ZW3D Turning Training Manual

Floor 4, NO.886, Tianhe North Road, Guangzhou 510635 P.R.China
(8620)38289780

ZW3D™ V2014 CAM Training Guide

Printed in the P. R. China.

Forward

The full version of the ZW3D documentation is available by selecting **Help Browser** from the ZW3D **Help Menu**. Go to the " **Program Folder \doc**" directory on the ZW3D program directory and open the file "**ZW3D.chm**".

This **Turning Training Guide** is available as a printed manual and in PDF format. To view or print the PDF version, install the Adobe Acrobat Reader and open the corresponding files in the " **Program Folder\PDF**" folder in the program directory. Refer to the **HELP Manual** for information relating to specific tasks.

NOTES:

1. For best results, view and print the PDF version of the documentation using the Adobe Acrobat Reader.
2. Since the **Turning Training Guide** is only a portion of the complete documentation, it is highly recommended that you utilize the **HELP Manual**. You will find additional topics, more information on command options, input options, additional command notes, and cross-links to related tasks and topics.
3. It should be noted that the HELP Manual will always be more up-to-date than any printed or PDF material. If you find any discrepancies between the two, always refer to the HELP Manual
4. While you are working with ZW3D, simply press F1 to view the documentation about the command you select.

To help you customize your CAD/CAM requirements, please contact our Customer Support department on our website http://www.zwsoft.com/ or call at (8620)38289780 ext 565. You can also contact your local sales representative or application engineer to discuss your requirements.

Thanks for being our customer!
Team ZW3D

Contents

Chapter 1 Introduction

Preface

The units used for this training are mm. It would be a good idea, if you are not already using mm, to make a copy of your "_config" file under the user directory as a backup, delete the "_config" file and restart ZW3D. You will be asked what units you want to work in and you should choose mm.

If at the end of this training exercise you want to return to your previous units, delete the current "_config" file, rename your backup to "_config" and restart ZW3D.

Requirements and Advantages

Some of the requirements and advantages of ZW3D Turning are listed below:

- ZW3D Turning generates tool path not only based on a Solid Model, but also a 2D sketch.
- The sketch must be closed and located in the XY plane.
- The solid model should be columnar part and the central axis should be X-axis.
- ZW3D Turning can generate the exact tool path and avoid overcuts according to the tolerance.
- ZW3D Turning uses feature recognition and can generate Tool path with and without a stock model.
- ZW3D Turning supports all the operations for 2 axis turning machining which includes Turing Drill, Face Cutting, Turning Rough, Turning Finish, Turning Groove, Threading, Parting and other operations.

Chapter 2 Basic Walk Through

This chapter will guide you how to create the turning tool path with a 2D sketch.
If you choose to use a solid model, then you also can use the same method introduced in this chapter to create the tool path. Now let's go to create a new CAM plan. Then we will create our Turning operations and tool path.

Step 1- Creating a new CAM plan

1. Pick **File** > **New** or pick the **Create a new file** icon  from the Quick access Toolbar or press (Ctrl+N) to display the **Create New File Form.**
2. Select the **CAM Plan** icon  in the Create new file form. Enter **My_Turning.Z3** in the **Unique Name** field as the file name and then pick **OK**.
   Both the new file and the new CAM Process Plan will be created. You are now at the CAM level. The new CAM Process Plan is created and activated. The ZW3D Title Bar indicates this with the description "File [My_Turning.Z3]".

Step 2 – Got to CAM setup manager

You will notice in the left of the screen is the **CAM Plan Manager**. In the Top of the Manager there are 3 icons: CAM Manager  ; View Manager  ; Visual Manager  .

In the CAM manager there is a Manager tree with a default name of Setup1 in the top of the tree. Unfold the tree we can find there are several branch options under it as follows:

SetUp1
- **--- Geometry (Undefined)**
- **--- Clearance**
- **--- Frame**
- **--- Tactics**
- **--- Operations**
- **--- Machine (Undefined)**
- **--- Output**

With the CAM plan manager tree you can finish all of the necessary operations to get the tool path.

Step 3—Inserting Geometry

1. Under **Setup1** left-click on the icon **Geometry,** then it will display the "**Shape Browser**".

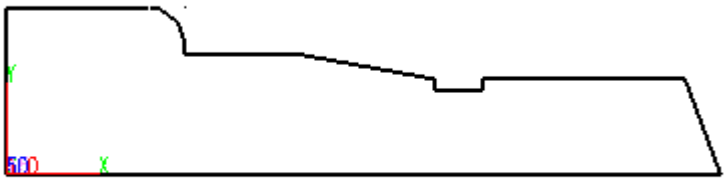
We need to select the geometry from the browser.

2. In the **Paths** list, locate the "\training" sub-directory of ZW3D program directory. If the directory is not listed, select the folder icon  below the list, locate and select the directory by using the File Browser, then it will appear in the list. (Lower case letters at the bottom of the list)

3. Then in the **Files** list shows all of the files located in the "\training" directory. Select the file "**Turning_Model.Z3**".

4. Select the file "**Turning Geom**" from **Shapes** list.

5. Select "**Yes**" to insert the geometry when the warning information pops up. Then the screen should display as following:



Note: What we added is a closed 2D sketch.

Step 4—Defining CAM Profile Feature

Now under Geometry a new part has been added in as following

SetUp1
--- Geometry
---Part: Turning Geom(1) <Turning Model.Z3
--- Clearance
--- Frame
--- Tactics
--- Operations
--- Machine (Undefined)
--- Output

1. Right click on the inserted geometry “Turning Geom”, then a context menu will display. Select **Add Feature** and then pick the **Profile** from the sub menu of it.

2. Then a dialog box for geometry selection will pop up; Set the **Input type** option to **profile**.

3. Then go to the Profiles option, pick the sketch as the profile.

4. The dialog box of the Profile Feature will display when you select ok to finish the selection of profile.
   In the dialog box of Profile Feature set the **Type** into **Part** and change the direction with option **Reverse Dir**. Make sure the direction of the profile is counterclockwise as following:

Note: The direction of the profile must be counterclockwise

5. Now we can find that a Profile feature has been added under the inserted Geometry “Turning Geom”.

SetUp1

--- Geometry

---Part: Turning Geom(1) <Turning Model.Z3

---Profile 1

--- Clearance

--- Frame

--- Operations

--- Machine (Undefined)

--- Output

6.**Creating cylindrical stock**

Now we will use the **Cylindrical Stock** command to create a cylindrical stock for the inserted part “Model”. The **Cylindrical Stock**  is located in the **Setup** tool bar.

Select Cylindrical Stock icon, the command dialog box will display.

Set the filter to “Profile” and then select the sketch, then the stock will be automatically created as shown below.

X axis is the default axis. It can be changed from the pull down menu or you can input the vector manually. We will accept the default axis value for this exercise. Also in this form we can set the Radius and Height for the Stock. For the “**Height (Z)**” input the value “126”. The size can be adjusted by the button“+/-”.

The radius of the stock in Z axis is the same as the sketch, but the height is 2 mm higher than sketch in +Z direction. Select “Ok” to create the Stock.

When this prompt “Hide[Model_Stock.1.2]?” appears select **No**. Then the stock will display as transparent like the image below:

Step 5 –Clearance and Frame

When creating a CAM plan, the system will automatically assign a Clearance value and a Frame for user. For now we use the default setup.

Step 6 – Creating an operation

1. Go to **Operations** under the Setup1 and double click on it, then the Operation Type displays.

2. Go to the **Turning** Tab and select the operation “Rough Turn”  .

3. Now a Rough operation named “Rough Turn1” has been added to Setup1 under **Operations:**

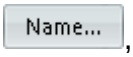
SetUp1

--- Geometry
---Part: Turning Geom(1) <Turning Model.Z3
---**Profile**
--- Clearance
--- Frame
---Operations
--- Rough Turn 1
--- Class (Rough)
--- Tool
--- Parameters
--- Feature (Undefined)
--- Machine (Undefined)
--- Output

Note: The operation can also be renamed by right clicking on the operation and picking “Rename” from the context menu.

Step 7-Defining the Tool

1. Double click on Tool item under the operation “Rough Turn 1”, and then the Tool manager will display. Here we can create and manage the cutting tools.

   In the Definition area, you can rename the Tool. In the **Name option** 

, if you leave this field blank then system will automatically assign a default name like Tool 1, Tool 2…etc to the tool after finishing the definition.

2. Input “my first tool” (without the quote marks) in the **Name** option.
   Note: Don’t use a comma in the name.

3. Set **Type** option to **Turning**. Select the Turning cutter from the **Subtype** option. ZW3D provides different shapes for each kind of tool, such as OD Turning Diamond, OD Turning Triangle, and OD Grooving Right and so on. Here let’s select the **OD**

Turning Diamond. In the Shape area the shape of cutter can be changed by adjusting the parameters provided according to your requirement.

4. After completing the definition, select “Ok” at the bottom of the Form. Then the Tool “my first tool” will be added to the operation **Rough Turn1**.

SetUp1
--- Geometry
---Part: Turning Geom(1) <Turning Model.Z3
--- Profile 1
--- Clearance
--- Frame
--- Operations
--- Rough Turn 1
--- Class (Rough**)**
--- Tool: my first tool
--- Parameters
--- Feature (Undefined**)**
--- Machine (Undefined**)**

Step 8- Setting Parameters

1. Go to the **Parameters** in Rough Turn 1 and double click on it, then the Parameters Form will display.

   Actually, the form contains all of the default parameters for generating tool path.

2. Then say ok to it and go to the Feature of Rough Turn 1

Step 9- Defining Feature

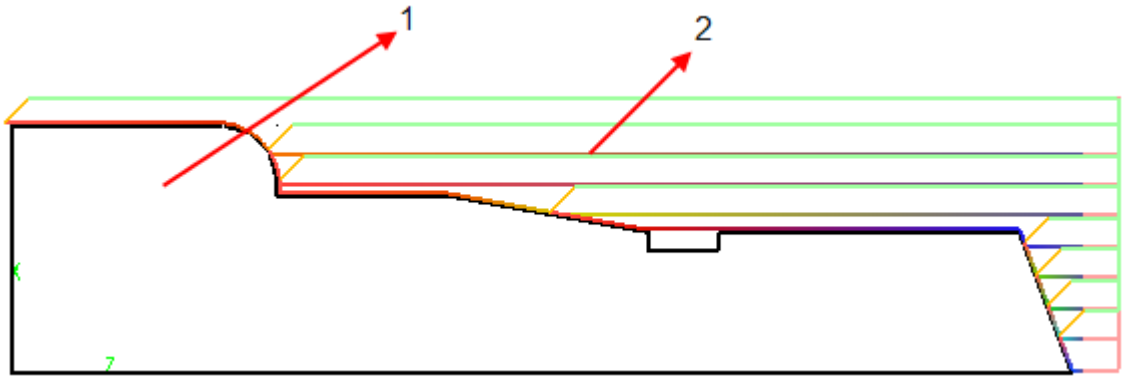
Under Rough Turn 1 the **Feature** now it is undefined. Next we are going to add a profile to it: Single click on it and then go to the inserted geometry, then double click the **Profile** feature which was created in step 4. Then the Profile feature will be added in as follows:

--- Operations
--- Rough Turn 1
--- Class (Rough**)**
--- Tool: my first tool

--- Parameters
--- Feature
---Contain:profile1
--- Machine (Undefined**)**

Step 10- Calculating the Tool Path

So far we have defined the necessary parameters for tool path. Now it is time to calculate it. Now right click on "**Rough Turn 1**" and select the "**Calculate**" from the context menu as shown in the image below:



1:Geometry 2:Tool path

Step 11- Defining the Machine

Next we need to define the Machine for proper output:

1. Go to the **Machine** under the Setup1manage tree and double click on it. The **Machine manager** will display.

2. From the manager we can find the default setup for output. For Turning we need to confirm the Post-processor is "**ZWPost**". Then go to the **Post Configuration** 

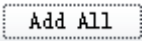
and left click on it then select the "**ZW_Turning_Fanuc**" from the list. Also you can select other post configuration for turning.

3. We can also define the machine by the option of **Machine Name**. Input the name "my first machine" in the **Machine Name** option. Then click the Apply button to have

the new name inserted in the field. Then “**my first machine**” will be added to the Rough Turn 1. Click the Apply button tocreate new machine.

4. Left click 

in options.

5. Left click 

in **Machine Tool Changer**.

6. Click **OK** to finish.

7. And click **YES** to sava changes.

8. Click **OK** to finish define.

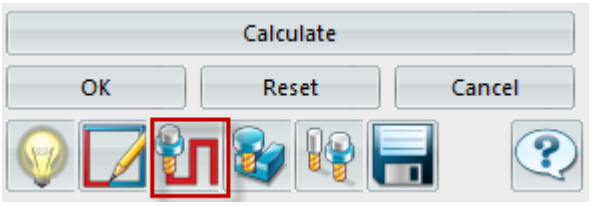
If the **Machine Name** field is left blank the system also will automatically assign a name like machine1, machine2 … etc to the new one.

Step 12—Rapid verification

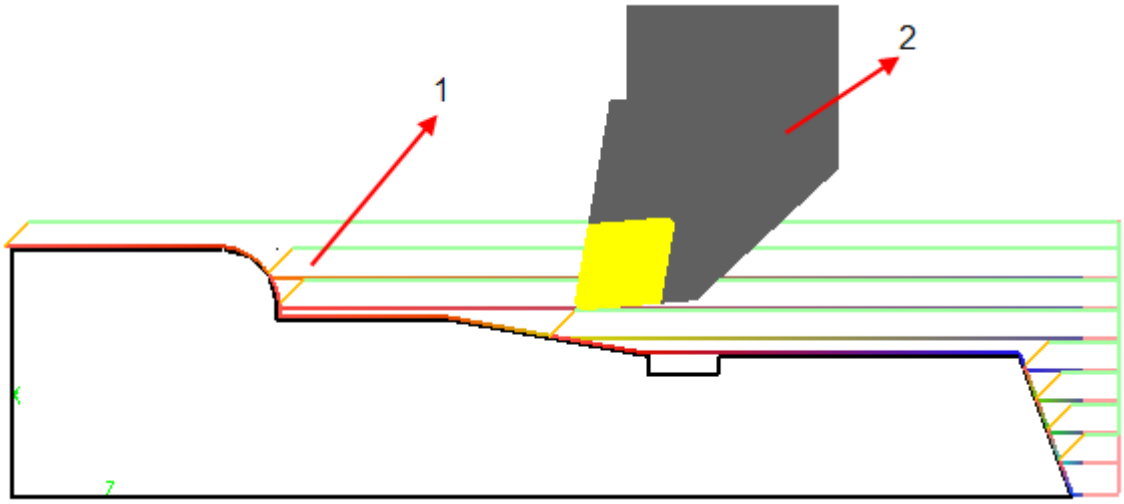
Congratulation!
Now you have finished the first fool path and also defined the machine.
From this process we can see it is really easy. Next we can create other required operations and get the tool path to finish our programming with the same method. But we haven’t finished yet. Normally each programmer wants to check the result.
There are 2 ways to check the result: the first one is to use the **Verify** to simulate the motions of the tool alone the tool path. Another one is to use the **Solid Verify** to simulate the real process of machining. Right click on the **Rough Turn 1** operation and select the “**Verify**” from the context menu.

Then the **Tool path Verify** dialog box will display**.**

It also can be selected with the button “**Verify operation**”  located at the bottom of the **Parameters Form**



Select the **Options** located in the bottom of **Verify Tool path** dialog box
Select an available frame from the **Frame** option. Here we will use a default frame.
Back to Verify Tool path dialog box and select the icon ">" to start simulating the movement of the tool.



1:Tool path 2:Tool

If you want to stop it, click the icon "||" to pause it first and then close the dialog box.

Step 13- Solid verification

Solid verification provides an accurate way to check your work before cutting your job.

1. Go to the **menu** option on the top. Then left click on **Output**, the Output dialog box will appear.

2. Under the **Organize** tab, all of the operations calculated will be listed. The options in the below of the list can manage the operations such as change the sequence of them and edit the tool path and so on.

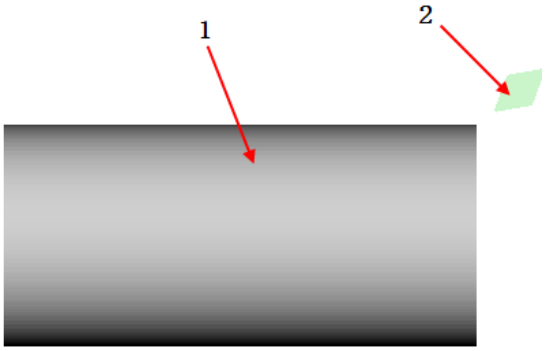
3. **Verify** tab**.** Selecting the **Verify** tab will bring up the solid verification interface. Once there we can assign the **stock** and **Target part**. When we click the **stock** button the

stock dialog box will turn up. Here we can select an existing stock or create a new one for solid verification. The shape of the stock can be a box, cylinder or a STL file.

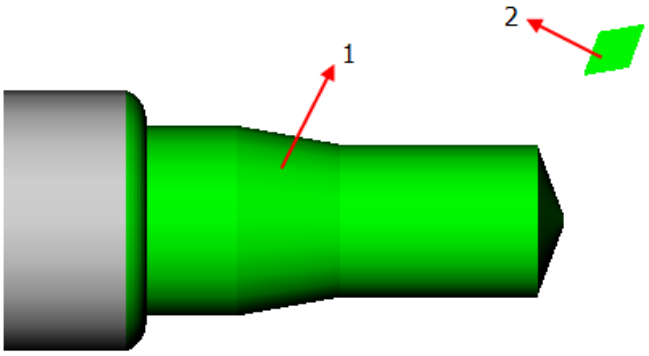
Note: Solid verification must have a stock.

4. Next click the **Verify** button, and the system will go to the solid verification Process dialog box. The solid verification process dialog box has the options similar to the options in the Rapid verification dialog box.

Click on the start simulation button  .Then the stock and the cutter will display as in the image below:



Start "Solid Verification"（1:Stock　2:Cutter）



Solid verification end (1: Target part 2: Cutter)

When the simulation is finished the cutter will return to the start point. Then you can close the solid verification dialog box.

Step 14-Output the CL file and the related documentation

1. Under **Setup 1**, right click on the **Operations** again, chose the **Output** and get **Output NC for all**.

2. Now double click on the **NC 1 file** again.

3. Check the box of **Display Output** and then CL file windows will be displayed. Now we can create all the documentation needed to complete the job. First, let's go to set some options.

    - In the **Part Id** field input "**my first tool path**".
    - Set **Tool Changes** into output.
    - Set **Speeds/Feeds** into output.
    - Set the **Tool Num** into Tool Id.
    - Ensure the **Display Output** is checked.

4. Click the **CL File** button then the CL file will be displayed in the output window.

5. Click the **NC Code File** button, the NC code will be displayed in the output window.

6. Click the **Tooling List** button. This will display a list of all the tools.

7. Click **Operation List** button. This will display a list of all the operations used.

8. Click the **Export Plan as XML** button to display the Operation list in an XML format.

9. Close the CAM output dialog box.

Save the file  .

All of the output files located in the "/output" folder in the user directory by default. All of the files named are the same name as "my first tool path" but with different extension name, these are .cl .nc, .tl, ,op and .xml.

HINT:
If you are not sure where your **USER** directory is located, Click on **Utilities > Configuration** and see the bottom of the **General tab**.

Congratulation! You have finished the basic demonstration

Chapter 3 Prepare for Turning

This chapter covers:

- **CAM Plan**
- **CAM Template**
- **Inserting Geometry**
- **Creating cylindrical stock**
- **Add CAM feature**

CAM Plan is used to create and manage Tool path for machining. It can be created with the same dialog box creating Part/Assembly or 2D sheet.

CAM template

We can load a Template file when creating a new part, 2D sheet or CAM plan. By default, the name of Template file is Template.Z3. The Template file is a standard ZW3D file which located in the "\resource" folder of the installation directory. It already has included the necessary setup when creating. For example, when creating a new drawing, we can load the template which automatically loads the drawing size or limitation; for a CAM plan, there are some standard machining methods which can load at startup and does not have to be input manually each time.

Next let's go to create the first CAM Plan.

Open the file "**Lathe_Part.Z3**" which located in the "\training" folder in the installation directory. Then create a new CAM Plan with the name "**Model_Turn_Plan**". For this exercise we will not use any Template. Now the CAM plan manager tree is displayed as follows:

SetUp1

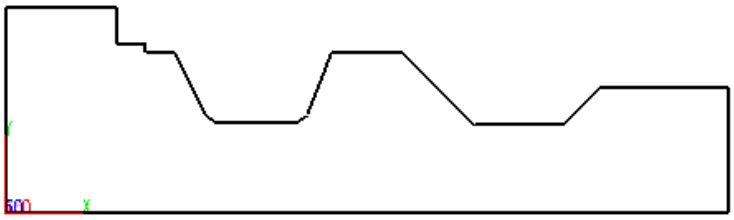
--- Geometry (Undefined)
--- Clearance
--- Frame
--- Tactics
--- Operations
--- Machine (Undefined)
--- Output

Note: Geometry and Machine are shown as undefined**.**

Inserting geometry

Go to **Geometry** and right click ->**Insert Component** on it then go to the **Shape browser.**

Select the part with the name of "Model" in the file "Lathe_Part.Z3" When the prompt "**Using the Lathe_Part.Z3, Model**" turns up, select "Yes". The part Model will be added to **Geometry.** The graphic area the geometry should appear as the image below:

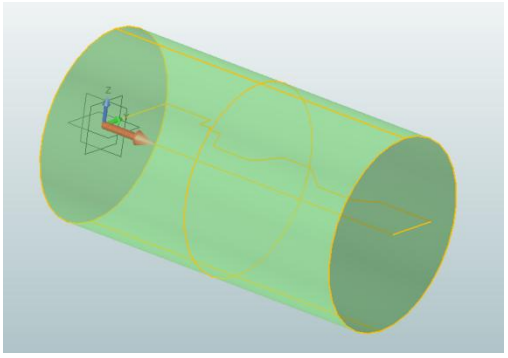


Creating cylindrical stock

Now we will use the **Cylindrical Stock** command to create a cylindrical stock for the inserted part “Model”. The **Cylindrical Stock**  is located in the **Setup** tool bar.

Select Cylindrical Stock icon, the command dialog box will display.

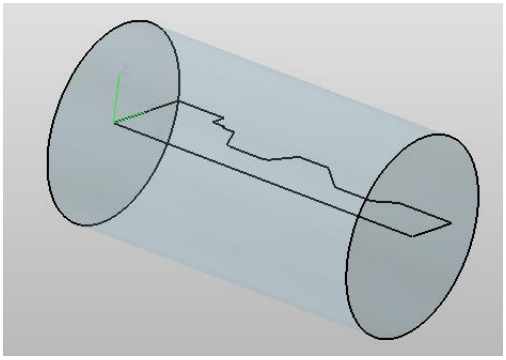
Set the filter to “Profile” and then select the sketch, then the stock will be automatically created as shown below.



X axis is the default axis. It can be changed from the pull down menu or you can input the vector manually. We will accept the default axis value for this exercise. Also in this form we can set the Radius and Height for the Stock. For the “**Height (Z)**” input the value “126”. The size can be adjusted by the button“+/-”.

The radius of the stock in Z axis is the same as the sketch, but the height is 2 mm higher than sketch in +Z direction. Select “Ok” to create the Stock.

When this prompt “Hide[Model_Stock.1.2]?” appears select **No**. Then the stock will display as transparent like the image below:



Now you can see the new stock has been added to **Geometry,** and the attribute of stock also has been assigned to it. The display mode can be changed by double left clicking on the stock.

Add the CAM feature

If the geometry is a solid model it can be used as a CAM feature for Turn cutting; If it is a 2D sketch, then we need to define a profile feature first.

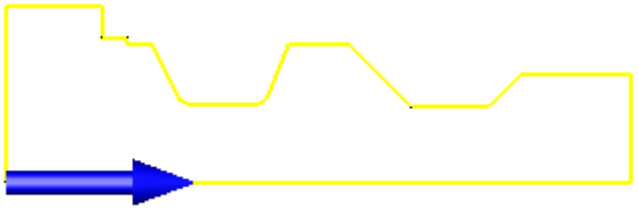
Left click on the part “Model”, and then select the “**Add feature**” option from the context menu. And then Select “**Profile**” from the pull down menu of “**Add feature**”.

Select the whole sketch as the profile feature, and then all of the selected geometry element will be highlighted as shown below:

Note: Set your filter to profile and then you can pick the whole sketch at one time.

Click the “Ok” button and then the dialog box of the profile feature will display.

Set the Type option to Part and reverse the direction of the profile if the direction is not the same as the image shown below:



Make sure the direction of the arrow is pointing Count clockwise.

Select OK. Then you will see the feature under the CAM Plan manage tree as shown below:

Setup 1

Geometry

---- **Part:** Model（1）>Lathe Part

---Profile 1

Now save the file.

We are now going to work thru the seven operations in Turning Module. These are Drilling, Facing, Rough Turn, Finish Turn, Groove, and Threading, and Part off.

Chapter 4 Drilling

This chapter covers

- **Creating Drilling operation**
- **Creating Tool**
- **Inserting the feature for operation**
- **Cutting Parameter Form**
- **Calculating Tool path**

First, go to the folder “\training” in the installation directory and to load the file “Lathe_Part.Z3”. Then open the CAM plan “Model_Cam_plan” which was created during the last exercise.

Creating Turn Drill

There is three ways to access the Turn Drill dialog box:
1) Go to the Operation and right click on it then select Insert operation from the context menu. Then we can get the **Operation Type**. Once there select **Turning** tab and pick Drill operation icon  under it to create a new Drill operation.

2) Double click on the Operation. The Operation Type will display, and then use the same process to create a new operation for drilling .

3) In the graphic area, right click and then select the **Insert operation** from context menu. Then click the **Operation Type**. Then go to Turning tab and select the Turn Drill operation 

Note: The system will assign names like Turn Drill 1, Turn Drill 2 to the new operation automatically if you don’t want give the operation custom name. To rename the operation, right click on it and then select “Rename” from the context menu. Here, we can use the default name.
At the same time, under “**Turn Drill 1**”, there are some options. For example: Tool is showed with the word “**Undefined**”. It is remind us there are still other parameters we still need to complete to get the tool path.
So next let’s go to define them.

Defining Tool
Right click the **Tool** of “Turn Drill 1” and then pick **Select** from the context menu.

This will bring up the Tool Manager.

1). In the **Name** option input the name “**Drill**”. The system will automatically assign a default name to each new Tool like Tool1, Tool2 and so on. Set the Type to Drill.
In this exercise we will drill a simple hole with the diameter of 10 mm and the depth of 20 mm. So here we can just select a normal twist drill.

In the right side of the Tool manager is the Tool library.
The folder “/resource” inside the installing directory includes a set of data for Tools which can be used. Also, there are other Tool libraries with different units such as the Metric and Inch. The turning cutters have not been included into this library yet.

We can change the library by the browser icon  .

Note: If you want to change units from Inches to Millimeters, it is recommended that you make copy the file “_config” in the user directory first as a backup. Then delete it and restart ZW3D again and select the Millimeters as the default units when system prompt asks you what units you want to work in.

For this exercise we will use the default setting for this Twist Drill
Select ok to add the Tool to Operation Drill 1.

Inserting the feature for operation
Go to the Feature option under the operation Drill 1and left click on it. Then go to Geometry and select the profile which is located under Geometry. Now the profile will appear under the Feature option of Drill 1. Repeat the steps above, and also add the stock into this operation.

Drill Parameter form
Go to the Parameter anddouble click on it.
The Drill parameters Form appears. It includes several options like following:

- Primary Parameters
- Pathsetting parameters
- Display parameters 

The following is an explanation of some common parameters. For the other parameters and more information about these parameters please refer to the help documentation. The parameters have not been specified in this training manual, all are at their default value.

Primary parameters

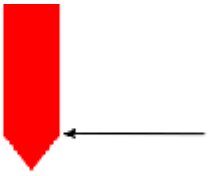
Speeds, Feeds
Specifies spindle speed and feed rate.

Drill Type
Turn Drill operation provides the following selections: Center Drill, Drill, Peck Drill, Chip Drill, Ream, Bore and Tap.

Select **Drill**.

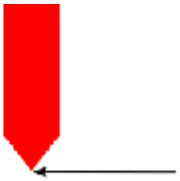
The Hole Depth options

The Depth option specifies the working depth of the hole.
The Tool Depth Ref option determines depth location on the tool. This location could be to the tip of the drill or the shoulder of the drill.
For the Tool shoulder, it ensures the Tool shoulder is down to the depth.



Tool Shoulder

For the Tool tip, it ensures the Tool tip is down to the depth.



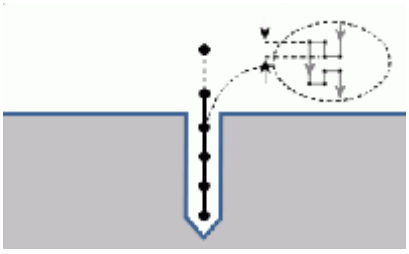
Tool Tip

Deep Drilling Options
Deep Drilling Options are available when PeckDrill or ChipDrill is selected.

Max Peck/Chip Depth and Min Peck/Chip Depth.
Specify the limitation for the depth of Peck or Chip drilling.

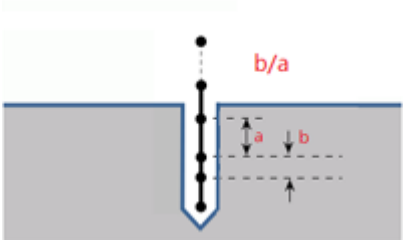
Retract Offset

This is the retracting distance between the current drilling and the previous one. Different Drill Type has different retracting path value. For PeckDrill, the tool will retract to the top face of the hole and then go to the retracting offset position; For the ChipDrill, the Tool retract to the specified distance to break the drill chip but does not retract completely from the hole as shown in the image below.



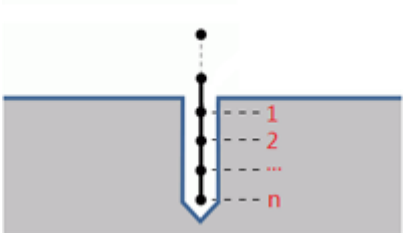
Reduction Factor

The reduction factor is a factor by which the new peck depth will be reduced. Refer to Reduction Start for a more detailed explanation. The effective value range of it is 0-1. The bigger the value is the more quickly it reduces. If the actual peck depth calculated is shorter than the min value, ZW3D will remove the stock left with the min peck depth. The chip cycle is the same as peck cycle in regards to this.



Reduction Start

This is a sequence number from which the Peck/Chip will cut at the reduced depth.
Drilling Depth =Max Peck/Chip Depth*(1-Reduction Factor* Reduction Start).For example, if the value is 3, for peck drill, the drilling depth for the previous 2 steps is at the max depth. All the pecks after that, the drilling depth will be at reduced depth, if the hole is deep enough to require 3 or more pecks to complete.



Path setting parameters

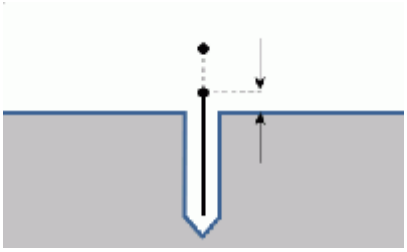
Dwell time(s)

Specify the dwell time that how long it will pause at the bottom of the hole before it retracts. The unit of dwell time is seconds.

Note: Both of the Rapid verification and Solid verification can't simulate the result of this value. It is just written to the CL file. Here set the value of the dwell time to "1".

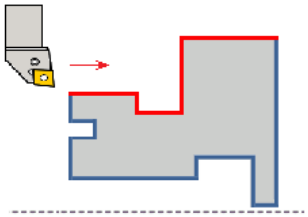
R-level Clear

This is the distance above the top of the hole at which the drilling operation will start.

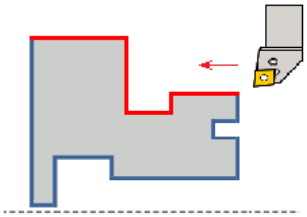


Cut Direction

Specify the movement direction of the tool during it machines the workpiece.



Left to Right



Right to Left

Display parameters 

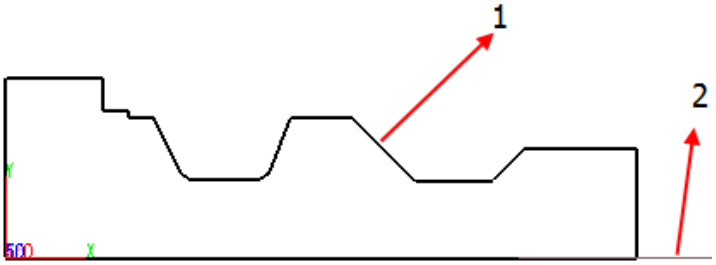
In this form we can set the style and color of the lines for displaying the tool path. For more information please refer to the ***Tool path analysis & display parameters*** in help documentation

Calculating the Tool path

There are 2 ways to calculate the tool path

First go to the operation "Turn Drill 1" and right click on it then selecting the calculation option from the context menu. Second in the Parameters Form, click the calculation button.

Now the drilling tool path has been created in the center axis of the sketch as shown in the image below:



1: Geometry 2: Tool path

Double left click on the operation “Turn Drill 1” . This will hide the tool path. Redo it again and the tool path will display once more.
Save the file.

Chapter 5 Facing

In this exercise, we will show how to use the Facing operation:

- **Creating Facing operation**
- **Creating Tool for this operation**
- **Add the feature**
- **Introducing Parameters Form**
- **Calculating Tool path**

First, go to the folder “\training” in the installing directory and to find the file “Lathe_Part.Z3”. Then open the CAM plan “Model_Cam_plan” which we created in the last chapter.

Creating Facing operation

The Facing operation is an operation only for cutting the face of the part with a simple function. Select the **Facing** operation  from the Operation Type Form. Then we can see an operation named **Facing 1** has been created under Operations in the Setup manage tree. Also, there are some options has not been defined yet, and are displayed with the word “undefined”. Next we must define these.

Creating Tool for this operation

Right click on Tool **Facing 1**. Then go to the **Tool manager**. First set the **Tool Type** in the definition area to Turning and then set the **Subtype** option to OD Diamond Right. Last assign the name “Face off” to it by using the **Name** option. If you want to change your Tool type you can change it in the subtype option.
Select Ok to the manager and then you will see that the Tool has been added to operation **Facing 1.**

Add Feature

We can use the same process we mentioned in last chapter to add the same Profile Feature to the Feature of **Facing 1**

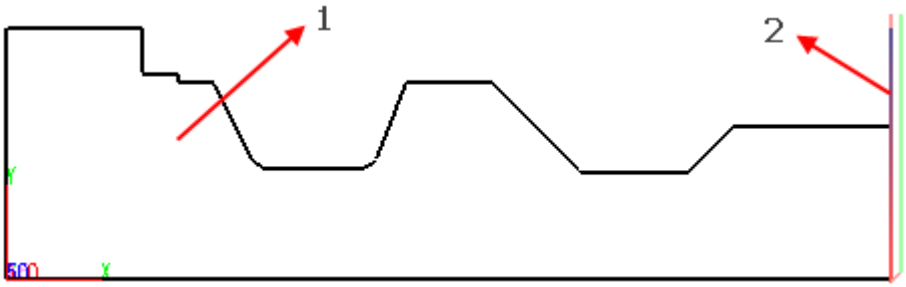
Parameters Form

Left click on the Parameter icon under **Facing 1** and then go to the **Parameters Form**. The options in this Form are as following:

- Primary Parameters

- Leads and link parameters 
- Display Parameters 

Don't change anything and calculate the tool path with the default parameters firstly, then we would get the result as shown below:



1:Geometry; 2:Tool path

Now change some parameters. Calculate it again and observe the differences.
Note: The parameters haven't been specified and we will use their default values.

Primary

Basic 

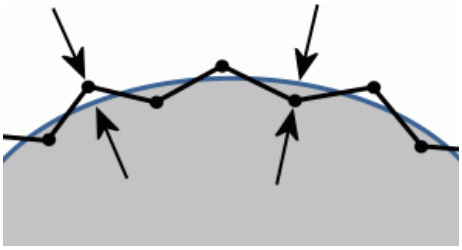
It can specify the speed and feedrate of the Rough, Finish, Engage and Retract and so on. Several different units are available to pick from. For the details please refer to the requirements of each machine.

Note: Here the setting for the speed and federate just affects this operation only. So if you want to specify a unit speed and feedrate to all of the operations you can use the **Speeds and Feeds** command from the Edit menu.

Tolerance and Steps

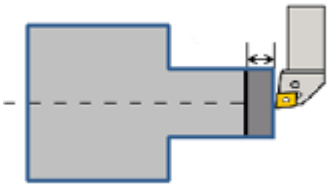
1.Path Tolerance

It refers to the maximum allow deviation distance between the tool path and work piece. This value is a symmetric tolerance value. So if the value is 0.1 it means that the tool path can move +0.1 mm or -0.1 away the part surface. For example, if we want to output a part with tolerance 0.1 then we need to set the Path Tolerance to 0.05. The smaller the value is the more time calculation takes and the more accurately the tool path follows the part.



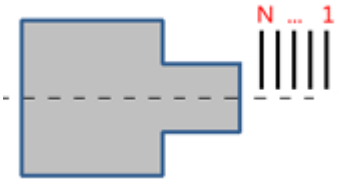
2. **Facing Thick**

How much material should be left remaining after the cut is complete. This value can't be a negative value.



3. **Numbers of Cuts**

This option specifies the numbers of passes as shown below. But it does not take the stock into account. The value input here stands for the number of passes the tool path will create.



4.Step Value

This is a value that determines how much material will be removed at each cut, but not included in the last cut. The default value is 0.5.

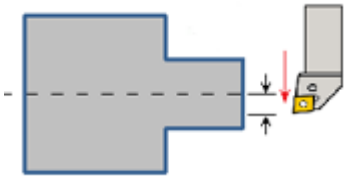
Path Setting

1. Cut Direction

Specify the movement direction of the tool during it machines the workpiece.

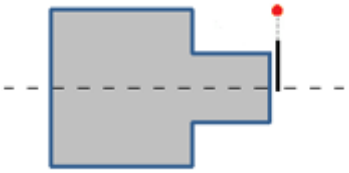
2.Overlap

This is a distance that tool travels past the center point of end face as shown below. This option ensures a complete cut on the face.



3.Station point

This specifies the station point for tool at beginning and end.



link and Lead parameter 

This option is used to set the engaging and retracting distance and angle.

After finishing your set up for this operation, calculate get the tool path again.
Double right clicking on the icon of operation “**Facing 1**” can hide the Tool path

Save the file.

Chapter 6 Turn Rough and Finish

This chapter covers:

- **Creating Turn Rough operation**
- **Creating Tool**
- **Adding Features**
- **Parameters Form**
- **Calculating Tool paths**
- **Rapid work through for Finishing**

First, go to the folder "**\training**" in the installing directory and to find the file "**Lathe_Part.Z3**" then open the CAM plan "**Model_Cam_plan**" which was created at in last chapter. Now under the Operation manage tree there are two operations created in last two chapters: **Drill** and **Facing**
In this exercise we will create another new operation and tool path.

Creating Turn Rough

Select the Turn Rough operation  the same way mentioned in the last two chapters. Then the new operation **Rough 2** will display under the existing operations

Creating a Tool
We will use the same method in the previous operation to create the tool for this operation.

Add the CAM feature
Add the defined profile feature to this operation. Use the same process here as in chapter two.

Parameters Form of Turn Rough
Go to the Parameters Form. The options are shown in the image below.

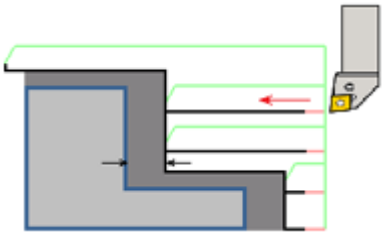
- Primary Parameters
- Limiting parameters 
- Path Setting Parameters
- Link and Lead parameters 
- Display parameters 

The display parameters options are the same as in the previous chapters. Any others that are different than the previous chapters we will explain here.

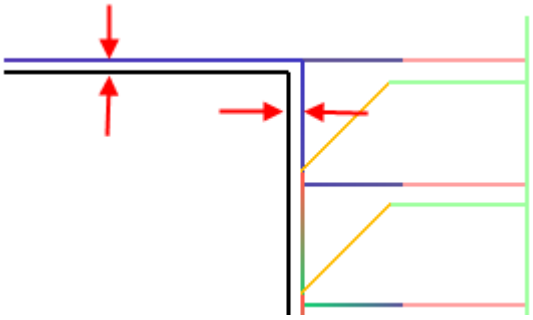
Parimary Parameters

1. Axial Thick/Radial Thick

This specifies how much materials should remain in lathe's Z-axis/X-axis after this operation. This value can't be a negative value.

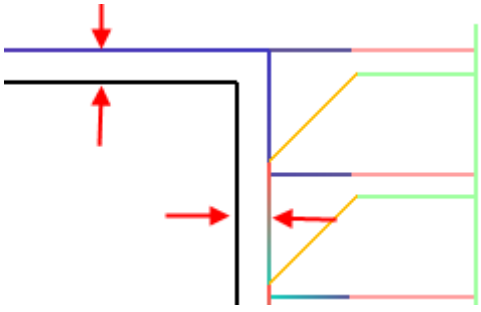


Now set the cut region =OD, cut mode=Horizontal, and leave the Axial Thick and Radial Thick options at the default value "0.2". Then select the Calculate button. Zoom in to the tool path above right area to get the tool path as shown in the image below. This will show the remaining material.



Axial Thick=0.2 Radial Thick=0.2

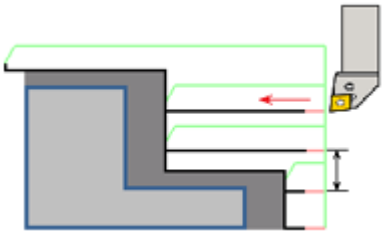
Change the thicks value to 0.5. Calculate again then zoom in to the same place. We can see the gap between the tool path and the part is increased to 0.5.



Axial Thick=0.5 Radial Thick=0.5

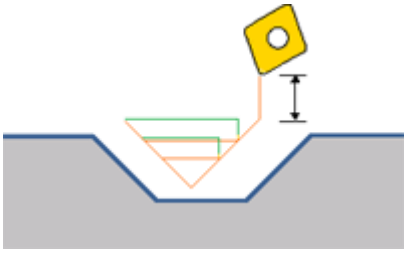
2.Step Value

This value determines how much material will be removed at each cut.
Note: The maximum value can't exceed the cutting length.



Clearance

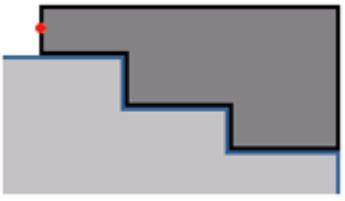
Specify the vertical distance when the tool begins to machine the concave.



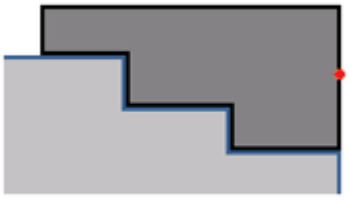
Limiting parameters  .

1.Left Trim Point/Right Trim Point

These parameters are used to limit the cutting region by assigning a left point and a right point.



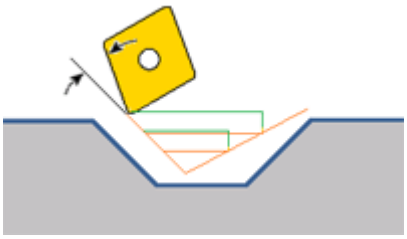
Left Trim Point



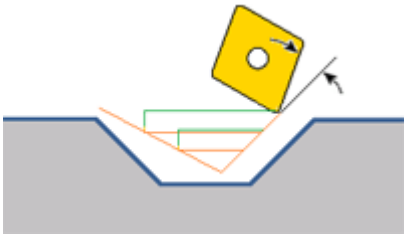
Right Trim Point

2.Front/Back

Specify the offset angle between the main cutting edge/deputy cutting edge and the stock. Using these two parameters, you can protect the cutter by adjusting the machine region.



Front



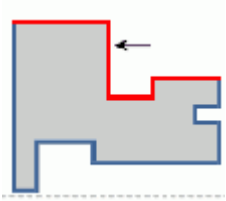
Back

Path Setting Parameters

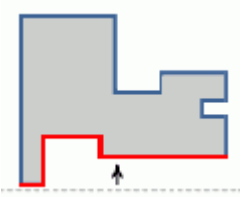
1.Cut Region

This is used to specify the OD (Outside Diameter) cutting or ID (Inside Diameter cutting) direction.

Note: Ensure the selected tool matches the type of cutting. For instance: the OD cutting should use an OD cutter. If the tool selected is not a match for the operation then no tool path will be created. Refer to the images below for an explanation of the different types.



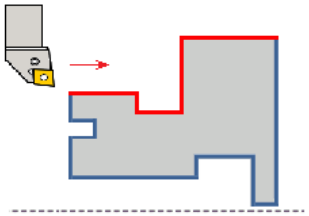
OD



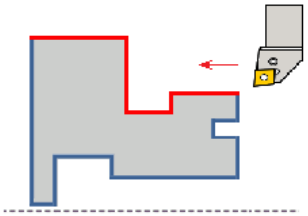
ID

2.Cut Direction

Specify the movement direction of the tool during it machines the workpiece.



Left to Right



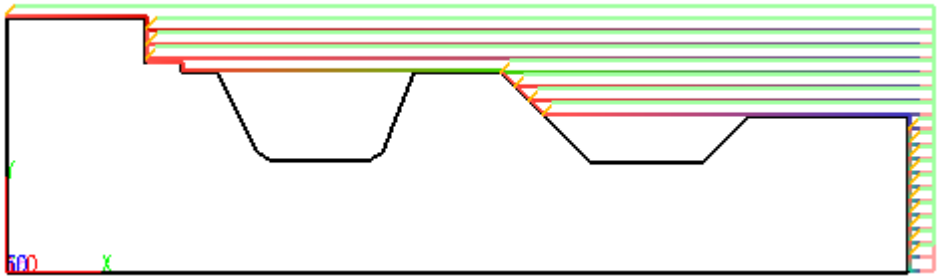
Right to Left

3.Cut Strategy

Turn Rough provides three modes for cutting. These are Horizontal, Vertical and Pattern Repeat.

Each cut mode provides different functions. Refer to the images below for an explanation of each type.

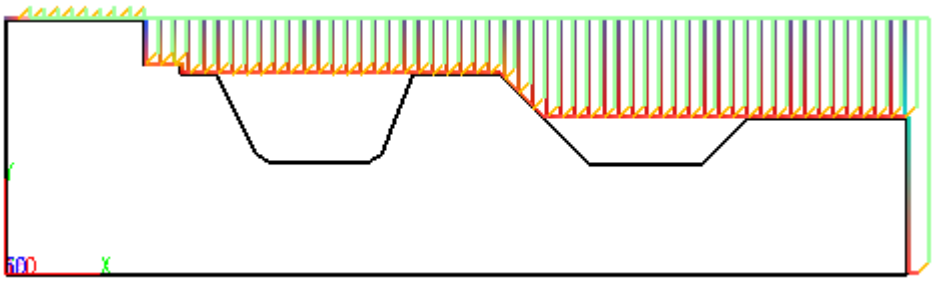
Horizontal



Cut Region=OD

Cut Direction=Right To Left

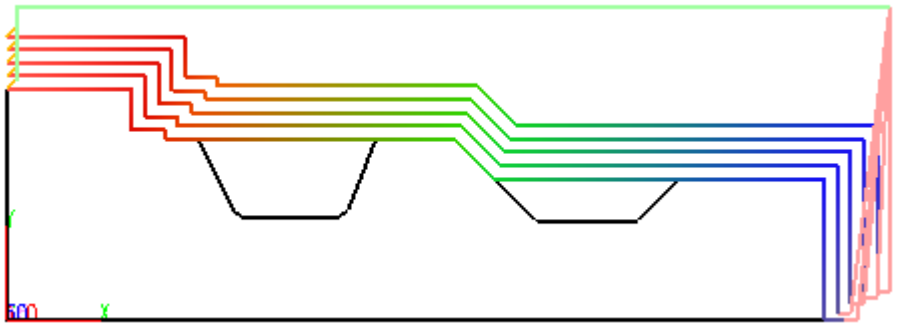
Vertical



Cut Region=OD

Cut Direction=Right To Left

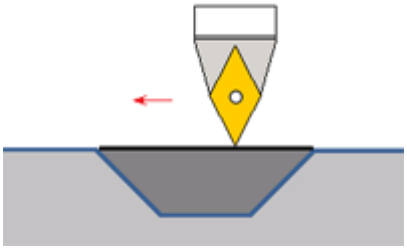
Pattern Repeat



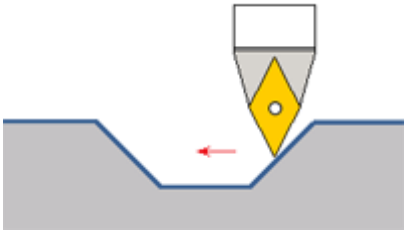
Cut Region=OD
Cut Direction=Right To Left

4.Into Concave

When set to Yes, tool will enter into and machine the concaves which is not in the face wall. Horizontal cutting in the Cut Mode (see below) field only machines the horizontal concaves, vertical cutting only machines the vertical concaves and pattern repeat cutting can machine the both. Please notice that, a appropriate for machining the concaves is needed. For this exercise, we set **Into concave = No.**



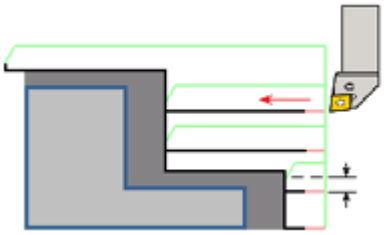
Ignore the Concave



Machine the Concave

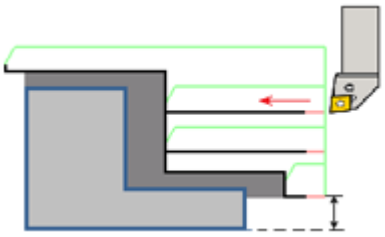
5.Overlap

It specifies the distance for overlap as shown in the image below.



6.Drill Diameter

Specify the diameter of the hole, and operation will not process these stock.

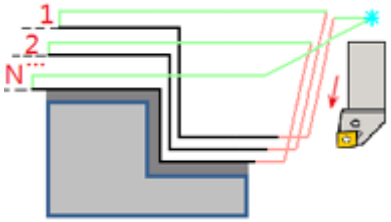


7. Station Point

Tool position at begin and end of the operation. This will let you send the tool to a specific location to allow for a safe tool change.

8. Number of Cuts

When using the Pattern Repeat mode this option is available. It is used to specify how many cutting passes to make in the operation as shown below.

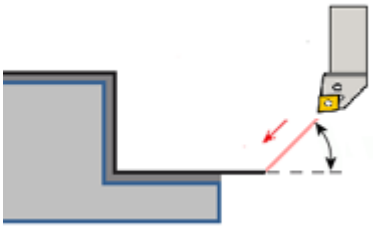


link and Lead parameters Tab

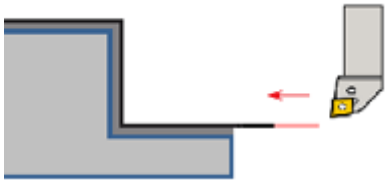
1.Engage Style

This allow the use to specify the path when tool engages the material. Three types are available.

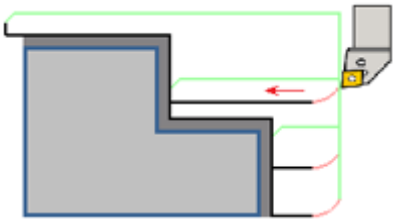
(1) Line+Angle - Engaging path is a line which will approach at an angle as shown in the image below. The angle and the distance of the line are defined by user.



(2)Tangent Line- The cutter engages the material by line tangent to the cuts of the operation as shown in the image below. The distance is defined by the user.

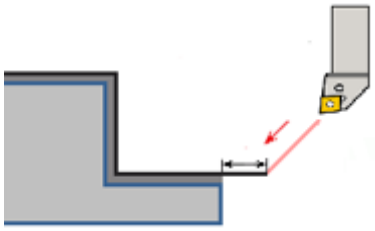


(3)Tangent Arc – The cutter engages the material by an arc tangent to the material as shown below. The Radius and the center of the arc is user defined.



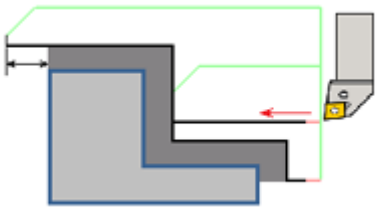
2.Extend Distance (Lead In)

This refers to the distance from the part at the beginning of the cut as shown in the image below. The value here can be positive or zero.



3.Extend Distance (Lead Out)

This refers to the distance from the part at the end of the cut as shown in the image below. The value here can be positive or zero.

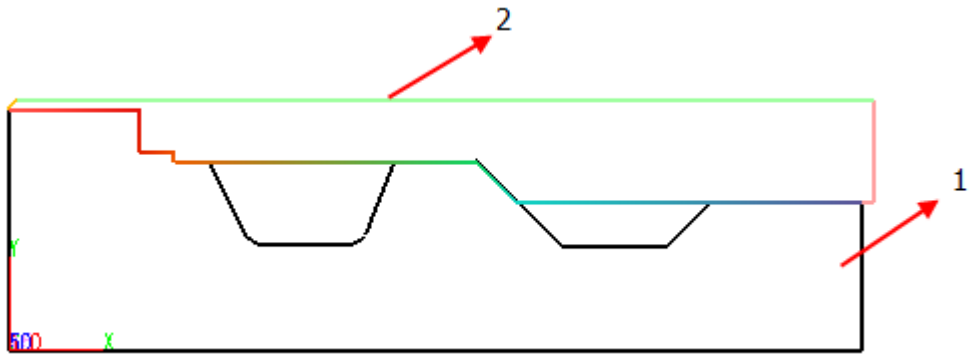


Change some of the parameters we have discussed here, and look at the difference in the tool path. Save your file.

Rapid walk through for Finish Turn.

Now let's create a **Finish Turn** operation. Pick the **Finish Turn** icon  from the **Operation Type** form.
Finish Turn 1 would be added to the list in the **Operation Tree.**
Next let's define the **Tool** and **Features.**
use the same process to define the Tool as before. Give the tool a different name other than the ones before. Just select the tool with the name of "Rough Turn" from the tool list. After the tool is loaded you can rename the tool again.

Add the feature to the operation as shown below.
Right click on it to calculate tool path using the default parameters. The results would be like as shown in the image below.



1：Geometry　2：Finish tool path

The parameters of the Finish Turn are similar to the Rough Turn's except for the option of the Patter Repeat cut mode is not used. So for the details please refers to the Parameters in Rough Turn.
Next we need to add a Turn Groove operation.

Save your file.

Chapter 7 Turn Groove

This chapter covers：

- **Creating Turn Groove**
- **Assigning a grooving tool**
- **Add the feature**
- **Parameters Form**
- **Calculating tool path**

In the previous chapters we have discussed the Drill, Facing, Turn Rough, and Turn Finish operations. In this chapter, we will discuss another important operation—Turn Groove. This operation is able to cut the grooves on the inside/outside of the part, or the End slot. The method used for creating the operation, the tool and the features are the same as what we have discussed in the last several chapters. We will not discuss these again here. If you are not clear on the process, please refer to the related exercise in second chapter.
Open the file that we saved in last chapter. Then follow the steps below to create the groove tool path.

Creating the Turn Groove operation 
Create the Tool for this operation and give it the name “Turn Groove”.
Note: Make sure the length of the shank of the turning cutter greater than the depth of groove being machined in this operation.

Assigning the feature to it.

Parameters Form of Turn Groove
This Form includes 6 Tabs. These are as follows:

- Primary Parameters
- Limiting Parameters 
-
- Path Setting parameters
- link and Lead parameters 
- Display parameters 

Now we will discuss the first 4 Tabs

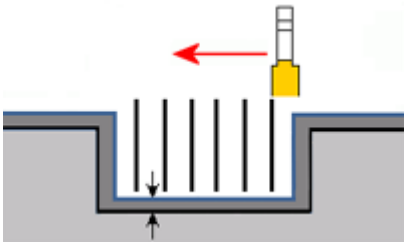
Select the **Parimary parameters Tab**

1. **Basic parameters** 

2. **Tolerance and Steps**

(1) Rough Thick

It refers to the thickness of remaining material after Rough Groove operation as shown in the image below.



(2) Step Type and Step Size

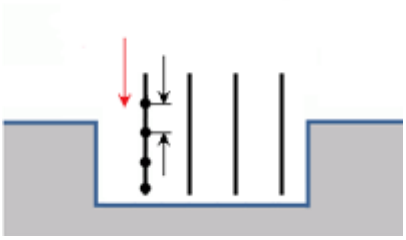
There are two kinds of options are available for Step type. These are " % of tool width" and step size. When selecting the option of "% of tool width", the step size will calculate with the % of the width of tool. For the option of "step size", it will be an absolute value.

(3) Enable Peck

Select "Yes" or "No" to determine whether or not to enable Peck**.**

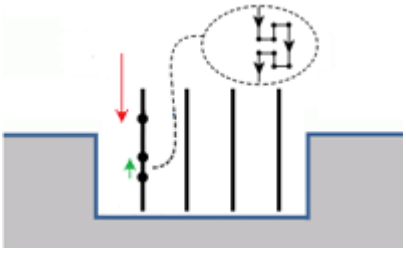
1) Peck Depth

Specify the cut depth for each cut but does not includes the last cut as shown in the image below.



2) Retract Offset

The distance for retracting each time.



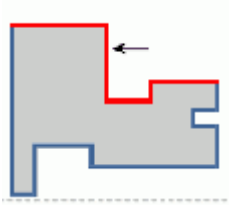
.Limiting parameter 

This tab is to set the cutting area by defining the beginning point and the ending point. Now let's set the parameters and calculate the tool path.

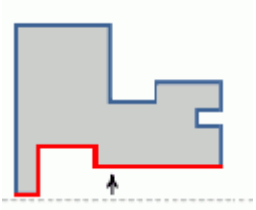
Path Setting parameter

(1) Cut Region

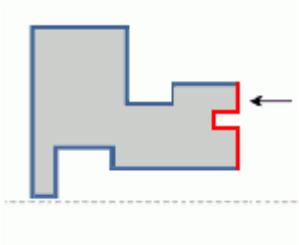
This specifies OD, ID or End slot as shown in the images below.



OD



ID

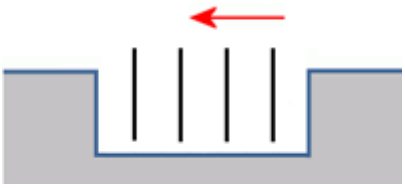


Face

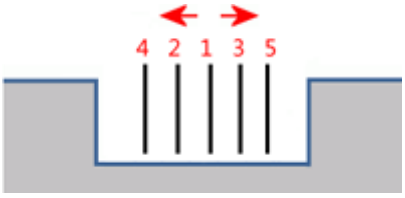
(2) Cut Strategy

Specify the cut strategy. This includes Zig and Alternate 2 kinds of strategies.

Zigzag – The tool cuts the stock along the direction.

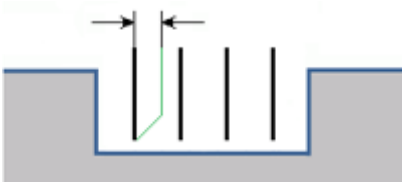


Alternate – The tool start cutting from the middle of the groove and then alternately remove the material from both sides as shown in the image below.



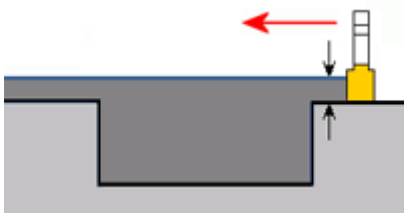
(3) Relief Amount

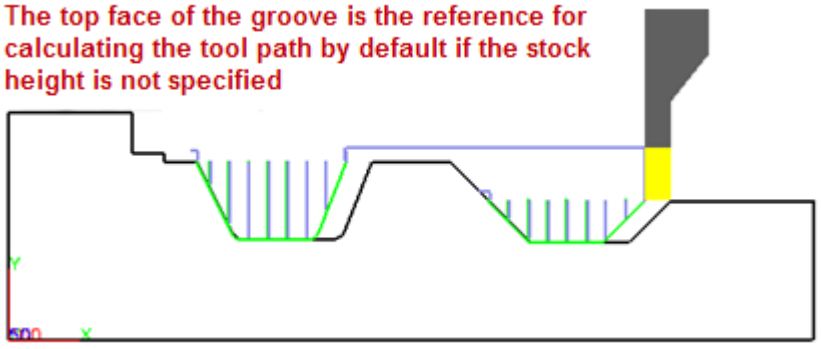
This is a distance that the tool will offset from the wall of the groove when retracting. It will help to avoid the possibility of collision between the tool and the wall by maintaining a safe distance between the tool and the wall.



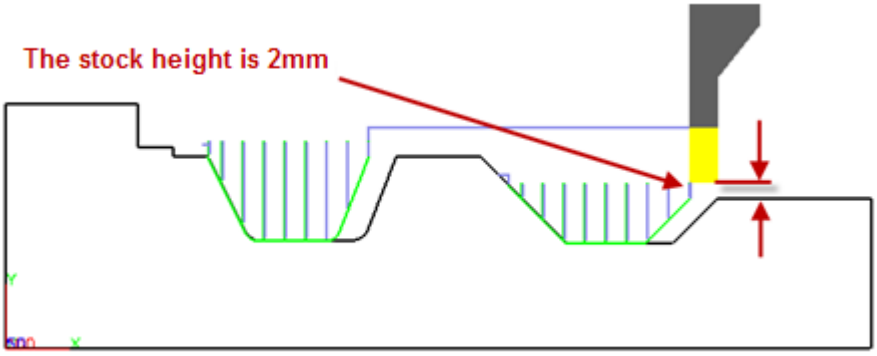
(4) Stock height

It is used to define the amount of material that above the groove. If this value is “0” then system will create the tool path to the top of the feature being machined. If there is additional material above the profile of the feature then you must use the stock height filed to avoid a collision between the tool and the material.





No stock height



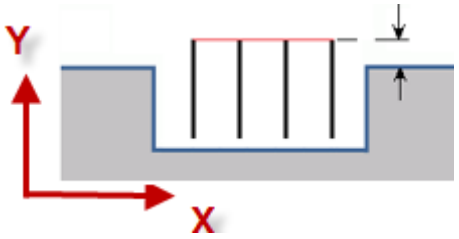
Stock height is specified

(5) Dwell time

This is the time that the tool will pause at the bottom of the groove when finishing. The default unit is seconds. This feature is not displayed during simulation but will be written into the CL file.

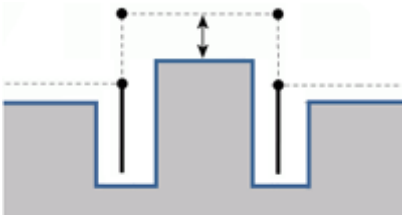
(6) Rough Clearance

It specifies the axial distance from the beginning position of the retract height to the right vertex for the first finish pass. If the input value is shorter than the actual tool width, then the finish grooving will be machined one groove at a time; otherwise it will be machined twice.



(7) Safe Distance

It is a distance between the top face and traversal plane at which tool moves from one groove to another location as shown in the image below.



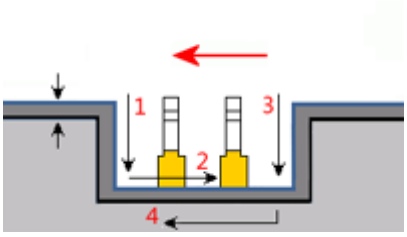
The following are the parameters for setting the **finish groove(Yes).**

Finish Groove

Yes or No. Select “Yes” to enable Finish Groove.

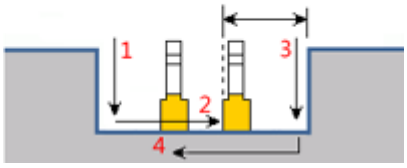
Finish Thick

The material left over in both X and Z directions.



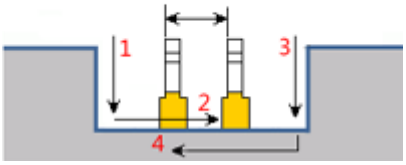
Retract Position

It refers to the distance in Y axis between the start point and top face of the groove.



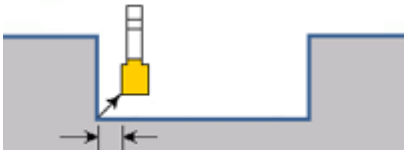
Overlap

It specifies the overlap distance between two contiguous 2 cuts



Back Off Distance

It is the distance in X axis between the start point and the end point of the retract movement. If there is only one pass then this option doesn't apply.



link and Lead parameters  .

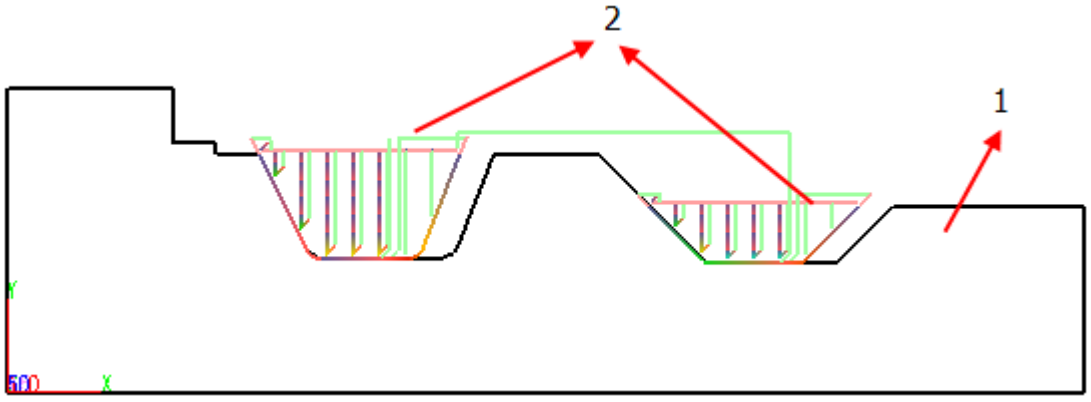
This tab is used to set the engage and retract type and the distance value for them. These parameters and their values were discussed in the last chapter.

Set up **Cutting Parameters** as following:

Rough Clear= 0
Safe Distance = 1
Rough Thick = 0.5
Dwell time= 1

Set up More cutting Parameters as following
Enable Peck = Yes
Retract Position= 12
Overlap= 1
Others use the default parameters.

Then calculate the tool path and the results will be as below:



1：Geometry　2：Groove tool path

Save your file.

Chapter 8 Thread

This chapter covers ：

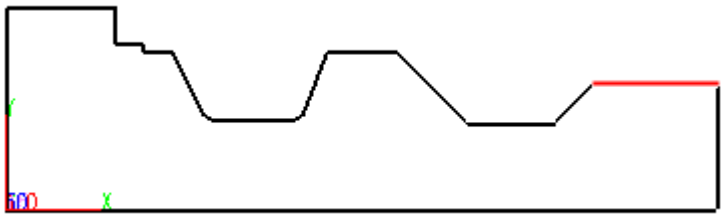
- **Rapid walk through for Thread**
- **Parameters Form of Thread**

the thread operation is similar to the Drill operation. It can generate the tool path for the OD thread and the ID thread in any position on the geometry. It doesn't need the thread feature. It can also can create the tool path for Straight thread, Tapered thread, Single or Multiple threads. Next let's go to create a tool path for a single thread in our geometry. Open the file saved in last chapter

Create the Thread operation. From the Operation type select the thread operation  . An operation with the default name of "**Threading 1**" will be added to the Operation manage tree.

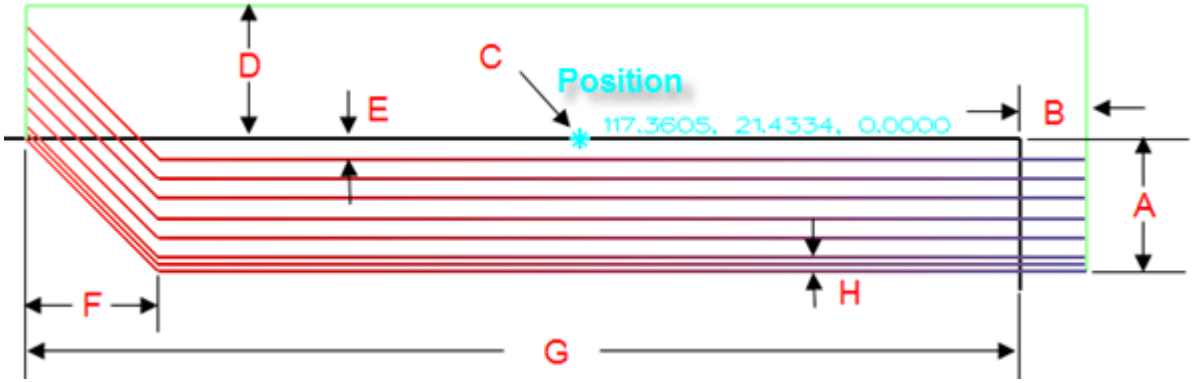
Create an OD Threading Triangle tool. From the **Tool manager** select an OD Standard Tool and name it with the name "Turn Thread". From the **Subtype** option we can select the tool type.
Add feature to this operation: Add the defined profile feature to this operation.

Set up the Parameters
There are two kinds of parameters. One is the Essential parameters and the other is the Additional parameters. First let's set the Essential parameters.
Cut Region=OD
Position = Select a point from the geometry. Select a point in the red line at random location as shown in the image below.



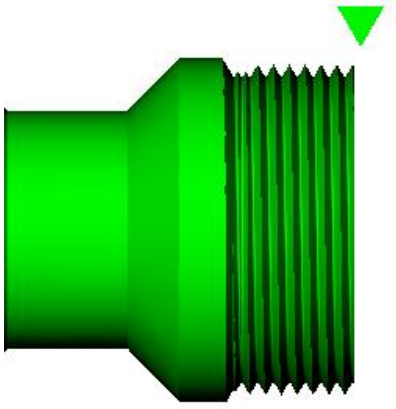
Start Length = 0
ThreadLen= 15
Cut Depth = 0.3
Finish Depth = 0.2
Numbers of Finish = 2
Then go to set the **Additional parameters:**
Clearance=2

PulloutLen= 2
Set up **Leads and link parameters**
Extension distance (Lead In) =1
Other parameters used the default value
Calculate the tool path and the result should be as shown in the image below.



A：Thread depth　B：Extension distance（Lead In）　C：Cut position　D：Clearance
E：Cut depth　F：Pullout Len　G：Thread length　H：Finish depth

It is hard to imagine the result of threading from the tool path. But we can use the solid verify function to simulate it. About how to use the verify function please refers to the related content in chapter two. Following is the verified result :



Parameters Form

- Primary Parameters
- Limiting Parameters
-
- Path Setting parameters

- link and Lead parameters 
- Display parameters 

Next we are going to explain the parameters in these tabs.

Primary parameters

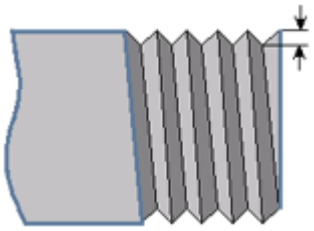
1.Basic parameters 

Speeds and feeds

2.Tolerance and Steps

(1) Thread Depth

Specify the depth of threading.

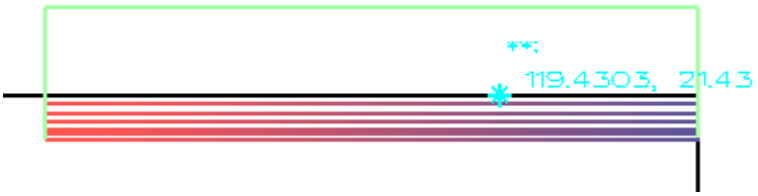


(2) Thread Type

It used to specify the type of cycle: Multiple Cycle or Basic Cycle.

- **Multiple Cycle** – In ZW3D CAM, the CNC(Computer Numerical Control) will finish most calculation of the multiple cycle toolpath, including the number of passes and cut depth for each pass, so ZW3D only shows the final pass.

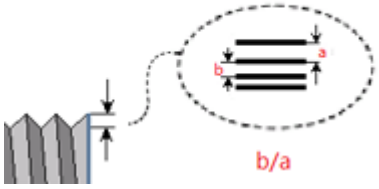
- **Basic Cycle** – ZW3D shows a complete toolpath.



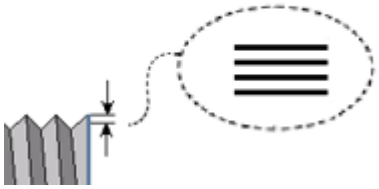
(3) Cut Type

This is refers to the step over for rough threading. Two options are available. They are “% remain” and “Constant”.

- If the Cut type set to “% remain”, the default value is 40. It means that the actual step over is 40% of remaining depth at present.

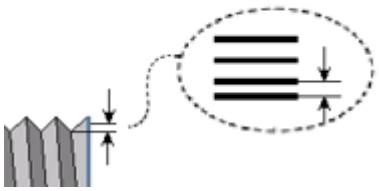


- If the cut type is set to “Constant” , the default value of the option of **Cut depth** is 0.2, and then the cutting will be implemented at the value we give to the option.



(4) Min Depth

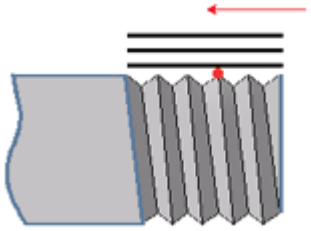
It specifies the minimum depth in cutting. If cutting depth is less than the Min depth then system will implement at the Min depth.



Limiting Parameters

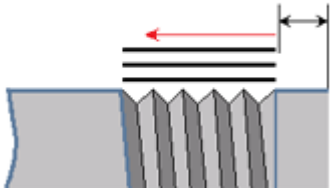
1.Cut Position

It refers to the starting point of thread. Click on the button and then go to the geometry to select a point for beginning.



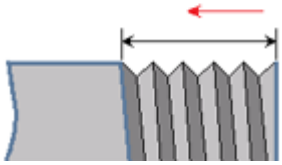
2.Start Length

It is an axial distance between the start point of the thread and the position point.



3.ThreadLen

This is the length of the thread including the pull out length. If this option blank then the whole line will be threaded.



Path Setting parameter

1. Cut Direction

There are two types of Cut Direction：Right to Left and Left to Right.

2. Cut Region

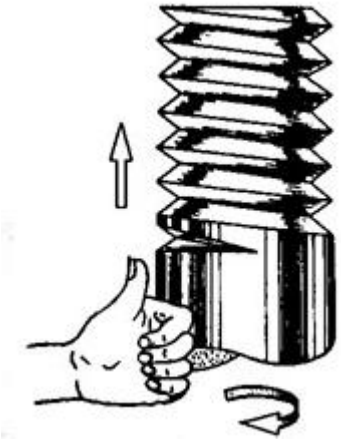
Specifies OD cut or ID cut for Threading.
Note: Different position require different style of tool to produce a proper thread.

3.Numbers of Spring

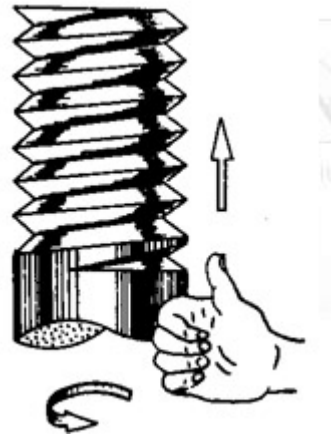
The default value is 1, means single thread. For the multiple thread passes input the corresponding value in the field. For example two means double threads.

4.Hand of Thread

This is used to select right or left hand threads as shown in the images below.



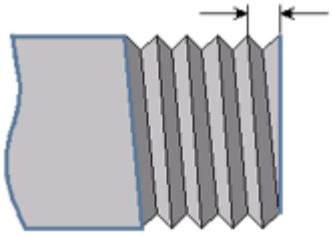
Left hand



Right hand

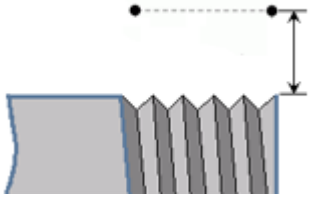
5.Pitch

Specify the pitch (the distance between each thread) for threading.



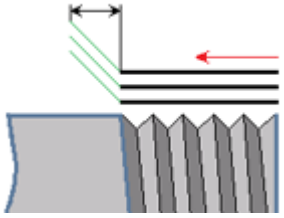
6.Clearance

This is the distance in the radial direction between the engagement point of each cutting pass and the crest of thread .



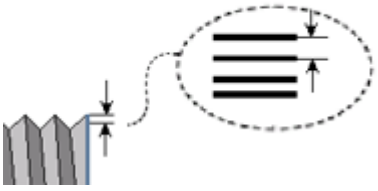
7.PulloutLen

It refers to the axial distance away from the end of thread when the tool pulling out of the part. **Note the Pullout length is a part of the length of thread.**



First Pass Depth

This is available when the G76 is selected**.** It specifies the depth for the first cut. Note, it will not reflect in the tool path but will be written to CL file



8.Finish Depth

This refers to the overall depth of the cut. If the value is 0 or left blank then system will only finish the bottom of the thread.

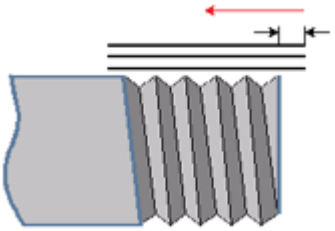
9.Number of Finish

This refers to the number of cutting passes for the finish. It should be larger than zero .

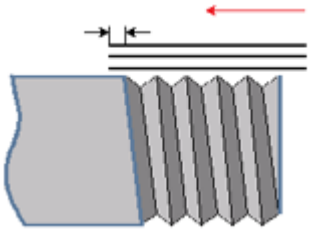
link and Lead parameters  .

Extension Distance

It specifies the extension distance for engaging and retracting the tool.



Extension Distance（Lead In）



Extension Distance（Lead Out）

Calculate the tool path again to check the differences.

Verify it again.

Save your file.

Chapter 9 Part Off

This chapter covers：

- **Preparation for Part Off**
- **Parameters Form**
- **Calculating tool path**
- **Solid verify**

This is the last operation in Turning. It is easy to get the tool path just by specifying a point on the geometry. Now let's take a look at how it works.
First open the file we saved in last chapter.

Creating the Part off operation

Select the Turning module in Operation Type and select the Part Off operation  to create a new operation. A new part off operation with the default name "**Part Off 1**" will be created under the **Operations** in the CAM manager tree.

Creating tool
Go to the **Tool manager** set the **Type** to Grooving and then select the **Parting Off Right** from the **Subtype** list. Then give a name "Part Off" to this tool.

Add the feature
Use the same process as before and add the same profile feature to the Features of Part Off operation.
Next we need to define the parameters.

Parameters Form
There are 4 Tabs:

- Primary Parameters
- Limiting Parameters 
-
- Path Setting parameters
- link and Lead parameters 
- Display parameters 

We have gone over the last two tabs before, so we will go over the **other Parameters**

Primary Parameters

1. **Basic Parameter**

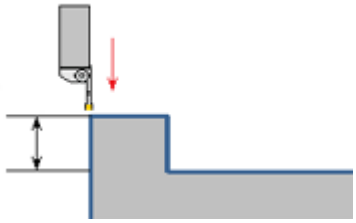
2. Tolerance and Steps

(1)Enable Peck

Determine whether or not to use the peck mode to cut.

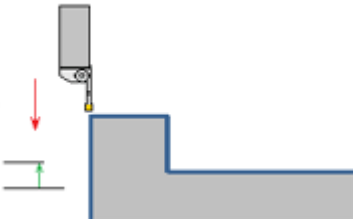
(2) Peck Depth

This will specify the depth for each cutting pass.



(3)Retract Offset

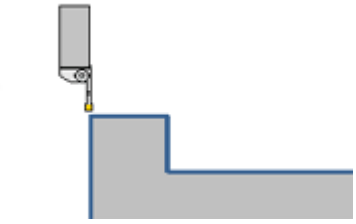
This refers to the retract distance after each pass.



Path Setting

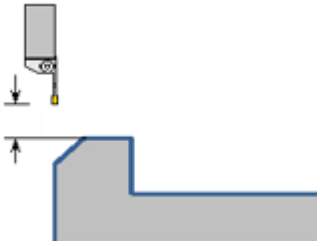
1.Cut Off Point

Select a point on the geometry as the point cut. Then the cutter will cut off the part from the point in the direction perpendicular to the axis of the part. The cutoff point must be specified otherwise the tool path will not be created.



2.Return Height

The position that the tool retracts to after cut off operation. It makes sure the tool can first go to a safe plane before it rapid returns to the start point.



3. Geometry Corner

Using cut off, we can specify the type of the corner geometry. These are as follows:

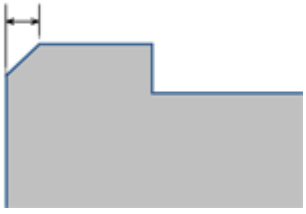
(1) None-It means the tool will cut off the part always in the direction perpendicular to the axis of the part.

(2) Chamfer – It will cut a chamfer as shown in the image below.

There are additional options to control the shape of the chamfer.

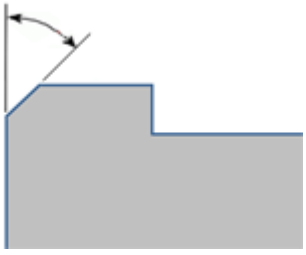
1) Chamfer Width

This specifies the width for chamfer as following. This value should not be bigger than 0.75 Tool Width



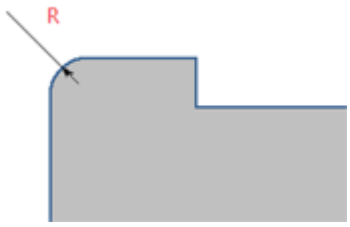
2) Chamfer Angle

This refers to the angle between the truncation face and the bevel edge of chamfer as shown in the image below.



(3) Fillet

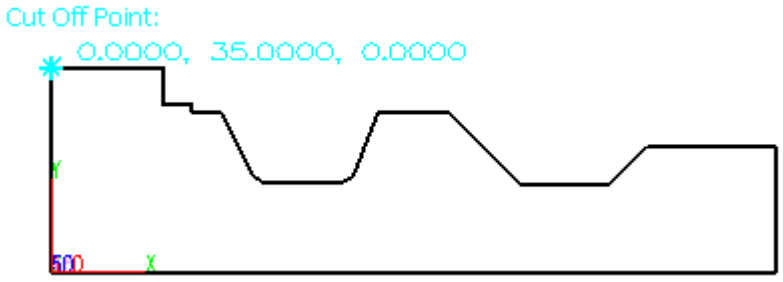
This option will produce a fillet cut as shown below. You need to specify the fillet radius. The radius of the fillet cannot be bigger than 0.75 Tool Width



Now go to the parameters for the tool path part off with a chamfer.

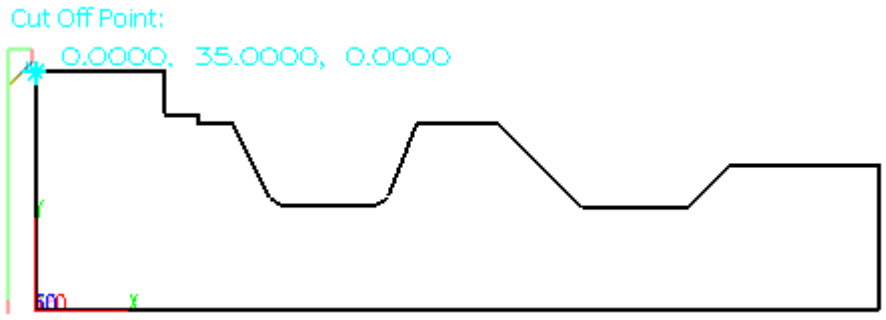
Set the parameters as following:

Cut Off Point= On the left side of the inserted sketch select the end point as shown in the image below:



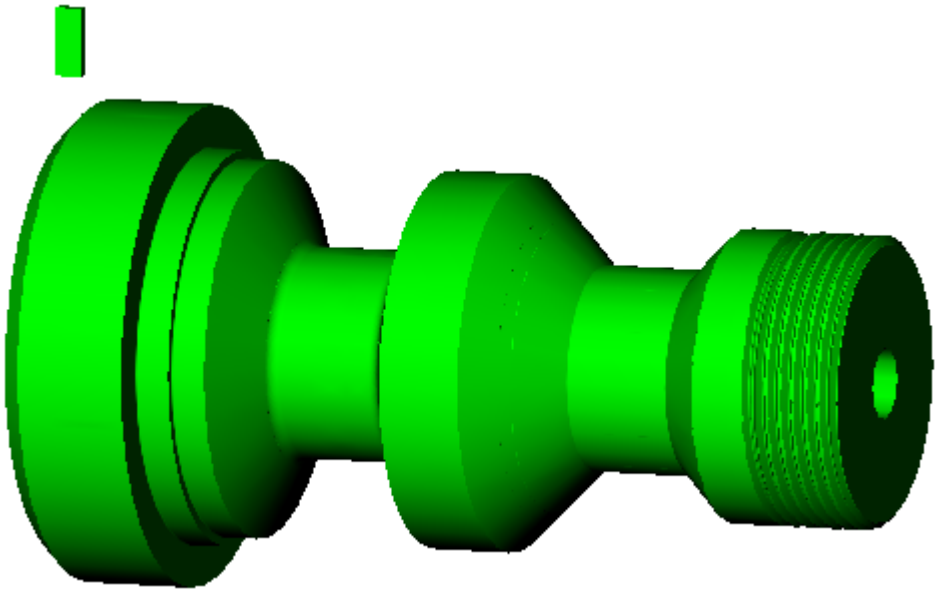
Corner Geometry =Chamfer
Chamfer width= 3
Set the Leads and links parameters
Extension distance (Lead Out)= 0
Leave the other parameters in their default values.

Click the calculation button and the tool path will generate as in the image below:



This completes the exercises for all the operations.

Next simulate all of the operations with **Solid Verify**
Go to **Output** and left click on it. Then select the **Verify** tab. Click on the **Start Verify** button and go to the solid simulation interface. Click the start button , then we can check the operations and see the results.



Save your file

Congratulation! You have finished the training on ZW3D Turning.