create a ***Spiral*** operation based on a ***Profile*** feature, using the tool ***R20D0.8***. Holes in this area are not shown for a better view.

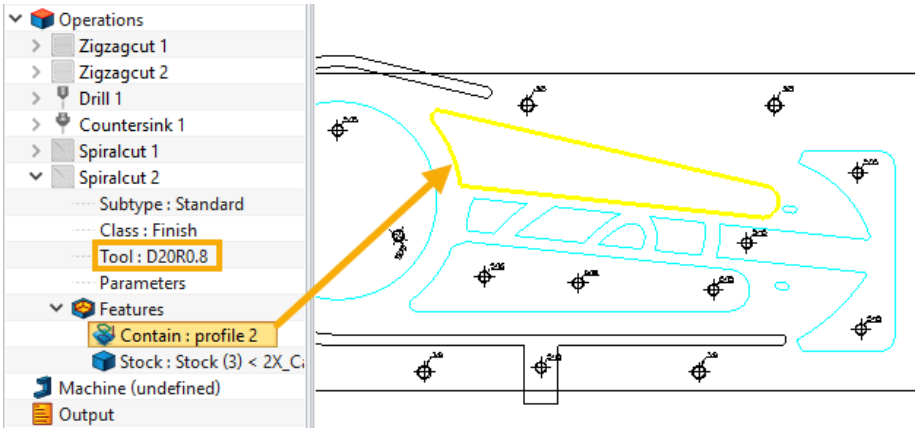


Figure 90 Create Sprial Operation (Spiralcut 2)

STEP 02 Set the parameters. The ***Primary*** and ***Limiting*** tabs are the same as Spiralcut 1. In ***Path Setting***, we pay attention to the ***Corner Control*** option. If select ***Standard***, there will be material remained after cutting (Figure 91). We can set the ***Corner Control*** to any one of the other three options to avoid this situation.

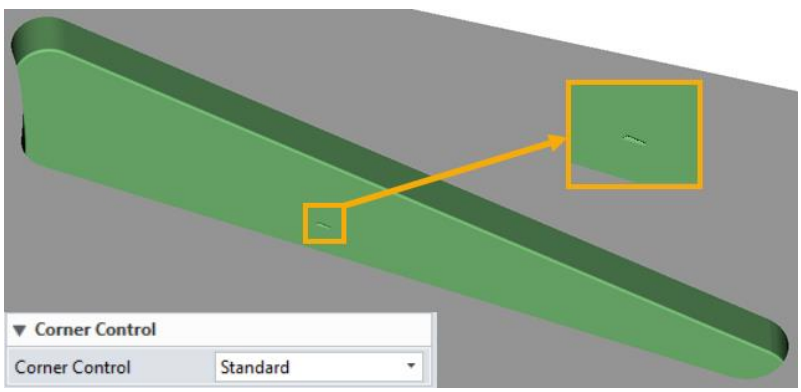


Figure 91 Material Remained By “Standard” Corner Control

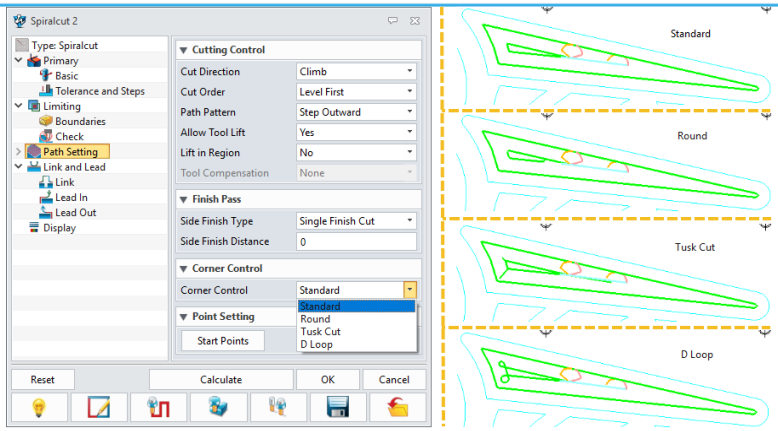


Figure 92 Toolpaths of Different Corner Control Options

STEP 03 Create a new ***Spiral*** operation by duplicate Spiralcut 2.

STEP 04 Double-click on the text ***Subtype*** under Spiralcut 3 to change it to ***Rest Mill***, and define a new tool ***D6R0*** as below Figure shows.

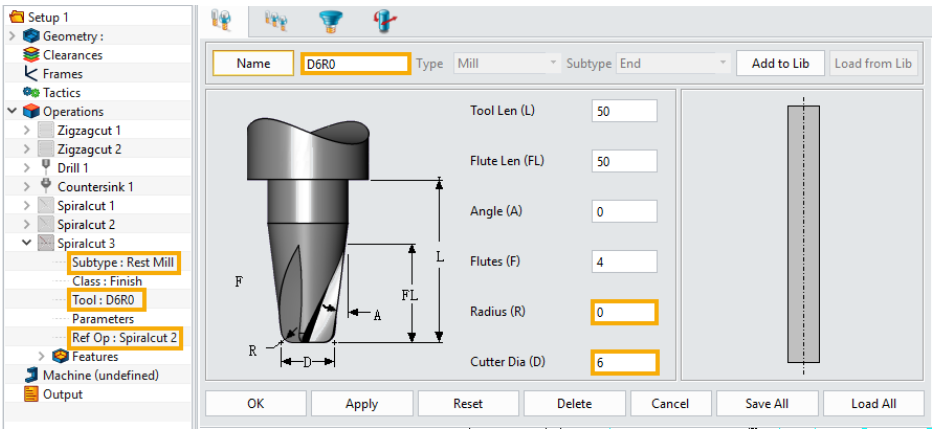


Figure 93 Create Rest Mill Operation “Spiralcut 3”

STEP 05 Change the parameters in the ***Path Setting*** tab.

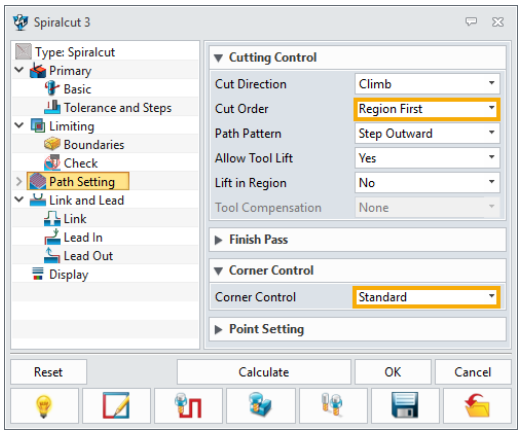


Figure 94 “Spiralcut 3” Parameters

STEP 06 Calculate the toolpaths.

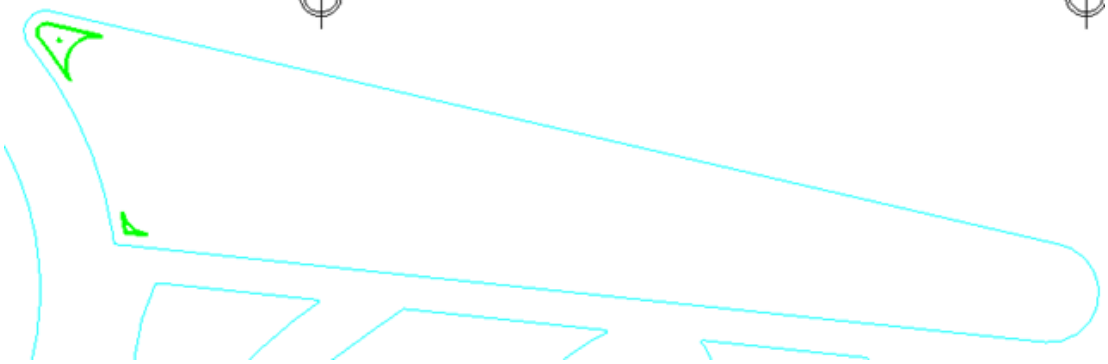


Figure 95 “Spiralcut 3” Toolpaths

4.1.6 Tool Compensation

STEP 01 Duplicate ***Spiralcut 3*** and edit the tool and parameters as below.

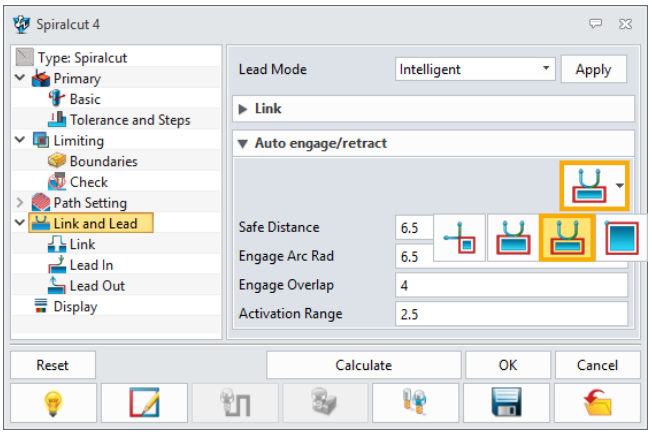


Figure 96 “Spiralcut 4” Parameters (1)

Notes:

In Spiral operation, ***Tool Compensation*** *option is not available if the engage/retract type is* ***Circular*** *or* ***None****.*

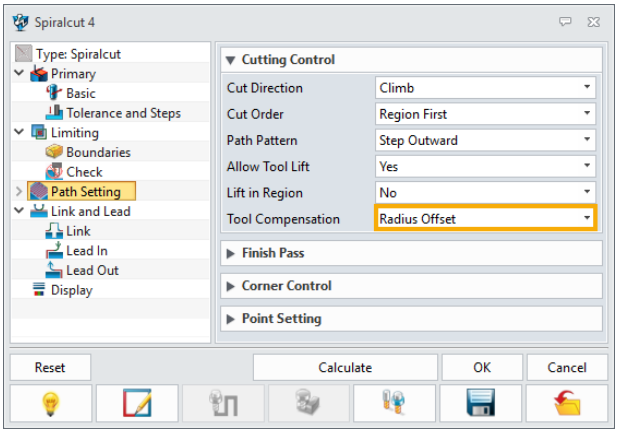


Figure 97 “Spiralcut 4” Parameters (2)

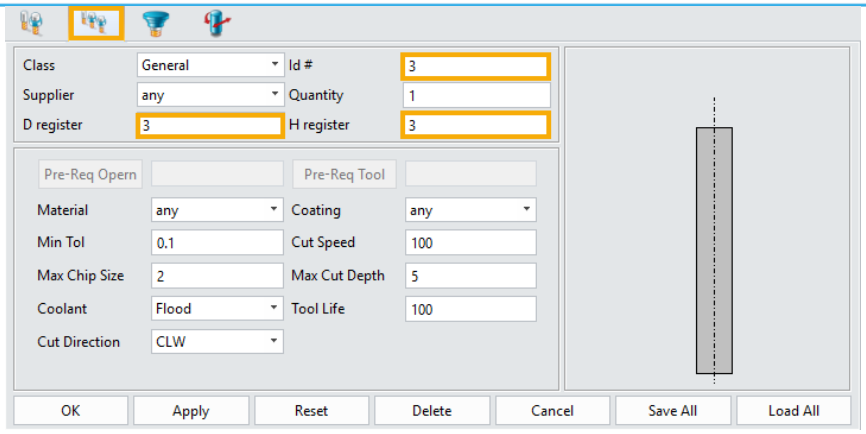


Figure 98 Edit the Tool “D6R0”

Notes:

ID #: *Tool ID*

D register: *Radius compensation ID*

H register: *Height compensation ID*

STEP 02 Calculate the toolpaths.

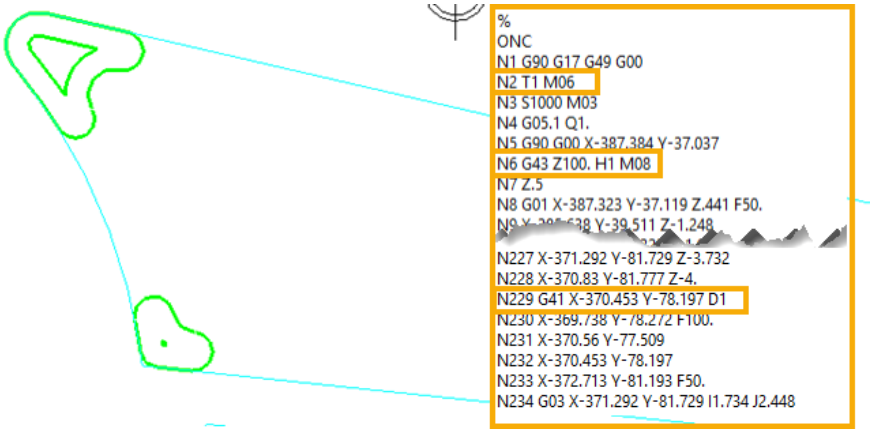


Figure 99 “Spiralcut 4” Toolpaths

4.1.7 Output

STEP 01 Insert a ***Frame*** on the bottom left corner of the part.

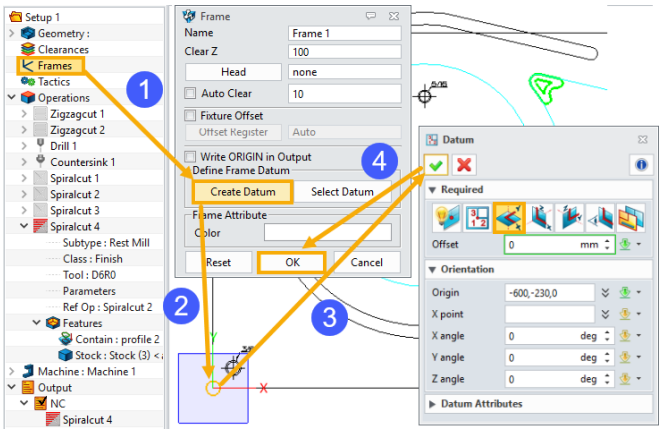


Figure 100 Insert a Frame

STEP 02 Define the machine.

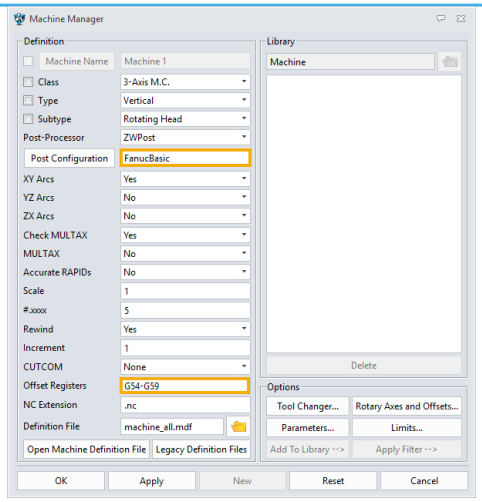


Figure 101 Machine Settings

Notes:

*The range of coordinate system IDs that input as **Offset Registers** can be used in **Frame** settings as below.*

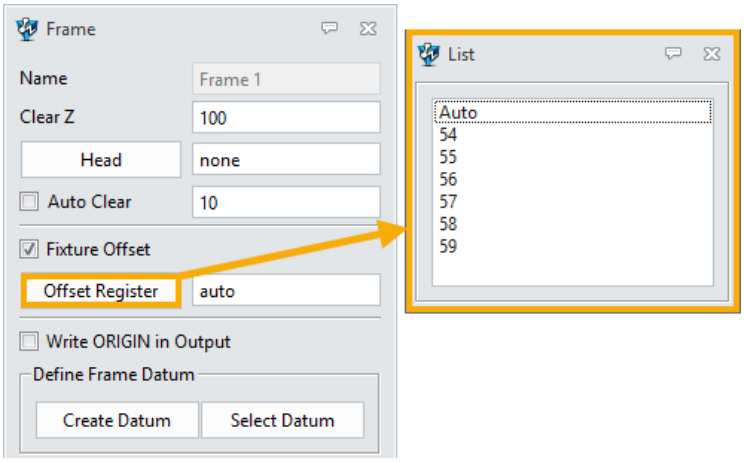


Figure 102 Frame Register Settings

STEP 03 Right-click on the text ***Output*** in CAM Manager to define output settings.

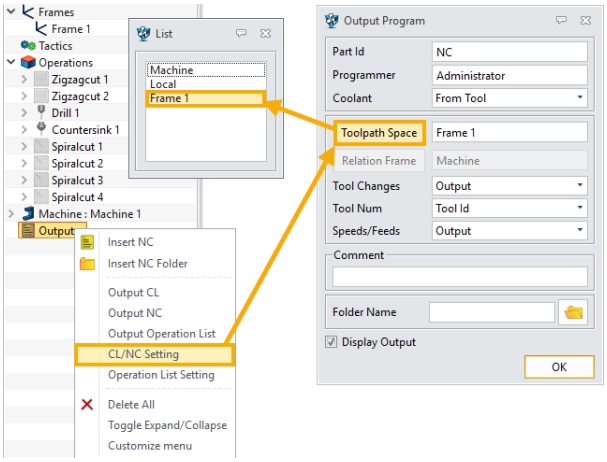


Figure 103 Define Output Frame

STEP 04 Right-click on text ***Operations*** in CAM Manager and select ***Output NC for All***

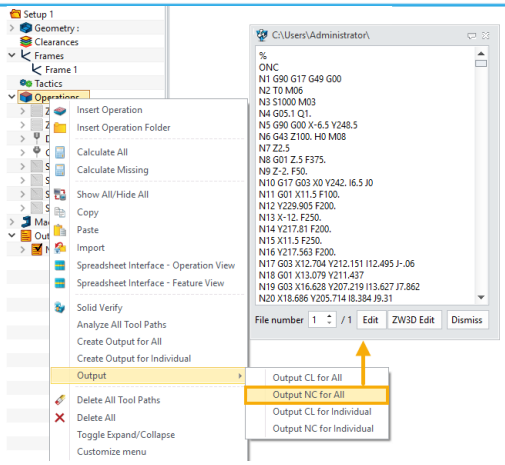


Figure 104 Output NC File

4.2 Case 2

In this chapter, we'll take a 3D part as an example, which is located in ZW3D installation folder. The typical location is "***C:\Program Files\ZWSOFT\ZW3D 201X Eng (x64)\training\2X machining\Slider.Z3***".

The part is shown below.

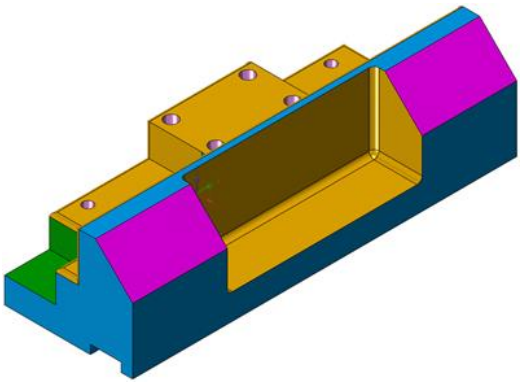


Figure 105 Part in File *Slider.Z3*

4.2.1 Sequence of Machining

1) Drill holes.
2) Cut the slopes in magenta.
3) 3.Cut the pockets in green.

4.2.2 Drill Holes

STEP 01 Create a Hole feature. There are 2 kinds of holes in this part. One is general and the other is thread hole. The detail parameters of the holes are shown in Figure 106.

STEP 02 Create a Hole Tactic based on the part and Hole feature.

STEP 03 Right-click on the text "Hole Tactic 1" and select ***Edit Tactic*** to load the tool library we created in Chapter 2.2.5.

STEP 04 Calculate the tactic to create drilling toolpaths.

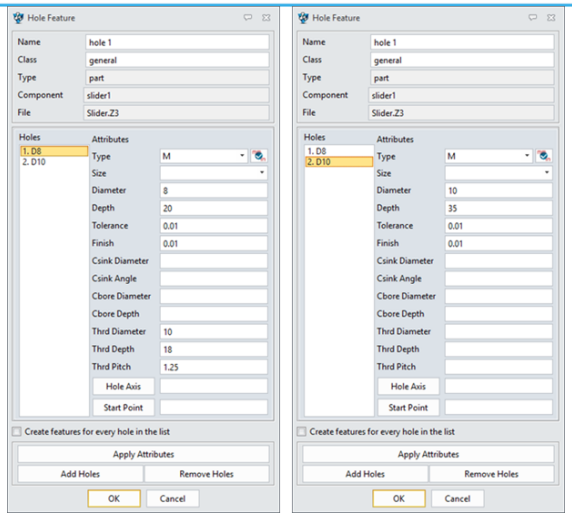


Figure 106 Hole Feature and Thread parameters

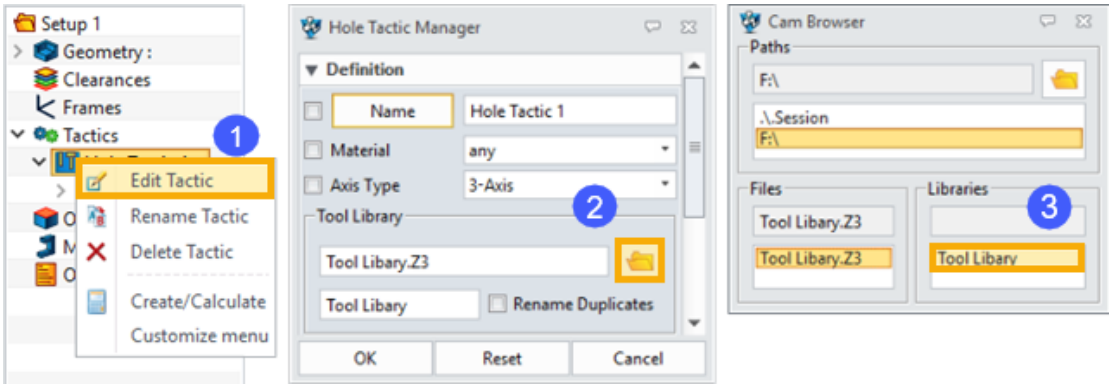


Figure 107 Edit Tactic to Load Tool Libary

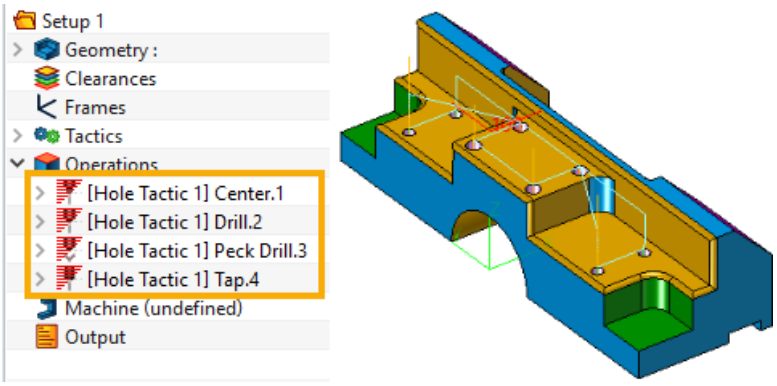


Figure 108 Hole Tactic Toolpaths

4.2.3 Cut the slopes

STEP 01 Create a ***Profile*** feature based on the top edges of the slopes. Click ***Modify Attributes*** button to set a draft angle to 45 degrees.

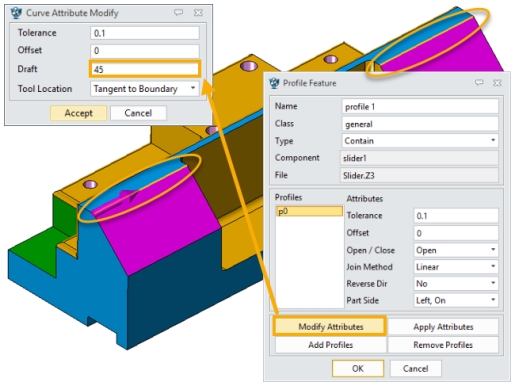


Figure 109 Create a Profile Feature

Notes:

*After clicking **Modify Attributes** button, you need to pick both edges and middle click to call out **Curve Attribute Modify** window.*

STEP 02 Create a ***Profile*** operation based on the ***Profile*** feature using the tool ***D25R2***.

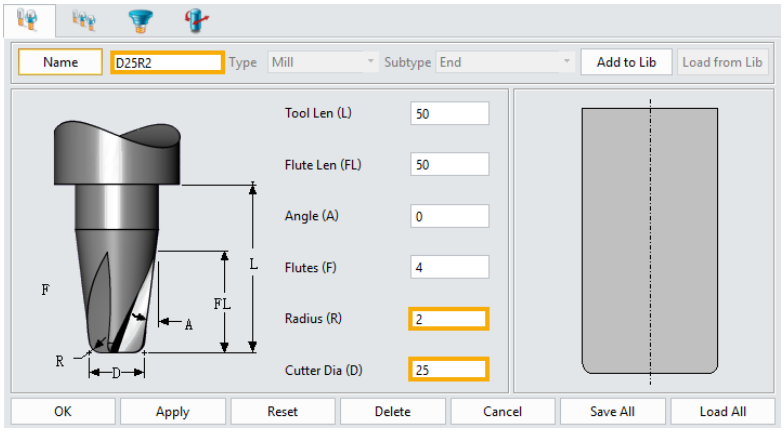


Figure 110 Tool “D25R2”

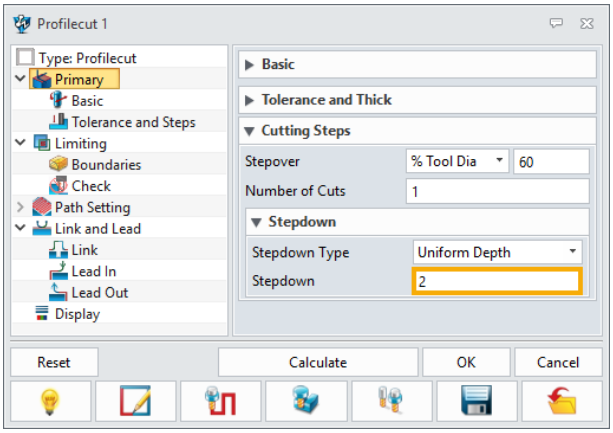


Figure 111 Parameters of “Profilecut 1” (1)

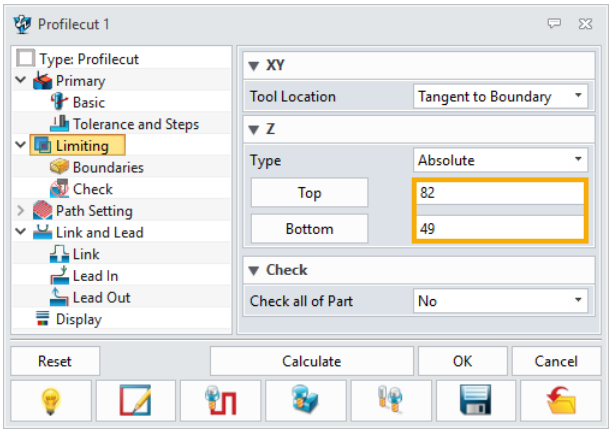


Figure 112 Parameters of “Profilecut 1” (2)

Notes:

The actual heights (Z value) of the top/bottom of the slop are 80 and 51. It is recommended to extend the range to get a better toolpath.

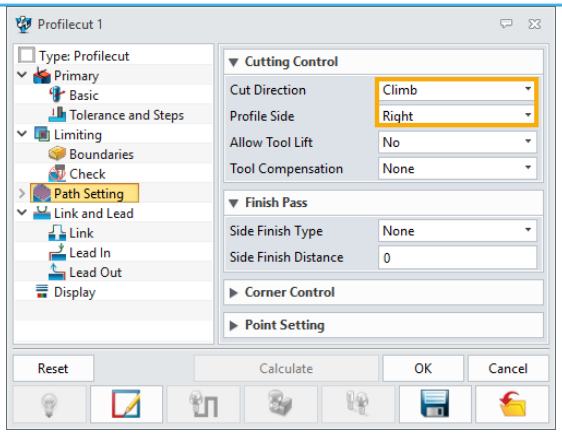


Figure 113 Parameters of “Profilecut 1” (3)

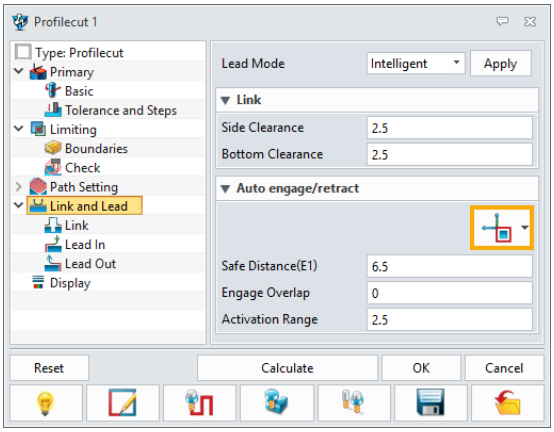


Figure 114 Parameters of “Profilecut 1” (4)

STEP 03 Calculate the toolpaths.

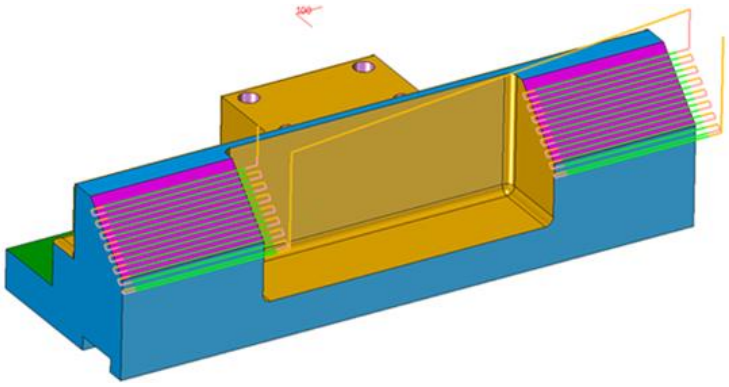


Figure 115 “Profilecut 1” Toolpaths

Notes:

*The option **Profile Side** in **Path Setting** tab is related to the direction of the Profile feature. So, if the toolpaths are on the other side of the slop, you can try to change the **Profile Side** to make it correct.*

4.2.4 Cut the Pockets

STEP 01 Create a ***Profile*** feature based on the pocket outlines. And modify the attributes of the two edges next to air.

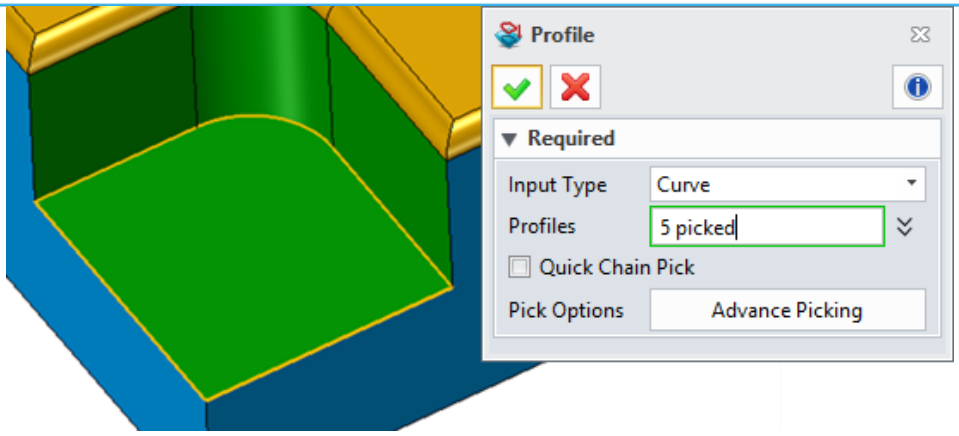


Figure 116 Select the Outlines of the Pocket

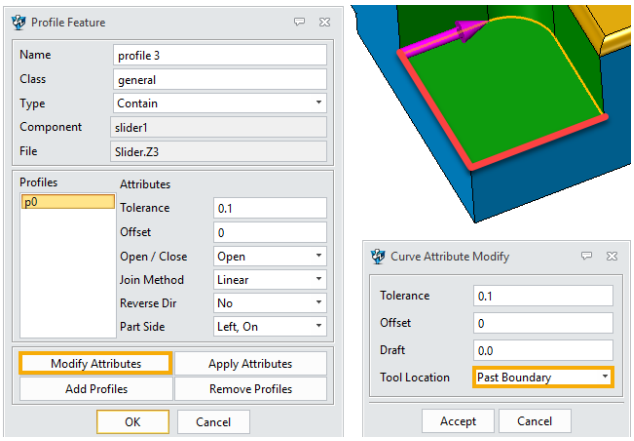


Figure 117 Modify the Attributes of the Two Edges Next to Air

STEP 02 Create ***Spiral*** operation using tool ***D16R0.8***, and add the ***Profile*** feature in.

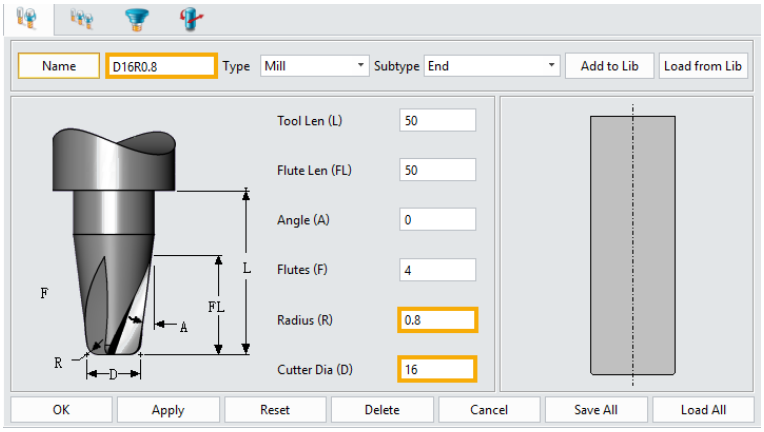


Figure 118 Tool “D16R0.8”

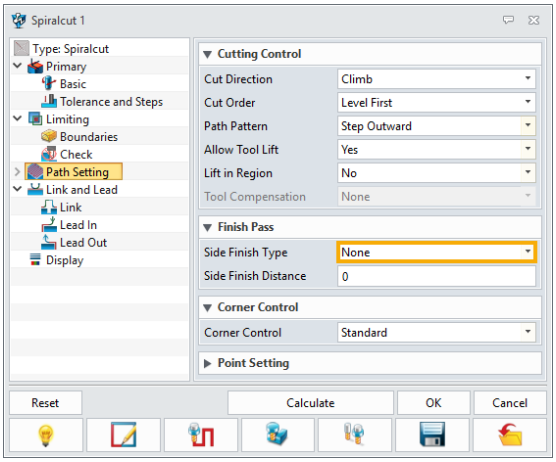


Figure 119 Parameters of “Spiralcut 1” (1)

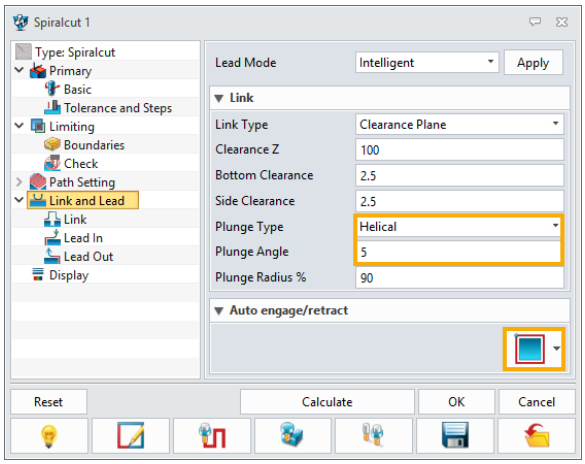


Figure 120 Parameters of “Spiralcut 1” (2)

STEP 03 Calculate the toolpaths.

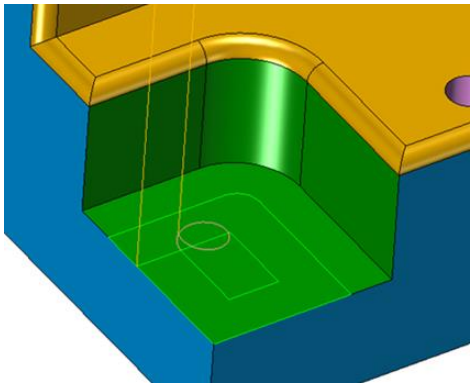


Figure 121 “Spiralcut 1” Toolpaths

5 Turning

Key Points:

- ✧ Key Points for Turning Profile
- ✧ Turning Operations

5.1 Getting Started

There are 2 key points beginners should keep in mind before creating a Turning toolpath.

- ➢ The X-axis of the global coordinate system should coincide with turning axis, and the origin should be located on the right side of the part.
- ➢ The most common feature used in Turning operations is *Profile*, which is made up of a closed anti-clockwise loop of Wireframes. The Wireframe loop is the section outline of the part, and ***Type*** of the Profile must be *Part*.

In this chapter, we’ll take another existing ZW3D part as an example, which is located in ZW3D installation folder. The typical location is **“***C:\Program Files\ZWSOFT\ZW3D 201X Eng (x64)\ training\2X machining model\Turning Case 1.Z3***”.**

5.1.1 Create a Profile

Open the file *Turning Case 1.Z3,* and go to the part Spindle. The geometry has already been located at the right place, what we have to do is to get the outline of the part.

STEP 01 Get the full section of the part with ***Planar Section*** command. Make sure the ***Plane*** is XY, ***Offset*** is 0.

STEP 02 Hide the part and erase all the unnecessary curves (dotted lines in red shown in Figure 5-2).

STEP 03 Go to CAM environment and create a Profile feature. Remember to define the Type of the Profile to “Part”, and the direction to anti-clockwise.

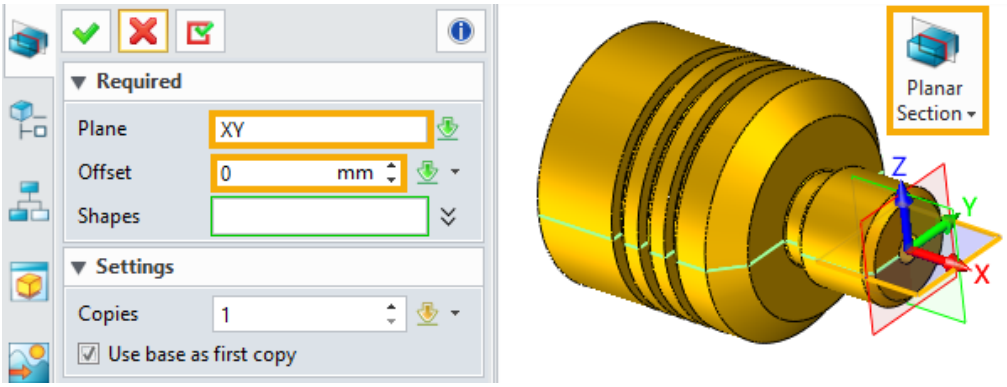


Figure 122 Section the Part

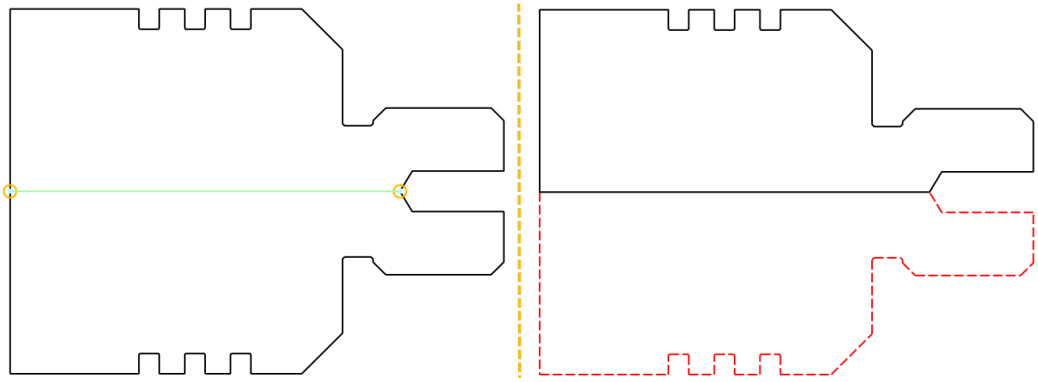


Figure 123 Erase Unnecessary Curves

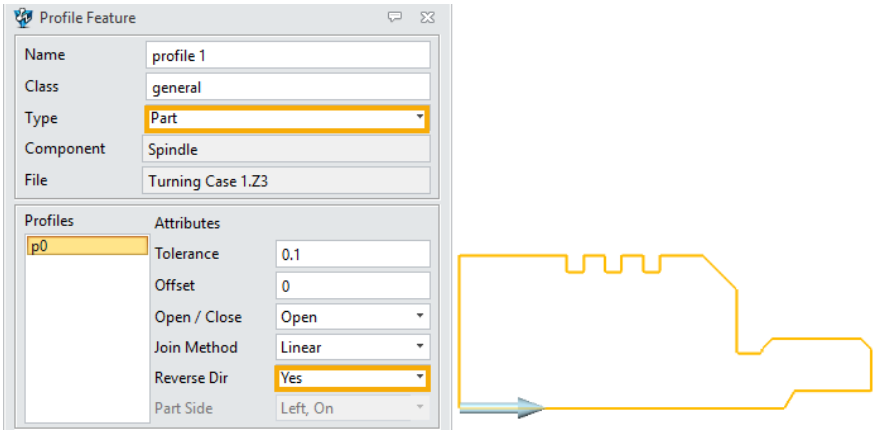


Figure 124 Create Profile

5.1.2 Overview of Turning Operations

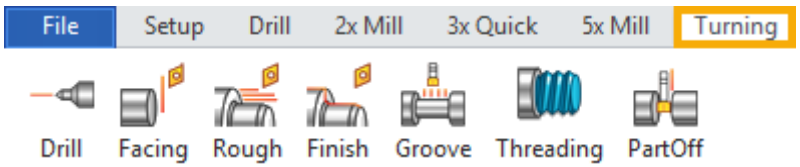


Figure 125 Turning Operations

Drill: This operation is used to drill holes in Turning.

Facing: This operation is used to machine the face of the workpiece

Rough: This operation is used to cut as much superfluous materials as possible.

Finish: This operation is used to cut allowance left by roughing operation.

Groove: This operation is used to cut grooves.

Threading: This operation is uesd to create threads.

PartOff: This operation is used to separate the objective from the workpiece.

5.2 Case Study

This case guides you through the steps to machine the part shown in Chapter 5.1. The Turning operations given above as well as their parameters will be explained during the execution of the steps.

5.2.1 Sequence of Machining

1) Face the right side of the part.
2) Rough and finish machine the outline of the part.
3) Cut the grooves.
4) Create the threads. Assume that there are threads on the surface of the right cylinder. Please see Figure 155 for detail.
5) Drill the hole.
6) Cut the part off.

5.2.2 Facing

It is assumed that a profile has been created as steps in Chapter 5.1. And the stock has been created as below.

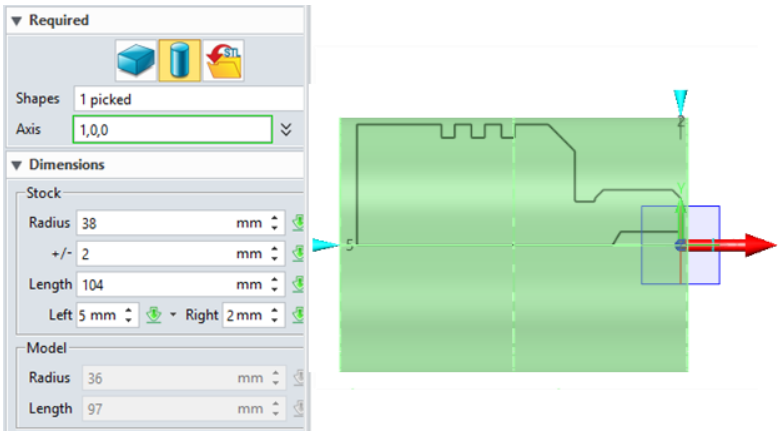


Figure 126 Stock

STEP 01 Click on the Ribbon ***Facing*** to add the operation in CAM Manager.

STEP 02 Add the profile created in Chapter 5.1 and Stock in ***Features***.

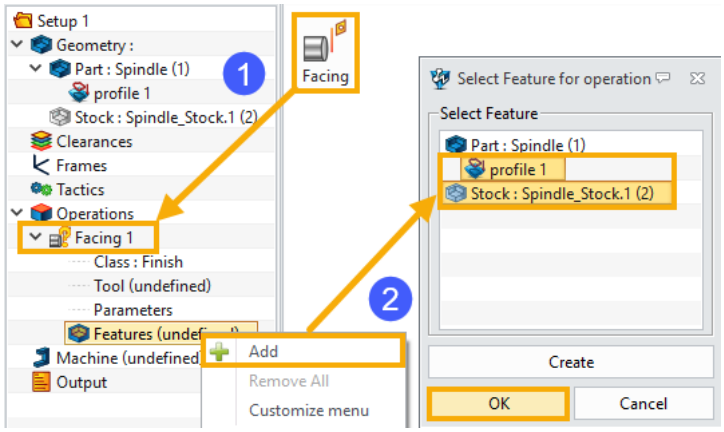


Figure 127 Create Facing Operation

STEP 03 Define the tool as below. For other parameters not shown in the figure, keep them as default.

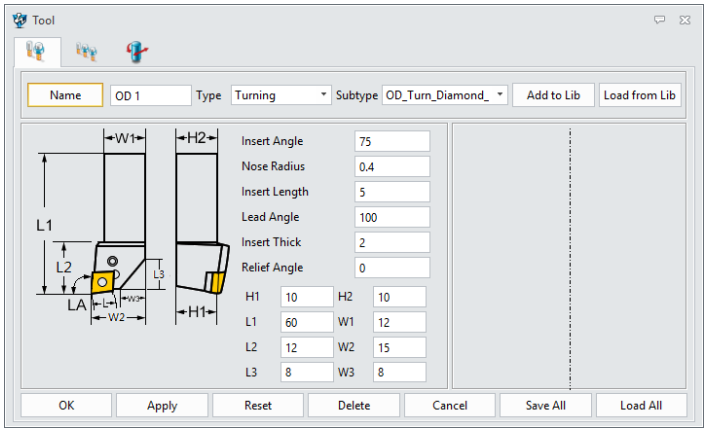


Figure 128 Tool Settings

STEP 04 Define the parameters as follows. The explanations of the options are listed below the figures.

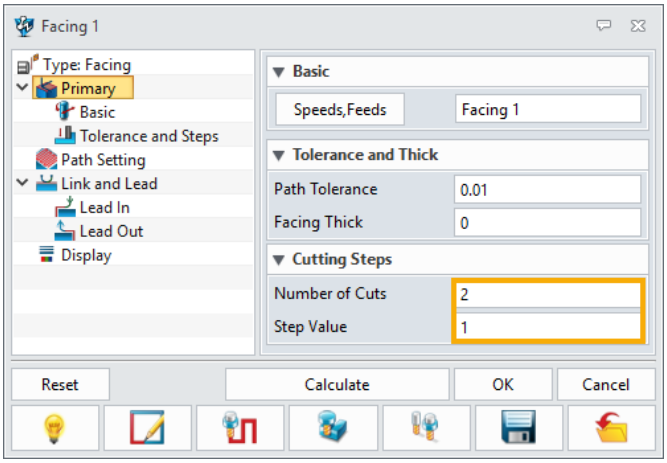


Figure 129 Parameter Settings (1)

Primary -Speeds, Feeds: Define Speed/Feed Table used in this Operation.

-Path Tolerance: Define toolpath tolerance.

-Facing Thick: Define the amount of material left after the operation is done.

-Number of Cuts: Define how many toolpaths to generate.

-Step Value: Define the amount of material to be removed by each cutting pass.

Notes:

*For options in **Primary** tab, keep **Speeds, Feeds**, **Path Tolerance** as default, and set **Facing Thick** to 0, **Number of cuts** to 2, **Step Value** to 1. This means that the tool will cut the stock twice, 1mm for each time, and there will be no extra material left after the operation is done.*

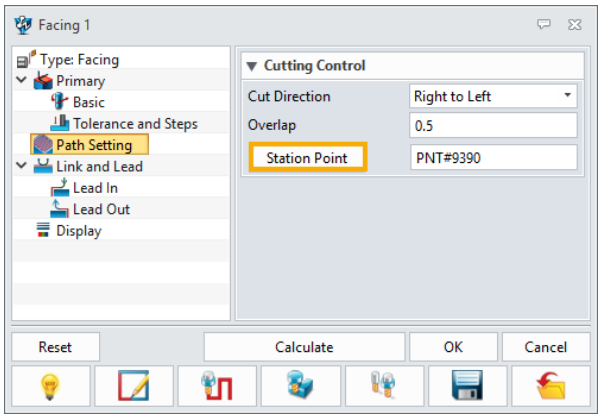


Figure 130 Parameter Settings (2)

Path Setting -Cut Direction: Define cut direction, right to left or left to right.

-Overlap: Define the distance that the tool cuts past the center of the part face.

-Station Point: Define the tool start and end location.

Notes:

*For options in Path Setting tab, set **Cut Direction** to **Right to left**, **Overlap** to 0.5, and pick the station point as you wish. This means that the tool will cut from right to left, starting from and getting back to the station point as you picked. And when the tool reaches the center of the part, it will continue cutting 0.5mm deeper.*

STEP 05 Calculate the toolpath and verify it. The toolpath is shown below.

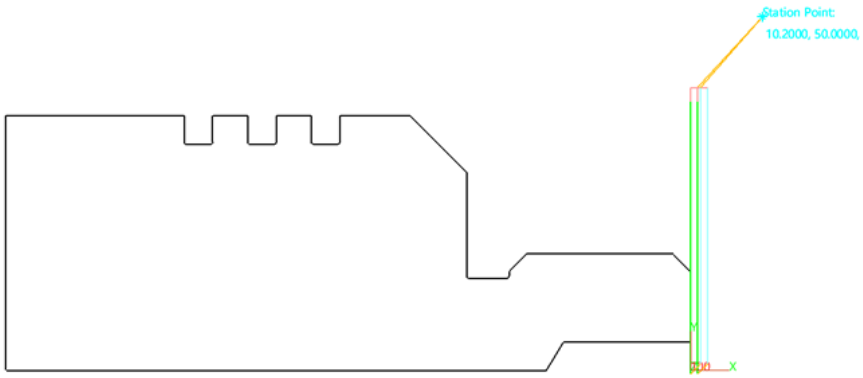


Figure 131 Facing Toolpath

5.2.3 Rough

The aim of the ***Rough*** operation is to cut the material as much as possible.

STEP 01 Click on the Ribbon ***Rough*** to add the operation in CAM Manager.

STEP 02 Add the ***profile*** and ***Stock*** in ***Features***.

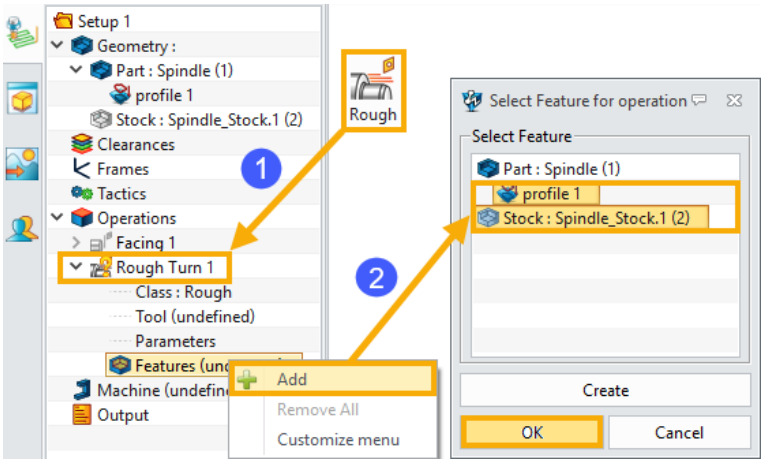


Figure 132 Create Rough Operation

STEP 03 Use the same tool (OD 1) as Facing.

STEP 04 Define the parameters. Options same as previous operations will not be listed here.

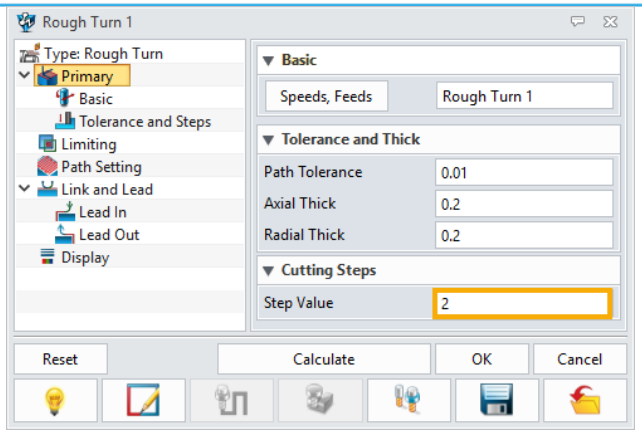


Figure 133 Parameter Settings (1)

Primary -Axial Thick: Define the amount of material left along the X-axis after the operation is done.

-Radial Thick: Define the amount of material left along radius after operation is done.

Notes:

*For options in Primary tab, keep **Speeds, Feeds**, **Path Tolerance** as default, and set **Axial Thick** and **Radial Thick** to 0.2, **Step Value** to 2. The tool will cut 2mm thick for each time, and there will be 0.2mm allowance along axial and radial directions left after the operation is done.*

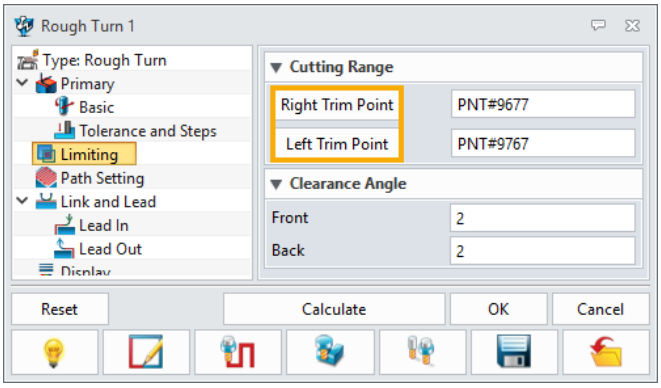


Figure 134 Parameter Settings (2)

Limiting -Right/Left Trim Point: Define the cutting range.

-Front/Back: Define the offset angle between the main cutting edge and the stock.

Notes:

*Set the **Right/Left Trim Point** at both ends of the part.*

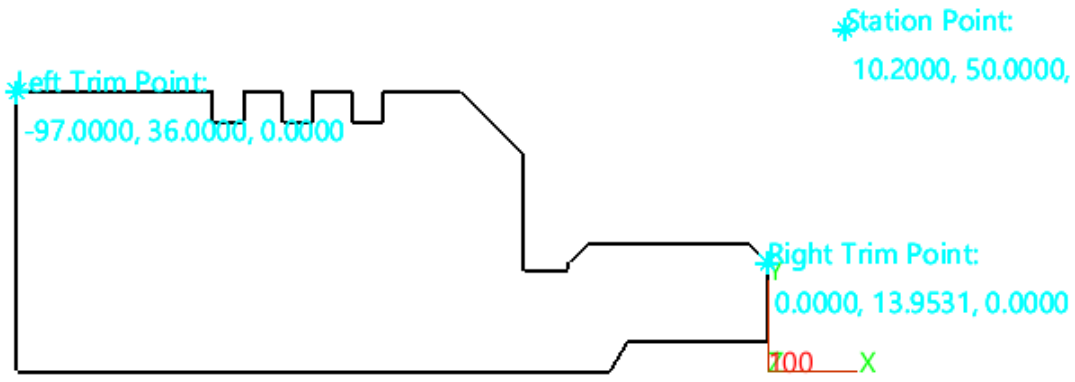


Figure 135 Trim Points and Station Point

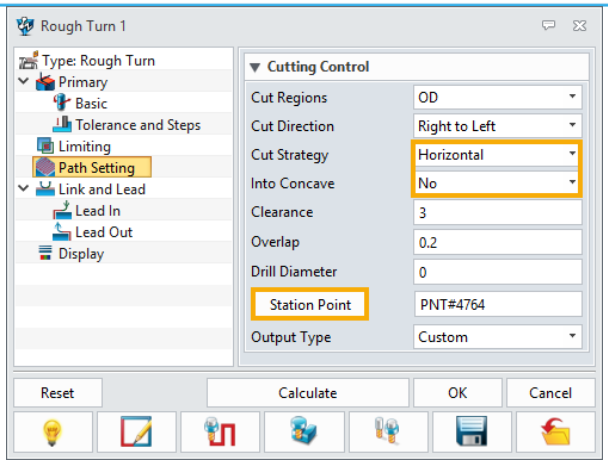


Figure 136 Parameter Settings (3)

Path Setting -Cut Regions: Define cut regions, outside (OD) or inside (ID).

-Cut Strategy: Define the strategy used to generate toolpath. Different Strategies are compared in Figure 136.

-Into Concave: Define whether the tool is allowed to cut into concaves. Different settings are compared in Figure 137.

-Clearance: Define vertical distance when the tool begins to machine the concave.

-Drill Diameter: Define the diameter of the hole in the part center, the toolpath will avoid machining the area.

-Output Type: Define whether the output contains circle commands.

Notes:

*For options in Path Setting tab, set **Cut Regions** to **OD**, **Cut Direction** to **Right to left**, **Cut Strategy** to **Horizontal**, **Into Concave** to **NO**, and pick the same station point as **Facing**. This means that the tool will cut outside surface of the part from right to left; the toolpaths are horizontal; and will not cut into concaves.*

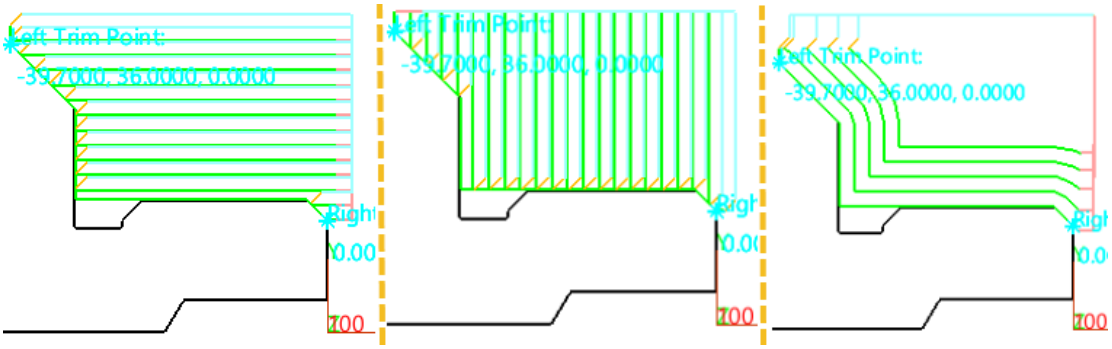


Figure 137 Horizontal Vertical Pattern

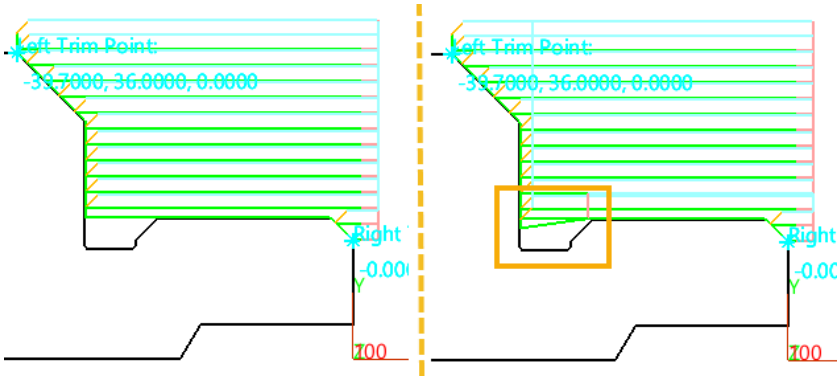


Figure 138 NO YES

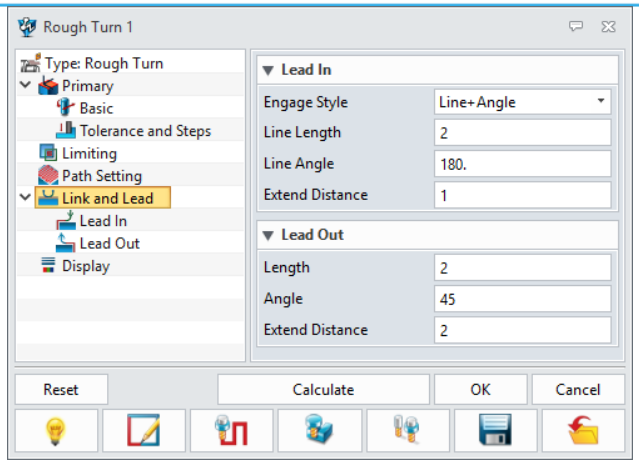


Figure 139 Parameter Settings (4)

Link and Lead-Engage Style: Define how tool engages into the stock.

-Link Length: Define distance of lead in toolpaths.

-Line Angle: Define the angle between lead-in and cutting toolpaths.

-Extend Distance (Lead In): Define extend cutting distance of toolpaths before cutting into stock.

-Length: Define the length of lead-out toolpaths.

-Angle: Define the angle between lead-out and cutting toolpaths.

-Extend Distance (Lead Out): Define extend cutting distance of toolpaths before retracting the tool.

Notes:

*For options in Link and Lead tab, set **Extend Distance (Lead Out)** to 2. This means that the tool will continue cutting 2mm before retracting.*

STEP 05 Calculate the toolpath and verify it. The toolpaths are shown below.

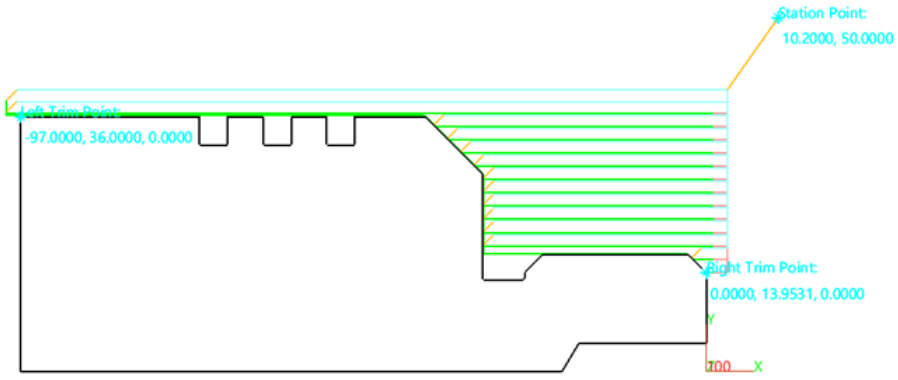


Figure 140 Rough Toolpaths

5.2.4 Finish

The aim of ***Finish*** operation is to cut the remained material left by Rough turning operation.

STEP 01 Click on the Ribbon ***Finish*** to add the operation in CAM Manager.

STEP 02 Add the profile in ***Features***.

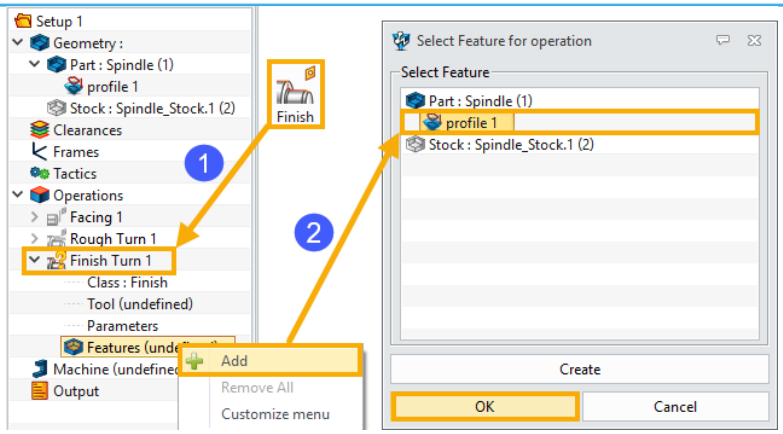


Figure 141 Create Rough Operation

STEP 03 Use the same tool (OD 1) as ***Rough***.

STEP 04 Define the parameters. Options same as previous operations will not be listed here.

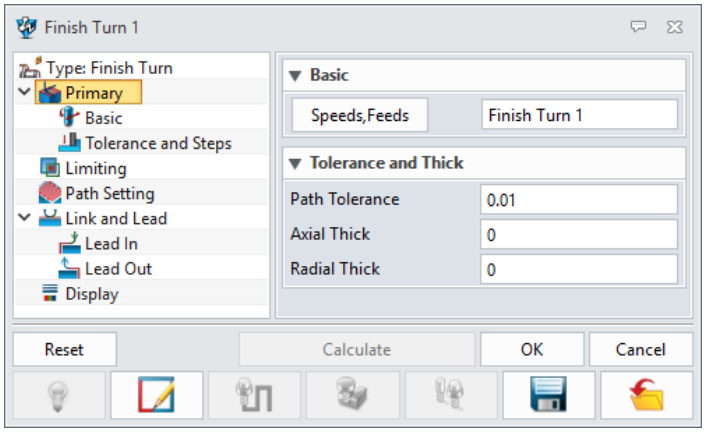


Figure 142 Parameter Settings (1)

Notes:

*For options in Primary tab, keep **Speeds, Feeds, Path Tolerance** as default, and set **Axial Thick** and **Radial Thick** to 0. This means that no extra material will be left after the operation is done.*

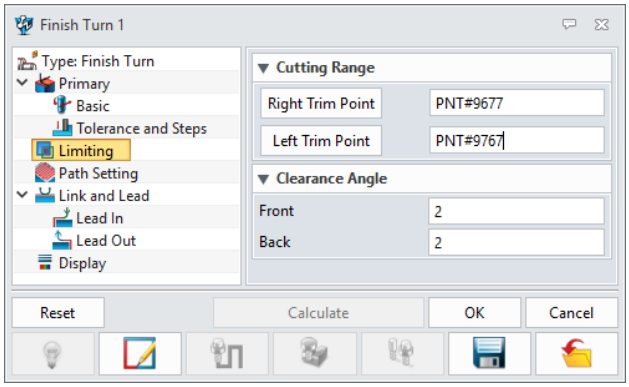


Figure 143 Parameter Settings (2)

Notes:

*All the settings in this tab are same as those in **Rough** operation.*

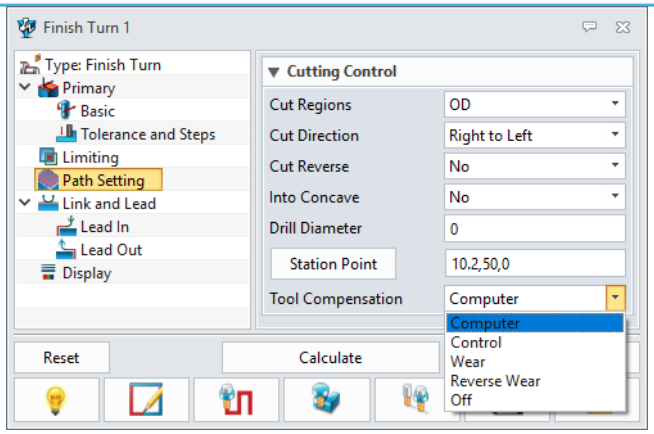


Figure 144 Parameter Settings (3)

Path Setting -Cut Reverse: Define whether to reverse cutting and don't change the toolpaths shape.

- Tool Compensation: Define the compensation type.

If set this to ***Computer***, the toolpaths will get compensation when calculating.

If set this to ***Control***, the toolpaths won't get compensation when calculating, but will output ***G42*** command.

If set this to ***Wear***, the toolpaths will get compensation when calculating and output ***G42*** command also.

If set this to ***Reverse Wear***, the toolpaths will get compensation when calculating and output ***G41*** command instead of ***G42***.

If set this to ***Off***, the toolpaths won't get compensation at all.

Notes*:*

*For options in **Path Setting** tab, set **Cut Reverse** to **NO** and **Tool Compensation** to **Computer**, the others are same as **Rough**. This means that the tool will cut outside surface of the part from right to left with compensation and will not cut into concaves.*

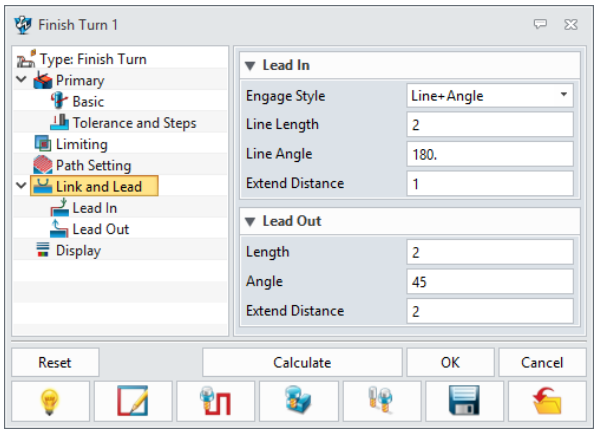


Figure 145 Parameter Settings (4)

Notes:

*All the settings in this tab are the same as **Rough** operation.*

STEP 05 Calculate the toolpath and verify it. The toolpaths are shown below.

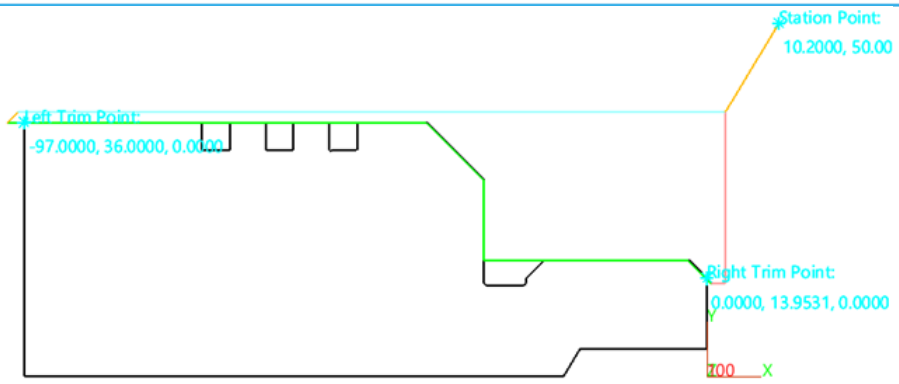


Figure 146 Finish Toolpaths

5.2.5 Groove

The aim of this operation is to cut the grooves of the part.

STEP 01 Click on the Ribbon ***Groove*** to add the operation in CAM Manager.

STEP 02 Add the profile in ***Features***.

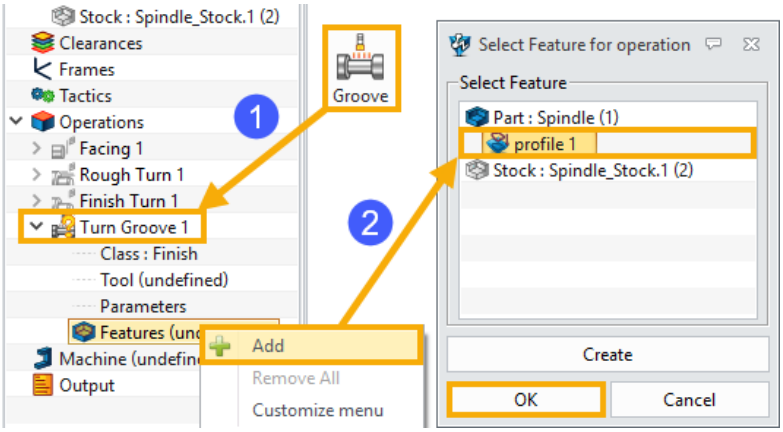


Figure 147 Create Rough Operation

STEP 03 Define the tool as ***GV 1*** shown in below Figure. The Insert Width (W) should not be bigger than the width of the grooves.

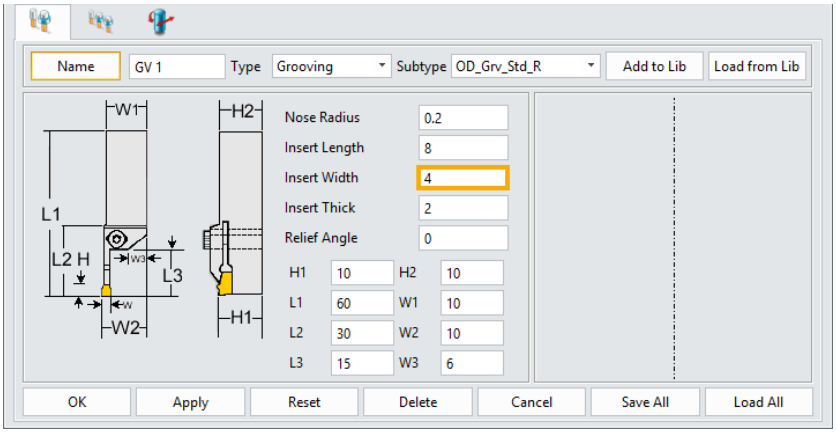


Figure 148 Groove Tool

STEP 04 Define the parameters. Options same as previous operations will not be listed here.

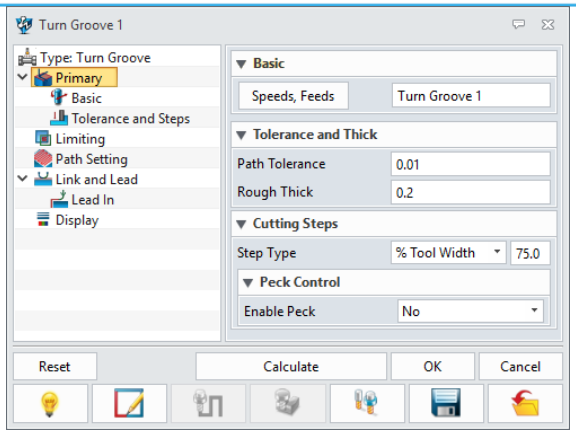


Figure 149 Parameter Settings (1)

Primary -Rough Thick: Define the amount of material left after rough cutting.

-Step Type: Define the step size.

-Enable Peck: Define whether to enable peck for rough cutting.

Notes:

*For options in Primary tab, keep **Speeds, Feeds, Path Tolerance** as default, and set **Rough Thick** to 0.2, **Enable Peck** to **NO**. This means that there will be 0.2mm material left after rough cutting.*

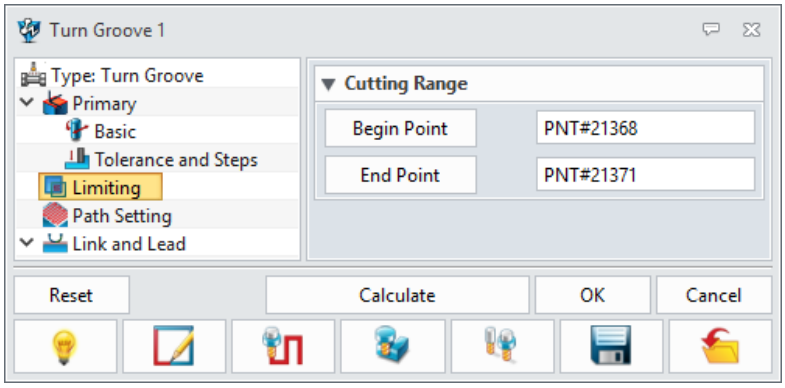


Figure 150 Parameter Settings (2)

Notes:

The begin & end points are picked as below.

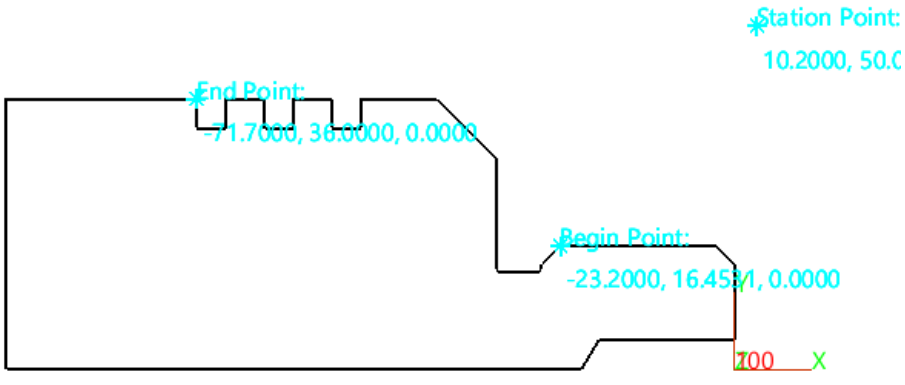


Figure 151 Begin Point, End Point, and Station Point

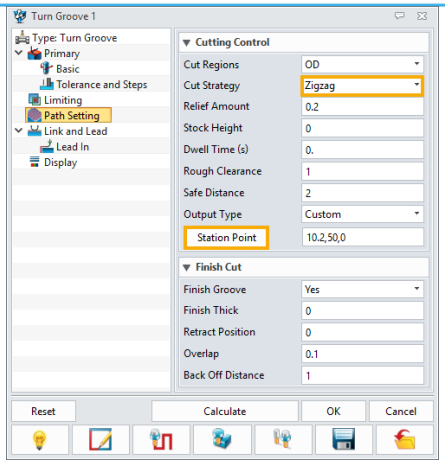


Figure 152 Parameter Settings (3)

Path Setting -Cut Strategy: Select the cut strategy from ***Zigzag*** and ***Alternate***. The difference between is shown in Figure 153.

-Relief Amount: Define the distance from the tool to groove side when retracting.

-Stock Height: Define the amount of stock above the top of the groove before the rough turning begins.

-Dwell Time: Define the dwell time the tool stays at the bottom of the groove in the rough grooving.

-Rough Clearance: Define the clearance above the top of grooves.

-Safe Distance: Define the distance above the part when the tool is moving from a groove region to another.

-Output Type: Define whether to output circle command in NC files.

-Finish Groove: Define whether to enable finish cut after rough grooving.

-Finish Thick: Define the thickness of material after finish grooving is done.

-Retract Position: Define the distance from the tool to groove side when retracting.

-Overlap: Define the overlap distance between two cuts.

-Back Off Distance: Define retract distance of finish grooving, only take effect when the second pass of finish grooving exists.

Notes:

*For options in **Path Setting** tab, set **Cut Strategy** to **Zigzag** and **Finish Groove** to **Yes**. This means that the tool will cut the grooves in a Zigzag shape of toolpath and enables a finish cut after roughing.*

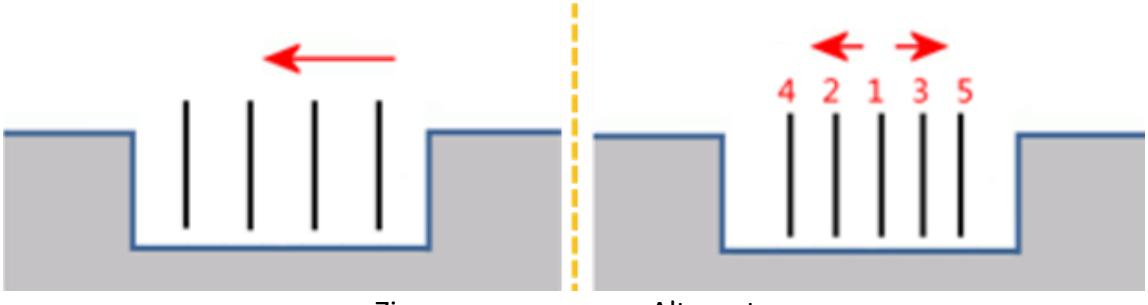


Figure 153 Cut Strategy Zigzag VS Alternate

STEP 05 Calculate the toolpath and verify it. The toolpaths are shown below.

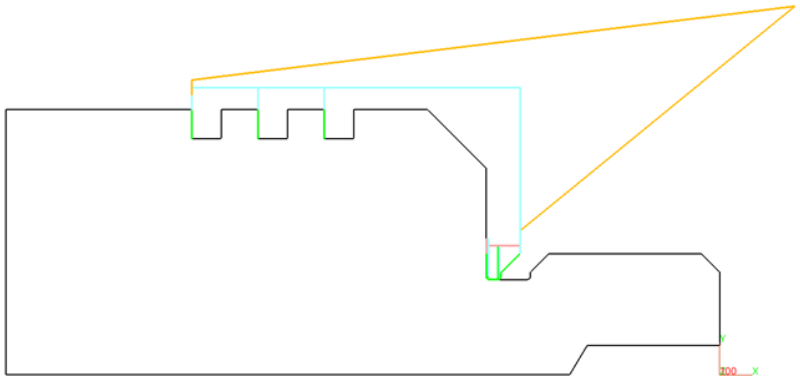


Figure 154 Groove Toolpaths

5.2.6 Threading

Assume that there are threads on the right cylinder of the part as below figure shows.

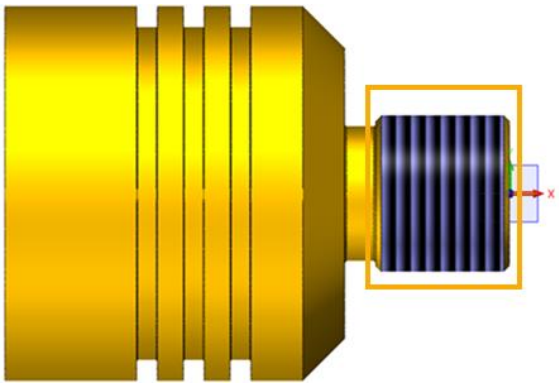


Figure 155 Threads location

STEP 01 Click on the Ribbon ***Groove*** to add the operation in CAM Manager.

STEP 02 Add the profile in ***Features***.

STEP 03 Define the tool as TD 1 shown in Figure 157.

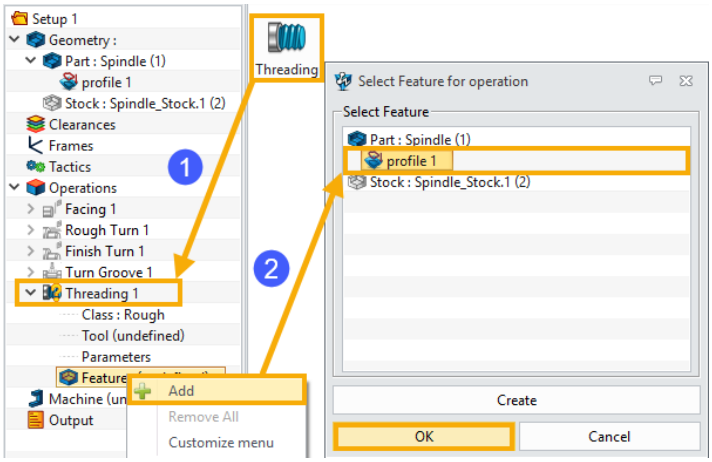


Figure 156 Create Threading Operation

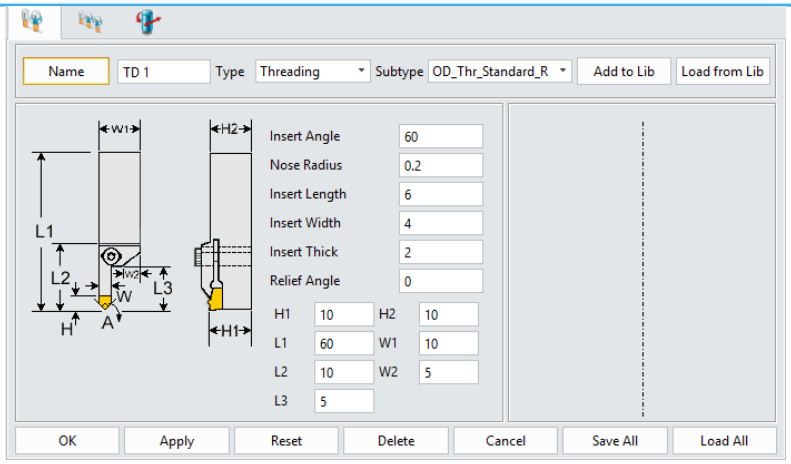


Figure 157 Threading Tool

STEP 04 Define the parameters. Options same as previous operations will not be listed here.

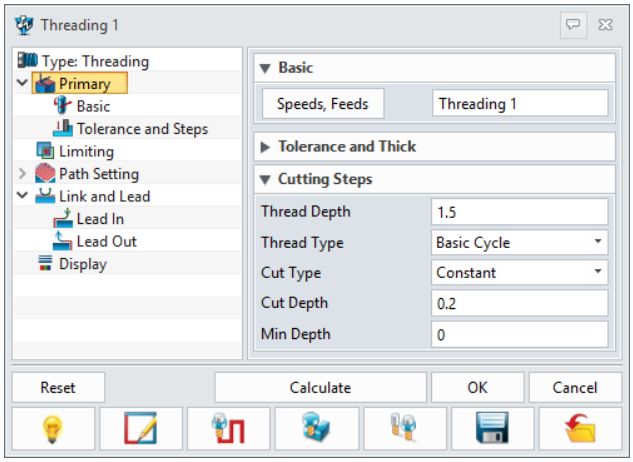


Figure 158 Parameter Settings (1)

Primary -Thread Depth: Define the vertical distance between the crest and root.

-Thread Type: Define the method to complete the rough threading motion.

-Enable Peck: Define whether to enable peck for rough cutting.

Notes:

*For options in **Primary** tab, keep **Speeds**, **Feeds**, **Path Tolerance** as default, and set **Rough Thick** to 0.2, **Enable Peck** to **NO**. This means that there will be 0.2mm material left after rough cutting.*

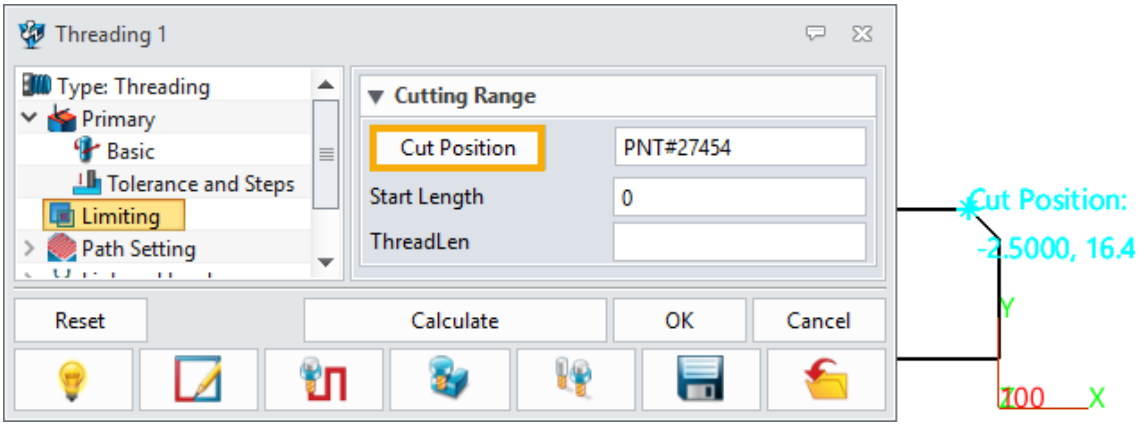


Figure 159 Parameter Settings (2)

Limiting -Cut Position: Select a point that locates on the thread line.

-Start Length: Define the distance from the start point and ***Cut Position***.

-ThreadLen: Define the length of threads.

Notes:

*For options in Limiting tab, select **Cut Position** as Figure 159 shows, and set **Start Length** to 0, **ThreadLen** to 0 or null. This means that threads will start from the selected point, ZW3D will calculate the thread length automatically.*

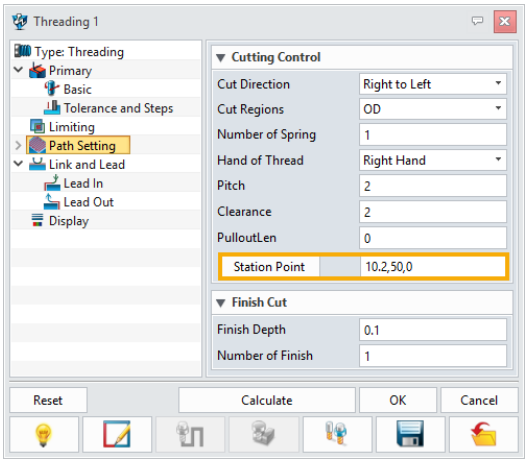


Figure 160 Parameter Settings (3)

Path Setting -Number of Spring: Define the number of starts.

-Hand of Thread: Define the rotating direction of threads.

-Pitch: Define the pitch.

-PulloutLen: Define the distance from the point that the tool begins to leave the part to the end of threads.

-Finish Depth: Define the finish cutting depth.

-Number of Finish: Define the number of finish toolpaths.

Notes:

*For options in **Path Setting** tab, set **Number of Spring** to 1, **Hand of Thread** to **Right Hand**, **Pitch** to 2, **Finish Depth** to 0.1 and **Number of Finish** to 1. This means the threads will be 1-start and right-hand, and there will be 1 finish cut after roughing.*

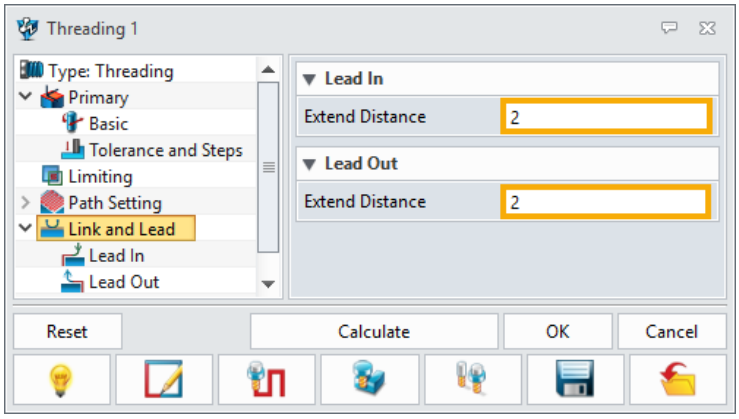


Figure 161 Parameter Settings (4)

Link and Lead -Extend Distance: Define an extra cutting distance before/after cutting the part.

Notes:

*For options in **Link** and **Lead** tab, set both **Extend Distance** to 2. This means the cutting will get started 2mm earlier before cutting into the part and continue cutting 2mm after finishing the part.*

STEP 05 Calculate the toolpath and verify it. The toolpaths are shown below.

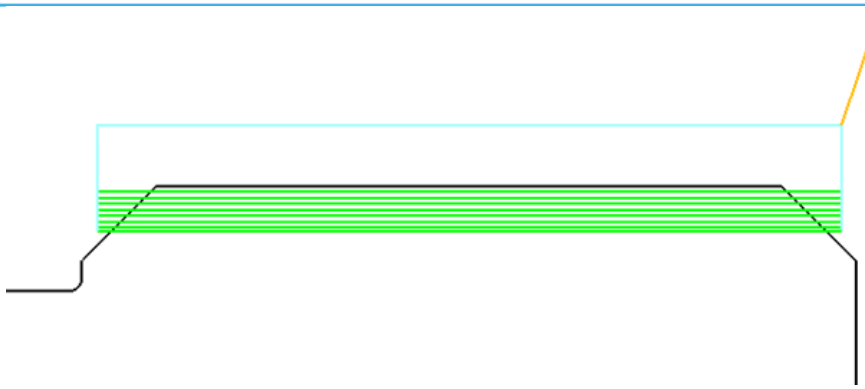


Figure 162 Threading Toolpaths

5.2.7 Drill

Before starting a drill operation, users should collect the hole information in advance, including the diameter, depth, etc.

STEP 01 Click on the Ribbon ***Drill*** to add the operation in CAM Manager.

STEP 02 Add the profile in ***Features***.

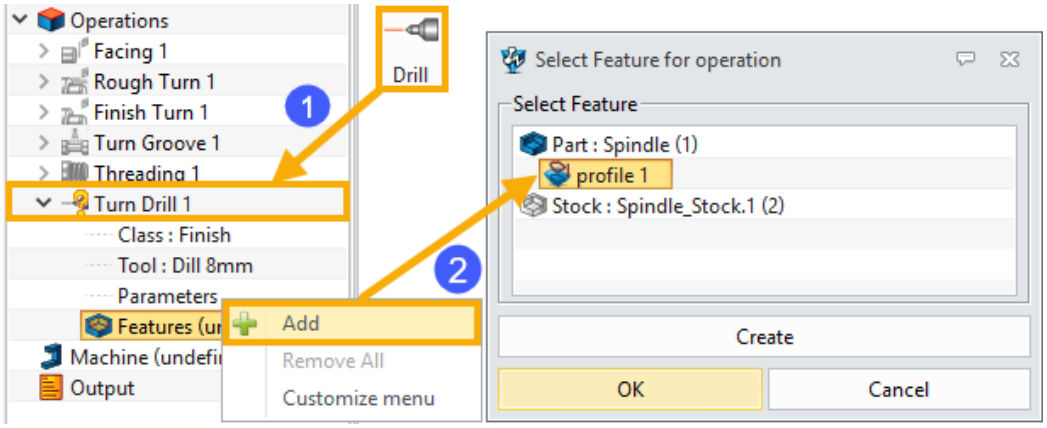


Figure 163 Create Drill Operation

STEP 03 Define the tool as ***Drill 8mm***.

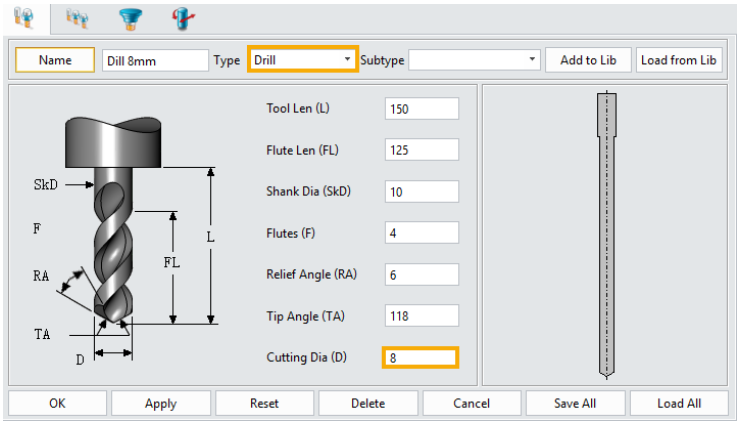


Figure 164 Threading Tool

STEP 04 Define the parameters. Options same as previous operations will not be listed here.

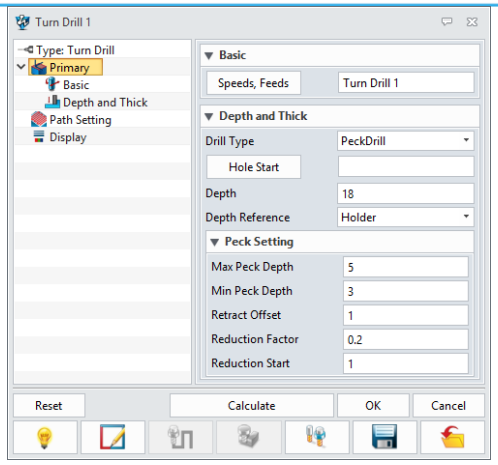


Figure 165 Parameter Settings (1)

Primary -Drill Type: Select the drilling method.

-Hole Start: Define the start point of the hole.

-Depth: Define the hole depth.

-Depth Reference: Define which location on the tool should be the reference point to specify the hole depth. When ***Drill Type*** is set to ***Center*** or ***Bore***, this option is unavailable.

-Max/Min Peck Depth: Define the Max/Min depth for each peck.

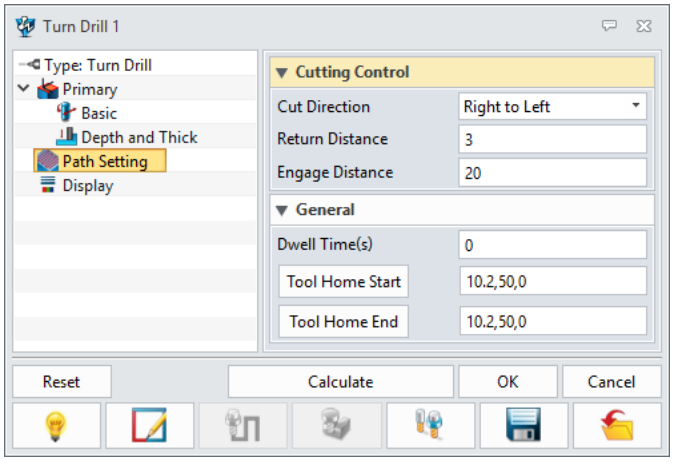
-Retract Offset: Define the retract distance before each peck.

-Reduction Factor: Define the factor by which the peck depth is reduced. Please see HELP document for detail.

-Reduction Start: Define the start number of pecks to use reduction factor.

Notes:

*For options in **Primary** tab, keep **Speeds**, **Feeds**, as default, and set **Drill Type** to **Peck Drill**, **Depth** to "18", **Depth Reference** to **Holder** and leave **Hole Start** to null. This means that the drilling is a peck type, and the distance from the beginning point to the tool holder is 18mm. ZW3D will find the start point of drilling automatically.*



Parameter Settings (2)

Path Setting -Return Distance: Define the distance from the start point of each peck to the top of the hole.

-Engage Distance: Define engage distance.

-Tool Home Start/End: Define the start/end point of drilling.

Notes:

*For options in **Path Setting** tab, set **Return Distance** to 3, **Engage Distance** to 20, **Dwell Time** to 0, and **Tool Home Start/end** to station point same as previous operations. Keep others as default. This means the drilling will start and end at the same point as station point.*

STEP 05 Calculate the toolpath and verify it. The toolpaths are shown below.

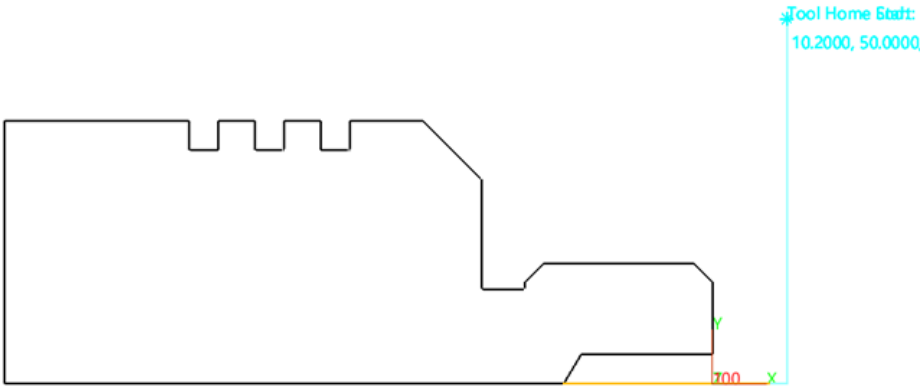


Figure 166 Drill Toolpaths

5.2.8 PartOff

STEP 01 Click on the Ribbon ***PartOff*** to add the operation in CAM Manager.

STEP 02 Add the profile in ***Features***.

STEP 03 Define the tool as ***PartOff 1***.

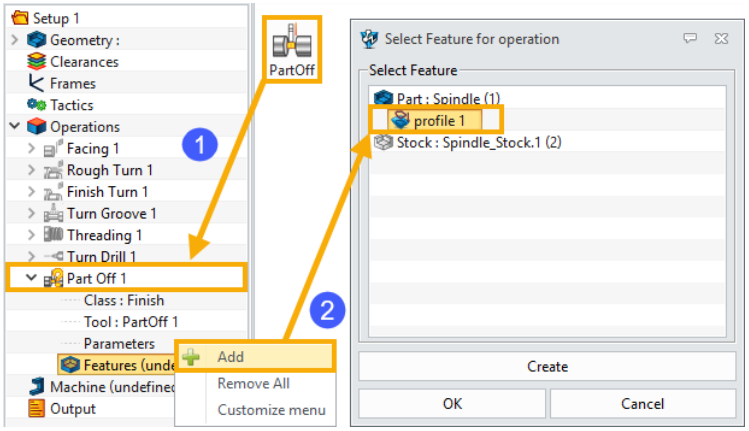


Figure 167 Create Drill Operation

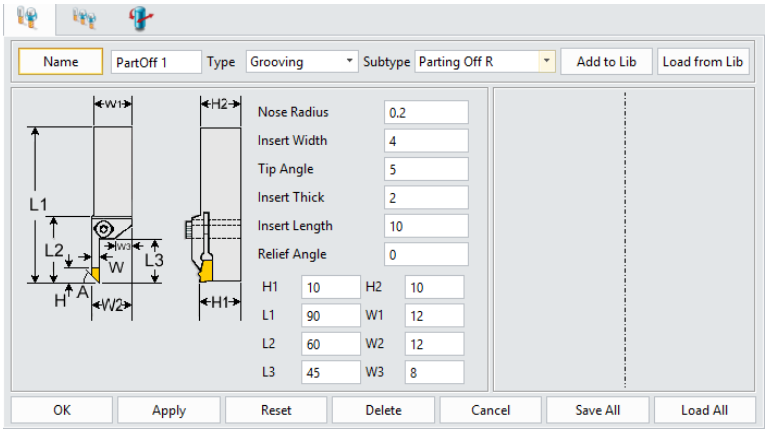


Figure 168 Threading Tool

STEP 04 Define the parameters. Options same as previous operations will not be listed here.

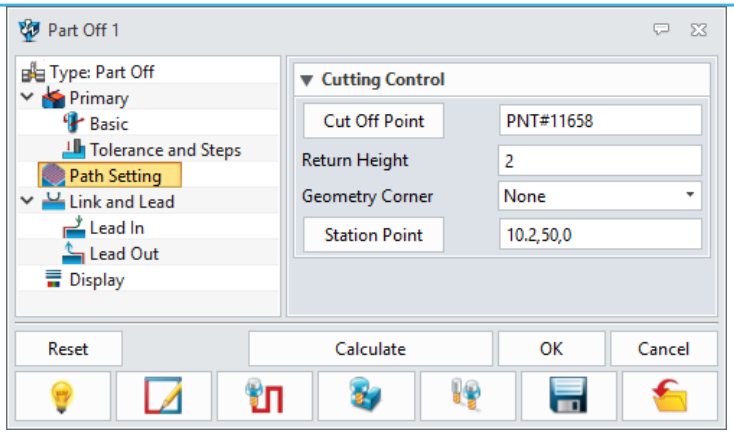


Figure 169 Parameter Settings (1)

Path Setting -Cut Off Point: Select the cutoff location.

-Geometry Corner: Define whether to add a chamfer or fillet corner at the cutoff position.

Notes:

Select the right end as the cutoff point and the same station point as previous.

STEP 05 Calculate the toolpath and verify it. The toolpaths are shown below.

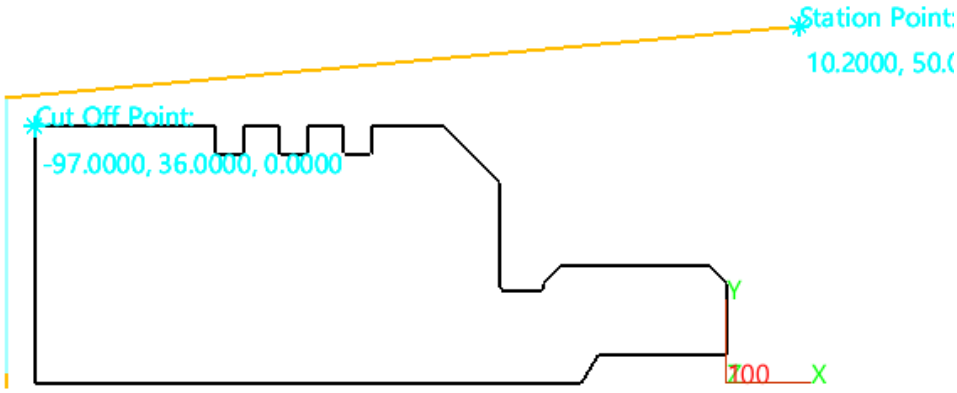


Figure 170 PartOff Toolpaths

5.2.9 Solid Verify All Toolpaths

It is recommended to check all the toolpaths by ***Solid Verify*** after calculation.

STEP 01 Right-click on the text ***Operations*** in CAM Manager and select ***Solid Verify***.

STEP 02 Click the play button to start the simulation.

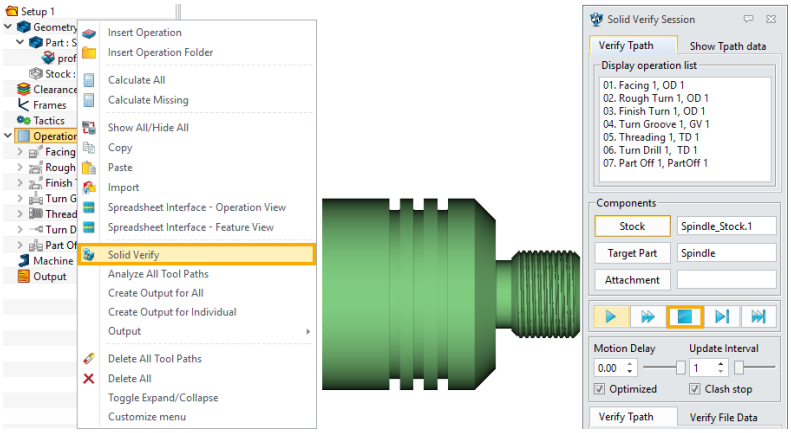


Figure 171 Solid Verify All Operations

5.2.10 Output NC File

STEP 01 Right-click on the text ***Machine*** in the CAM Manager and select ***Manager*** to call out Machine

Manager.

STEP 02 Define the ***Class***, ***Post Configuration*** and ***Tool Changer*** as below.

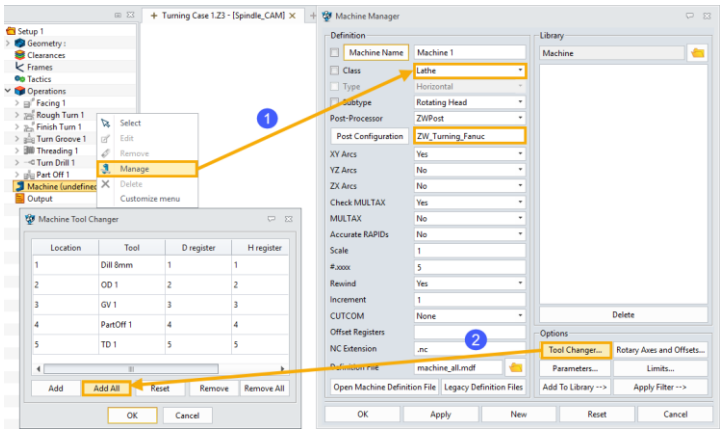


Figure 172 Machine Manager

STEP 03 Double-click on text ***Output*** in CAM Manager to add an output ***NC***.

STEP 04 Right-click on text ***Output*** in CAM Manager and select ***CL/NC Setting*** to call out ***Output Program*** dialog. Then define output file location.

STEP 05 Right-click on text ***NC*** in CAM Manager and select ***Add Operations*** to include all the operations we have created.

STEP 06 Right-click on text ***NC*** in CAM Manager and select ***Output NC***.

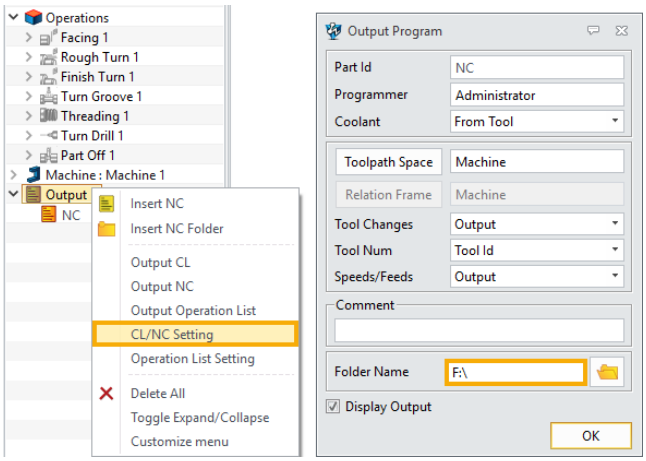


Figure 173 CL/NC Setting

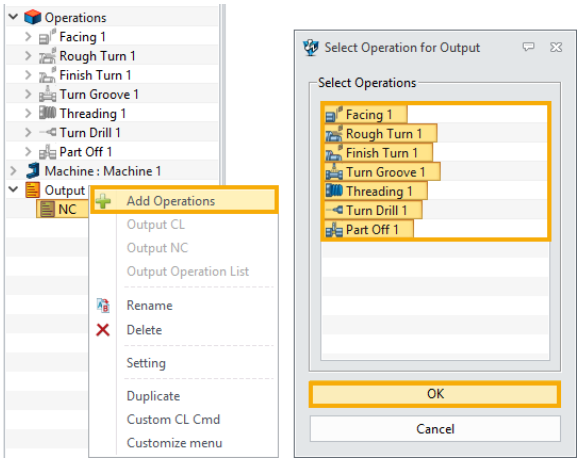


Figure 174 Add Operations in

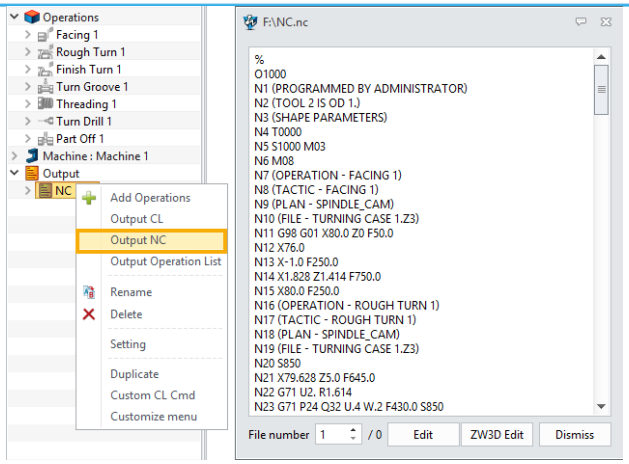


Figure 175 Output NC File

6 Epilogue

Thank you for your valuable time.

In this tutorial, we've shown you how to create 2X milling and turning tool paths with the key parameters explained in detail. We hope this tutorial can help you understand the way to apply 2X 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