ZW3D Turning Training Manual

II

Forward

The full version of the ZW3D documentation is available by selecting Help Browser from the ZW3D Help Menu. Go to the "\ZWSOFT\ZW3D 2012 Eng\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 Adobe Acrobat Reader and open the corresponding files in the "\ZWSOFT\ZW3D 2012 Eng\PDF" folder in the program directory. Refer to the HELP Manual for information relating to specific tasks.

1. For best results, view and print the PDF version of the documentation using Adobe Acrobat Reader.

2. Since the Turning Training Guide is only a portion of the complete documentation, it is highly suggested that you utilize the HELP Manual. You will find additional topics and more information on the commands, 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.

ZW3D Training recommends that users print or make an extra copy of Appendix B and Appendix C to use as a quick reference guide for ZW3D milling operations

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!

The ZW3D Team

Contents

Chapter 1: Introduction

Preface

The units used for this training are in mm. We recommend that if you are not already using mm for your default units, you should make a copy of your "_config" file as a backup, delete the "_config" file and restart ZW3D. You will then be asked what units you want to work in and you should choose mm for the purpose of using this training manual.

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.

➢ 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.

Note:

➢ Sketcher must be closed

➢ The part should be columnar part

Chapter 2: Basic Walk Through

This chapter will guide you thru 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

Open a new file ![New] and then select the CAM plan in the Create new file dialog box to create a new **CAM plan**. We also can give it a **Unique Name** in the dialog box as shown below:
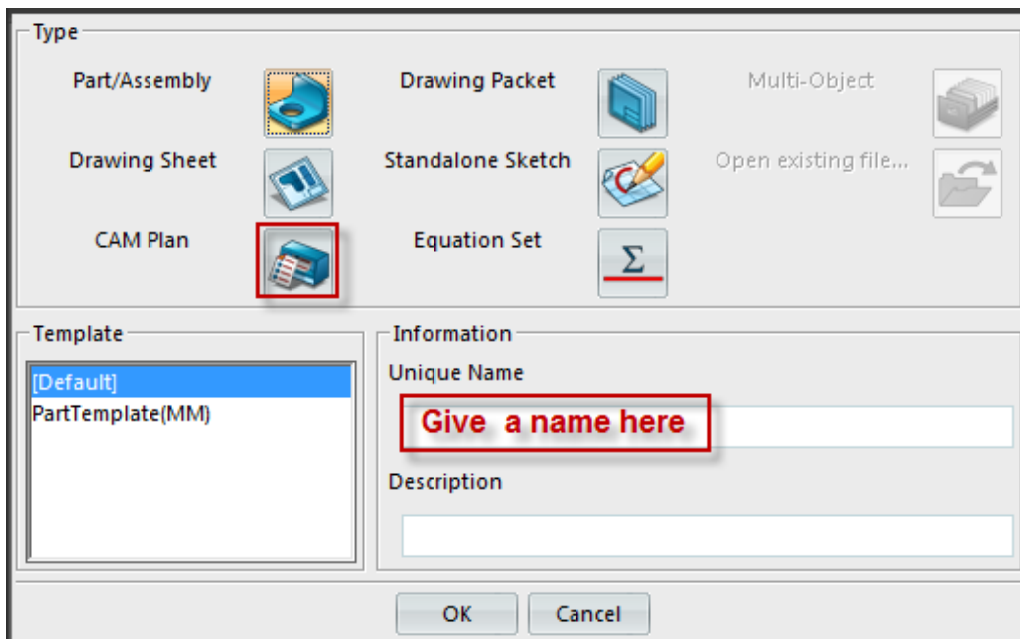
Step 2 – Got to CAM setup manager

After selecting Ok in the dialog box, we can go to the CAM level. Then in the left of the screen, we can find the Manager. In the Top of the Manager there are 3 icons as shown in the images below:

CAM Manager ![icon], View Manager ![icon], Visual Manager ![icon].

In the CAM manager there is a Manager tree with a default name of Setup1 in the top of it. Expand the tree and we can find there are several branch options under it as follows:
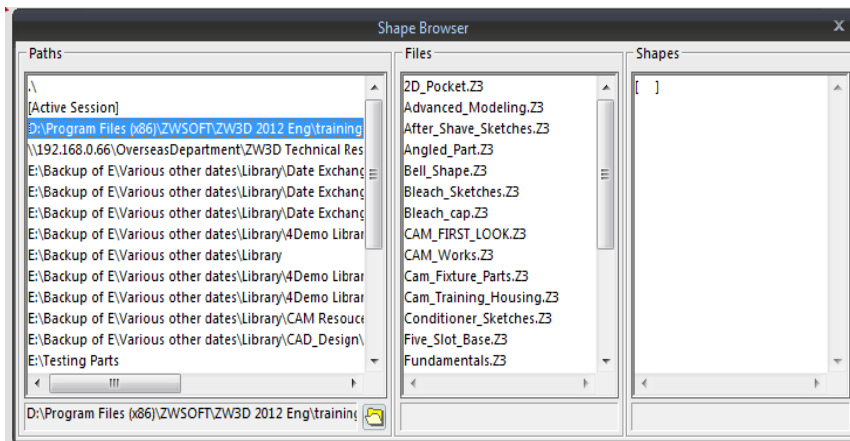
> **SetUp1**
>> **--- Geometry (Undefined)**
>> **--- Clearance**
>> **--- Frame**

<div align="center">

--- Tactics

--- Operations

--- Machine (Undefined)

--- Output

</div>

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

Step 3—Inserting Geometry

Under **Setup 1** left-click on the icon **Geometry,** then it will display the "Shape Browser" as shown in the image below.


We need to select the geometry from the browser that we are using here.

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.

Then in the **Files** list shows all of the files located in the "\training" directory. Select the file "Turning_Model.Z3". Select the part"Turning Geom" from **Shapes** list. Select "Yes" to insert the geometry when the warning information pops up. Then the screen should display as following:
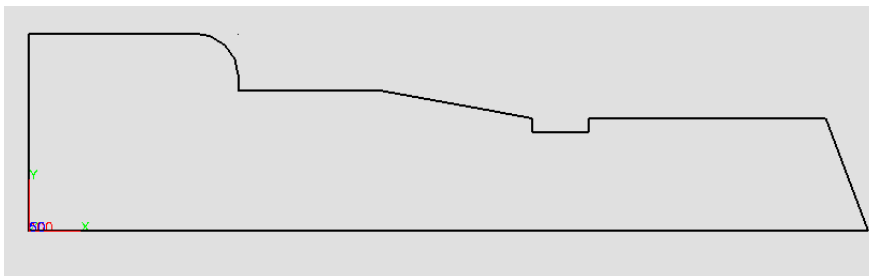

Note: What we added is a closed 2D sketch.

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

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 as shown in the image below.
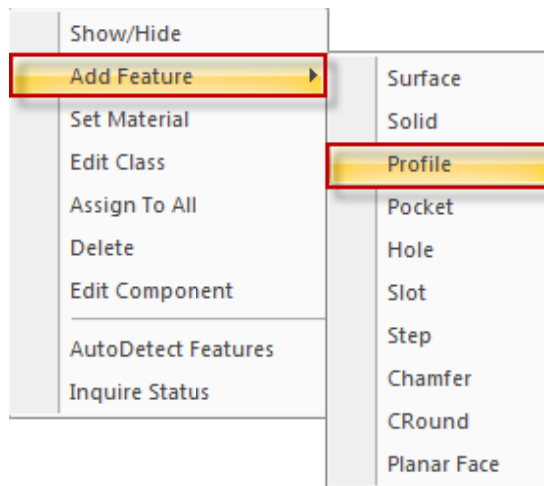

Then a dialog box for geometry selection will pop up. Set the Input type option to profile as shown in the image below.
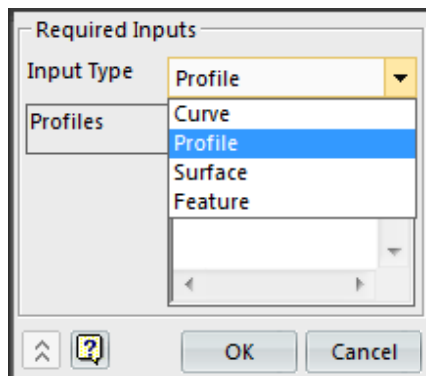
Then go to the Profiles option, pick the sketch as the profile as shown in the image below.
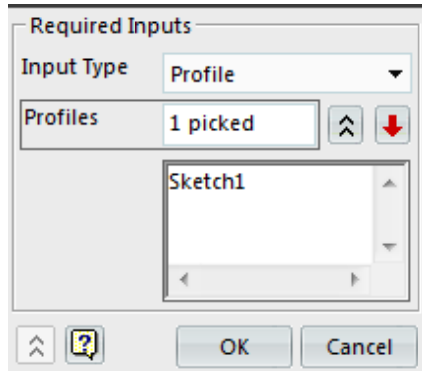
The dialog box of the Profile Feature will display when you select ok to finish the selection of profile.

In the dialog box of Profile Feature set the **Type** to "Part" and change the direction with option "**Reverse Dir**" as shown in the image below.
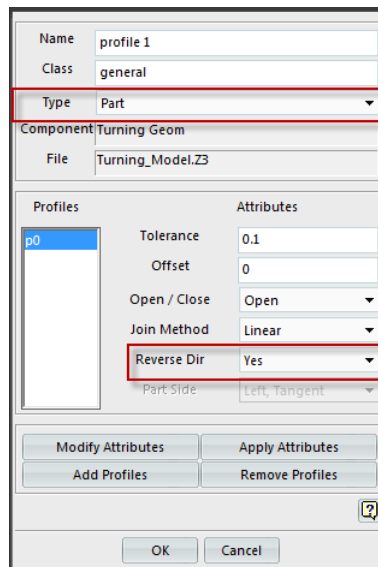
Make sure the direction of the profile is counterclockwise as shown in the image below.

Note: The direction of the profile must be counterclockwise

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

--- Clearance

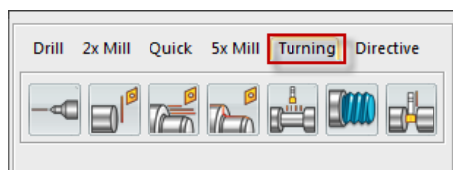--- Frame

--- Operations

--- Machine (Undefined)

--- Output

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 can use the default setup.

Note: Now ZW3D Turning use the XY Datum as the working Datum. So before programming, make sure the XY Datum is the working Datum.

Step 6 – Creating an operation

Go to **Operations** under the Setup1 and right click on it, then the Operation Type displays as shown in the image below.

Go to the Turning Tab and select the operation "Rough Turn" .

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

Left 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. Please refer to the following picture:
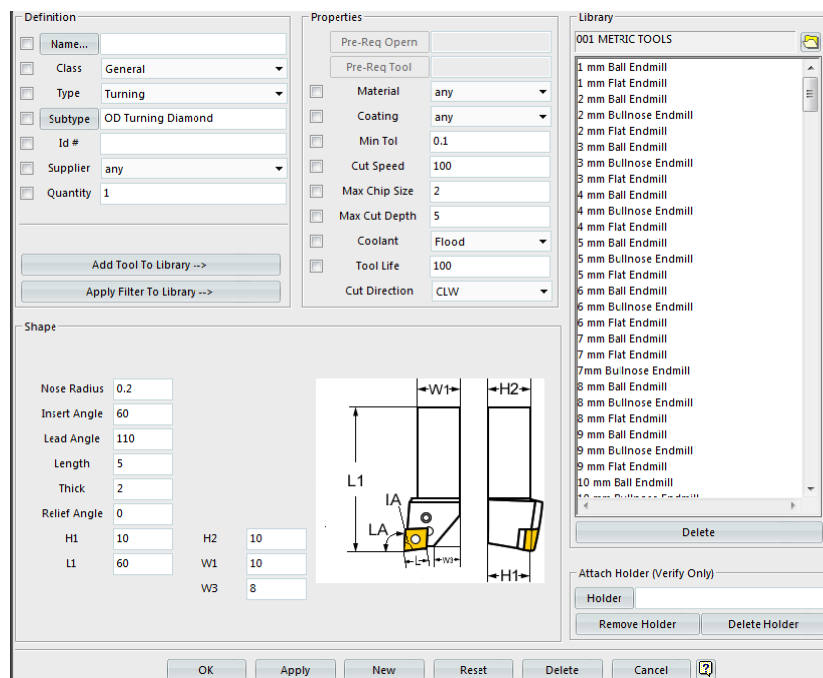
In the **Definition** area, you can rename the Tool. In the **Name option** Name... , if you leave this field blank then system will automatically assign a default name like Tool 1, Tool 2…etc to the tool after finishing the definition.

Input "my first tool" in the **Name** option.

Note: Don't use a comma in the name.

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.

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

Go to the **Parameters** in Rough Turn 1 and left click on it, then the **Parameters Form** will display as shown in the image below.

Actually, the form contains all of the default parameters for generating tool path except for the parameters of the cutting region. We first need to define the cutting region parameters. Go to the limiting parameters .

You can define this by selecting the first point as the start point and the second point as end point on the sketch to limit the cutting region as shown in the image below.
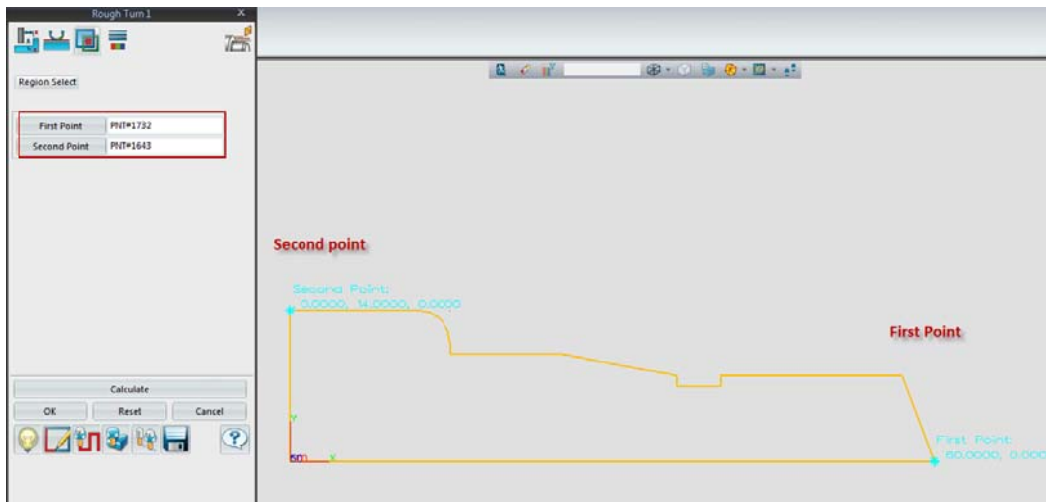
Now select ok to it and go to the Feature of Rough Turn 1

Note: The cutting region must define

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 select 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**
>   **---**Profile1
> **--- Machine** (Undefined)

Step 10—Calculating

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:

13
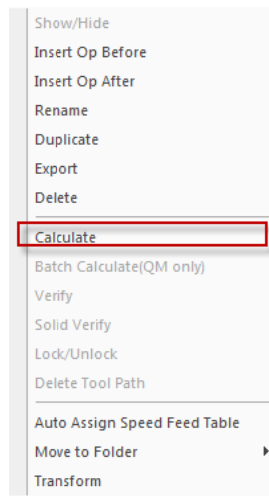
Your results should be similar to those shown in the image below.
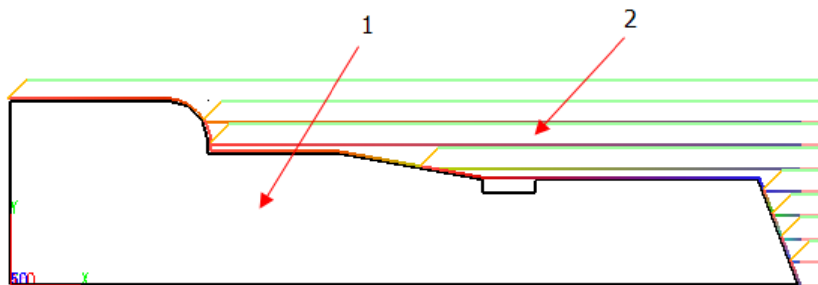
1: Geometry 2: Tool path

Step 11- Defining the Machine

Next we need to define the Machine for proper output:

Go to the **Machine** under the Setup1manage tree and left click on it. The **Machine manager** will display. From the manager we can find the default setup for output. For Turning we need to change the Post processor to Turning. Currently ZW3D Turning module provides 2 kinds post processors for user: one is the "Fanuc" another is the "Simense802d" just as displayed as the image below.

Then Go to the **Post Configuration** Post Configuration and left click on it then select a post processor from the list as shown in the image below.
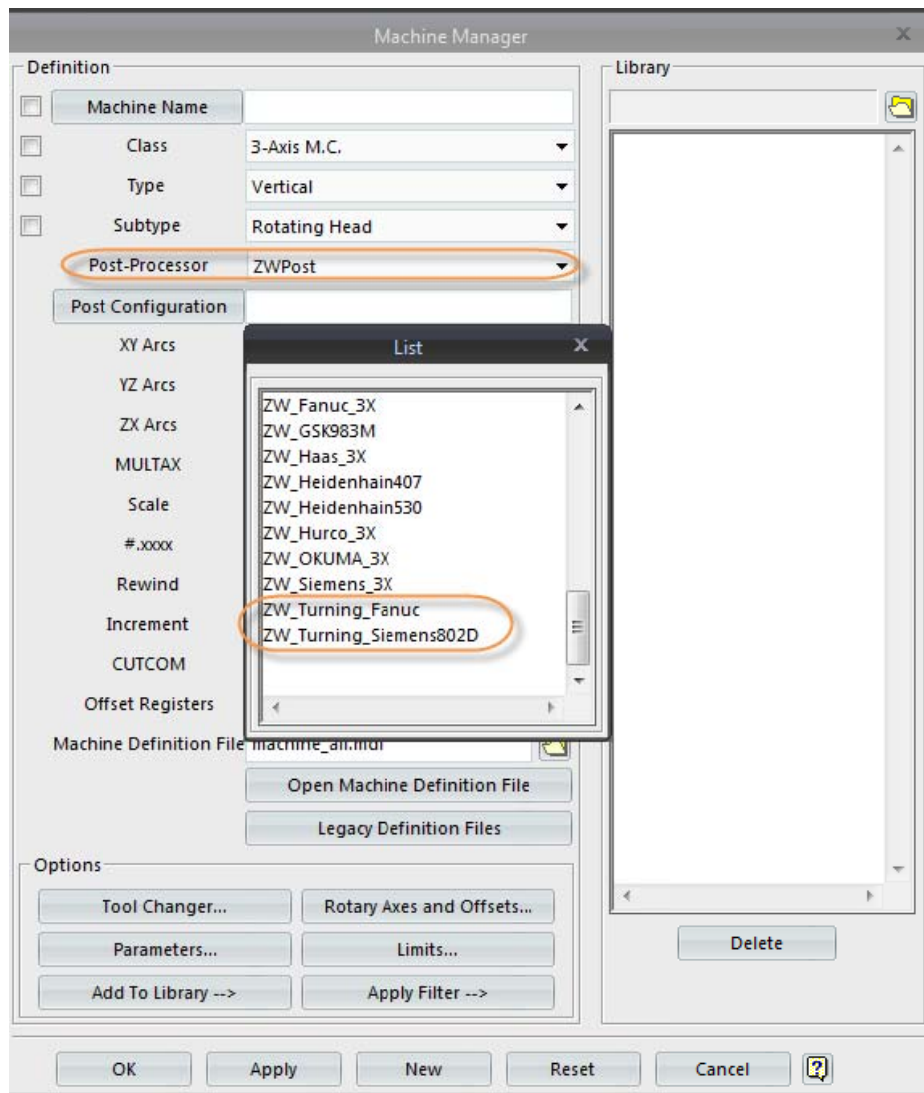
We can also define the machine by the option of **Machine Name** [Machine Name]. If this field is left blank the system also will automatically assign a name like machine1, machine2 … etc to the new one. Input the name "my first machine" in the **Machine Name** field. Then click the apply button to have the new name inserted in the field. Then "my first machine" will be added to the Rough Turn 1. Click the Ok button to close the form.

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 as shown in the image below.
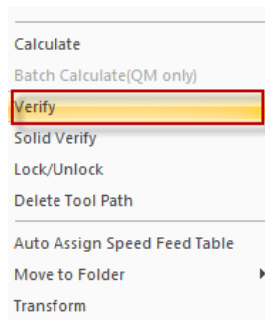
Then the **Tool path Verify** dialog box will display**.**
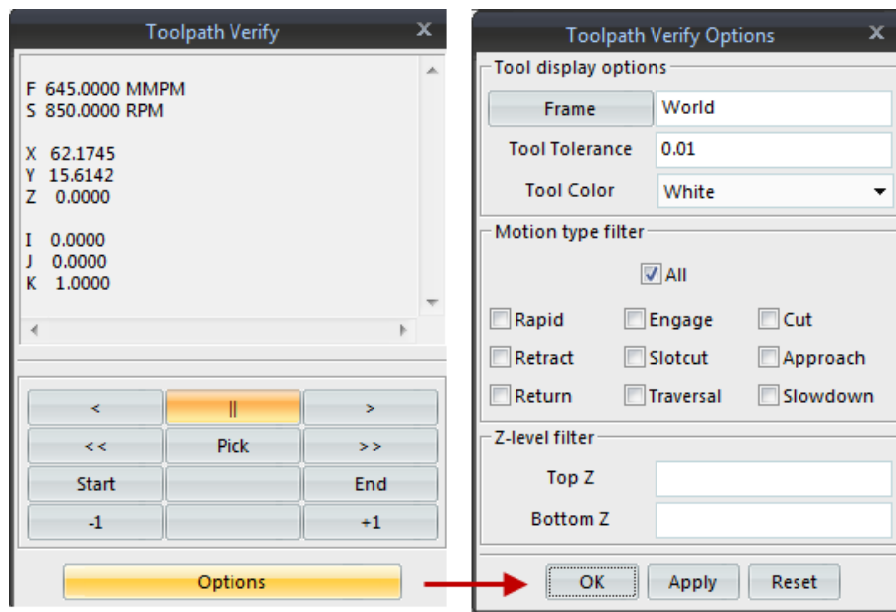
It also can be selected with the button "**Verify operation**" located at the bottom of the **Parameters Form as shown in the image below.**

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.
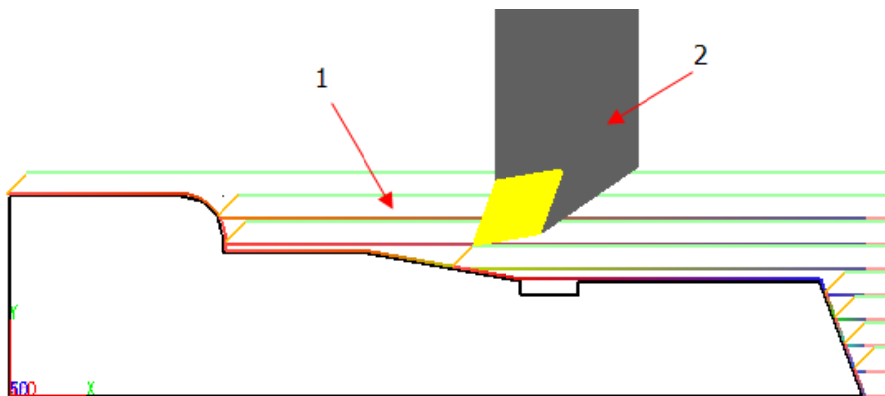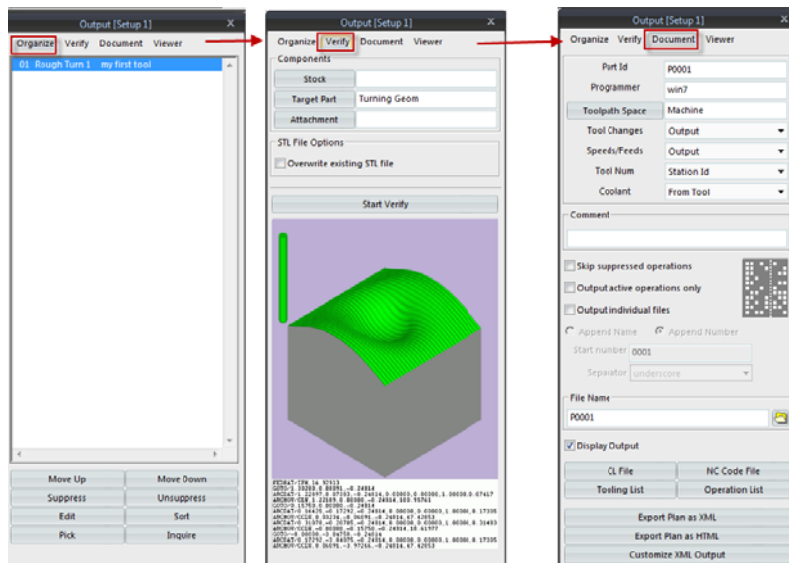

1: Tool path 2: Tool

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.

Go to the Output option in the bottom of the Setup manage tree. Then left click on it and the Output dialog box will appear as shown in the image below.

17

Under the **Organize** tab, all of the operations calculated will be listed. (Refer to the image above). Selecting the **Verify** option 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** interface **dialog box** will pop 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 as in following:
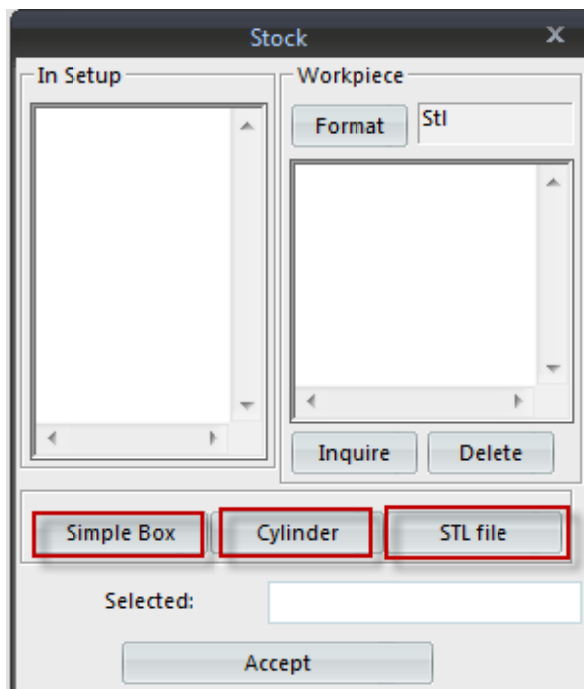
**Note: Solid verification must have a stock**

Clicking the **cylinde**r button, the Cylinder stock dialog box will display as in the image below.
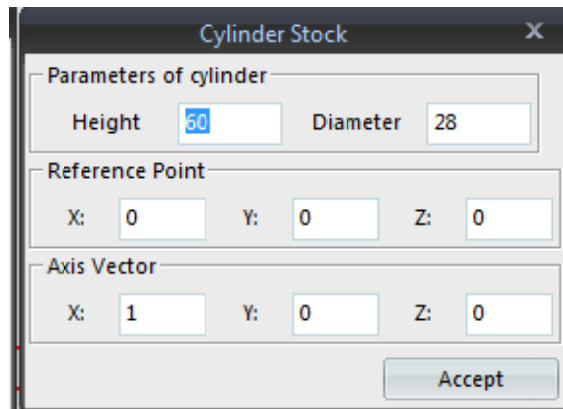
Select the Accept button and the cylinder stock will be added to the geometry in the graphic area.

Next click the Start 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:
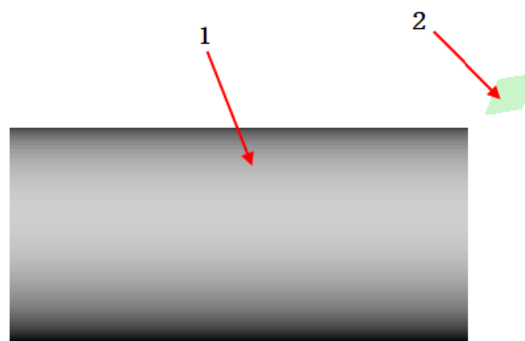
**Start "Solid Verification"（1: Stock 2: Cutter）**
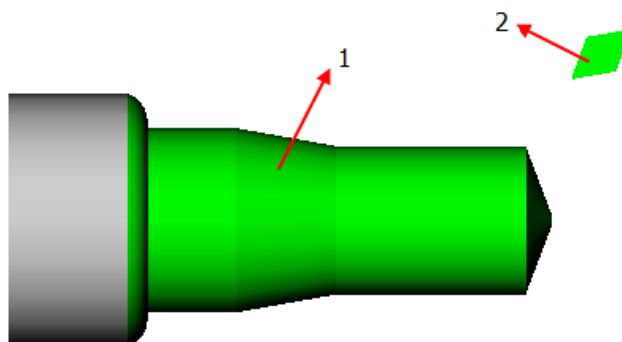
Solid verification end (1: Target part 2: Cutter)

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

Go to the **Documentation** tab.

Select Display Output and then CL file windows will be displayed. At this point in the process the window is blank. Now we can create all the documentation needed to complete the job.
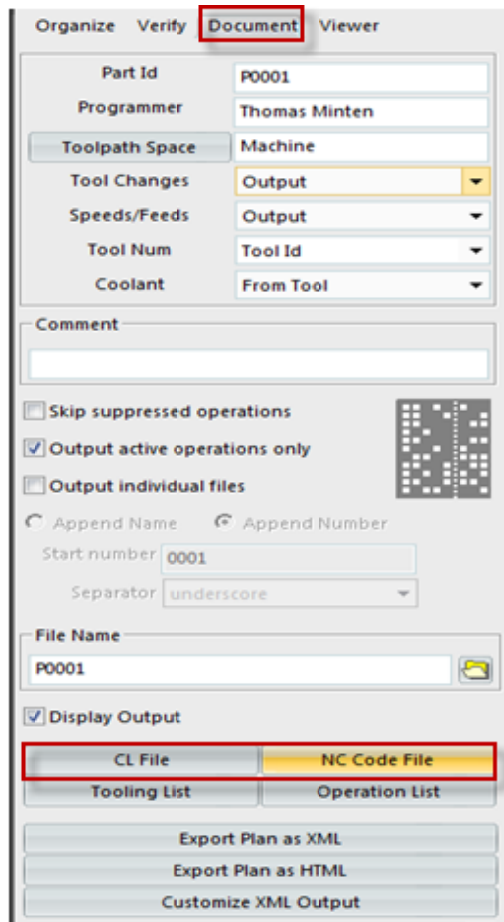
In the **Part Id** field ⬚Part Id input "**my first tool path".**

Set **Tool Changes** to output.

Set **Speeds/Feeds** to output.

Set the **Tool Num** to Tool Id.

Ensure the Display Output is checked.

20

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

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

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

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

Click the **Export Plan as XML** to display the Operation List in an XML format.

Close the CAM output dialog box.

Save the file by clicking the button.

All of the output files located in the "/output" folder in the default user directory. 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 .

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 default Template file is located in the "\resource" folder of the default 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 each time 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 default installation directory. Then create a new CAM Plan with the name "Model_Turn_Plan". For this exercise we will not use any Templates. 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

Now right click on the **Geometry** tab. Then select insert on the **Shape browser.**

Select the part named "Lathe_Part.Z3". Highlight the part named "Lathe_Part.Z3", and select "Yes". The part Model will be added to **Geometry.** The graphic area the geometry should appear as the image below.
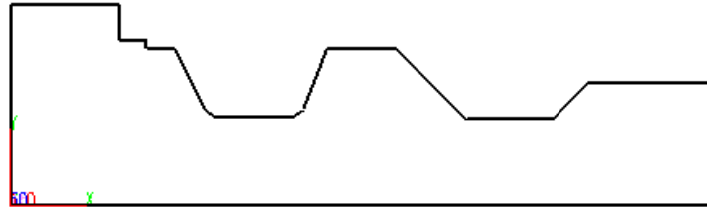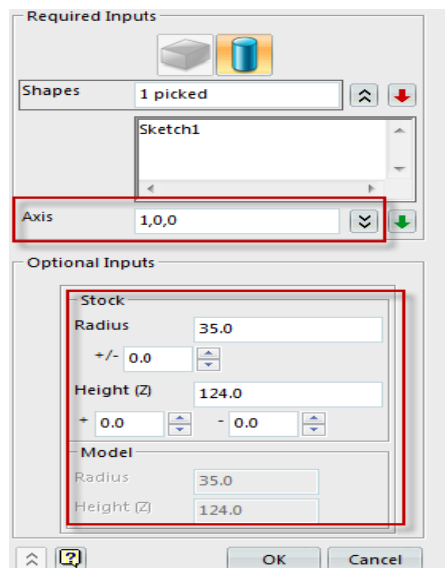

Creating cylindrical stock

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 as shown in the image below.

Select Cylindrical Stock icon, the command dialog box will display as shown in the image below.

Set the filter to "Profile" and then select the sketch, then the stock will be automatically created as shown below.
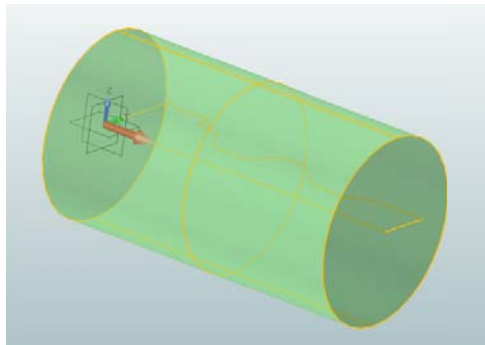
X axis is the default axis. It can be changed from the pull down menu or you can input the vector manually. We will accept the default axis value for this exercise. Also in this form we can set the Radius and Height for the Stock. For the "Height (Z)" input the value 126. The size can be adjusted by the button"+/-.

The radius of the stock in X axis is the same as the sketch, but the height is 2 mm higher than sketch in +Y direction. Select "Ok" to create the Stock.
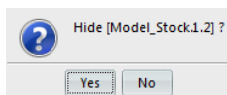
When this prompt appears select no. Then the stock will display as transparent like the image below.
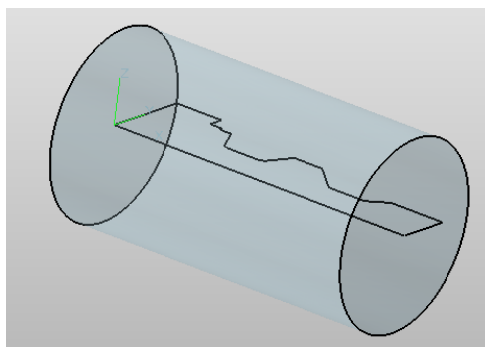
Now you can see the new stock has been added to **Geometry,** and the attribute of stock also has been assigned to it. The display mode can be changed by left clicking on the stock.

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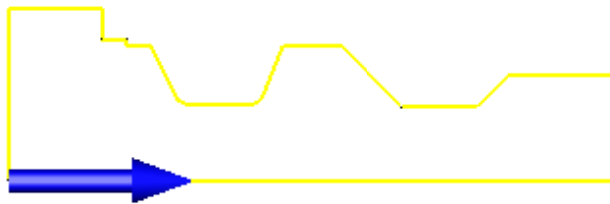
Make sure the direction of the arrow is pointing Count clockwise.

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 operation

This chapter covers

Creating Drilling operation
Creating Tools
Inserting the feature for operations
Cutting Parameter Form
Calculating Tool path

First, go to the folder "\training in the default 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** Turning and pick Drill operation icon under it to create a new Drill operation.

2) Left 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** tap and select the Turn Drill operation .

Note: **The system will assign names like Drill 1 and Drill 2 to the new operation automatically if you do not want give the operations custom names. 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 "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.

Defining Tool

Right click the Tool of "**Drill 1**" and then pick "**Select**" from the context menu as shown below.
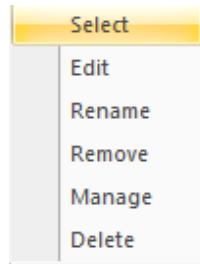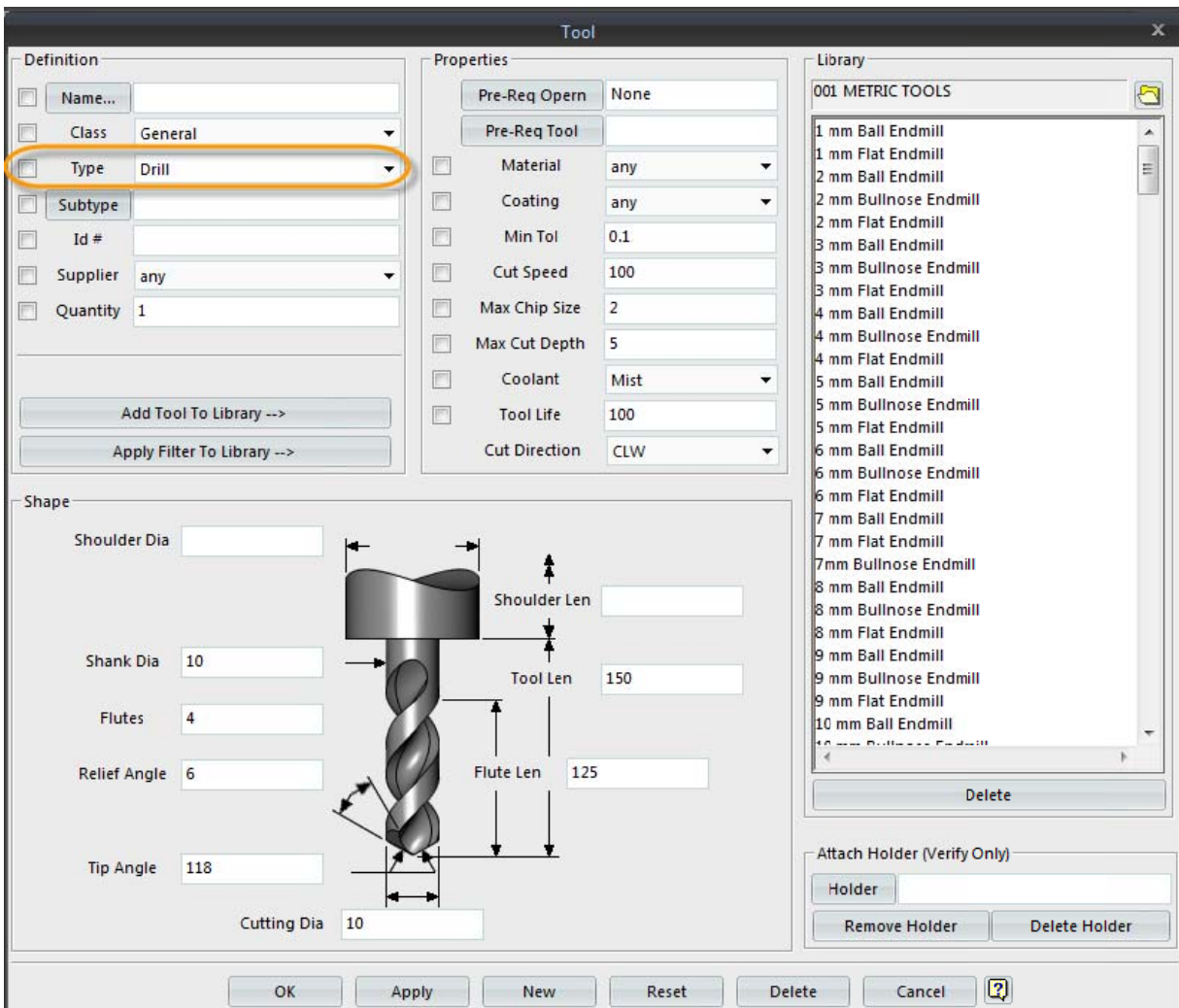
This will bring up the tool manager.

1). In the Name option input the name "Drill". The system will automatically assign a default name to each new Tool like Tool 1, Tool 2 and so on. Set the Type to Drill.

In this exercise we will drill a simple hole with the diameter of 20 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 just like the picture above.

29

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 restart ZW3D again and select the Millimeters as the default units as shown below.

Default units?
⊙ Millimeters ○ Inches

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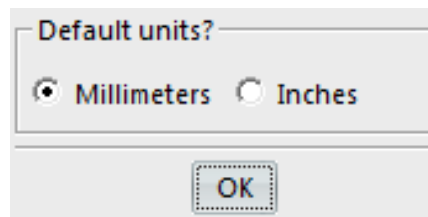
For this exercise we will use the default setting for this Twist Drill
Select ok to add the Tool to Operation Drill 1.

Adding the feature to the operation
Go to the Feature option under the operation **Drill 1**and 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 and left click on it.
The Drill parameters form appears. It includes several options like following:

Cutting Parameters

More cutting parameters

Display parameters

The following is an explanation of some of the 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 values.

Cutting parameters Tab

Drill Type

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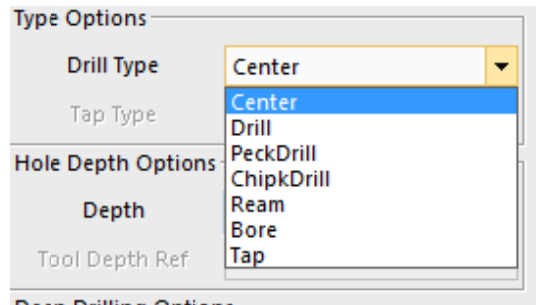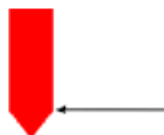

Select Drill.

The **Hole Depth** options as shown below**.**
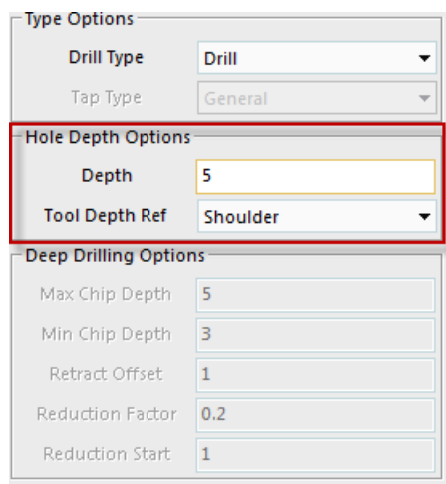
The Depth option specifies the working depth of the hole.

The Tool Depth Ref option determines depth location on the tool. This location could be to the point of the drill or the shoulder of the drill

For the Tool shoulder, it ensures the Tool shoulder is down to the depth specified in the form.

Tool Shoulder

For the Tool point, it ensures the Tool point is down to the depth specified in the form.

31

Tool Tip

Deep Drilling Options

Deep Drilling Options are available when Pick Drill or Chip Drill is selected.
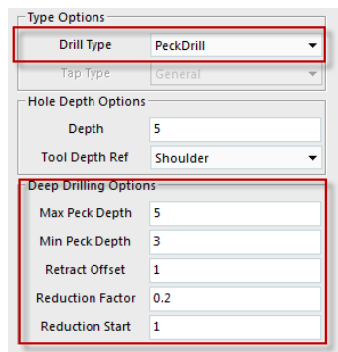
Max Peck/Chip Depth and Min Peck/Chip Depth.

This is used to define the parameters of Peck or Chip drilling.

Retract Offset

This is the retracting distance between the current drilling peck depth and the previous one. Different Drill Type has different retracting values. For peck Drills, the tool will retract to the top face of the hole and then go to the retracting offset position; For the Chip Drill, 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..
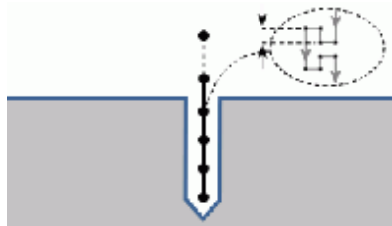
Reduction factor

The reduction factor is the 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.
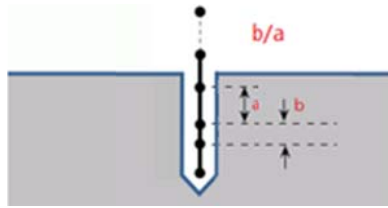
Reduction Start

This is a sequence number from which the Peck/Chip will cut at the reduced depth.

New Drilling Depth = Max Peck/Chip Depth *(1-Reduction Factor)

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.
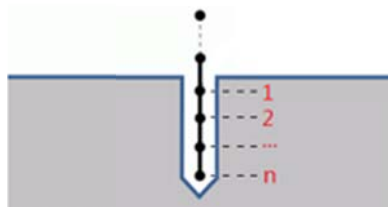
**More cutting Parameters**

Speed and Feed

Specifies spindle speed and feed rate.

Dwell time

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.
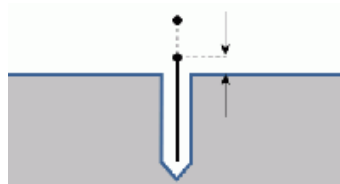
Selecting **Display parameters Tab**

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 and display parameters** in help documentation   .

Calculating the Tool path

There are 2 ways to calculate the tool path.

First go to the operation "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 as shown in the image below.
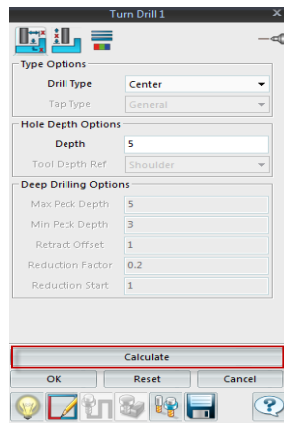
Now the drilling tool path has been created in the center axis of the sketch as shown in the image below:
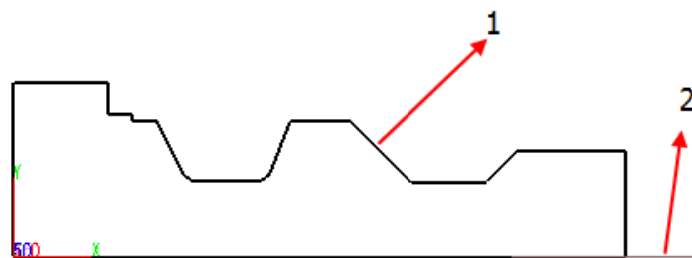
1: Geometry 2: Tool path

Double left click on the operation "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 a Facing operation
Creating a 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 a 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 Turning Diamond.** Last assign the name " **Face off**" to it by the 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:

Cutting Parameters

Leads and link parameters

Feed 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.
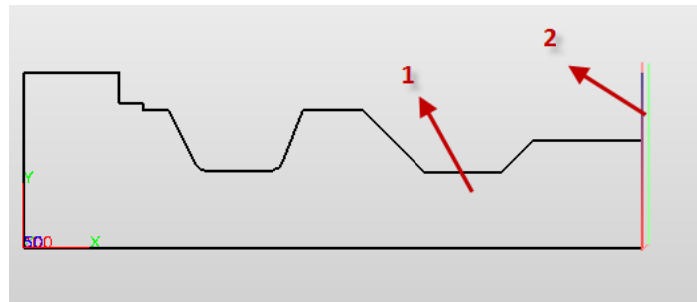
1: Geometry: 2: Tool path

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.

Select the **cutting parameters Tab** again and return to the cutting parameters form. Next let's take a look at some other options here.

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.
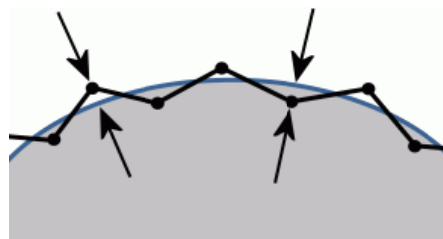
Step over

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.

Over cut

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.
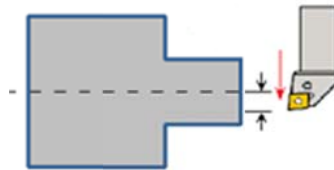
Surface thick

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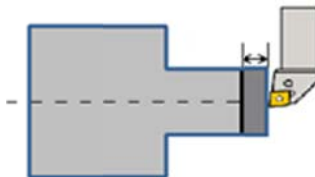
Numbers of cutting passes

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.
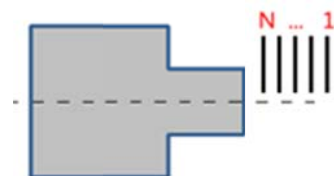
Start point

This specifies the starting point at the beginning of the operation.
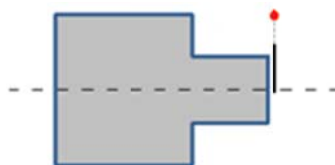
Leads and link parameter

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

Speeds Parameters

This option specifies the speed and federates of the Rough, Finish, Engage and Retract operations. 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 feed rate to all of the operations you can use the speed and feed rate command from the context menu of each operation.

Dynamic View

Speeds and Feeds Manager

Set origin | Allows you to copy speed feed tables from a "library"

Zoom All | F1 to more details

Insert Operation

Insert Hole Tactic

Insert Mill2 Tatic

Insert Mill3 Tatic

Sketch ▶

Spreadsheet Interface - Operation View

Spreadsheet Interface - Feature View
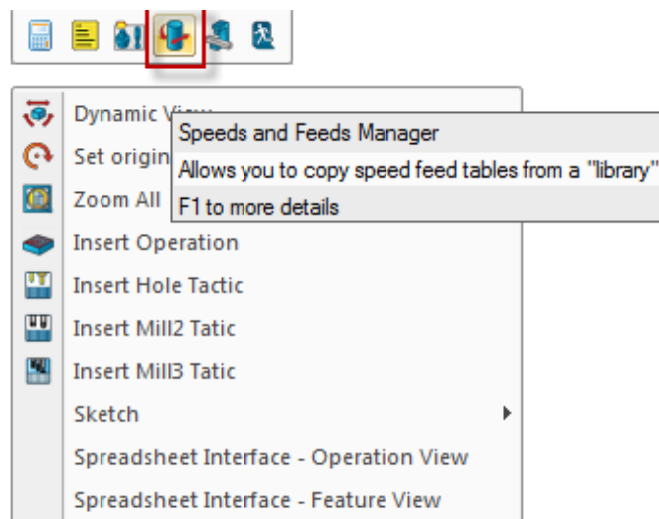
After finishing your set up for this operation, calculate the tool path again.

Double right clicking on the icon of operation "**Facing 1**" can hide the Tool path

Save your file.

Chapter 6: Turn Rough and Finish

This chapter covers:

    Creating Turn Rough operation
    Creating Tools
    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 "M**odel_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 in the tree.

Creating a Tool

We will use the same method in the previous operations 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.

Cutting Parameters

Leads and links parameters

Limits parameters

Display parameters

41

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.

Cutting Parameters Tab

Cut direction

This is used to specify the **Outside Diameter** cutting or **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.
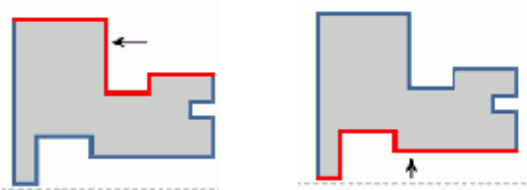
Out Diameter   Inner Diameter

Cut Mode

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 mode
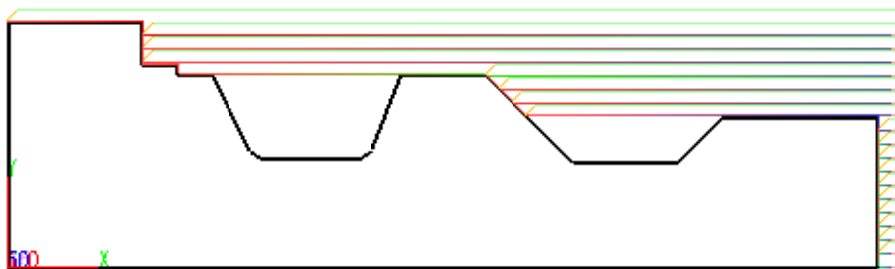
Cut direction: Outside diameter

Vertical mode
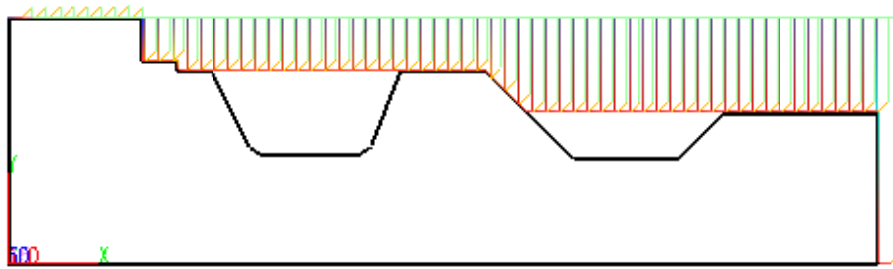
Cut direction: **Outside diameter**

Pattern Repeat
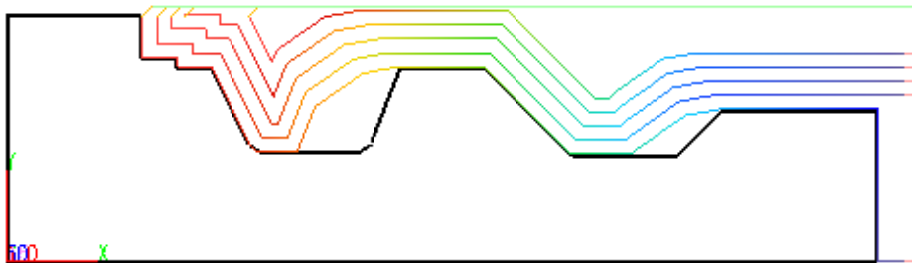
Cut mode: **Outside diameter**

Step size

This value determines how much material will be removed at each cut.

Note: **The maximum value can't exceed the cutting length.**
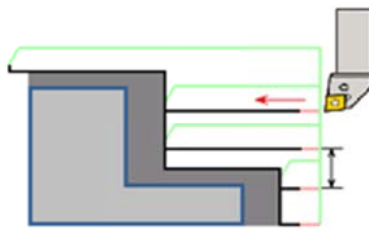
Thick

This specifies how much materials should remain in the XY plane and Z level after this operation. This value can't be a negative value.
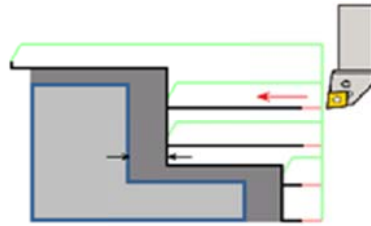
Now set the cut direction to OD, cut mode to horizontal and leave the Thick option at the default value "0.2". Then select the Calculate button. Zoom in to the tool path above right area to get the tool path as shown in the image below. This will show the remaining material.
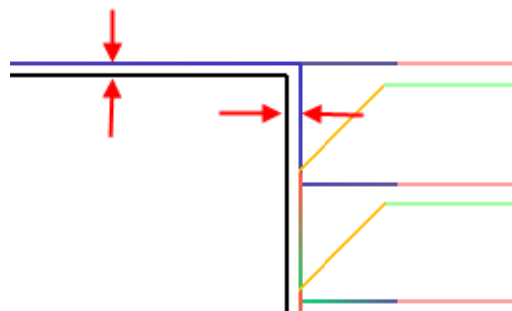
Thick=0.2

Change the thick value to 0.5. Calculate the operation again then zoom in to the same place. We can see the gap between the tool path and the part has increased to 0.5.
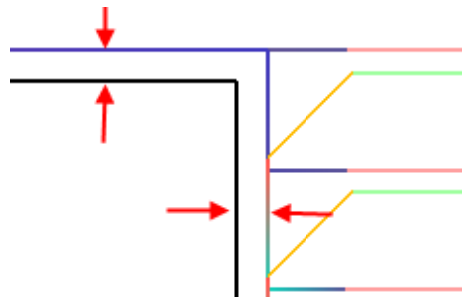
Thick=0.5

Overlap

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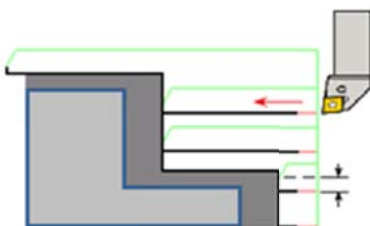
Number of Passes

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.
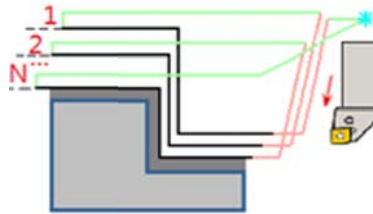
Station Point

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.

Leads and link parameters Tab

Lead type

This allows the user to specify the path type when tool engages the material. Three types are available.

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.
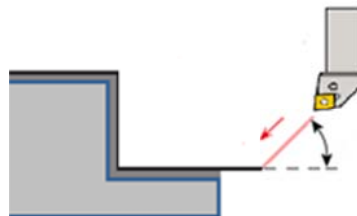
Tan line- The cutter engages the material by line tangent to the cuts of the operation as shown in the image below. The distance is defined by the user.
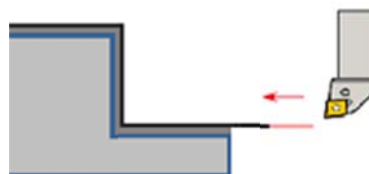
Tan Arc – The cutter engages the material by an arc tangent to the material as shown below. The radius and center of the arc is user defined.
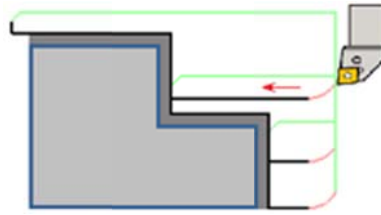
Extension Distance (Engage)

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.
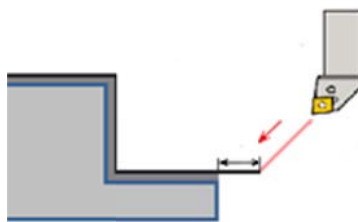
Extension Distance (Retract)

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.
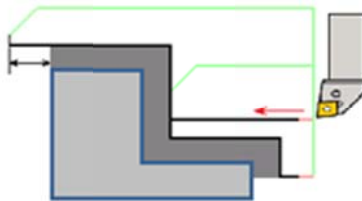
Limiting parameters 。

These parameters are used to limit the cutting region by assigning a start point and an end point as shown in the image below.
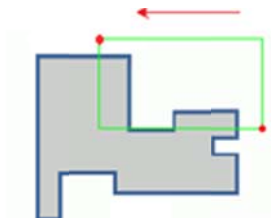
Change some of the parameters we have discussed here and look at the differences in the tool path.

Save the 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 **Tools** and **Features.**

Use the same process to define the Tool as before. Give the operation 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.
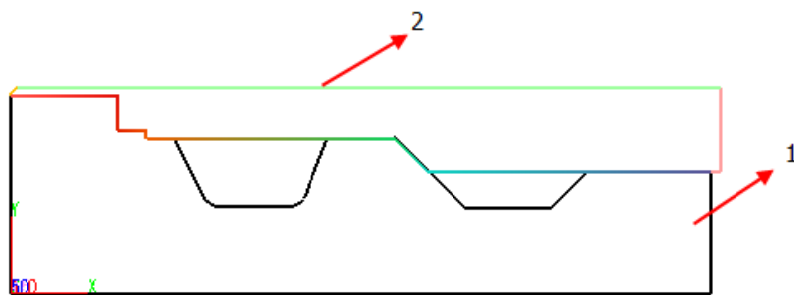
1：Geometry 2：Finish tool path

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 out grooves on the inside and outside of the part. 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 grove 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.

Add the feature to it.

Parameters Form of Turn Groove

This Form includes 6 Tabs. These are as follows:

    Cutting Parameters

    More cutting parameters

    Leads and link parameters

    Limiting Parameters

    Speeds parameters

    Display parameters

Now we will discuss the first 4 Tabs

Select the **cutting parameters Tab** 。

Cut direction

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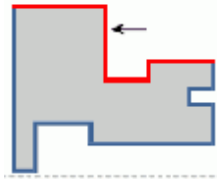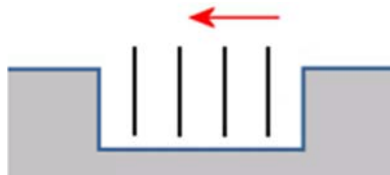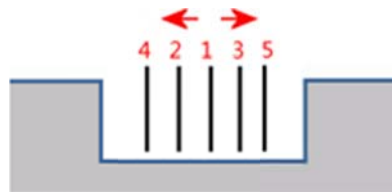

OD

ID
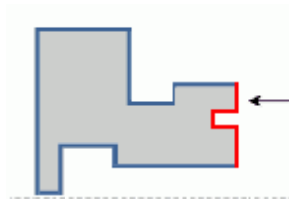
End slot

Cut strategy

Specify the cut strategy. This includes Zig and Alternative 2 kinds of strategies as shown below.

**Zig:** The tool cuts the stock along the direction as shown in the image below.

Alternative: The tool start cutting from the middle of the groove and then alternatively remove the material from both sides as shown in the image below.

Safe Distance

It is a distance between the top face and traversal plane at which tool moves from one groove location to another location as shown in the image below.
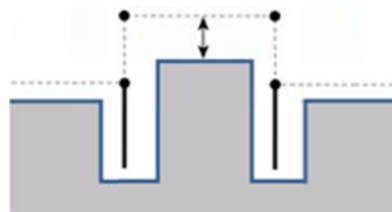
Stock height
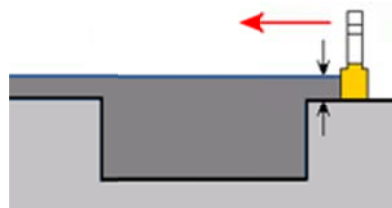
It is used to define the amount of material that must be removed. If this value is "0" then system will create the tool path to the top of the feature being machined. If there is additional material above the profile of the feature then you must use the stock feature to avoid a collision between the part and the material.
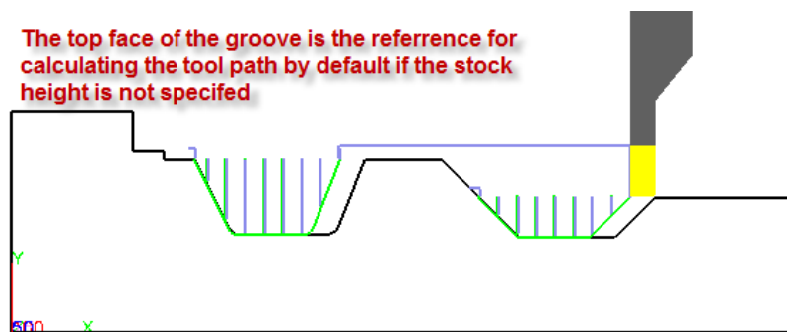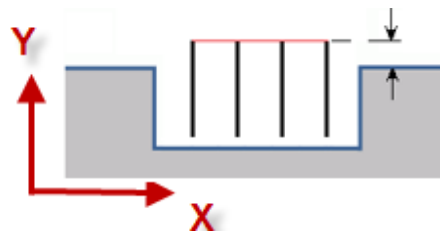

No stock height
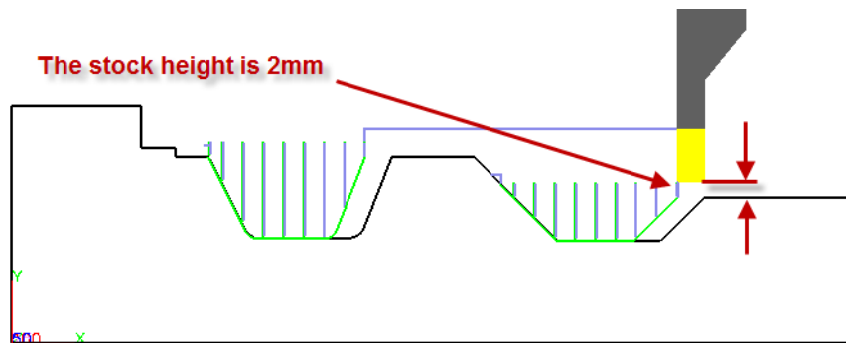
Stock height is specified

Rough clear

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 grove at a time. Otherwise it will be machined twice.

Step Type and Step Size

There are two kinds of options are available for Step type. These are "%" of tool and "width". 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.

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.

Rough Thick

52
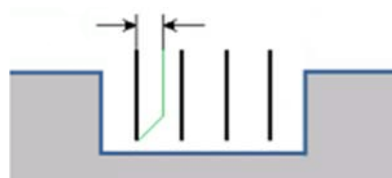
It refers to the thickness of remaining material after Rough Groove operation as shown in the image below.
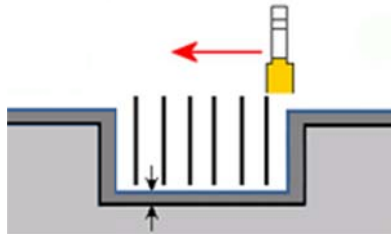
Dwell time

This is the time that the tool will pause at the bottom of the grove when finishing. The default unit is seconds. This feature is not displayed during simulation.

More cutting parameters Tab。

In this Tab we can select whether or not to enable Peck motion as the cutting mode and whether or not to enable a Finish groove after rough groove.

The three parameters for the Pecking options are as follows:

Peck parameters:

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

Peck step: Specify the cut depth for each cut but does not includes the last cut as shown in the image below.
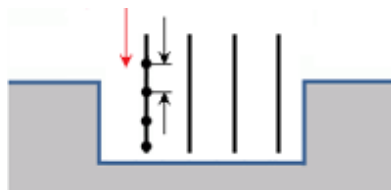
Peck return amount: The distance for retracting each time.
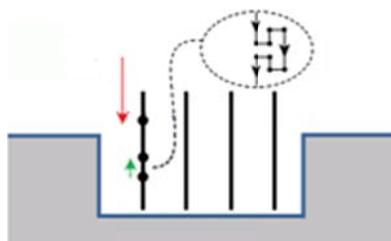
The following are the parameters for setting the finish groove.

Finish Groove: Yes or No. Select "Yes" to enable Finish Groove.

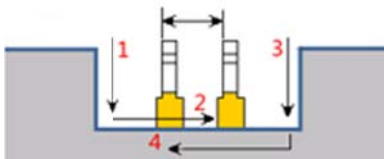**Finish Thick:** The material left over in both X and Z directions as shown in the image below**.**
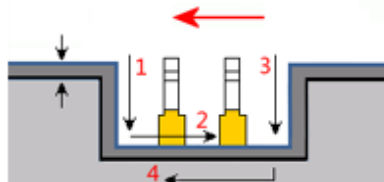
Return position

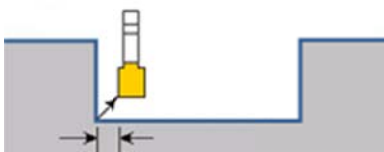It refers to the distance in Y axis between the start point and top face of the groove.

Overlap

It specifies the overlap distance between two contiguous 2 cuts as shown in the image below.

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 does not apply.

Leads and link parameters 。

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

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.

Set up **Cutting Parameters** as following

Rough clear= 0

Safe distance = 1

Rough Thick = 0.5

Dwell time= 1
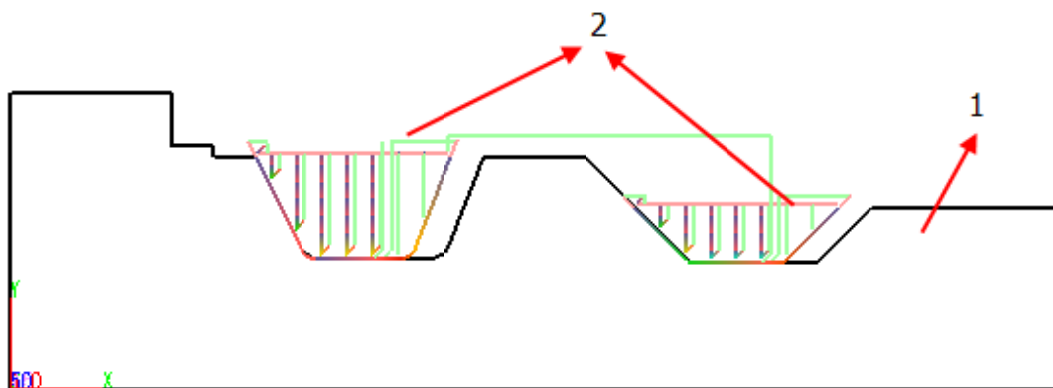
Set up More cutting Parameters as following

Enable Peck = Yes

Return Position= 12

Overlap= 1

Use the default parameters

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

1：Geometry 2：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 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 and then the 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 Threading Triangle 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 required parameters and the other is in the Additional Parameters. First let's set the required parameters.
Cut Position=OD
Position = Select a point from the geometry. Select a point in the red line at random location as shown in the image below.
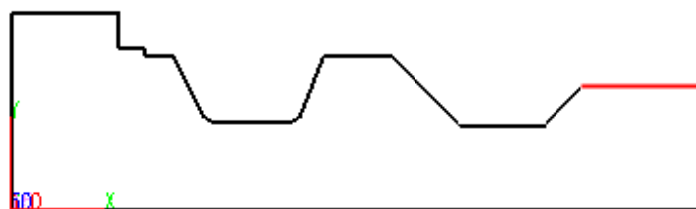
Start length = 0
Thread len= 15
Cut depth = 0.3
Finish depth = 0.2
Numbers of Finish = 2

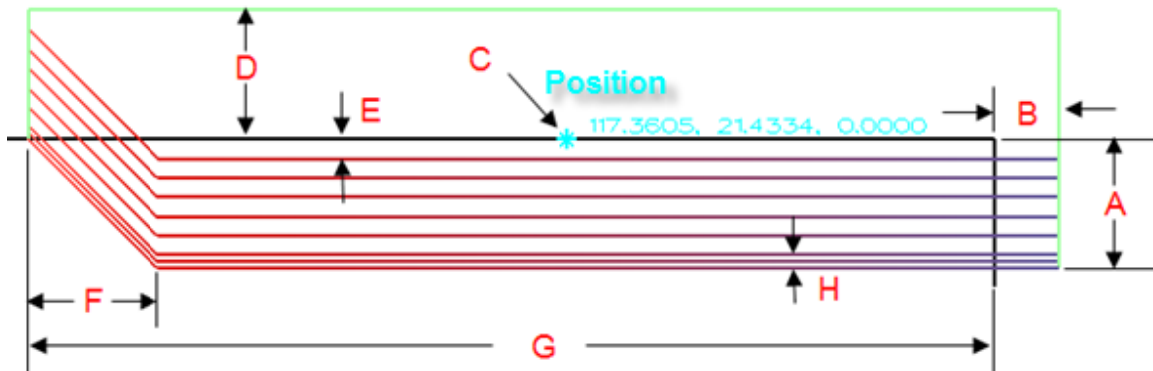Then go to set the **Additional parameters:**

Clearance=2

Pullout len= 1

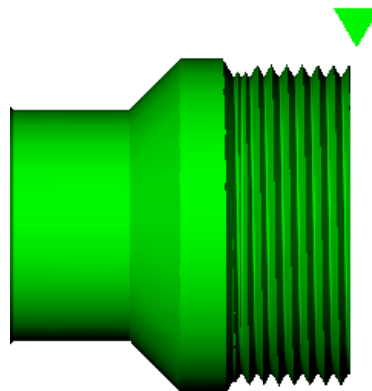Set up **Leads and link parameters**

Extension distance (Engage) =1

Other parameters used the default value

Calculate the tool path and the results should be as shown in the image below.

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

Now simulate the operation with Solid Verify. The results are shown in the image below.

Parameters Form

Cutting Parameters

Leads and link parameters

Speeds parameters

Display parameters

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

Cutting parameters 。

Cut Position

Specifies OD cut or ID cut for Threading.

Note: Different position require different styles of tool to produce a proper thread.

Pitch

Specify the pitch (the distance between each thread) for threading as shown in the image below.
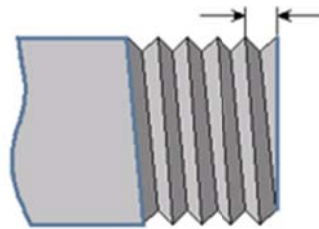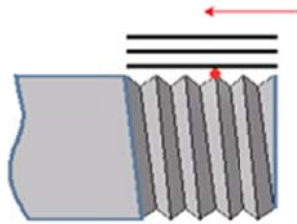
Thread depth

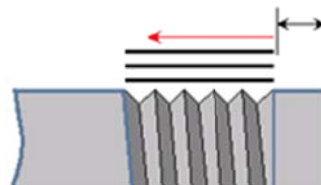Specify the depth of threading as shown in the image below.
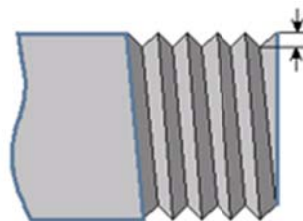
Position

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

Start length

It is an axial distance between the start point of the thread and the position point as is shown in the image below:

Thread len:

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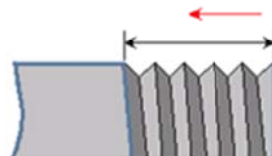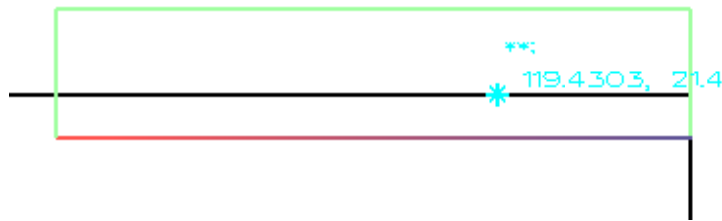

Thread Type

It used to specify the type of cycle: G92 or G76.

G76（Compound cycle） this is the most commonly used threading cycle. G76 can create roughing and finishing threading cycles.
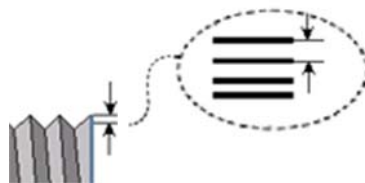
G92（**Simple cycle**）- It is a simple cycle that consists of 4 actions: "Engaging-Threading-Retract-Return" for each cutting level.


First pass depth

This is available when the G76 is selected**.** It specifies the depth for the first cut.

This option will not appear in the Solid Simulation.

Cut Type

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

If the Cut type set to "% of remain", the default value is 40. It means that the actual step over is 40% of remaining depth as shown in the image below.
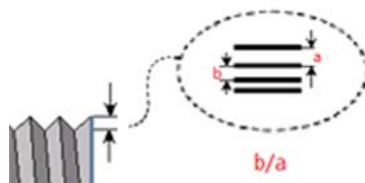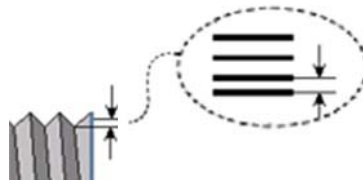

If the cut type is set to "constant", then the default value of the option of **Cut depth** is 0.2 as shown in the image below.
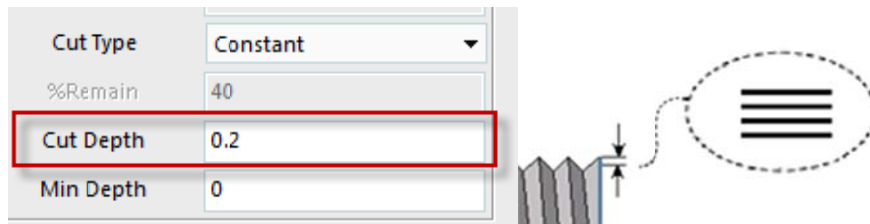
Min Depth

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

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.

Number of Finish

This refers to the number of cutting passes in the tool path. It should be larger than zero.

Numbers of spring

The default value is 1. For the multiple thread passes input the corresponding value in the field. For example two means two additional passes. This is used to get the flex out of the tool to ensure the part is cut to the programmed depth.
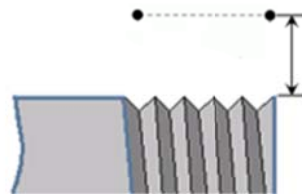
Hand of Thread

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

Left hand   Right hand
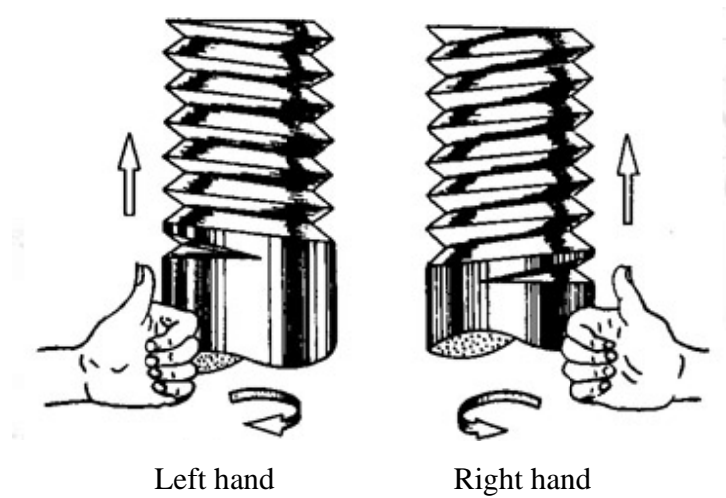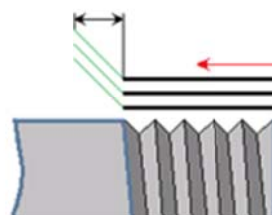
Clearance

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

Pullout Len

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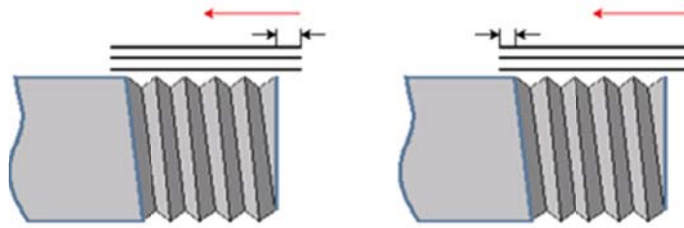
Note the Pullout length is part of the length of thread.

Leads and link parameters 。

Extension distance:

It specifies the extension distance for engaging and retracting the tool as shown in the images below.

Extension Distance（Engage）　　**Extension Distance（Retract）**

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 then 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**" from the **Subtype** list. Then name the tool **"Part Off".**

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.

Cutting parameters

Leads and link parameters

Speeds parameters

Display parameters

We have gone over the last three tabs before so we will only go over the **Cutting Parameters** 。

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.
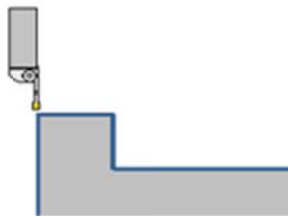
Return height

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.
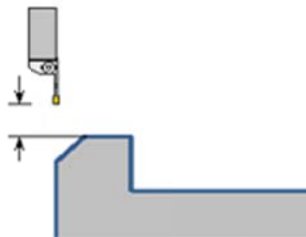
Corner Geometry

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

Non-It means the tool will cut off the part always in the direction perpendicular to the axis of the part.

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

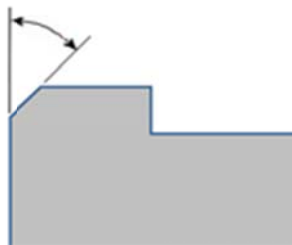There are additional options to control the shape of the chamfer.

Chamfer Width

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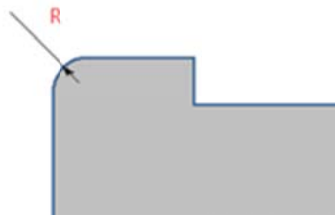

Chamfer Angle

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

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
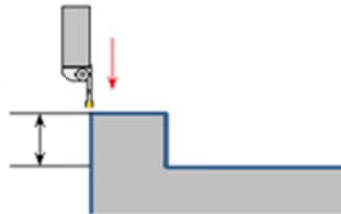

Peck parameters

Enable Peck

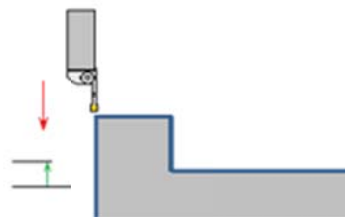Determine whether or not to use the peck mode to cut.

Peck step

This will specify the depth for each cutting pass as shown in the image below.
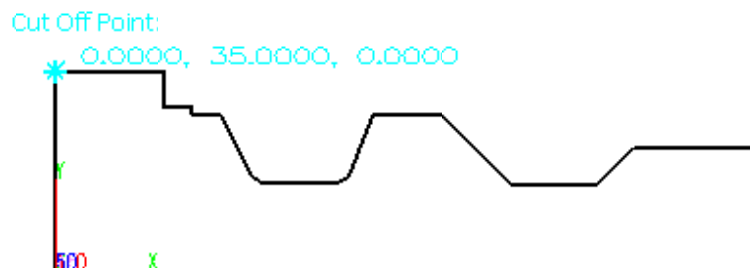
Peck Return Amount

This refers to the retract distance after each pass.

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

Set the parameters as following:

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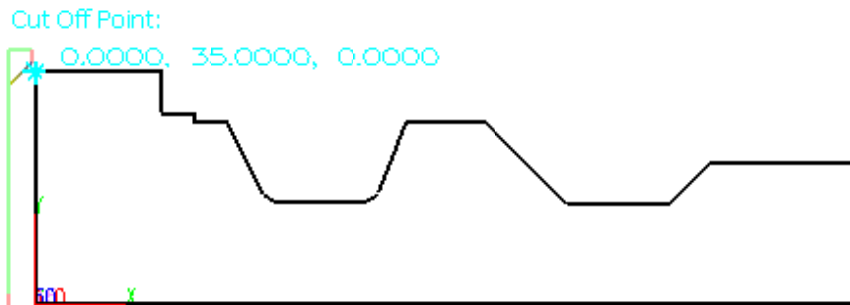

Corner Geometry: Chamfer

Chamfer width: 3

Set the Leads and links parameters

Extension distance (Retract): 0

Leave the other parameters in their default values.

Click the calculation button and the tool path will appear as in the image below.
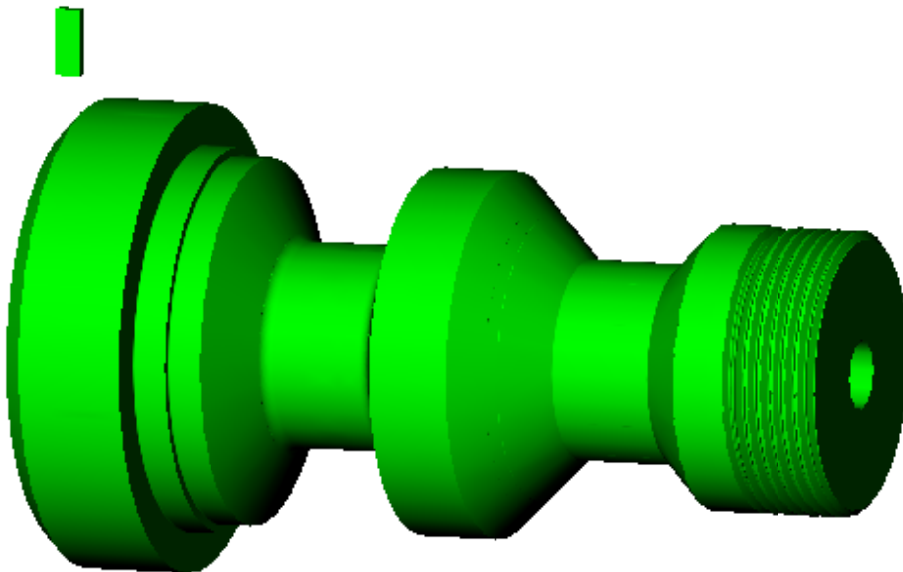
Cut Off Point:
0.0000, 35.0000, 0.0000

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.