

ZW3D CAM Training Guide

Copyright and Trademarks

©Copyright 2014ZWCAD Software Co., Ltd. All rights reserved.
Floor 4, NO.886, Tianhe North Road, Guangzhou 510635 P.R.China
(8620)38289780

ZW3D™ V2014CAM Training Guide

This documentation may be reproduced provided it complies with the terms presented on the LICENSE AGREEMENT supplied.

ZWCAD Software Co., Ltd and the program authors have no liability to the purchaser or any other entity, with respect to any liability, loss, or damage caused, directly or indirectly by this software and training materials, including but not limited to, any interruptions of service, loss of business, anticipatory profits, or consequential damages resulting from the use of or operation of this software.

Updates may be made to this documentation and incorporated into later editions.

ZW3D™ is a registering trademark of ZWCAD Software Co., Ltd.

The ZW3D™ logo is a registering trademark of ZWCAD Software Co., Ltd.

ZWCAD™, ZWSOFT™, the ZWCAD™ logo, and the ZWSOFT™ logo are all trademarks of ZWCAD Software Co., Ltd.

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 Folder2014\doc**" directory on the ZW3D program directory and open the file "**ZW3D.chm**".

This **CAM 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 FolderPDF**" folder in the program directory. Refer to the **HELPMannual** 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 **CAM Training Guide** is only a portion of the complete documentation, it is highly recommended that you utilize the **HELPMannual**. 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 HELPMannual 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 HELPMannual.
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!
Team ZW3D

Contents

Chapter 1 Introduction	4
Chapter 2 Basic Walk Through	6
Chapter 3 Creating Stock	18
Chapter 4 Getting Ready to Mill	32
Chapter 5 Rough Milling Basic Walkthrough.....	41
Chapter 6 Rough Offset 2D Cutting Parameters.....	49
Chapter 7 More Roughing Capabilities	66
Chapter 8 Finishing HSM Lace Milling	81
Chapter 9 Finishing Leads and Links.....	96
Chapter 10 More Finishing Options.....	116
Chapter 11 Special Finishing Methods.....	129
Appendix A 3 Axis Machining	141
3Axis CAM First Tool Path.....	142
2-1/2 Axis Milling	152
CAM 2-1/2 Axis Pocketing	158
3 Axis Milling	175
3 Axis Milling Exercise.....	183
Appendix B Operations Form Finishing Parameters	196
Appendix C Operations Type - Icon Maps	206

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 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 ZW3DQuickMill are listed below.

- ZW3DQuickMill is based on an internally calculated STL triangulated representation of your part, therefore the geometry must be based on surface or solid data. Wireframe geometry is only used to limit the toolpath to specific regions.
- ZW3DQuickMill produces gouge free accurate toolpaths based on user specified tolerances.
- ZW3DQuickMill parts do not have to be “Solids”. The input data could consist of multiple surfaces with small gaps, overlaps and inconsistent surface normal directions.
- ZW3DQuickMill is fast, toolpaths are calculated in stages. Each stage is saved in a special area called “cache”. The advantage to the user is that if a toolpath is recalculated only the parameters that are changed will be actually recalculated and not the entire toolpath. ZW3DQuickMill will look in the cache area for the latest existing toolpath data and reuse it wherever possible, so that the same calculations are never unnecessarily performed twice.
- ZW3DQuickMill supports a variety of modern milling techniques such as:
 - High-speed milling.
 - Rest machining based on previous cutter shape.
 - Sequential rest machining based on previous operations.
 - Adaptive feed rate control based on volume of material removed and current cutting condition.
 - A wide variety of lead in/out options combined with advanced linking between toolpath regions.
 - Advanced limiting based on clipping and containment, allowing the user to choose shallow (flatland) milling, or steep regions only, etc.

Chapter 2 Basic Walk Through

STEP 1 - Open a New File

In this step we will open a new file and create a new CAM Process Plan.

1. If you are currently in an active part, pick  to return to the object level.
2. Pick **FileSave**  or **Save All** to save any files you have worked on in the active session.
3. Pick **File>New** or pick the **Create a new file** icon  from the Quick access Toolbar or press (Ctrl+N) to display the **New File/Object Form**.
4. We can use the **New File/Object Form** to create a new ZW3D file and our CAM process plan at the same time. Select the **CAM Plan** icon  , enter **My_CamPlan.Z3** as the file name and , if you like, **Description** 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_CamPlan.Z3]" .

STEP 2- The CAM Plan Manager

You will notice on the left side of the display the **CAM Plan Manager**.  All CAM activity is performed using this manager.

Items such as **Geometry, Clearances, Frames, Tactics, Operations, Machine and Output** should be displayed.

The CAM Plan Manager

A default setup is already created for you containing all of the elements needed to get started creating tool paths. Functions are performed by either selecting an element directly (e.g., Geometry, Machine, etc.) or by right-clicking on an element and selecting from the Object Editor popup menu. Elements that require further definition are indicated as (undefined). Definitions should flow from top to bottom.

You can toggle the [CAM Plan Manager](#) on and off using the **Manager** button on the [Status Bar](#).

STEP 3 - Inserting the CAM Geometry

Let's begin by inserting the CAM geometry that we'll use for this 'walk through'. It's an assembly consisting of a part, clamps and stock. Each is component within the

assembly and each have the appropriate CAM class attributes assigned to them. Refer to the **Part Attributes Form** for more about assigning CAM class attributes at the part level.

1. Under **Setup1** in the CAM Plan Manager, Double Left-click on **Geometry**. This will display the **Shape Browser**.

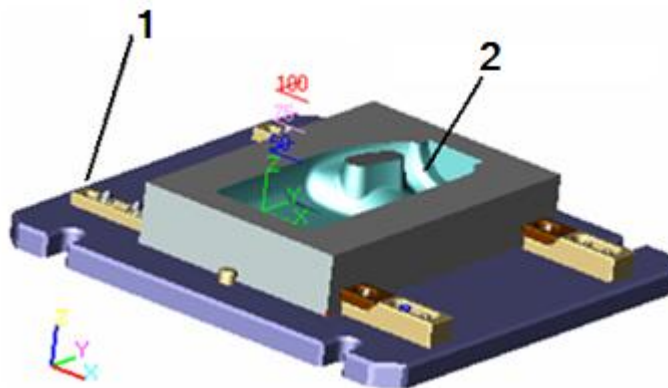
Notes:

Items in the CAM Plan Manager tree are context sensitive.

Right-click to access menu. Left-click to select.

Double Left-click on Geometry will automatically execute the first command(Insert) in the menu.

2. In the **Paths** list of the Shape Browser, locate the "**training**" sub-directory of your ZW3Dprogramme directory. If the directory is not listed, select the folder icon  below the list and use the File Browser to locate and select it. It will then show up in the **Paths** list for selection. (Lower case letters at the bottom of the list)
3. Now the **Files** list shows all of the ZW3D files located in the "**training**" directory. Select the file "**millbottle_Cam.Z3**."
4. From the **Shapes** list, select "**camassy**".
5. Pick "**Yes**" to insert the assembly. It should look like the figure below. Notice that the part, stock and clamp components were added to the **Geometry** tree in the CAM Plan Manager.



CAM geometry
1:Clamps(Includes Table) 2:Part (Stock not Shown)

You can double click on each component's icon to toggle its blanked state.

STEP 4 - Clearances and Frames

You will notice in the CAM Plan Manager that Clearances and Frames are listed under Setup1. These are already assigned default values and we will not change them here. A set of default clearance planes were automatically created when the CAM assembly was inserted in STEP 3. Refer to the geometry figure in that step.

Setup1 will automatically use the world XY datum plane as the default Frame. We can use these clearances and frames for now. You can customize them in future CAM plans. Let's continue.

STEP 5 - Creating an Operation

As you can see, the CAM Plan Manager makes it quite easy to create the elements necessary to create a setup. Our geometry, machine, clearances and frame are all specified. Now let's create an operation.

1. Under Setup1, doubleLeft-click on Operations.
This will display the CAM Operation Type Form. All ZW3D CAM milling operations from each tool path group are listed on this form. Optionally, you can select the operation from the toolbar.
2. An operation which can be applied to whole CAM Components, without any other features present, is "Lace."  It's in the Finishing group on the Quick tab.
3. Select it and the form will close automatically.
4. Now notice that a new operation called **Lace1** was created with default parameters and added to the **Operations** tree. Parameters that are initially undefined for a new operation are identified with "(undefined)".

STEP 6 - Specifying a Tool

You will notice that the **Tool** for the Operation **Lace 1** is currently (undefined). Let's specify a tool to use for this operation.

1. Under **Lace 1**, doubleleft-click on **Tool**. This will display the **CAM Tool Manager Form**. It allows you to create, manage and select tools for operations.

Notice that by default, ZW3D creates an end mill tool. Again, you don't have to name your tools since ZW3D CAM will assign names automatically but it helps to recognize them when you have quite a few. So let's name our first tool.

2. Type "**my first tool**" (without the quote marks) in the **Name** field.

3. **Tool properties** are listed in the top center of the form. It contains the default properties for our end mill tool.
4. These properties are fine for our demonstration.
5. The **Shape** section is located at the lower left corner. These properties will be assigned to the active tool (i.e., "my first tool"). As you can see, **10mm Ball End Mill is the default tool.**

Again, these properties are fine for our demonstration.

6. Now pick **OK** from the **CAM Tool Manager** to create our end mill tool.

You will notice in the CAM Plan Manager that our end mill tool has been assigned to the **Lace 1** operation.(See **Tool** option in Lace 1 Operation)

STEP 7 - Tool Path Operation Parameters

It's time to set the parameters for our **Lace 1** operation.

1. Under **Lace 1**, select **Parameters**. This will display the **Lace Operation Definition Form**.

This form contains default parameters that we can use for our tool path. In the future, you will use these operation definition forms (referred to as Opdefs) to customize the cutting parameters for your operations. Each operation type has an Opdef form similar to this one.

2. For our tool path the default parameters are fine. Pick **OK** to close the Opdef form.

STEP 8 - Assigning the Features to Cut

Now everything is set for our **Lace 1** operation except for defining what part features to cut. You're doing excellent so far and this is an easy step so let's do it.

1. Under **Lace 1** in the CAM Plan Manager, you will notice that **Features** are currently (undefined). **Right-click or double left click on Feature(undefined) to create.** We will select geometry from the popup form "Select feature for operation".

For this demonstration we want to machine the entire part. In the future you may want to machine individual CAM features such as pockets, steps, etc. You would do so by creating additional CAM features from part geometry. So let's continue.

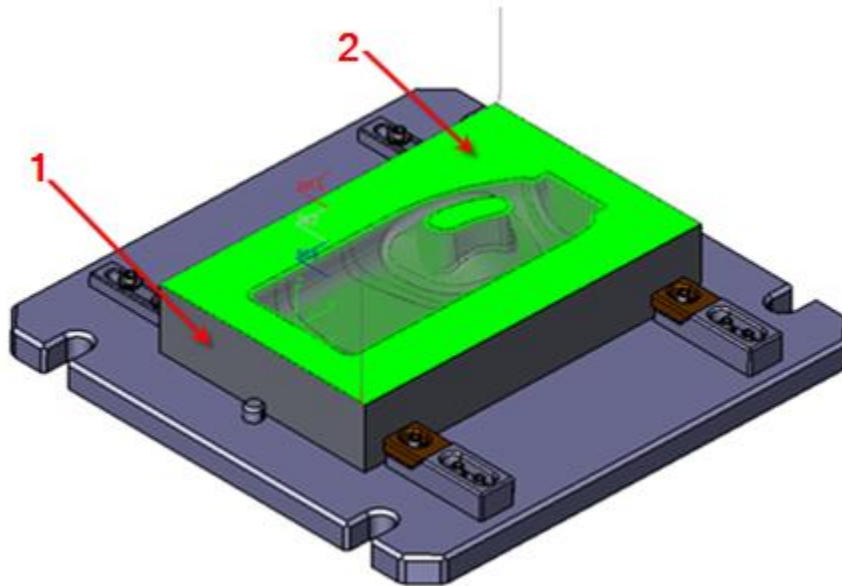
In the form

2. Select **Part :camblank (1) < millbottle_Cam.Z3** and notice that it has now been added under **Features** as well.

STEP 9 - Calculating the Tool Path

Now that we have the **Lace 1** operation completely defined, it's time to calculate the tool path. This again is an easy process.

1. Under **Operations**, right-click on **Lace 1** and select **Calculate**.



1:Part 2:Toolpath

A **Quick Mill Progress** status bar will be displayed in the bottom left corner of the display while the tool path is calculated. When complete, the tool path will be displayed on the CAM assembly in the graphics window as shown above.

STEP 10 - Creating a CAM Machine

Now that the CAM toolpath has been generated, we need to specify the machine parameters used for this Setup. These parameters include: X,Y and Z limiting, Tool Changer, Machine Definition Files, Cutter Compensation, Rotary Axes and Offsets and Post-Processor. Notice that **(undefined)** appears to the right of **Machine** in the manager tree. Let's create a new machine.

1. In the CAM Plan Manager Double-click on **Machine**. The **CAM Machine Manager Form** will be displayed.
2. You will notice that a default machine is already specified. It's a vertical 3-axis machining center with a rotating head. The post processor is ZW3D Post and there is an extensive list of configurations predefined. Pick one that you are familiar with.

If you select **Options**, **Limits**, you will notice that all of the parameters have default values. We don't even need to give the machine a name but let's do so anyway.

3. In the **MachineName** field enter "**my first machine**" (without the quote marks).
4. Now pick **Apply** to create the new machine.
5. Pick **Tool Changer** in the Options part.
6. Click on **Add All** in the **Machine Tool Changer** form.
7. Click OK to finish adding the tools to the changer.
8. Click Yes to save the modifications to the tool changer.
9. Click OK to finish the Machine definition.

*Notice that the name of our machine now appears to the right of **Machine** in the manager tree. (Names and values appear in the manager tree to ease recognition and navigation.)*

STEP 11 - Rapid Verify the Tool Path

Congratulations! You have now created your first tool path and defined the machine that will do the cutting using the new ZW3D CAM interface. As you can see, it's a very simple process that uses the power of the CAM Plan Manager to assemble, define and calculate tool paths. In the future, you can use this same process to create multiple tool paths within a setup using entire parts or CAM features.

But we're not done yet. Let's perform a rapid verification of the tool motion just to make sure it looks ok. This is called a rapid verify because the tool and tool path are displayed in wireframe mode as the tool rapidly moves through its motions. Here we go.

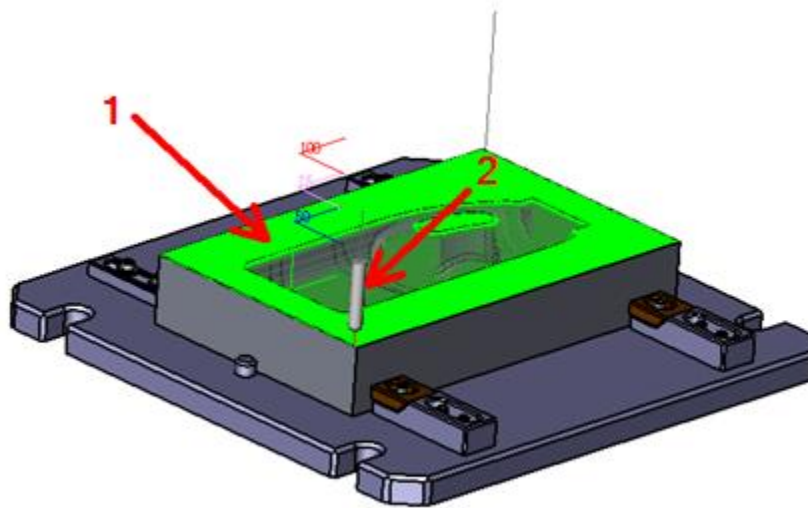
1. Under **Operations**, right-click on **Lace 1** and select **Verify**. This will display the **Toolpath Verify Form**. (shortcut to 'Verify' at the bottom of the **Parameters** form )

This form has controls to move forward and backward through the tool path. Overall tool motion can be verified or specific motions within the path. The coordinates and tool normal are displayed for each tool position in 'World Space' (by default) of **Setup 1**.

2. Select **Options** at the bottom of the Toolpath Verify form.
3. Select **Frame** to select from available frames or local coordinates. In this case we don't have any.

4. Select the ">" button to set the tool in motion. It will move through its approach and retract motions as well as each cut in the tool path.

5. You may not want to wait for the tool to complete its entire path. You can pick || from the Tool Path Verify Form to stop the tool and then pick the close box  in the top-right corner of the form to close it.



1: Toolpath 2: Tool

STEP 12 - Solid Verify the Tool Path

You can also perform a more visual solid verification. The components (part, stock and clamps) in our CAM assembly are solid models (although you can also use open surface models). During a solid verification, the display is changed to shaded mode and the actual material removal is simulated and displayed.

(shortcut to 'Solid Verify' at the bottom of the **Parameters** form )

1. Under **Setup 1**, you will see **Output** at the bottom of the tree. Select it now and the **CAM Output Form** will be displayed.

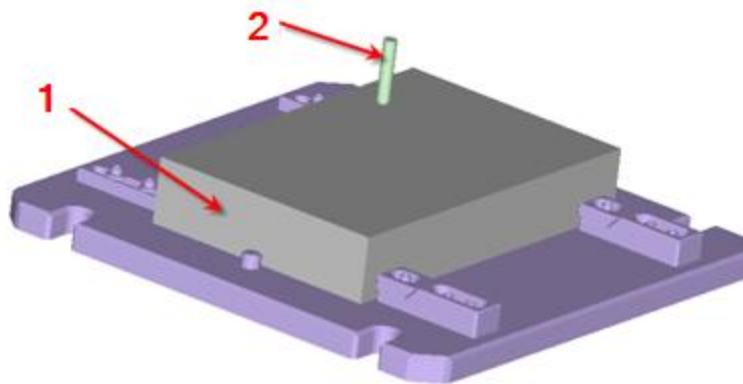
Notice that our **Lace 1** operation is listed. This form contains **Organize, Verify, Document and Viewer** tabs. You can organize your operations using the functions at the bottom of the form (e.g., **Edit, Suppress, Inquire**, etc.). You can verify your tool paths using rapid and solid operations. You can also output documents related to the selected tool paths such as a tooling list, operation list, CL File and NC code File.

Let's perform a solid verification.

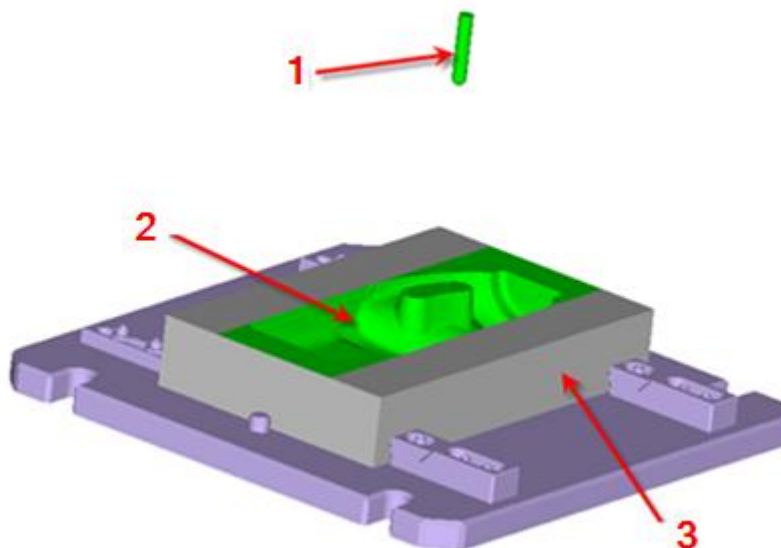
2. Since there is only one operation, notice that the tool path in the graphics window is already highlighted.
3. Now pick the **Verify** tab and the **Start Verify** button.
4. Pick the **Run Simulation** button  on the Solid Verify Session form. The % complete status will be displayed in the text input field. Sit back and watch.

Our CAM Assembly contains a component named **stock** and whose CAM class attribute is defined as **stock**. It is recognized as a stock component and is listed in the **In Setup** list. Solid Verify requires stock to run.

The Solid Verify Session Form contains functions similar to the Rapid Verify Form that we used in the previous step. There is a row of buttons in the middle of the form to quickly change to rapid verify without recalculating the session. There are display and tool movement controls and access to additional options and settings with the **Options** button.



Beginning the Solid Verify Session(1:Stock 2:Tool)



The Solid Verify Session Complete

(1:Retracted Tool 2:Operationbeing verified 3:Stock)

5. When the session is complete, the tool will retract to its initial Z-level. You can now pick the close box in the top-right corner of the Solid Verify Session Form to close it.

STEP 13 - Output CL File and Related Documents

Now that we have verified our **Lace 1** tool path it's time to actually output some documents. The **Documents** of the **CAM Output Form** contains all of the information and options we need to generate a CL File, G and M Codes, tooling list and operation list for our tool path. We're almost done so let's take a look.

1. Right click on Operation under CAM manager tree, it will display a context menu.
2. Select "Output All operations" option from this menu. Then a folder named NC will be created under Output item.
3. Right click on the "Output" under the CAM manager tree, and then select the CL/NC output setting, then the "Output Setting" form will be displayed

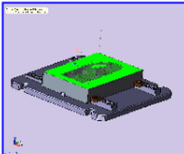
- In the **Part Id** field, enter: **1000** This is output in the NC code and must be a four digit number. Most machine controllers will error with letters.
- In the **Programmer** field, enter your name.
- **Tool Changes** button should already be toggled to **output**.
- **Speeds/Feeds** button should already be toggled to **output**.
- Select the **Tool Num** button and then select **tool id** from the list.
- In the FileName field enter: **my first toolpath**. In the future, you can pick the **File Name** button to select an existing file to overwrite if you wish.
- Now make sure the Display Output box is checked.

4. Right click on the NC folder and then pick the output option: select output CL to display the output from the ZW3DPost post-processor. Different configurations can be created to customize the output for different controllers. Refer to the Machine Programming Details Form for more information.

5. 5. Next choose the Output **NC**. The output will be displayed.
6. 6. Pick the **Tooling List** option from the context menu to output a list of the required tools.

7. 7. Pick the **Operation List** to display and output a detailed listing of all the operations in the setup. This listing's format and its contents are fully customizable.
8. 8. Pick **Output XML** button to show the setup sheet.

All Details		Opdef List		Machine Details		Tool Details	
CAM PLAN OPERATION SHEET SUMMARY							
SEQ #	Name	Type	Tool Name	Total Distance	Total Time	Z Min	Z Max
1	Lace 1	Standard	my first tool	100398.72	401.49	-37.9007	100.0000
TOTAL DISTANCE AND TIME FOR ALL				100398.72	401.49	Click c	

Operation Parameters for Lace 1			
	Milling Parameters		Speed Feed
	Name	Lace 1	Spindle Speed
	Class	Finish	Cutting Feed
	Op Type	Lace	
	Tool	my first tool	
	Process	Setup 1	
	Subtype	Standard	
	Ref Opn		
	Path Tol	0.01	
	XY Surface Thick	0	
Z Surface Thick			
Cutting Mode		Zigzag	

MACHINE DETAILS	
Name	my first m
Class	3-Axis I
Type	Vertic
SubType	Rotating

Tool Changer Details		
Station Id	Tool Name	D Register
1	my first tool	D1

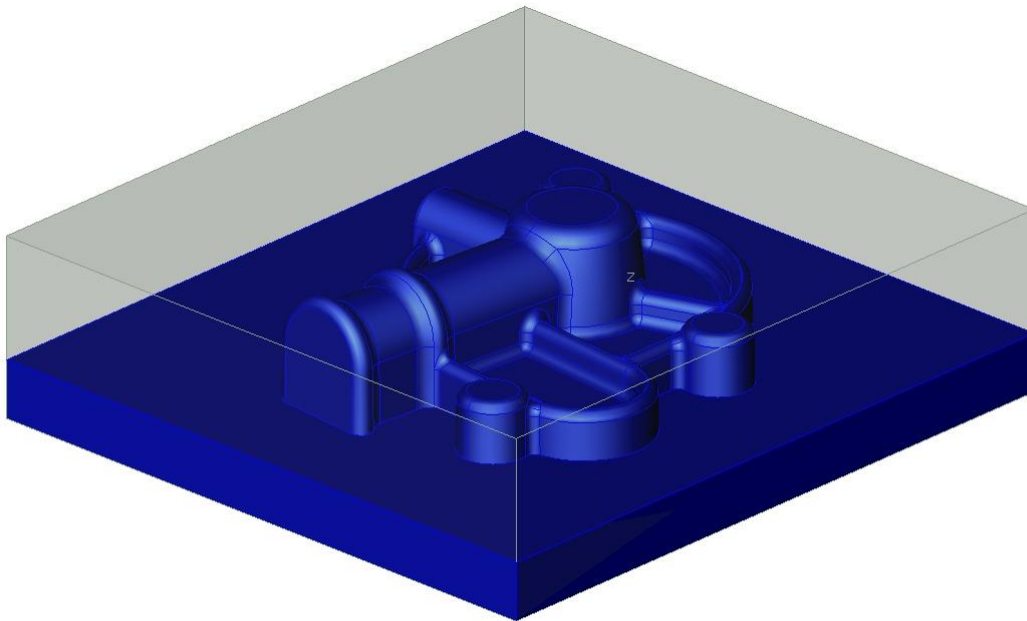
9. Pick **Dismiss** to close the **CAM Output Forms**.
10. Pick **File>Save** or pick the **Save File** icon from the **Quick Access Toolbar** or press (**Ctrl+S**) to save your ZW3D file and CAM plan.
11. The Message Area should say: "**Save**" was successful.
12. If you wish to close ZW3D now pick **File> Exit**.
13. We just produced 5 output files that are now located by default in the **/output** folder of your user directory. All are given the same file name "**my first toolpath**" with the extensions: **.cl**, **.nc**, **.tl**, **.op** and **.xml**. You can use Windows Explorer and go to your user directory now to locate your output files.
14. Congratulations! The **basic walk through** of the CAM interface is now complete.
15. HINT: If you are not sure where your **USER** directory is located, Click on **Utilities > Configuration** and see the bottom of the **General** tab.
16. User folders: C:\Documents and Settings\All Users\ZWSOFT\ZW3D 2014Eng

17. Program folders: C:\Program Files\ZWSOFT\ZW3D 2014Eng

Chapter 3 Creating Stock

In this Chapter we will learn a number of different ways of creating stock around our part.

- Stock is the material around the part prior to machining.
- Stock is used in all Quick Milling roughing operations.
- Stock is required for Solid Verify.
- The stock can be any shape from a simple rectangular billet to a more complex casting or forging. In our first example we will create extruded stock based on the outside shape of our part.



The image above shows the part surrounded by the stock. This example is based on extruding the outside envelope.

NOTE: This exercise is done in mm units. If not already set, you will want to set the default units in your config file to mm and possibly set the default part template to 'PartTemplate(MM)'.

Exercise 1: Creating Stock

Locate and open the file “Cam_Training_Housing.Z3”. This file will be found in the training directory of the program directory.

You can pick existing objects to open by clicking on the object name on the right side of the menu. If there is only one it will be opened by default.

This is the part we are going to create stock for.

Exit part. 

Click out , Create a new Part and name it as “Housing_Extruded_Stock”.

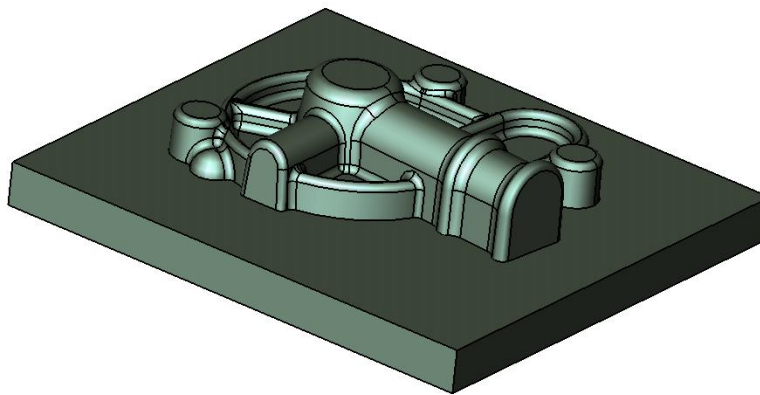
Insert the original part geometry.

<Right-click> and select “**Insert Component**”

Select the part “**Housing Geometry**”.

Click the “**Anchor component**” option.

Locate the component at **0,0,0**.

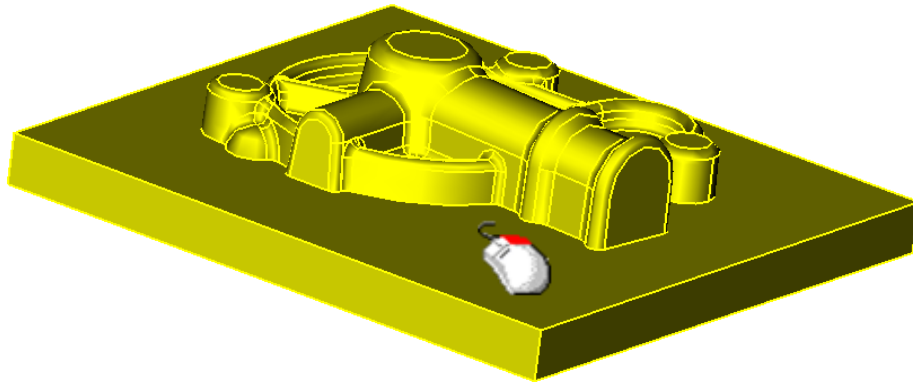


Currently the housing part is a component. We need to merge the component so that the actual geometry of the part becomes available.

Make sure the **pick filter** is set to:**All**.

Then place the mouse pointer over the part until it becomes highlighted, **<right-click>** then select the **Merge** option and complete the command.

Alternatively you can pick the merge command from the “Shape Tab” . 



The part geometry is now available to use and is associated with the original Housing part. We can now use this geometry to create the extruded stock.

Pick the “**Extrude**” icon  from the shape Tab. Select the base option then **<Right-click>** and select the **Insert Curve List** option.

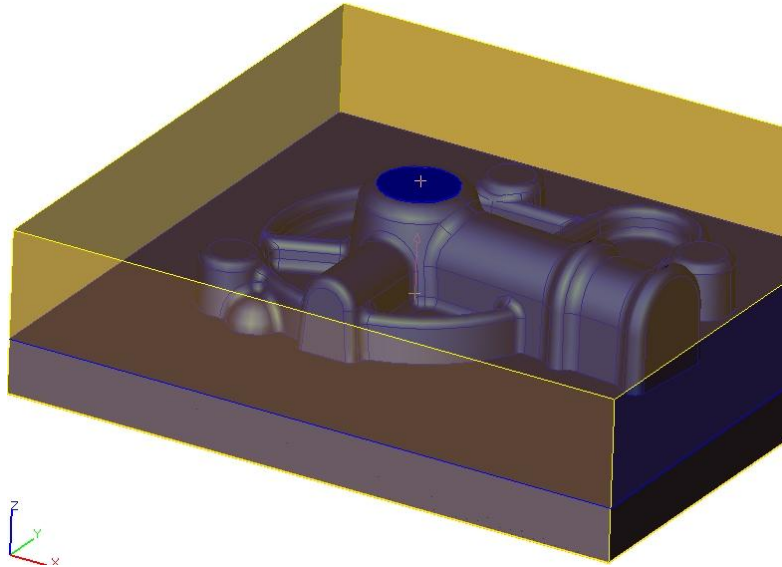
The curve list is created by picking the bottom four edges at the rectangular base of the part.

Complete the command.

The start value is **0**. (Middle-click for the default value of zero)

The end value is obtained by **<Right-clicking>** the “**To Point**” option and picking the top face of the part as shown below. Pick OK to complete the command.

If you have picked the top face successfully the end value of the extruded base will be **75**.



The outside width and length of the part is exactly the same as the original part geometry, if you wish to make the X and Y values of the stock larger, use the Offset option in the extrude command.

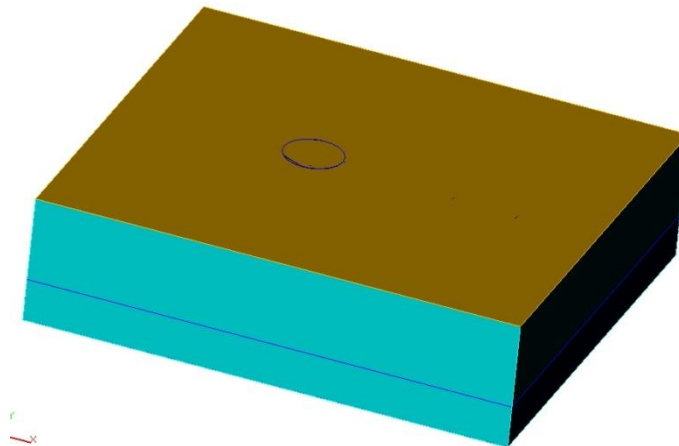
The Z value or height of the part is also the same as the original geometry, 75mm. Let's assume we wish to increase the height of the part by 3mm. We should not simply change the end value of the extrude command to 78mm. The end value of the extruded base was achieved by picking the top face of the part; this ensures that if the height of the part changes, the height of the stock will also change when the stock part is regenerated. Therefore simply inputting an end value of 78mm will mean that the link between the height of the part and the height of the stock will be lost.

To keep the stock height the same as the part plus 5mm, pick **"Face Offset"** from the shape Tab. 

Pick the top face of the stock as shown below and offset this face 5mm.

NOTE: Use Inquire > Curve > Length to check the size of the stock.





This keeps the link between the part height and the stock height.

Next we will change the attribute of the stock.

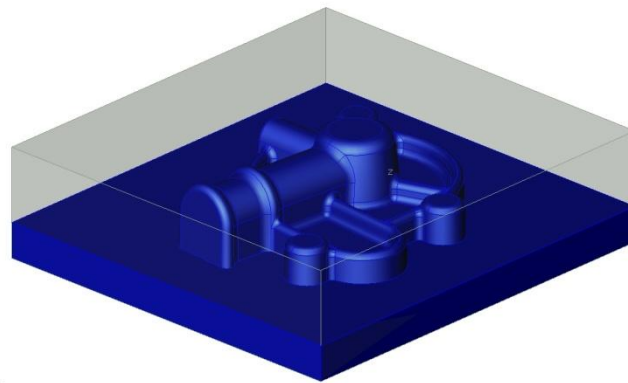
Select the **Attributes** pull menu and pick the **EDIT** option.

Make sure the **pick filter** is set to shape, pick the stock shape and Middle-click.

From the **Face Attribute** form choose a face color.

Also from the attribute form set the transparency to a value of about 80.

The stock shape should now look similar to the stock shape below.



We have to change the attributes once more. This time we will change the “Part” attributes. Eventually we will want to import the different types of geometry into Cam. ZW3D Cam can recognize the difference between part geometry and stock geometry and also other types of geometry such as clamps. This is achieved by specifying the relevant “Cam Class” in the Part attribute form.

Select the Attributes pulldown and pick **Part...**

Pick the **Cam Classes** button and from the list select **Stock**.

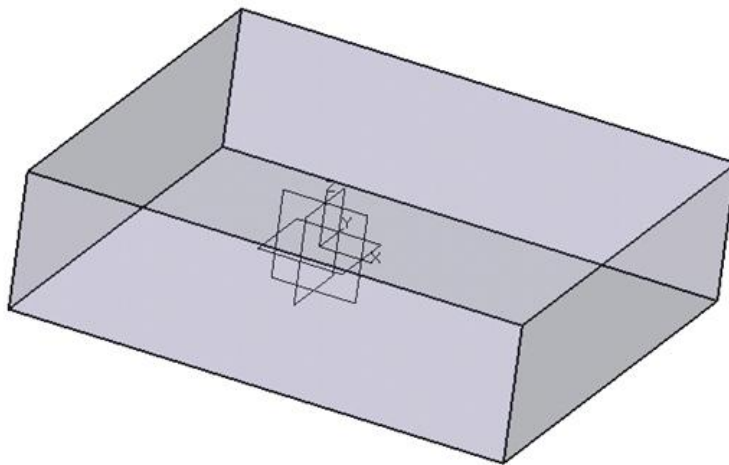
The stock option is then placed into the class field. This will tell ZW3D Cam that this geometry is classed as “Stock” and will be used in rough milling operations as the starting shape to be machined.

You should always specify what type of class the part has. However if you do not specify any class description the default class will always be “**Part**”.

Finally to complete the exercise we will have to erase the Housing shape so that the only geometry present in the part is the stock.

Pick the **erase** icon. 

Click out the “Housing shape” in “History” so that only the stock geometry is left.



The image below shows the stock without the housing geometry present. However, the history list shows that if the part were to be modified, regenerating the history will update the stock.

Save the file.

Exercise 2: Creating the Stock using the “Stock” feature.

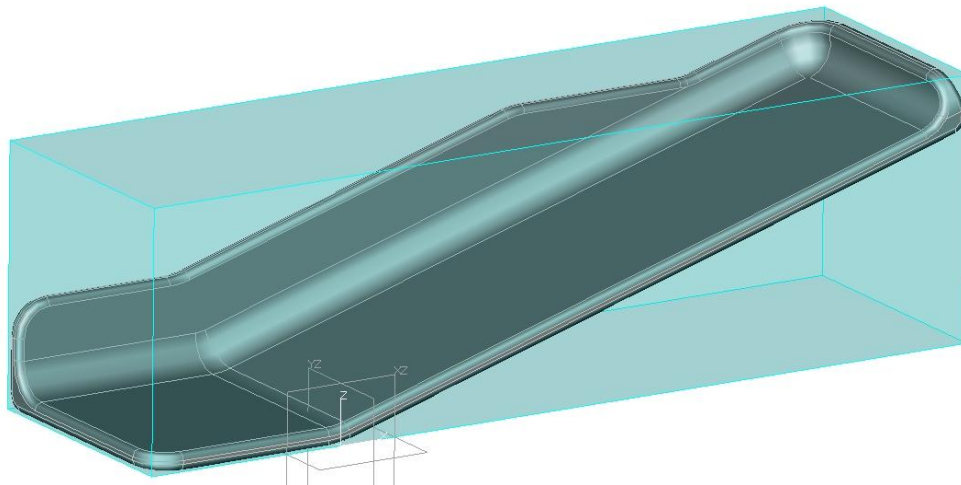
The previous exercise created stock by extruding the outside edges of the part. Using this method any ZW3D modeling commands can be used to create the stock. If

however the stock is rectangular in shape then you can use the “Extruded Stock”  from the Mold and Die Tab.

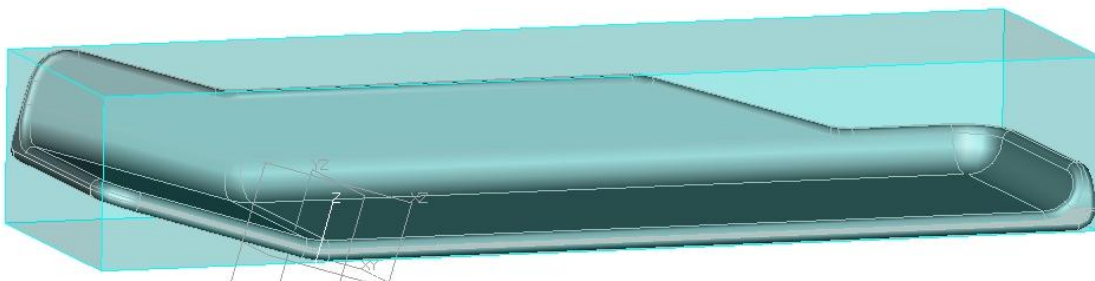
Let’s complete the second exercise by creating a stock feature around a new part. Load the file “**Angled_Part**”.

We will create stock using the stock feature around the example part in two ways.

In the first example the stock will be created parallel to the XY plane as shown below.



Then we will create a smaller size stock where the bottom face of the stock is parallel to the angled bottom face of the part.



Start by creating a new part called “stock” and inserting the “Angled Bracket” part at zero as a component, exactly like we did in the first exercise.

Next, merge the component so that we can use the geometry.

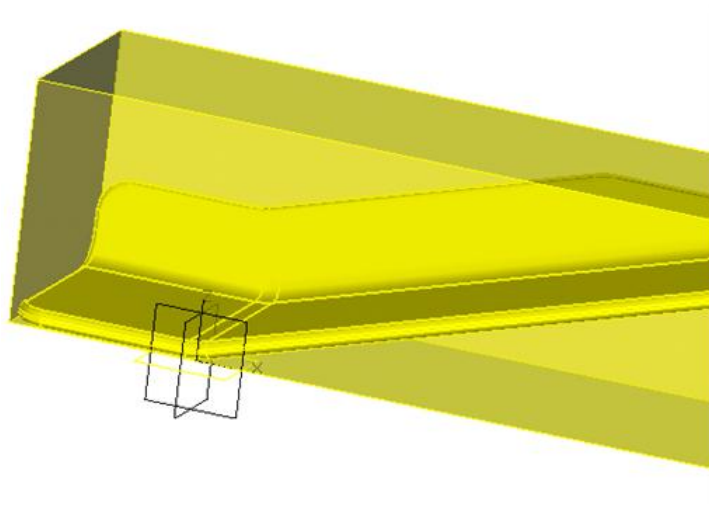
Pick the “Block” feature.

At the prompt “Select Shapes to Enclose”, Pick the part ensuring the **pick filter** is set to “Shape”.

At the prompt “Select Reference Plane”, the XY datum is the default so just Middle-click.

The stock should be previewed as shown below with the base of the stock parallel to the XY plane.

Complete the command.

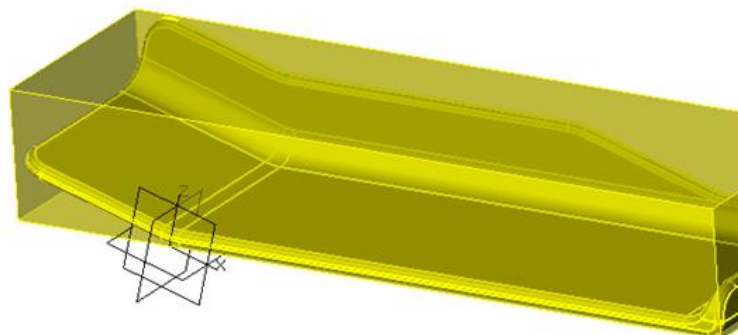


Undo or **redefine last** command and create the stock again but this time parallel to the brackets longer angled bottom face.

At the prompt “Select Shapes to Enclose”, Pick the part as before.

At the prompt “Select Reference Plane”, pick the bottom angled face of the part.

Complete the command.



This configuration requires only about 2/3 the material for the stock.

To complete the “stock part” don’t forget to erase the “Bracket” Geometry and to set the “Part Attributes” to “Stock”.

Let’s take a closer look at the Stock Feature Form.

The minimum size of the stock based on the reference plane or planar face you choose is automatically calculated. The actual sizes of the stock, is shown in the **Length(X)**, **Width(Y)** and **Height(X)** fields. You can overwrite these values by either typing in a new value directly to the relevant field, or by using the increment/decrement buttons.

The “**Side Increment**” option allows the user to specify a standard round up value which can be used to obtain a more nominal size of stock. For example if the side increment was set to 1 the 265.55 X value shown above would be rounded up to 266. Or if the side increment value was set to 10 then the 265.55 value would be rounded up to 270. In other words the stock sizes are allowed to increase in steps by the value entered into the “Side Increment” field.

The numbers listed under “**Model**” are the exact size of the model and are there for reference only.

If your requirement is for a rectangular stock shape then the Stock feature is the preferred choice. The stock will automatically update if the original geometry is changed providing the user remembers to regenerate the “Stock part”.

We can automate this however by using the “Link Manager”.

First **save** the part.

The Link Manager

Now that we have saved the our part let’s take a look at the capabilities of the “Link Manager”

Load the file “Angled Part.Z3” and load the part “Angled Bracket”

If we were to change the size of the part in some way we could update the stock to the new size by going into the stock part and performing a regen. However, we can update the stock while we are still in the “Angled Bracket” part by using the link manager.

From the “**Tools**” **pulldown menu**, pick “Link Manager”.

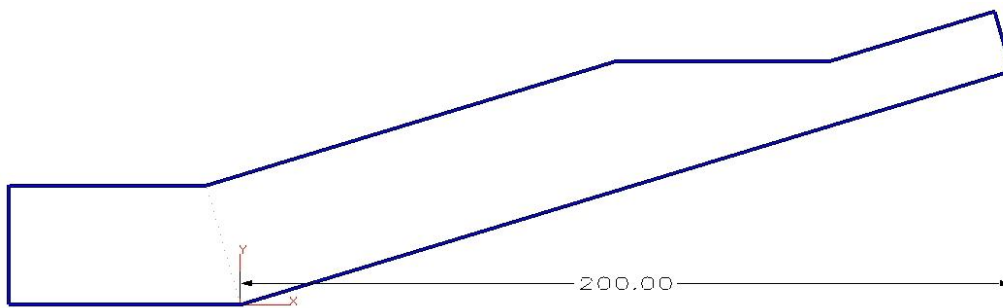
To use the Link Manger first create a new ZW3D Object link. For our example we will use the “stock” part that we created previously.

Pick the “Insert” Button next to the “ZW3D Object links” window.

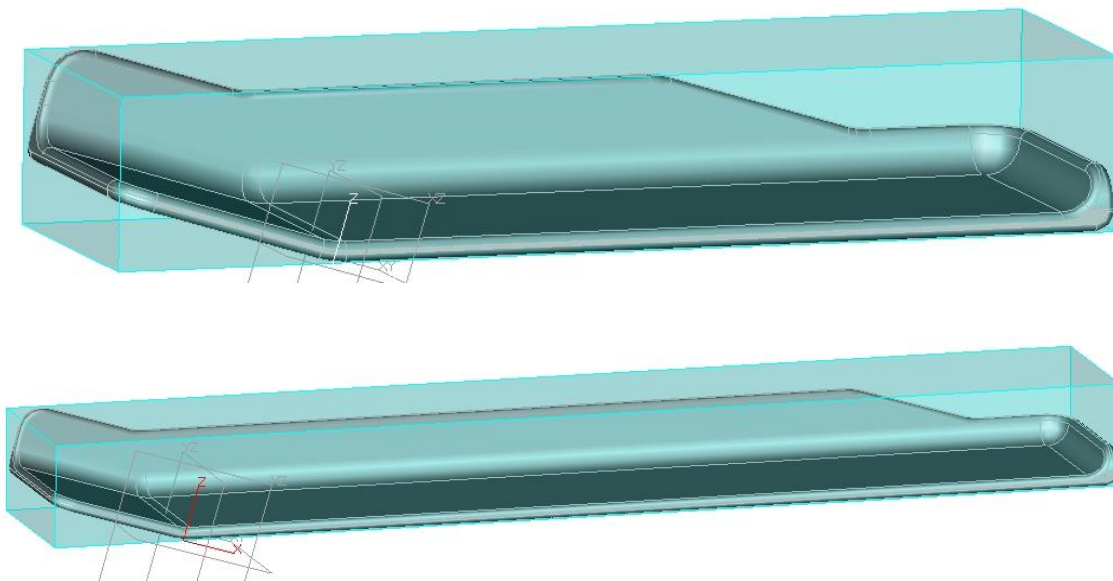
From the “Objects” form, **select the part “Stock”**. This will be entered into the ZW3D object Links in the Link manager form. At this time if we wish we could regen the stock by simply pressing the **Regen button**. Other links could be added to the Link Manager at this stage, such as a drawing sheet for example, this could also be regenerated from the Link Manager. To make this process even more automated select the “**Auto Regen**” tick box. Now every time the Angled bracket is regenerated the Stock part will also automatically regen. If you require more information on the Link manager please refer to the **HELP Manual**.

Let's change the length of our Angled Bracket part.

Edit the 200mm sketch dimension. Change the value to 400mm.



The part history is automatically regenerated because the “Auto regen on edit” option is selected in the ZW3D configuration. The part is regenerated and the stock has also been updated. The images below show the original stock length and stock updated to the new length. Both show the original geometry. Don't forget to always erase the original part geometry from the “Stock” part.



Save the Part.

Creating Stock for the “Mill_Bottle” Blowmold and Model

We will next create two stock parts for the example parts that we will be using with ZW3DQuickMill.

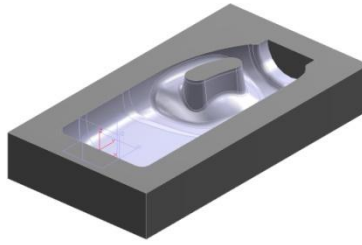
First however let’s review the method of creating Stock.

- **Create a new part**; give it a meaningful name, “Stock” for example.
- Using the **Insert component** command, insert the geometry to be machined.
- **Merge** the component.(Shape tab) 
- **Create the stock** geometry using standard ZW3D modeling techniques, the **extrude** command for example, **or** if the stock is rectangular use the “**Stock Feature**”.
- When creating the stock also create an allowance or plus material on any relevant faces as required. If you require an allowance on the top face only of the stock, then you could use the face offset command.
- Erase the original part geometry from the stock part.
- Edit the Face attributes of the stock to make the faces transparent (optional).
- Edit the Part attributes of the stock part and change the “Cam Class” to “Stock”
- Leave the stock part and go in to the actual Cam part geometry, create a link to the Stock part using the “Link Manager” so that with any changes to the cam geometry a regen of the cam part will also trigger a regen of the Stock part.

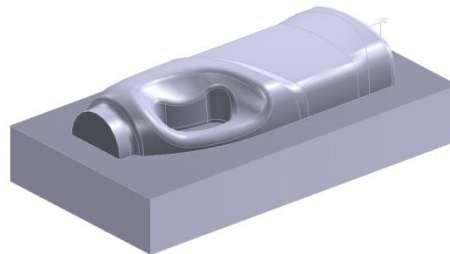
Clear the Session and load the part “Mill_Bottle.Z3”.



original bottle design geometry



CAM geometry for a blow mold



CAM geometry to produce a model of the bottle

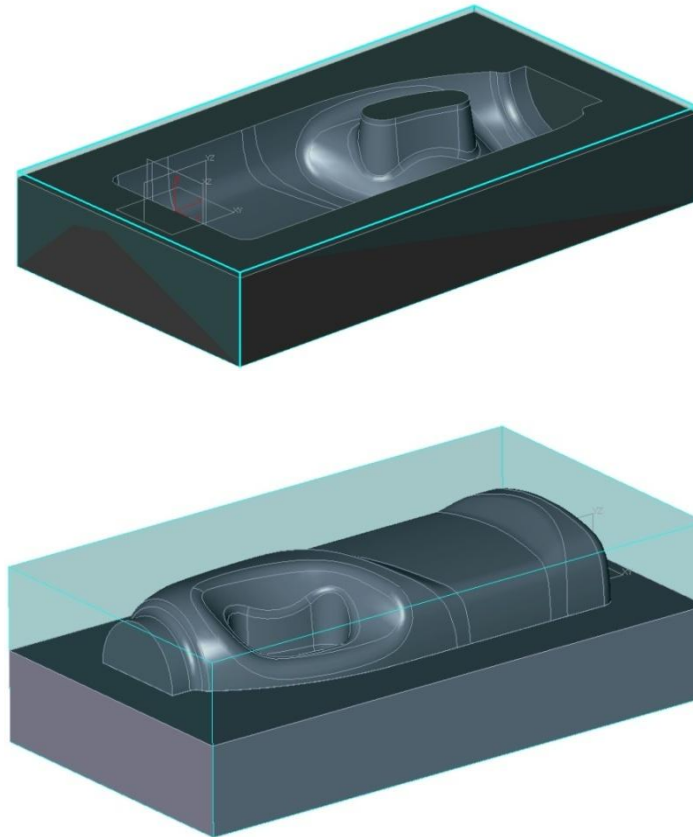
We will use the Blow mold part for cavity type milling and the Model part for core type milling.

In the Mill_Bottle object file create two stock parts for the “blowmold” cam geometry and the “model” cam geometry.

Call the blow mold stock “**Stock_Blowmold**” and the model stock “**Stock_Model**”.

- Create both stock parts using the “Extruded Stock”  (mold and die tab)
- Make the width and the length of the stock parts the same size as the cam geometry.
- Add a 3mm offset to the top face of both stock parts.
- Use the **Link Manager**(**Tools** pull down menu) to create the relevant links required to ensure the stock parts will be automatically regenerated if the **Bottle** part gets changed. **Hint**, you will need four links in the bottle part and make sure you insert the links in the correct order in which you want the relevant parts to regenerate.

The stock of both parts is shown in the figures below, please note that the cam geometry has been deliberately left in the images, you **must** erase the cam geometry from your actual stock parts.



Save your File.

There is a third and simpler way of creating stock.

Exercise 3: Creating Stock from the CAM Plan

ZW3D has shortcuts to creating a CAM Plan and Stock for your model.

Open the File “CAM_Training_Housing.Z3” and pick the Housing Geometry part from the preview

Right-click in the background area and select “**CAM Plan**” near the bottom of the menu.

This will open a new object called “Housing Geometry_Cam” and automatically add the part as a geometry feature to the CAM.

Select the “**Stock**” command from the “Setup” tab and select OK.

Stock is created and added to the CAM Plan. You will be asked if you want to hide it. That is your choice. If you don't, you will notice that the stock has a transparent attribute already added so you can see through it and has the correct CAM Class, Part Attribute.

Chapter 4 Getting Ready to Mill

In this chapter we will cover,

- Cam Plans
- Cam Templates
- Loading the geometry into the cam plan
- Setting up machines
- Setting up clearance planes
- Creating and loading Cam Assemblies.

CAM Plans or Cam process plans are used to create and manage NC toolpaths. CAM Plans are created from the same form as you would use to create a part/assembly or

drawing and so on. The CAM Plan form is available in the **Create New File** form  . Creating a CAM Plan allows the user to gain access to the Cam User interface.

Cam Templates

Object Template files can be loaded when a new part/assembly, drawing or CAM Plan is created. By default, the template file name is "Templates.Z3". A default Templates.Z3 file is located in the "\resource" directory of your program directory. The Templates file is a standard ZW3D file that contains objects that you may wish to load when a new object is created. For example when you create a new drawing sheet you may wish to load a particular template file that contains the size of the drawing and perhaps the drawing border. In cam the requirement may be for your standard methods of cutting certain parts. The Cam objects in the template file can contain any amount of information that you feel is useful for you. An example would be perhaps a template file designed to cut small core type parts. This could contain the Machine definition, various cam operations, Speeds and Feeds, Tools and so on. Templates speed up the process of creating cam plans by avoiding having to create the same information every time you create a new cam plan. Also you can standardize on your methods of cutting parts. Further information on template files is available in the on line help.

Let's create our first CAM Plan.

Load the file "Mill_Bottle.Z3"

Create a new CAM Plan called "Model_Cam_plan". At this time we will not use any cam template.

This will take us into the Cam user interface. The Cam tree looks initially as shown below.

```

Setup1
  Geometry(undefined)
  Clearance
  
```

Frames
Tactics
Operations
Machine(undefined)
Output

Notice that the Geometry field is marked as (undefined).

Our first task is to insert the relevant cam geometry that we will need to cut the part. The cam geometry we require for this exercise is the “model” part and the “Stock_Model”.

Place the mouse pointer over the “**Geometry**” icon and hit the right mouse button, select the option “**Insert.**”

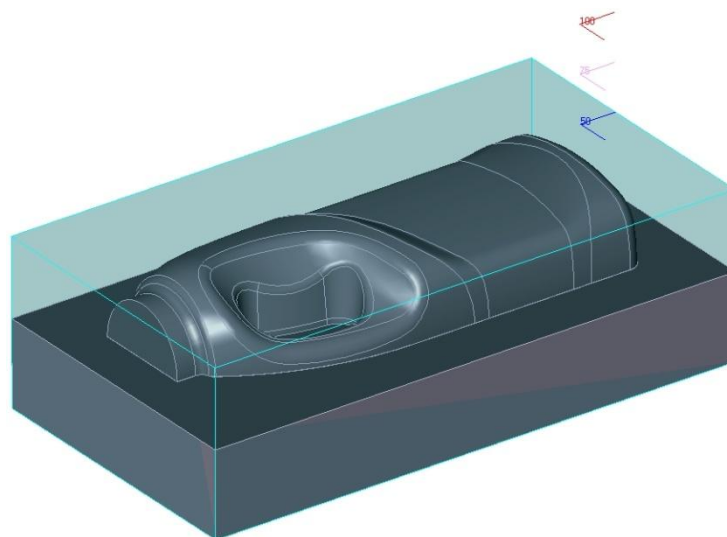
Next you will be placed in the Shape Browser. Here you can browse for the geometry that you wish to insert into your current CAM Plan. Highlight the shape you wish to insert, which in this case is the “model”.

At the prompt “Use Mill_Bottle.Z3 model?”, hit the “Yes” option and the “model” part will be inserted into your CAM Plan.

Repeat the same process and insert the “Stock_Model”.

Once the Cam geometry for the model and the model stock is inserted the Cam tree Geometry branch will show the correct names for the parts. Notice that the “**model**” has the class “**Part**” and the “**Stock_Model**” has the class “**Stock**.” This is because the part attributes have been set correctly in model space.

The graphics window displays the parts as shown.



Clearances

Place the mouse pointer over the “**Clearances**” icon in the cam tree.

<Right-click> and select “**Edit**,” or double click with the left mouse button to activate the clearance form.

The Icons on the cam tree all use the double left mouse click and right mouse menu select options. The <double Left-click> is a short cut tooften used <Right-click> menu choices.

The clearance options that are used in Quickmill are:-

Clear Z This is used when the tool retracts up to “clearance” such as the start and end of an operation.

Auto Clear. This is used to retract the tool to a “Safe” distance above material when the tool moves in “rapid” mode for example”

A more detailed description will be given when we discuss “Leads and Links” later in this manual.

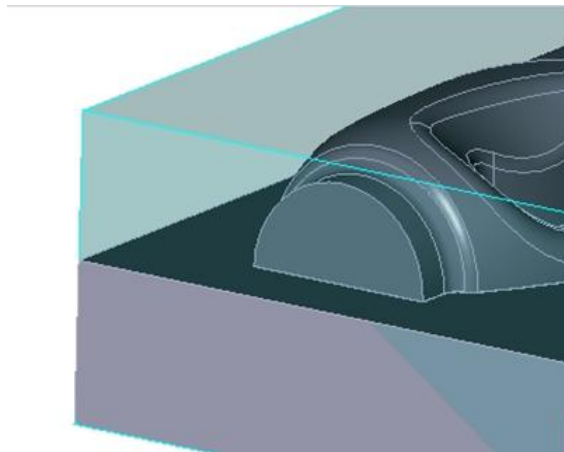
Enter the clearance values as following:

Clear Z =50, Approach Z = 50, Retract Z= 50, Auto Clear = 10.

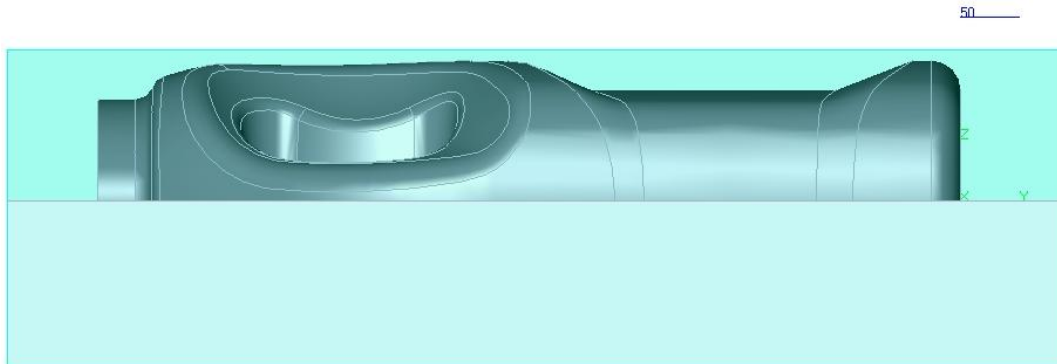
Make sure you check the **Auto clear** tick box.

We have used a clear value of 50. We can use the **Inquire pull down menu > Coordinate** command to find out the top height of the stock as shown below.

The result should be : (-80, -265, 41.10177).



The clearance plane at Z 50 is then created as shown below.

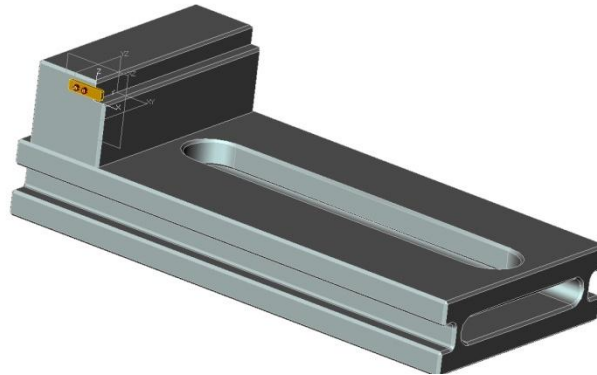


Save your file.

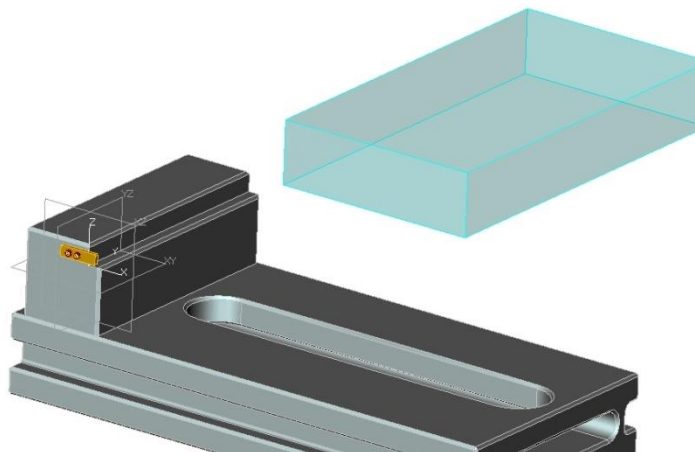
Cam Assemblies

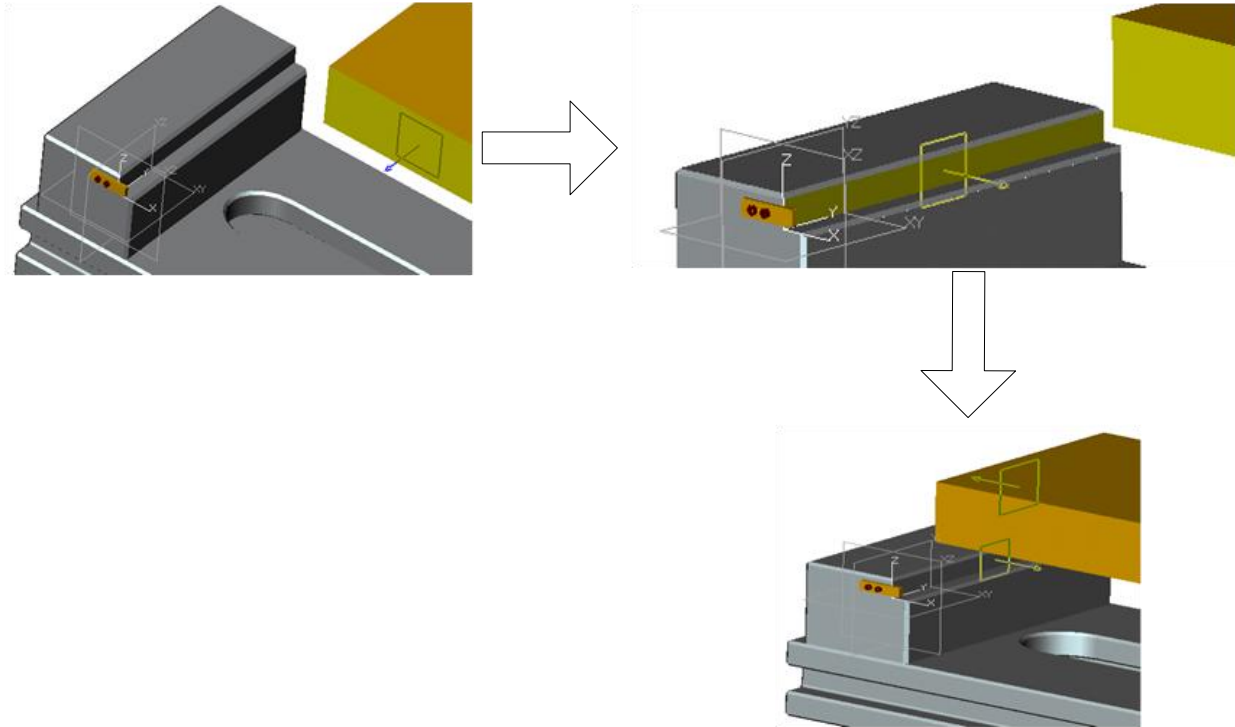
In the previous example we created a Cam Plan for the “Model” part and loaded the stock and the part into the Cam Plan. We can also load fixtures, clamps and tables into a CAM Plan if we wish. We can also utilize ZW3D’s powerful Cad assembly tools to first create an assembly which consists of all the relevant geometry such as parts, stock, clamps etc. and load this entire assembly into a CAM Plan.

- Create a new Part/Assembly in the “Mill_Bottle.Z3” file called “Blowmold_Cam_Assy”. 
- Right mouse click in the graphics area and select **“Insert Component”**.
- Use the browser icon in the Insert component form. 
- Locate and load the file “Cam_Fixture_Parts”.
- Check the “Anchor component” tick box.
- Select the “Fixed Vice Jaw” component from the list.
- At the prompt “Specify Insertion Point” enter 0 and then hit the <enter> key. This locates the component at 0,0,0



- Use the **Insert component** command to insert the next component (Middle-click recalls the last command).
- Press the down arrow next to the browser icon in the form.
- Load the Mill_Bottle.Z3 file and then select the “Stock_Blowmold” part.
- Position the part to roughly the same place.
- Align the first position of the stock into the correct place.
- Tip: Set the **pick filter** to “Face”.
- Tip: you will need to toggle the **Same Facing/Opposite Facing** options during Align command as required.



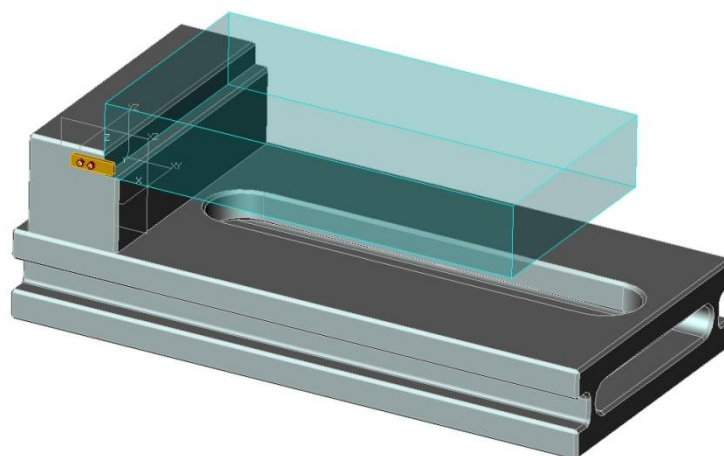


Figures above show the stages for aligning the first two faces of the Stock and the Vice.

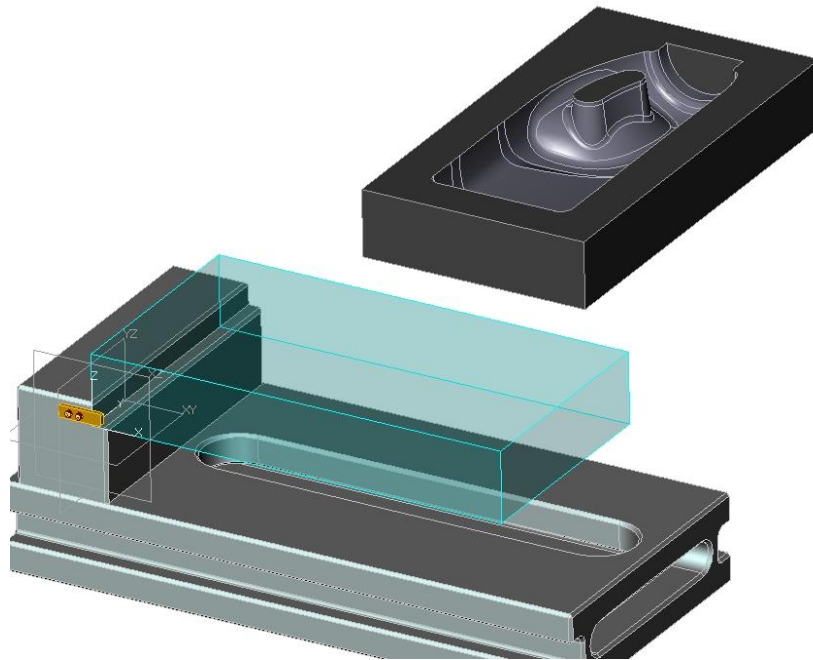
Use the same method to locate the next two faces and then the subsequent last two faces.

Generally speaking you are going to make 6 face picks before selecting 'OK'.

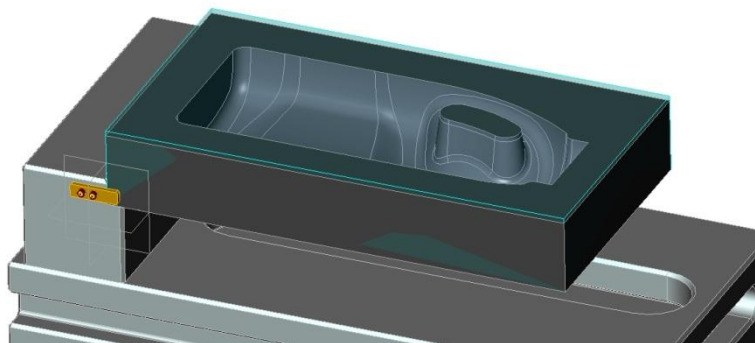
The fully aligned stock to the vice is shown in figure below.



Insert the next component which is the "blowmold" part.

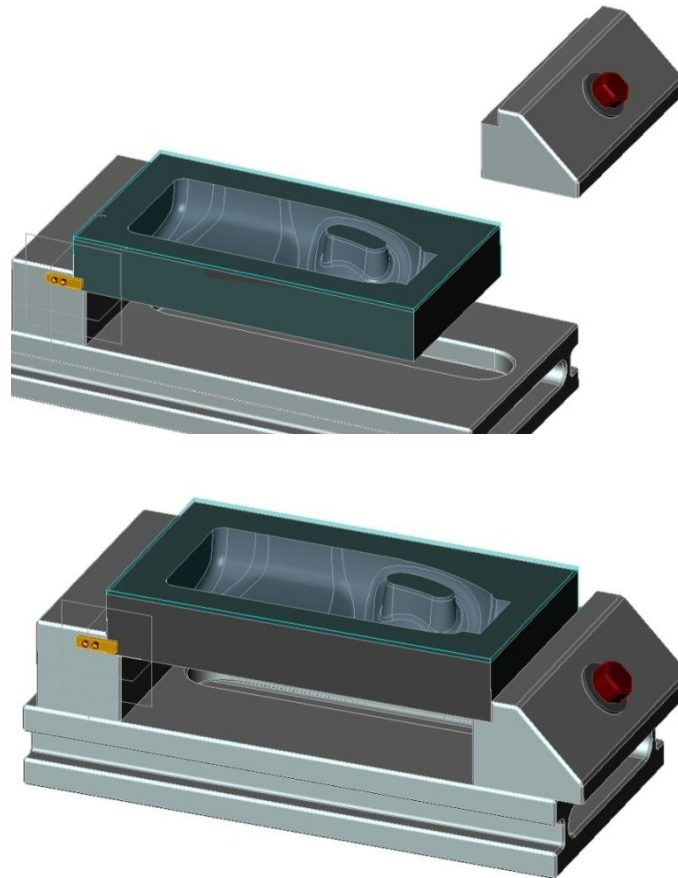


Align the blowmold into position. (Remember to set the blowmold in the bottom of the stock so the extra material can get machined away.)



Load the file “Cam Fixture parts” directly from the Insert component form. 

Use the same method once more to insert and align the Moving Vice Jaw.



The cam assembly is now ready to be loaded into a CAM Plan.

Note: Cam assemblies must only contain components, merged parts are not allowed.

Save your file.

Create a new CAM Plan in the “Mill_Bottle.Z3” file and name it “Blowmold_CAM_Plan”.

From the CAM tree “**geometry**” select “**Insert**”.

From the Shape Browser select the “Blowmold Cam Assy”.

This is then inserted into the current cam plan. Notice how the correct Cam Class is shown for the different components. The cam class is defined in the **Part Attribute** form when the original parts are created.

Change the values of each **Clearance** plane to be 60mm and the **Auto clear** value 10mm with the auto clear check box ticked.

Save the file.

Chapter 5 Rough Milling Basic Walkthrough

In this chapter we will cover,

- Basic QuickMill walkthrough, which will include:-
- Creating Cam operations
- Creating Cam Tools
- Referencing Cam feature into an operation
- A quick look at the operation parameters form
- Calculating our first toolpath
- A brief look at verifying

Basic QuickMill Walkthrough

Load the file “Mill_Bottle.Z3” and then load the CAM Plan “Model_Cam_plan”.

This will load the CAM Plan from the previous chapter. Displayed in the graphics area are both the stock, which is transparent and the part geometry. Let's hide the stock.

Place the mouse pointer over the “Stock” icon in the Cam tree

Right click and select the “Show/Hide” button, which toggles the display on/off.

Only the part geometry is displayed in the graphics window.



Creating CAM operations

Let's begin our walkthrough by creating a Quick Mill (QM) Cam Roughing operation.

The Cam operation form can be accessed in three ways.

- Option 1. <Right-click> with the mouse over the operations icon on the Cam tree and select “Insert”.

- Option 2. <Right-click> with the mouse in the Graphics window and select “Insert Operation”.
- Option 3. <Double Left-click> the operation icon in the cam tree.

The first time you go into the **Operation Type** form the **2X milling** options are shown.

Select the **Quicktab**. We will only be using this form (QUICK MILL Operations) in this training lesson.

We are going to create a “Rough Offset 2d” roughing operation. This is a 2D area clearance toolpath that removes the rough material in constant Z steps. This toolpath has a spiral type pattern that is projected onto the cam geometry.



Select the **Rough Offset 2D** icon from the top left.

The Operation branch of the Cam tree now includes our first operation “**Rough Offset 2d 1**”.

Notice that several of the operation fields, such as “Tool”, “Ref Op” and “Features”, are currently shown “(undefined)”.

The operation has the default name “**Rough Offset 2d 1**”.

You can choose to accept this name or you can rename it to a name of your choosing.

If you wish to rename the operation, place the mouse over the “**Rough Offset 2d 1**” icon and press the right mouse button.

Select the “**Rename**” option and enter the new name of the operation.

In our example we will accept the default name.

Creating Cam tools

The Cam Tool Manager can be accessed from each tool path operation in the operation section of the Cam Plan Manager tree. Tools can be defined, stored and retrieved from tool libraries and selected for operations. The Tool form is your primary interface with the Cam Tool Manager. With it you can define the shape and physical properties of a cutting tool. You can also access tools from tool libraries. A tool library is any ZW3D CAM Plan containing tool definitions. For further information on the tool manager refer to the **HELP Manual**.

<Right-click> the mouse over the “**Tool**” icon and select the “**Select**” option. Use the

browser icon  to load an example tool library that is included with the ZW3D installation. This library is located in the “Resource” folder which in turn is located in your “ZW3D User” folder. The Name of the Library is called “**TOOLS_METRIC_and_INCH.Z3**.” This library contains an example selection of tools specified in two CAM Plans. 1. INCH TOOLS and 2. METRIC TOOLS. This tutorial uses metric tools. Select the “**METRIC TOOLS- Legacy**” library and then select the correct tool from the list

We are going to use an 18mm diameter 2 corner radius End Mill.(Endmill D18/2)

Once you have selected the tool from the list, press the **OK** button to load the tool into the CAM Plan and load the 18mm end mill into the Rough Offset 2d 1 operation.

Referencing Cam Feature into an operation

The current operation that we are defining does not have any geometry associated with it. We need to reference the correct geometry required by the roughing operation. This operation is going to rough the entire part so we need to reference the part geometry. We also need to reference the stock geometry. **QUICK MILLroughing** operations **always** require the stock geometry. Even if the part has been partly machined, **always specify the stock geometry on roughing operations.**

Adding the Cam Features:-

- Make sure the “Geometry” branch of the Cam Tree is expanded.
- <Double Left-click> the **Feature** icon on the Cam Tree, Rough Offset 2D 1. It display the form of select feature for operation.
- Select the “Part” in the list of this form And then pick ok button to exit.
- This references the part geometry into the feature branch of the current operation.
- Repeat the above method, <Double Left-click> the **Feature** icon and so on, and reference the stock geometry into the current operation.
- The CamTree shows the Part and also the Stock geometry referenced into the current operation.

Setup1
 Geometry:
 Part: model (1)
 Stock:Stock_Model(2)

```

Clearance
Frames
Tactics
Operations
    Rough Offset 2d 1 (Rough Offset 2d)
    Class : Rough
    Tool :Endmill D18/2
    Parameters:
    Ref Op( undefined)
    Features
        Part: model(1)
    Stock :Stock_Model (2)
Machine(undefined)
Output
    
```

- Remember to **always** reference stock geometry when creating **roughing** operations.
- When you have more than one operation specified in the Cam tree, you do not have to select the cam feature from the Geometry branch. You may instead select the Cam Feature geometry that you wish to reference into the current operation from a previously defined operation.

Let's take a quick look at our operation parameters.

Operation Parameters

<Double Left-click> on the operation **Parameters** icon in the Cam tree.

The operation parameter form for the rough offset 2d operation type is shown. The form is divided into separate topics.

Take a moment to visit each of the parameter topics using the buttons at the top of the form. Do not change any of the options at this stage. Return to the "Cutting" tab.

As you can see there are many options available to you. In fact there are so many different options available that the thought of having to learn them all can be quite daunting.

Fortunately, you do not have to. QUICK MILL has been carefully designed so that the majority of the time you can accept the default values. You can even specify what those default values should be, refer to the "Cam Operation Config File Editor" in the HELP Manual for further information.

Also all the various QUICK MILL forms for the different types of operations are very similar in appearance and where possible share the same options, speeding up learning.

Cam processes however by their very nature need to be flexible and allow the user to choose the exact method of cutting the parts.

QUICK MILL offers the best of both worlds enabling the user to use many of the default values most of the time but with the added capability of fine tuning the operations to suite specific requirements.

At this stage we will not change any of the parameters, so exit the form by pressing the “**OK**” button.

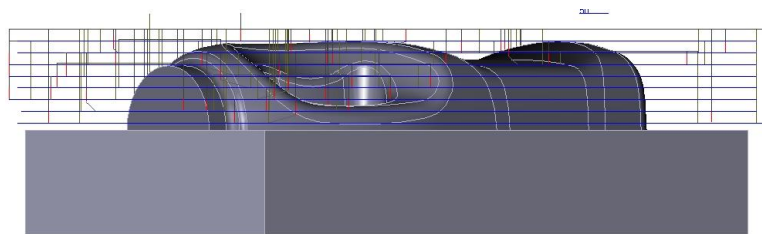
Calculating our first toolpath.

<Right-click> the “Rough offset 2d 1” icon on the Cam Tree.

Select the **Calculate** button. This calculates the toolpath.

<Right-click> the “Rough offset 2d 1” icon on the Cam Tree once more. Select the “Verify” option to use the Wireframe Verify capabilities.

More information on this type of verifying can be found in the on line help.

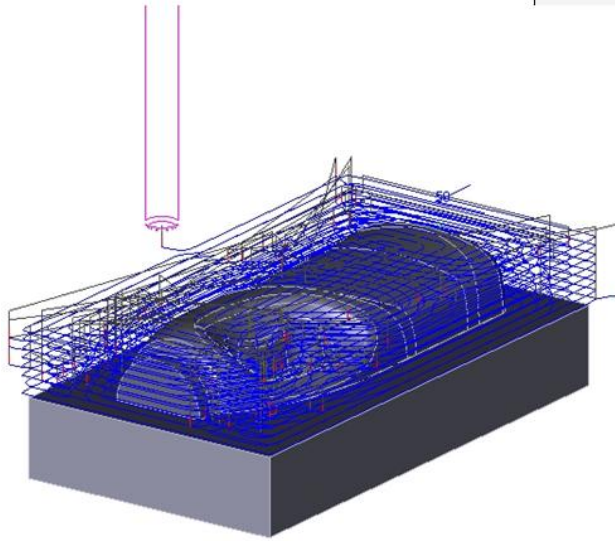


```

F 250.0000 MMFPM
S 1000.0000 RPM

X -97.8628
Y -117.4998
Z 50.0000

I 0.0000
J 0.0000
K 1.0000
    
```



Solid and Rapid Verify.

Solid Verify gives the user a more accurate representation of the virtual milling toolpath. The cutter (and optionally the tool holder and Fixtures) is displayed along with the stock and eventually the cut part. The material removal and tool motion is dynamic.

Rapid Verify, is less accurate, but allows the part to be verified without having to view each toolpath motion. The final toolpath is displayed for each tool in the CAM Plan.

The user can swap between each verify options as required.

Partially cut parts can also be saved as virtual “Stock” input into subsequent verify sessions.

Right -click on the **Operation**” icon on the Cam Tree. Ensure that the **“Solid Verification”** option on the Contextmenu is displayed. Select it.

Press the start arrow (**run simulation**) on the Verify session form.

We will be using the verify capabilities further on in this manual. For a more complete description refer to the on line help.

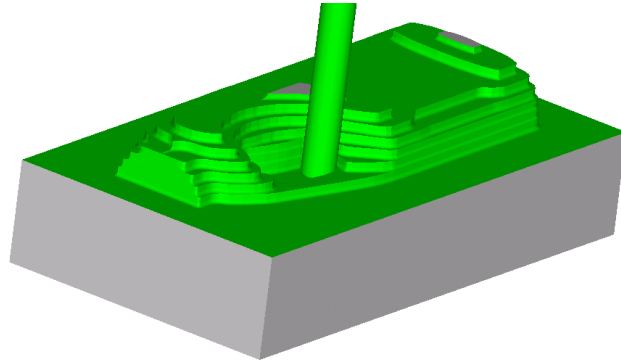
The verify buttons from left to right are:

- Run solid simulation. ▶
- Single step forward solid verification. ▶▶
- Reset. ■
- Run rapid simulation to the end of all operations. ▶|

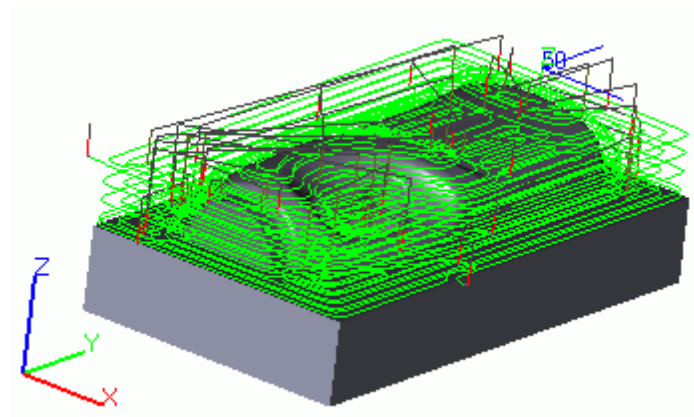
- Run rapid simulation to the end of the current operation. 

Save your file.

Solid Verification



Rapid Verification



Save the file.

Chapter 6 Rough Offset 2D Cutting Parameters

In this chapter we will cover,

- The operation Form Parameters, with emphasis on:
- Z level roughing with Interlaced Z steps.
- Different cut patterns.
- SynchroZLevel.
- Enabling Spirals to produce a near constant load on the cutter.
- Analysis Accuracy.

Load the file “Mill_Bottle.Z3” and then load the CAM Plan “Model_Cam_plan”

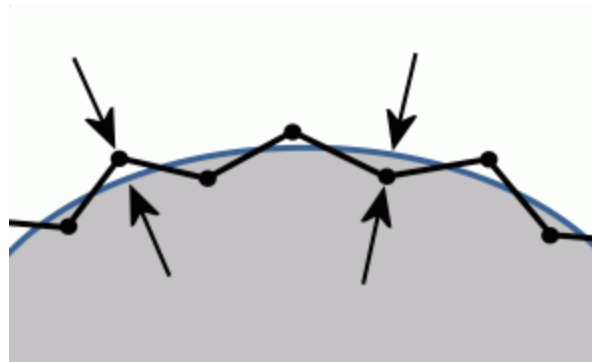
Our CAM Plan currently has one operation “Rough Offset 2d 1” Let’s begin this lesson with a closer look at the operation parameters.

DoubleLeft-click on the **Parameters** icon in the Rough Offset 2d operation to open the parameter form. We will first take a tour of the “Cutting” section. Many of the parameters discussed here also apply and are used by other operation types.

Select the “**Primary**” option

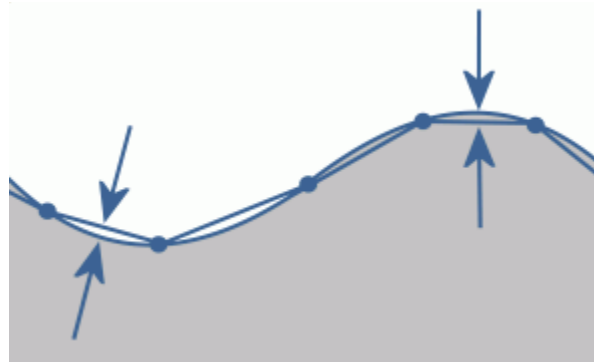
Path Tolerance

This is the toolpath tolerance. See the graphic at the bottom of the toolpath form. The value is the maximum distance that the tool will be positioned from the part between each toolpath move. The value is a plus and minus condition, so if you specify 0.1mm tolerance, the move can be up to 0.1mm plus metal condition and up to 0.1mm minus metal condition. To set an “Outtol” (plus metal condition) 0.1mm for example. Set the tolerance value to be 0.05 and the surface thick (see below) field to be at least 0.05. Decreasing the “Path Tol” value will increase the toolpath computation time.



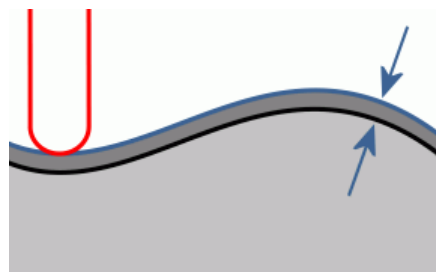
Surface Tolerance

QUICK MILL toolpaths are calculated on a tessellated representation of the part geometry (like an STL or faceted model). The “Surface tol” is the tolerance of the “STL” tessellated part. If this field is left blank, then the value will be the same as the value specified in the Path Tol.



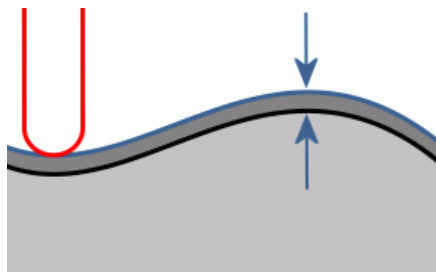
Surface Thick

This is normally used to specify a plus offset distance from the tool to the part and is usually used in roughing and semi finishing operations. A plus or minus value can be used. However the maximum minus value that can be used must not be bigger than the radius of the tool being used. When using QUICK MILL roughing operations; ensure that the Surface Thick value is at least three times greater than the Path Tol value. Do **not** use QUICK MILL roughing operations to finish parts.



Z Surface Thick

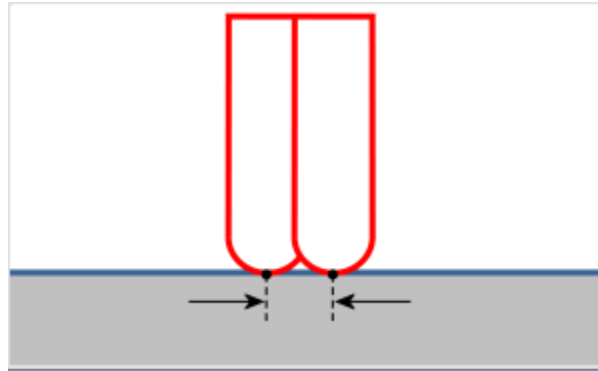
This provides control over the floor stock to be left on the workpiece. If it is not defined, it will be the same as the **XY Surface Thickness** parameter. This **Z Surface Thickness** parameter can be a negative value.



Stepover

<Left-click> on the Stepover to gain access to the options, Absolute, % Tool Diameter and Scallop.

Absolute option shown below.



Stepover Size

Enter a value in the field. This is the step over value that will be used by the tool. The actual value will depend on what the Stepover is set to. So if for example the stepover type is set to % Tool Diameter, then the default value of 45.0 refers to an actual step over value of 45% of the current Tool Diameter. If the tool Diameter is changed the step over value will change accordingly. QUICK MILL is designed so that the maximum step over is no more than 45% of the Tool Diameter.

Stepdown

<Left-click> on the Stepdown to gain access to the options, Step Size or % Flute Length. The step down in Z will be based on an actual value or on a % of the flute length of the current tool being used.

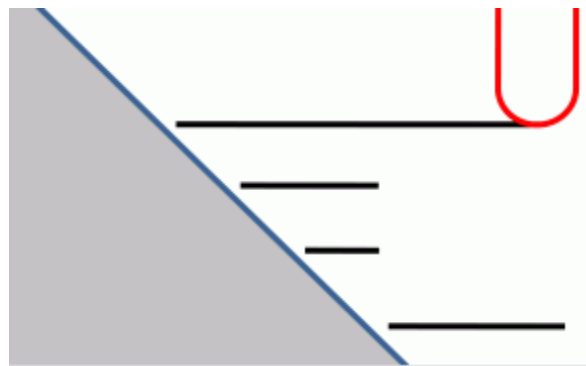


Stepdown Value.

Enter the value in the field. The actual step value depends on which Z step Type option has been selected.

Number of Cuts

This is the number of extra or interlaced cuts that will be applied between each Z level area clearance in the roughing cycle. The interlaced cuts have a constant Z value. Normal constant Z roughing operations remove all the material between the part and the stock at each Z level of the operation, performing an area clearance. The tool will then move down to the next Z level and calculates the next area clearance material removal between the part and the stock, or just the part if it is a cavity or pocket type cut. Our example is a core type part where most of the material removal is between the part and the stock. This can leave Z steps on the part that can be difficult to remove using finishing operations. The Z Intercut option removes these steps by performing extra cuts between each of the normal Z level area clearance passes in the rough operation. The extra passes only remove material local to the part and do not perform unnecessary fresh air moves. The resulting tool path still has steps but they are much smaller and easier for the finishing operations to deal with.



Z Intercut Progress

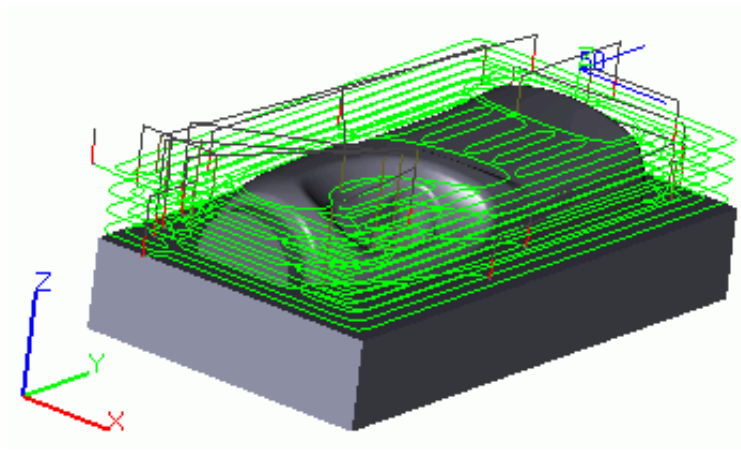
This is the Z direction of the steps when using the interlaced cut option. Click on the button to access the options, Top to Bottom or Bottom to Top. Currently the field is unavailable. It becomes available only when you enter a value in the previous “# of Steps” field.



Let's compare our part, cut first without Z Intercut passes and then with.

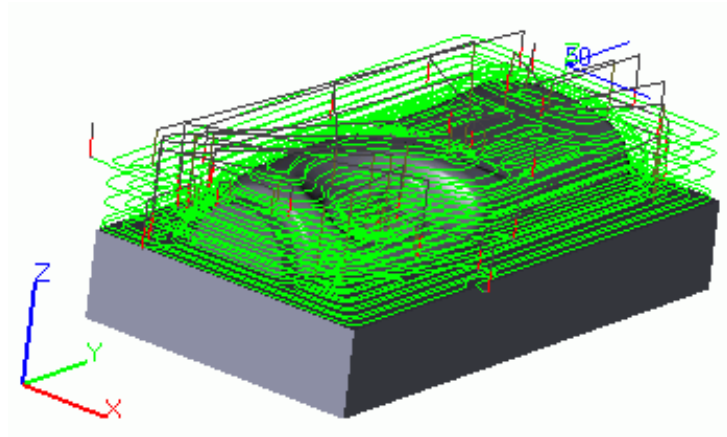
- First make the Z step size larger and also use a rough allowance of 1mm.
- Change the value of the Z step size from 5 to 10.
- Press "ok" on the operation parameter form, to accept the edit and close the form.
- <Right-click> on the "Rough Offset 2d 1" operation and select calculate.
- Use the Rapid verify option to verify the part.

The image below shows the part cut with 10mm Z steps and without using Interlace Z cuts. The resulting steps in Z can be difficult to remove with finishing operations. We could reduce the value of the Z steps in order to make the steps on the part smaller and more manageable. This would however increase the cutting time dramatically as each Z level would have to perform a complete area clearance cut at each of the new Z levels.



We can use the Z Intercut option to do the bulk of the milling at a Z step value of 10 and yet still end up with smaller steps on the part. The tool will remove more material local to the part without the extra unnecessary cuts you would get if you simply made the Z step smaller.

- Change the **Number of Cuts** to 3.
- This will keep the main milling to Z steps of 10 and create additional Intercut levels between each of the normal Z levels of the roughing operation.
- Calculate the tool path.



Edit the operation parameters once more and set the **Number of Cuts** option to 0, and it will remove the Z Intercut moves

Calculate the tool path once more.

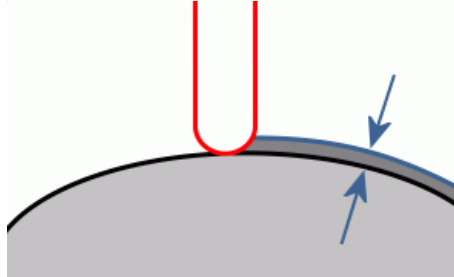
Notice how quickly QUICK MILL performs this calculation. This is because QUICK MILL uses a technology called “Advanced Caching”.

We will be looking into caching in much more detail later.

Select the “**Limiting**” option

Cast Offset

This parameter allows you to enter stock thickness to be considered during tool path calculations.

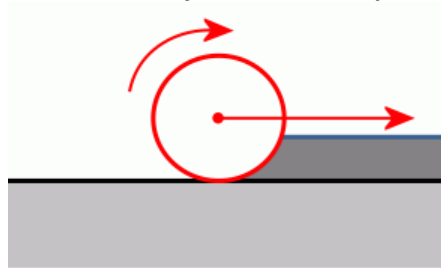


Select the “**Path Setting**” option

Cut Direction

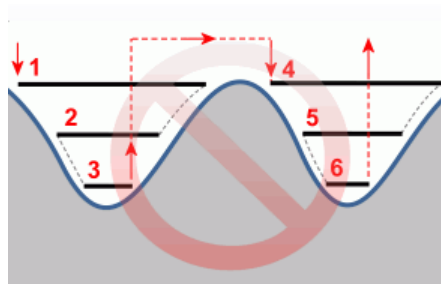
The pull down window shows the various cutting mode options.

- **Zigzag** which allows the tool to move in any direction climb or conventional.
- **Climb** which forces the tool path to be using climb milling in one direction.
 - Shown opposite.
- **Conventional** which forces the tool to cut using conventional milling in one direction.
- Other options (but not available in this particular operation type) are:-
- Top to Bottom, forcing the Tool to always cut with a downwards motion.
- Bottom to top, forcing the tool to always cut in an upwards motion.



Cut Order

Choose the options “Region first” or “Level First”. The option “Region First” forces the tool to cut local regions down to depth. For example if the part contained pockets, each pocket would be cut down to depth, whereas if the option “Level First” was selected all the regions at a particular Z level would be machined and then all the regions in the next Z level and so on.



Path Pattern Guide

The pop up box has the following options:

- Part.
- Stock
- Both.
- Longest Part Span.

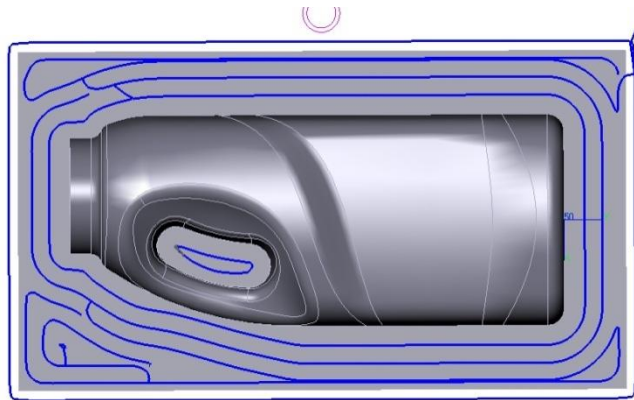
The path pattern Guide controls the shape of the flow of the tool when cutting. The actual path pattern that is performed at each Z level of the Offset 2d roughing operation

is based by default on the shape of the part. This can be changed to any of the options shown above.

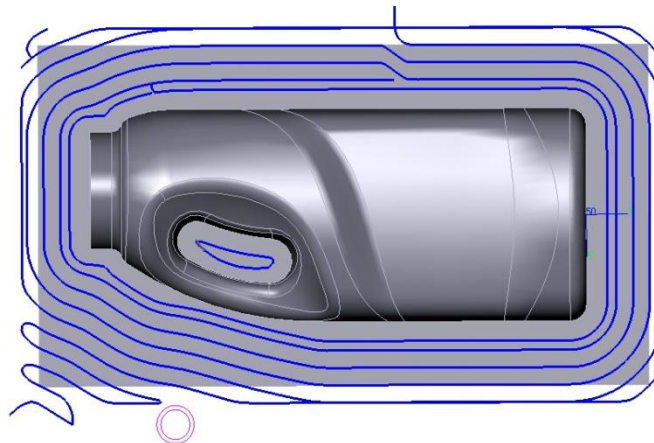
Rough Offset 2d Path Pattern Guide Examples.

Two examples of possible Path Pattern options are shown below. The actual path pattern in our example changes at each Z level. The figures below have had all of the Z level cuts except the very last one removed to avoid ambiguity.

Path pattern flow based on the shape of the part.



Path pattern flow based on the longest part span.

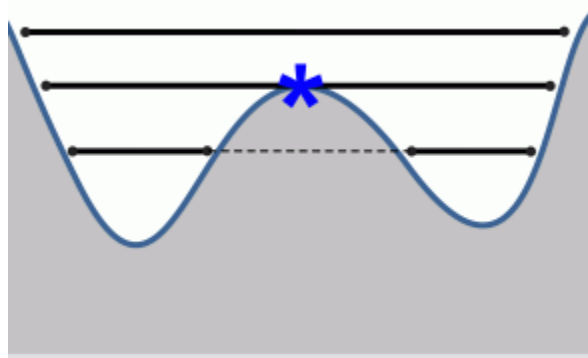


If you have altered the path pattern guide, please reset it to “Part” and re-calculate the tool path.

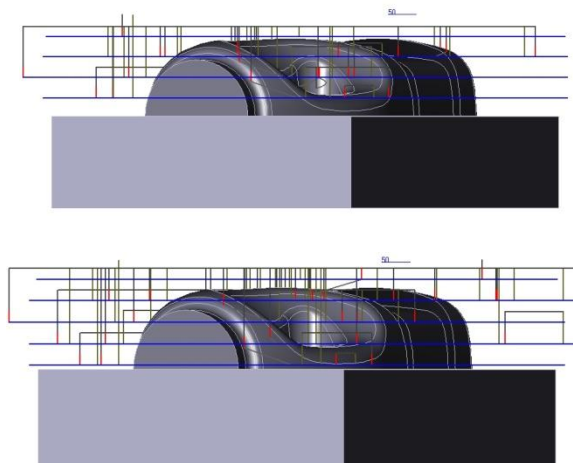
SynchroZLevel

Use this value to generate the cutting loop at the specified Z level. The cutting at Z level is synchronized to meet this level.

Enter the Z value or pick a point from the graphics window. The standard set of point input options are available.

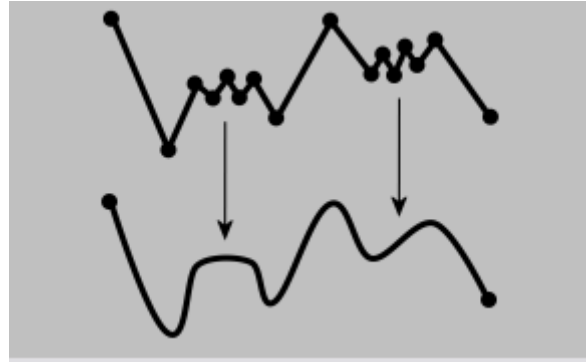


The image below left shows our current part without using the SynchroZLevel option. The last pass is a bit high up in Z. The final Z level depends on the starting Z height and the step size used (10mm in our example). If we use the SynchroZLevel option and pick a point at the base of the bottle shape (Z 0 in this case) the starting height will be based on this Z value increased in steps of 10 (the Z step). The image below right shows how the last pass in Z is at the same level as the bottom of the bottle shape (plus 1mm because of the 1mm Surface thick)



% XY Smoothing

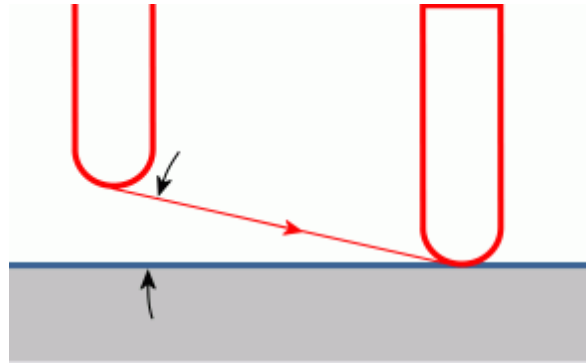
Smooth all tool paths based on a percentage of the Step Size.



Select “**Link and Lead**” option

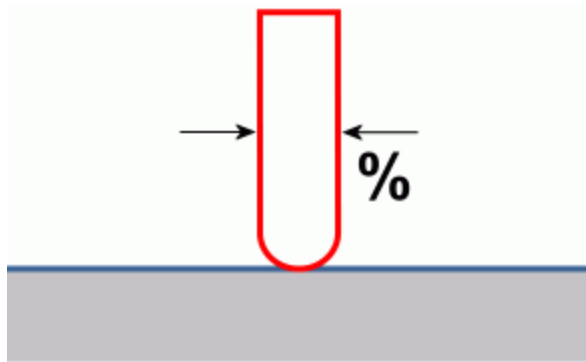
Ramp Angle

If this value is > 0 , the tool will ramp from the current cut depth down to the next. Use of ramping motion removes the need for access holes.



% Ramp Extent

This refers to the minimum length of a plunge move. It is measured as a percentage of the Tool Diameter. Range = is 50 % to 100% of tool diameter. Default value = 80 % of tool diameter.



Modified QUICK MILL rough offset2dcut operation definition.

WARNING: Minimum ramp distance 10.0000 less than tool diameter.

WARNING: Minimum ramp distance value to 18.0000 (=tool diameter) is used.

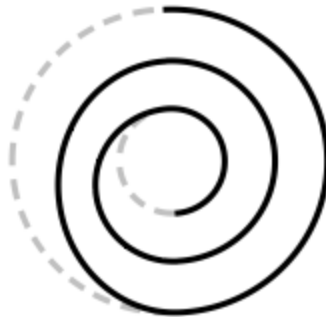
Safe Ramp

(Yes or No) This will check Ramp moves to avoid any tool collision (and breakage) with the workpiece.

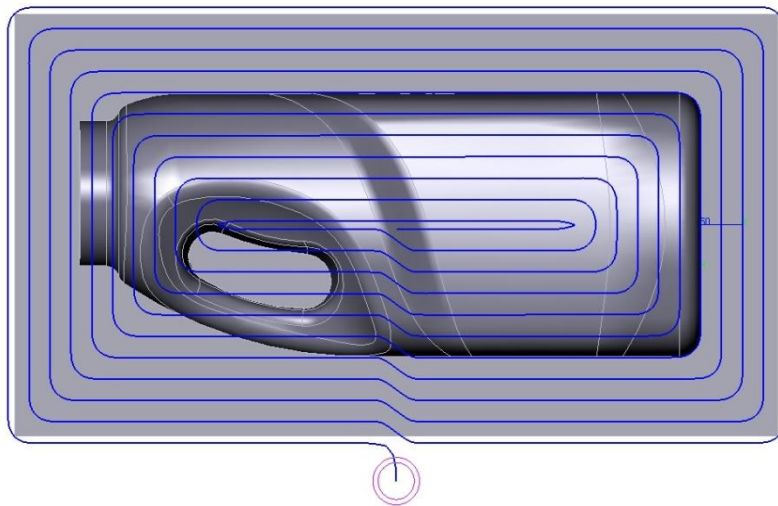
Select “**Advanced**” option

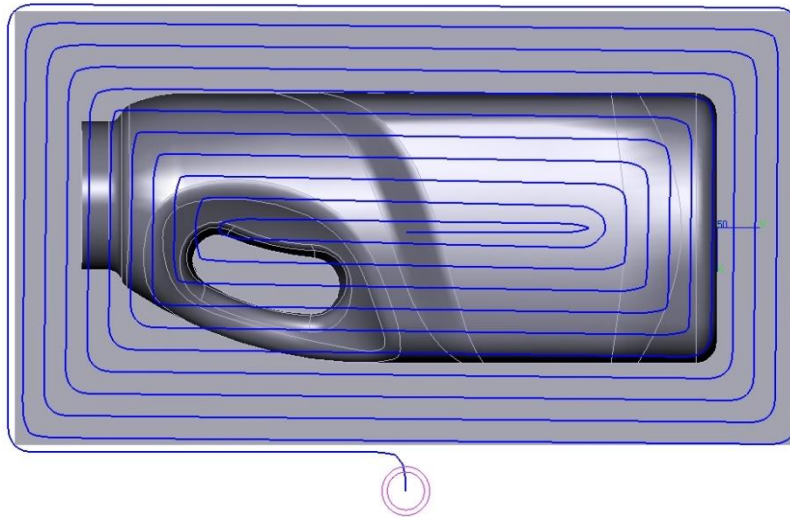
Spiralize

This option (available from the “Advanced/Spiralize” tab) forces the tool to cut each region in a continuous spiral like pattern. The first image below shows the first Z level of current operation (all engage and retract move have been removed for clarity). The toolpath is spiral in shape but if you look carefully you can see that the tool moves over to cut each concentric pass. It is a smooth stepover but nevertheless the tool does perform stepover move. The second image uses the “Enable Spiral” option. This time the toolpath is a continuous spiral cut with no sudden stepover. This is a very efficient toolpath and the load on the cutter on the entire toolpath is almost constant.



Over Mill = Half

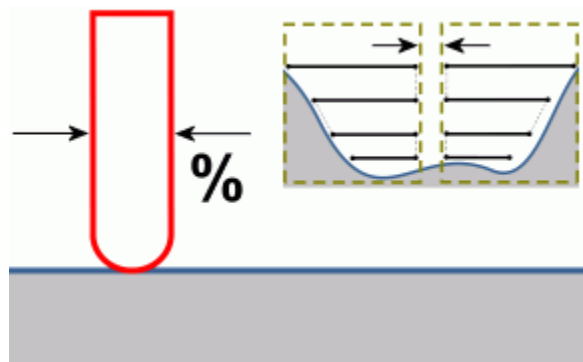




If you have tried the **enable spiral option**, please set it back now recalculate the toolpath.

% Interference

This is an inter-region interference. So the distance that is safe between 2 regions of toolpath that can or can't be linked by region.



Analysis Accuracy

The **Analysis Accuracy** section of the cutting parameter form defines the amount of analysing that will be performed. Analysis accuracy has a dramatic impact on the toolpath computation time. If the value of analysis accuracy is too fine the toolpath will take a long time to calculate.

The analysis accuracy has no affect on the accuracy of the cut part. That is controlled by other factors such as "Path Tol" and "Surface Tol".

Analysis accuracy is used to detect what to be cut and what to avoid. Analysis accuracy is also used by the Adaptive Feed Control" (AFC) capability.

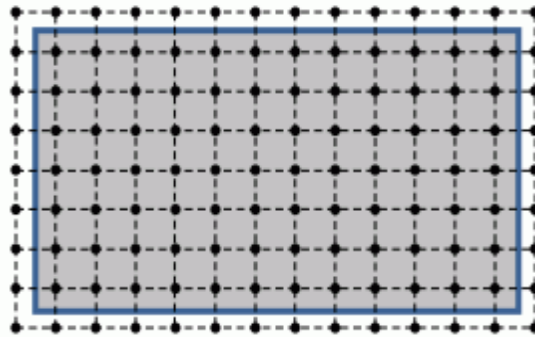
The following list illustrates some of the areas that are affected by analysis accuracy.

- Many types of collision detection, rapid moves, tool holders, fixtures, clamps etc.
- Rest material detection.
- Uncut and un-cuttable material, used for example in sequential roughing operations.
- Shallow and steep angle region detection.
- Adaptive feed rate control.
- Silhouette detection.

Don't worry about understanding all of the previous examples, we haven't covered them yet. Just think of analysis accuracy as detecting what bits to cut and what bits to avoid.

The analysis accuracy is entered into the form in one of two ways.

Option 1. User defined GridStep



On the bottom of the form there is a tick box called “**User Value**”. If this is ticked the greyed out field becomes available and the user can enter the desired value of the “GridStep” in mm or inches. This will set up a grid that the software follows and is used to detect what to cut or not cut, as explained above. The larger the value of GridStep the faster the toolpath calculation will be. **Do not use a value which is larger than the current tool diameter/10.**

The smaller the GridStep value the slower the toolpath calculation will be. So the question is what we should set the value of GridStep to be. Well it depends on a number of factors, the most important one being the size of the part.

If the part is for example approximately 50mm square, then a GridStep value of 0.5mm for example would be reasonable and would calculate the toolpath quickly.

However if you specify the same or an even smaller GridStep value on a part that is 500mm square, then we would have a much larger grid, the toolpath would take much longer to calculate because of lots of unnecessary analysis accuracy detection. There are other factors that require the user to set the correct value, for example if the part contains tiny features. Then it might be necessary to make the GridStep smaller in order

to detect all the tiny features. If you decide to use this option a default value of 0.5mm is a good place to start.

This option however should only be used by the more experienced user. The second option is much simpler to use and the default value will take care of about 90% of all parts.

Option 2.Coarse/Fine. Setting the analysis accuracy with a sliding scale

This is the more automated and easier to use option. Use this method and 90% of the time you will not have to change anything, just use the default.

Use the slider to set the grid step from Coarse to Fine. Unlike the previous option this does not set a step value to define a grid but rather uses a different method to calculate how dense the grid should be automatically, based on the size of the part. The software determines the size of the part and sets up the grid depending on where the slider is positioned. The default is set to the “Coarse” position. This will sample 250,000 points which creates a grid of 500 x 500. If you position the slider at the other end of the scale and set it to “Fine” the software will sample 100,000,000 points which is a grid of 10,000 x 10,000.

Use the slider to tweak the results. For example if a rest material operation has not detected all of the rest material to your satisfaction, move the pointer slightly towards the fine setting and it will improve, but the toolpath calculation will be slower.

This is by far the most preferred method and requires very little intervention from the user, as stated previously the default value of “Coarse” takes care of 90% of all parts regardless of size.

Chapter 7 More Roughing Capabilities

In this chapter we will cover,

- Rough Lace Operation
- Speeds and Feeds
- Copying existing Speeds and Feed tables
- Saving the current workpiece in a Verify session
- Sequential Rest Roughing
- Loading a previously cut workpiece into a verify session
- Plunge Roughing
- A quick look at Limiting
- Wave Propagation Roughing



Load the file “Mill_Bottle.Z3” and then load the CAM Plan “Model_Cam_plan”.

Right-click on the Rough Offset 2D 1 operation and hit the “Show/Hide” toggle to switch off the display of the current toolpath.

The Rough Lace operation.

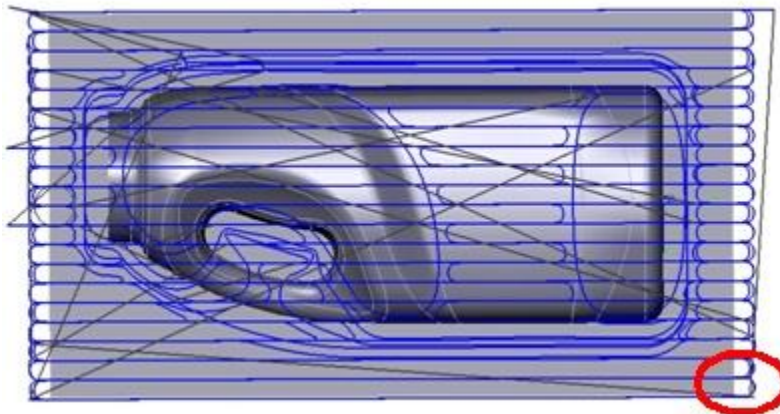
Let’s create another Roughing operation. This time we will be using a roughing operation based on a “Lace” cutting pattern. Lace cuts can either be in one direction or the tool can move back and forth in a Zigzag manner.

- Double Left click on the **Operations** option in the CAM tree to open the Operation Type form and select the Quick Mill operation. Then create a new **Rough Lace** operation by selecting the icon. 
- Once the operation has been created we need to insert a tool. Use the same tool as in the first operation.
- The tool already exists in the CamPlan, so simply Right-click the mouse over the “Tool” icon in the Rough Lace cam tree operation to activate the “Tool List.”

- Now select the “Endmill D18/2”.
- Next Double left-click on the feature icon and insert the part geometry “model”.
- Now insert the Stock geometry “Stock_Model”. Remember QUICK MILL roughing operations **always** require that the stock is referenced in the operation feature.
- The operations involving Features are model (1) and Stock_Model (2).

Go into the Parameters/**cutting parameters** form of the Rough Lace cut operation. Notice how similar this form is to the previous operation form. QUICK MILL has been designed so that whenever possible all the forms will have the same look and feel.

- Make sure the **Surface Thick** is 1mm.
- Change the **Stepdown Size** to 10mm.
- Select the “**Path Setting**” Tab hit the “**SynchroZlevel**” button and select a point on the top planar face.(as shown below)
- Choose Climb as **Cut Direction**..
- Calculate the toolpath.



Cutting Direction

- **Zigzag** (Default) this allows the tool to move back and forth bi-directionally utilizing both conventional and climb milling techniques.
- **Climb** Forces the tool to cut in a unidirectional climb milling mode.
- **Conventional** Forces the tool to cut in a unidirectional conventional milling mode.

Feeds and speeds

Speeds and Feeds are defined individually for each operation and take the same name as the operation.

- Press the **“Feed parameters”** tab in the parameters form on the Rough Lace operation
- The name of the Speeds and Feeds is displayed.
- The Advanced Feed Control will be discussed in a later chapter.
- Press the **Speeds/Feeds** button to open the Speed and Feeds form.

The Speeds and Feeds table contain a base **Finish** and base **Rough** field. These are the base values that are used in general cutting conditions. Which one is used, Finish or Rough, depends on the setting of the Class switch in the Cam operation tree

The Speeds and Feeds types listed are shown. Use only the fields that are **not** greyed out. The other usable (not greyed out) fields may be defined separately from the base feed rates. You have the option of defining an explicit rate (just a number), by typing the word “rapid” or by defining these values as a percentage (%) of the base rate.

You cannot use the % option on the base rates. These must be absolute numbers that the other rates may be calculated from.

- Type in the word “rapid” in the two fields “Retract and Traversal” in the “Feed rates” section.
- Press “OK” to save and exit the form.

Make sure the Rough/Finish class type on the previous Rough Offset 2D operation is set to “Rough”. If it is set to “Finish” change it now.

Do not calculate the toolpath yet.

Copying Speeds and Feeds tables from other operations

We have made some changes to the Speed and Feed form on the Rough Lace operation. We can copy this form into the Offset 2D operation.

- Open the Feed and Speed form in the first operation(pick the “Feeds,Speeds” button).
- Press the button **“Copy Values from Another Table”**.
- Select “Rough Lace” from the list.

- All the values from the other operation Speed and Feed table is copied into the current one.

We need to update both toolpaths. This time we will perform a batch toolpath calculation instead of individually calculating each toolpath separately.

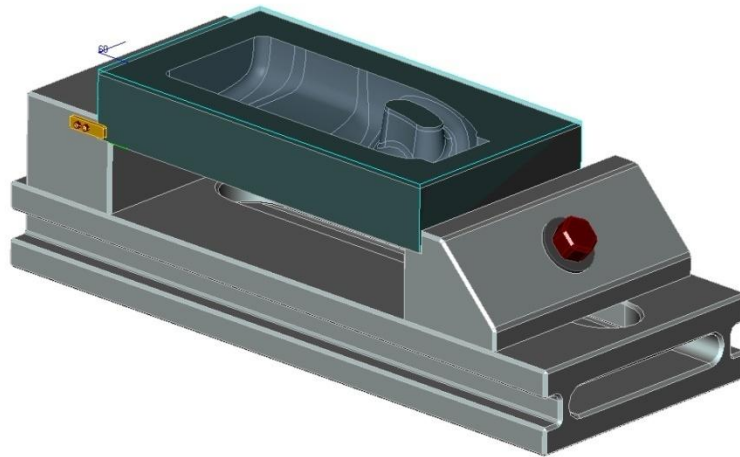
- Right-click on the **Operations** icon in the Cam tree.
- Select **“Calculate All”**.
- All toolpaths in the current CamPlan will be calculated.

Save the File and then clear session “File > Manage Session> Clear Session.” If the “Manage session” menu is inactive, select “Utilities > Configuration” and check the “Enable Session Management” box on the “General” tab.

Rest Roughing

Using QM, rest roughing with small tools is easy to achieve. QUICK MILL automatically detects left over material and generates the appropriate rest roughing toolpath.

Load the file “Mill_Bottle.Z3” and then load the CamPlan “Blowmold Cam plan”.

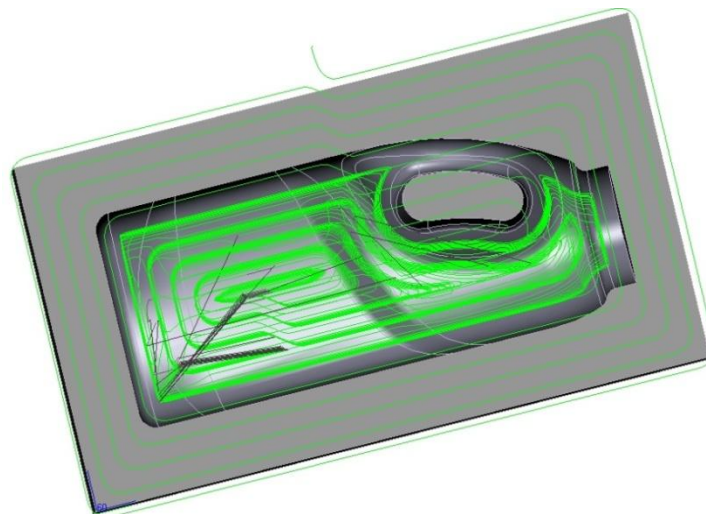


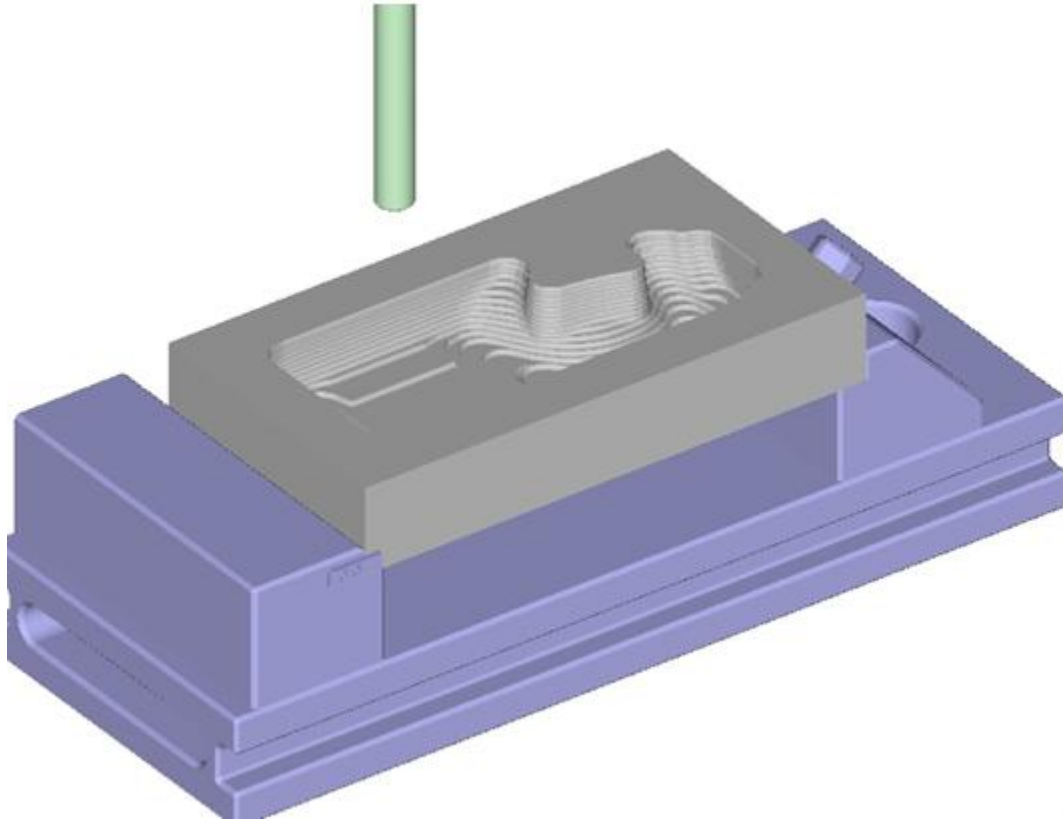
- Hide the fixtures and the stock.
- Define a **Machine**. (Name it “Machining center”)

Create a Rough Offset 2D operation to the following specification

- Use a 20 Diameter - radius 2 Endmill (Endmill D20/2).

- Load the cutter from the example tool library supplied with the software.
- Apply a roughing allowance of 1mm by setting the Surface thick to be 1.
- Set the stepdown size to be 2.5mm.
- Don't forget to add the part and the stock feature to the operation.
- **Calculate** the toolpath
- **Solid verify** the toolpath
- After the verification is complete select the **Options** button on the Solid Verify session form to open up the Solid Verify Options form.
- We will save the current “machined” part and use this partly machined workpiece as stock in subsequent verification operations.
- Hit the “**Stock**” tab.
- Press **Apply** in the form to save the current workpiece.
- Close the Verify session.





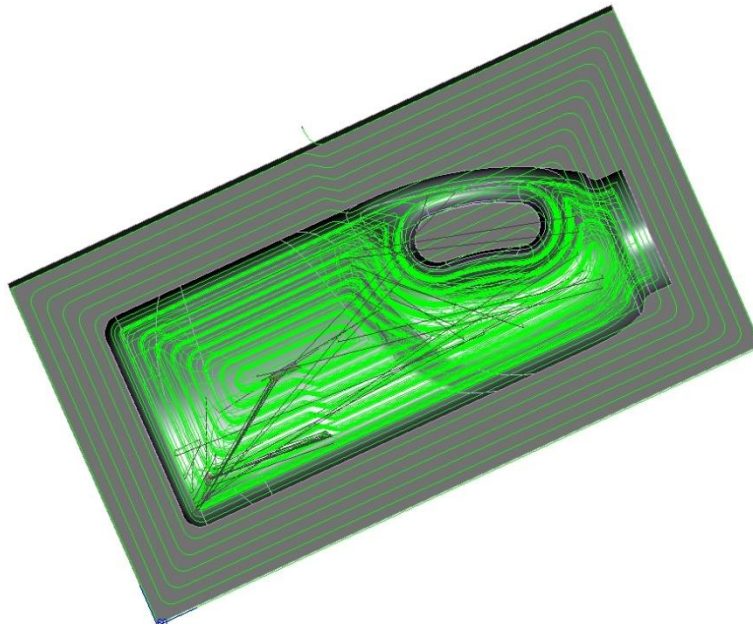
Creating a sequential rest roughing operation

Create a new Rough Offset 2D operation with the following options:

- Use a 12 Diameter Ball Endmill.(Ball Endmill D12).
- Load the cutter from the example tool library supplied with the software.
- Apply a roughing allowance of 1mm by setting the **Surface thick** to be 1.
- Set the **stepdown size** to be 2.5mm.
- Add the part and the stock feature to this operation.
- Hide the previous toolpath.
- Do not calculate the toolpath just yet.

If the toolpath were to be calculated at this stage, the toolpath would be very similar to the first roughing operation, with the exception of more cuts because of a smaller tool. What we really need of course is for this toolpath to cut only those areas that the

previous larger cutter was unable to machine. We can accomplish this by using a technique called Rest Roughing.

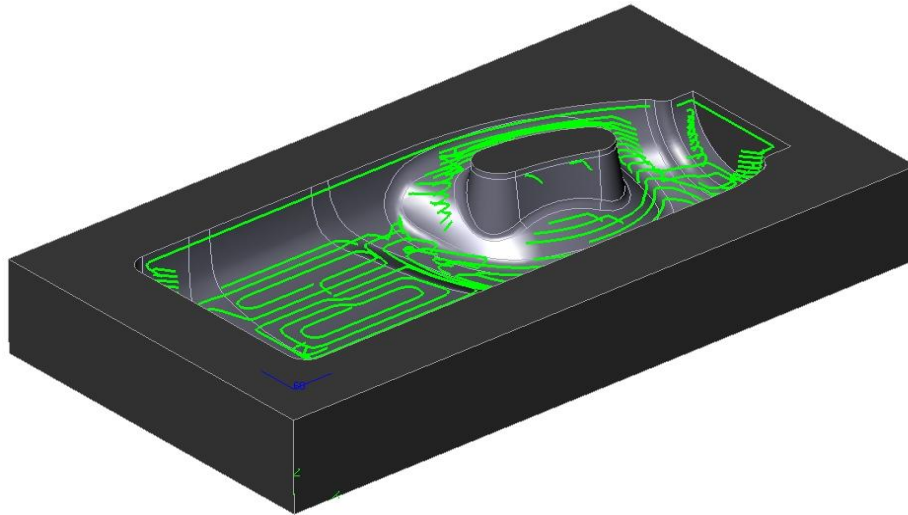


Users can program QUICK MILL to use related toolpath combinations producing optimum cutting results. These cutting combinations can use various finishing and roughing cycles with an assortment of tools and cutting parameters. As each cycle completes QUICK MILL dynamically updates the machined stock for use in the next sequential cycle.

We can use Rest Roughing on our new operation by simply referring to a previous operation in the Cam tree.

- Right-click or double left click on the Ref Op icon on the second roughing operation in the Cam tree, then the form of Select Reference Operation will be displayed
- Pick the first rough operation
- The second rough operation now has a Ref Op attached to it. (it shows “ Ref Op : Rough Offset 2d 1”)
- Note: Even though we are referencing the first operation to determine the current machined stock, we must not forget to include the original stock feature in the second operation.
- Now calculate the tool path.

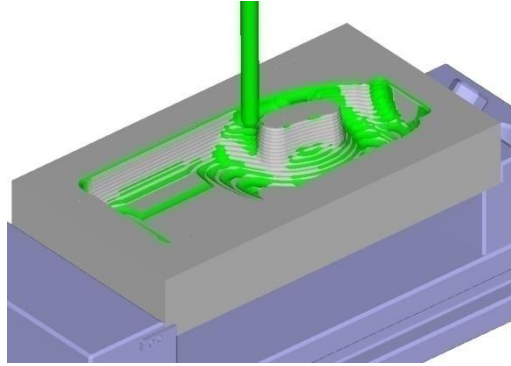
- The calculated toolpath may not look exactly like your example. This is because in the image shown the toolpath has been fine tuned by using a feature called Limiting and Analytic Analysis. These features and techniques will be explained in a later chapter.



Loading an existing workpiece in a verify session

Verify the Rest Roughing operation as follows:

- Open the **Output** form.
- Hit the **Stock** button, to open the stock form.
- Select the “Rough Offset 2D 1” stock.
- Hit **accept** to load this stock into the verify session.
- Press the “**Start Verify Button**”
- Check the box for “**Verify active operation only**”
- Pick the second operation(02 Rough Offset 2d Endmill D12/2) and pick ‘**Run simulation**’.
- Once the Verify is complete save the workpiece as before.
- **Save** the File.



Plunge Roughing

Load the file “**Mill_Bottle.Z3**” and then load the CAM Plan “**Model_Cam_plan.**”

Hide any toolpaths that are currently displayed.



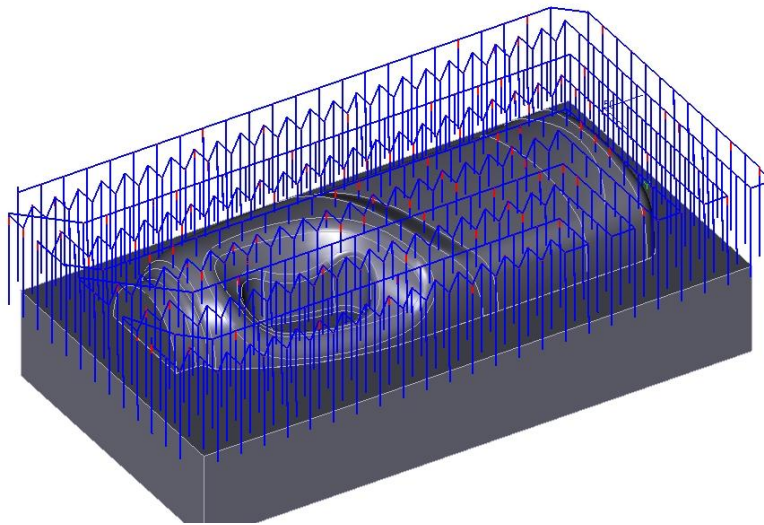
The Rough plunge cut operation is a method used to rough out parts by using a vertical drilling process. Parts requiring long tools will benefit from this operation. These can include deep cores, cavities, high shoulder slots and high straight or sloped walls.

QUICK MILL Plunge Roughing offers choices such as waterfall or single point plunge. When used with special machines and tools, the new roughing technique is reputed to offer maximum cutting productivity.

Create a QUICK MILL rough plunge operation

- Load an 18 Diameter Flat Endmill from the example tool library(Endmill D18/0).
- Load the Stock_Model (2) and the model(1) into the operation **feature**.
- Open the operation parameters form.

- Select the “**Primary**” option.
- Change the **Surface Thick** value to be 1mm.
- The first Rough Plunge operation that we will carry out will be a “Single point plunge”.
- Currently the default value of “**Stepdown Size**” is 20mm.
- Delete this value to leave the field blank. This forces the tool to cut all the way down to the maximum Z distance for each cut in the rough sequence.
- Note that the default XY Step Size is larger than the recommended maximum of 49% (of the current cutter diameter). This is fine for this type of toolpath, but use the maximum 49% rule on other roughing operation types.
- **Calculate** the toolpath

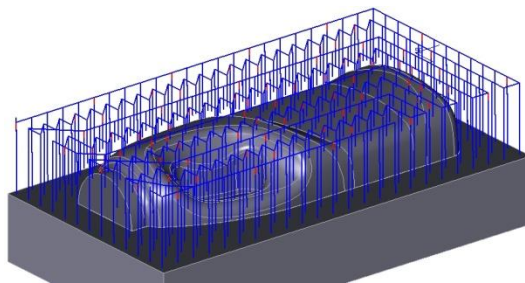


A quick look at Limiting

With the limiting capabilities built into QUICK MILL you can use a variety of options to limit what will be cut and what not to cut. For example in finishing operations you can apply angular limiting to mill only flat areas or conversely only steep areas this is easily achieve using the limiting tools. We will be discussing the Limiting capability and options in detail when we start to use QUICK MILL finishing operations. But for now let's take a quick look at using limiting to improve the previous rough plunge toolpath. The tool has cut the part correctly however we can save some time by not cutting out past the stock quite so much. In fact one cut has gone down too deeply in one corner.

- Open the Plunge Rough operation parameter form.
- Go into the “**Limiting**” tab as shown.
- The toolpath is automatically limited in the X and Y direction by a containment which is based on the “Containment Type”.
- The default containment type is “Simple Box.” This is used when the outside shape to be cut is rectangular in shape. The outside shape in this example is our stock shape however this type of containment is **always** based on the part not on the stock.
- Other Containment types are available, but we will leave those for later discussion.
- The outer rectangular shape is therefore the limit at which the centre of the tool will move to in X and Y when cutting the part. Clearly however the tool in our example has moved passed this boundary.
- Note the “%**Offset**” field, the default is 300.0. We can make this value smaller which will bring the tool centre inside the containment.

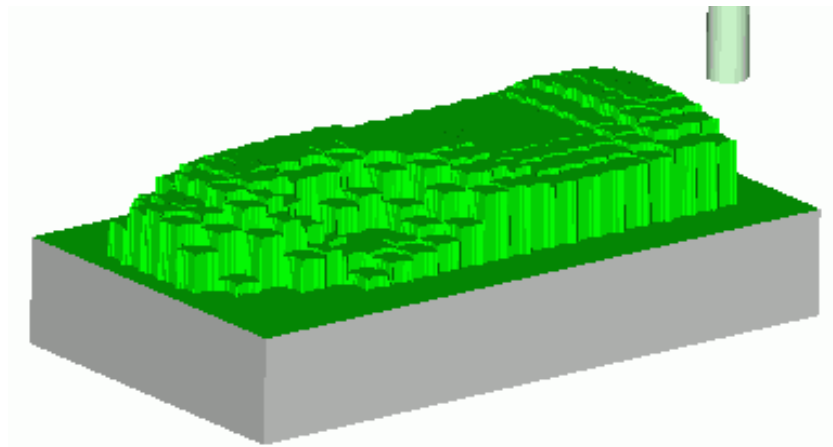
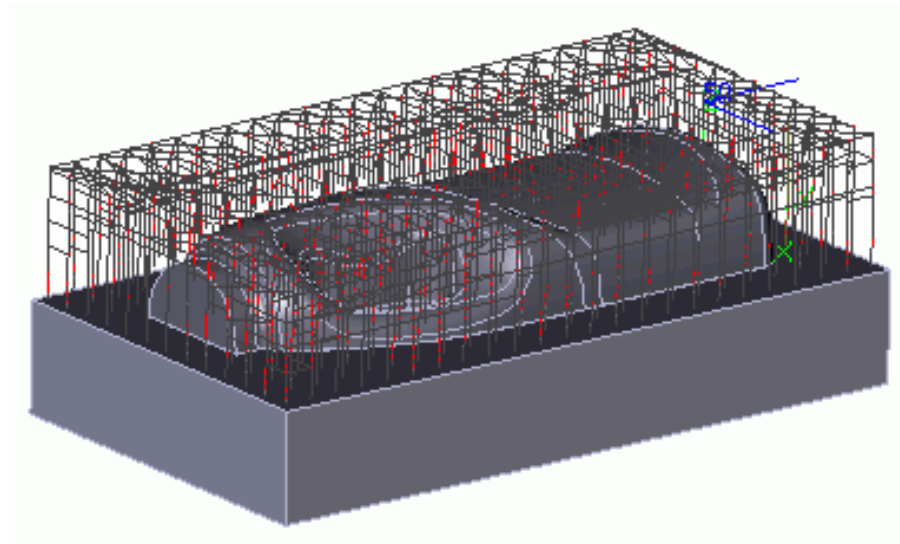
Change the % **Offset** value to 30 and recalculate the toolpath.



Rough Plunge with the waterfall option.

- Let's modify the Plunge rough operation to change from a single point plunge to a waterfall cut.
- Enter the value 10 in the **Z Step** field.
- **Calculate** the toolpath as shown.

- Previously the tool cut down to the maximum Z distance for each cut in the rough sequence.
- This time the tool is limited to cut down a maximum of 10mm at each position in the cut sequence, gradually cutting the “holes” to eventually produce the cut shape as shown.



Save the file and then clear the session.

Wave Propagation Roughing

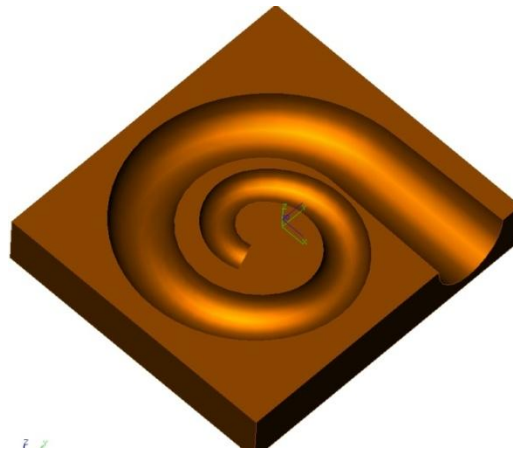
This particular type of roughing is extremely powerful, removing material in a very efficient and sophisticated way producing an almost constant load on the tool. It works by using a technique called “Wave Propagation.” Think of what happens when you

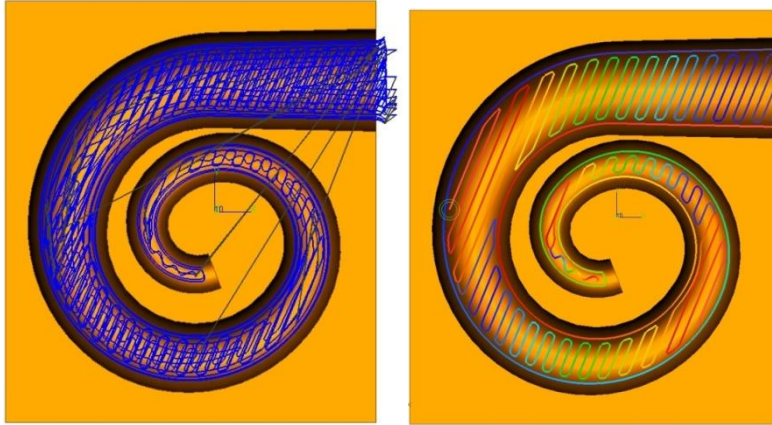
throw a small pebble into a pool of water. Small waves are formed and move out to the boundary shape of the pool irrespective of the shape of the pool. The wave roughing sequence behaves in a similar manner.

Load the file “**Wave_Propagation.Z3**” and then load the CamPlan “**Spiral_Cam_Plan**”.

First let’s cut the part using a normal Rough Lace operation.

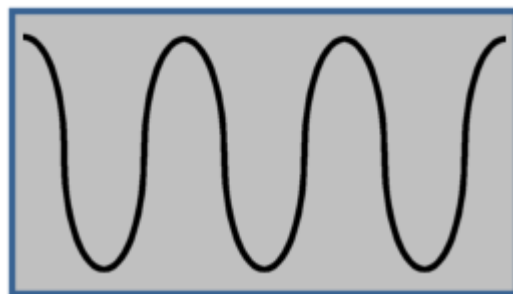
- Create a **LaceRoughing** operation. 
- Use a 16 Diameter Radius 2mm tool(Endmill D16/2).
- Set the **surface Thick** to be 1mm.
- **Calculate** the toolpath.
- Note, the image below shows just one level of the cut for clarity, also notice the color display, showing in this case the start and end of a cut span, more about this quick analysis tool later.
- The cut direction in this example is the longest span.



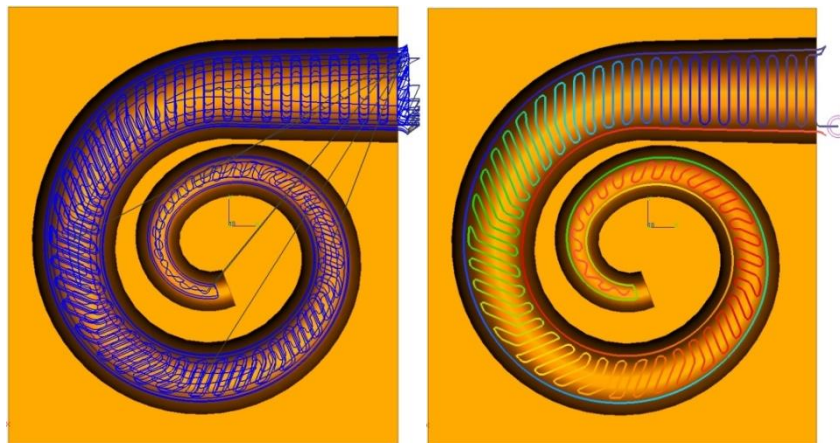


Modify this operation to cut the part with a propagating wave pattern.

- Open the **Parameters** form and select the **Advanced/Path Pattern/Waves** tab.
- Set **Enable Wave Path** to **Yes**, and Toggle the “Wave Type” to “Trochoidal”.



- Calculate the toolpath.



Save the File and clear session.

Chapter 8 Finishing HSM Lace Milling

In this chapter we will cover

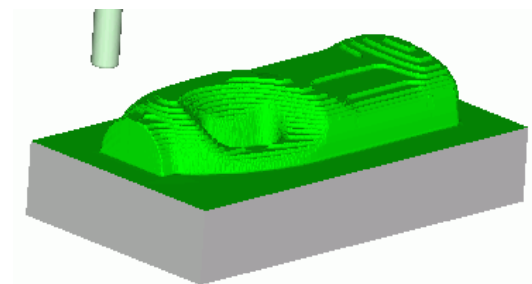
- Preparing our existing Cam Plan for finishing operations.
- Quick Mill Cache.
- HSM Lace Finishing operations.
- Cutting Parameters, Cleanup Pass, Finishing Distance and Z Corner Radius.
- Limiting Parameters.
- Cut Direction, Longest Span and Flat area detection.
- Copying existing operations.
- Lace Combo milling with steep wall detection.

Load the file “Mill_Bottle.Z3” and then load the CAM Plan “Model_Cam_plan”.

Prepare the existing Cam Plan for finishing.

- We do not require all three existing roughing operations. We will keep only the first operation “Rough Offset 2D 1.”
- We can either delete the other two operations or we can delete the other two toolpaths.
- If you wish to delete the operations Right-click on the second rough operation name and select “**Delete**” . This will delete the operation. Repeat this for the third operation.
- Or if you wish to keep the operations for reference, Right-click on the operation name and select “Delete ToolPath”, which will remove the calculated toolpath, but not the actual operation.
- Edit the remaining operation “Rough offset 2d 1” and change the Parameters/cutting form to add 3 intercut steps (# of Steps, see Chapter 5 for a description of Interlace rough milling).
- **Calculate** the toolpath.
- **Verify** the toolpath.
- Save the verified work piece, which we will load as “stock” in the subsequent finishing verify sessions.

Save the file.



QuickMillCache

Before we start with finishing operations let's discuss a powerful feature of Quick Mill.
QuickMillCache.

Much of the toolpath calculation created during initial QUICK MILL operations will be reused in further toolpath generation by QM's advanced caching control.

QUICK MILL toolpath calculation is achieved by calculating separate incremental steps. The steps are gradually more and more specialized until the final toolpath is achieved. An example of some of the incremental toolpath calculation steps are, "STL" creation, Leads and Links and so on. When a toolpath operation parameter is changed, QUICK MILL first looks for the most recent version of the cache entry that wasn't affected by the change and begins processing from there. This feature means that we don't have to calculate the same steps unnecessarily over and over again. This can reduce the time required to calculate modified toolpaths by as much as 30% to 90%. You have probably already noticed how quickly you can recalculate toolpaths when you perform an operation edit.

The cache data is not stored with the CamPlan but is instead stored in a cache folder. The default Cache folder where QUICK MILL stores the incremental steps in toolpath creation is the "temp" folder in your ZW3D user directory. The cached data will be still available to you even if you save a file or when you clear a session because the cache folder is not emptied on file save or session clear. This means that the cache folder can quickly grow in size if not managed correctly. The maximum allowed size of the cache folder is 5000MB. The QUICKMILL cache folder can be managed using the following tools.

Open up the **Tools** pull down menu.

There are three options that are used to manage the QUICK MILL cache folder.

- Clean Quickmill Cache.
- Clean QuickmillDir.
- Show QuickmillLog.

Clean QuickMillcache

Remember that QUICK MILL toolpaths are based on internally calculated STL files and that these are triangulated representations of your part. The "Clean QuickMillcache" command removes all of the QUICK MILL cache files (for all parts not just the current active one) but does **not** remove the STL files.

Clean QuickMillDir.

This command removes all of the QUICK MILL cache files **including** the STL files.

Show QuickMill Log.

This command displays an XML formatted QUICK MILL log file. The file logs all QUICK MILL activities and the log is displayed in a Windows Explorer window.

Lace Finishing Operations

Lace cuts are amongst the QUICK MILL operation types that fall into the projected toolpath category. Projected toolpaths are where the toolpath pattern is first calculated in X and Y only (2D) and then projected onto the part. The two types of lace finishing operations available in QUICK MILL are: **Lace** and **High Speed Mill Lace (HSM)**.



Projected toolpath type **Lace**.



Projected toolpath type **HSM Lace**.

There are important differences between Lace and HSM Lace that we will be discussing later, but first let's create a HSM Lace operation and check out the options that are available.

- Create a new HSM Lace operation with the following options.
- Load a 10mm Ball Endmill from the example tool library.(Ball Endmill D10)
- Add the feature "model(1)".
- Add the feature "Stock_Model(2)".

Open the HSM Lace Parameters form.

Select the "**Primary**" option.

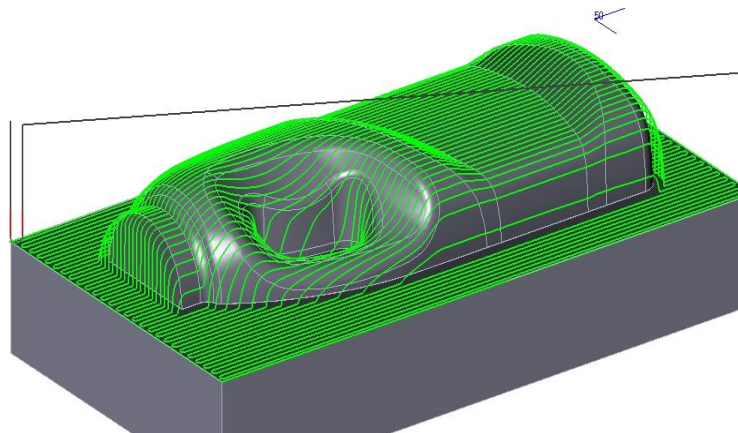
The form is similar in appearance to the roughing forms that we have used in the previous operations. There are some differences and also new options that we need to discuss.

First notice that the default value for the Path Tol is set to 0.01mm. The roughing default is 0.1mm. Clearly with finishing operations we normally need a tighter tolerance than with roughing parts. However when you calculate toolpaths with a small Path Tol value,

the toolpaths will take longer to calculate. We will be changing the Path Tol value to 0.1mm for **all** finishing examples in this tutorial.

In fact it's a good idea to do this with finishing operations on your actual parts. Start with a larger tolerance, say 0.1mm. Calculate the toolpath, and check it out. Tweak the toolpath to suit your requirements and then set the Path Tol to the desired tighter value and recalculate the toolpath.

- Set the **Path Tolerance** to 0.1mm.
- Change the **Stepover** to "Absolute".
- Change the **Stepover Size** to 3mm.
- **Calculate** the toolpath.

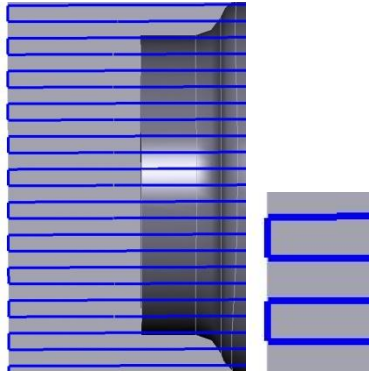


Select the "**Path Setting**" option.

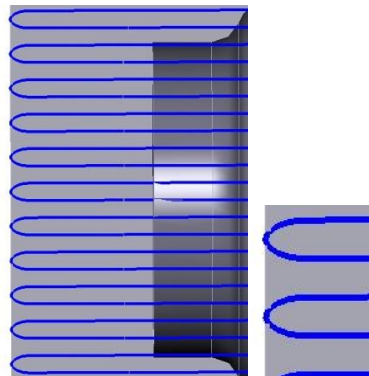
% XY Smoothing

This is used with high speed milling cycles to smooth the step over. The actual value is based on a percentage of the current stepover. Try the following % XY smoothing values and finish up with a smoothing % of 50.

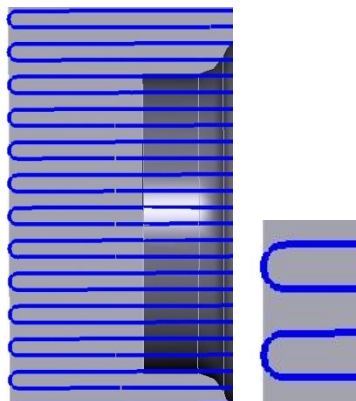
% XY smoothing set to **0**.



% XY smoothing set to **100**.



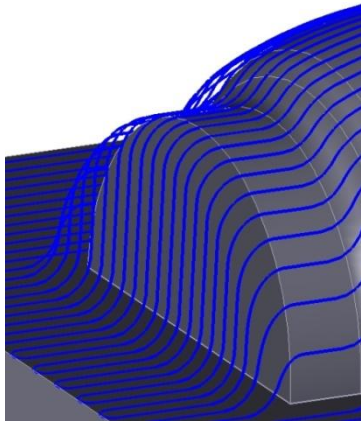
% XY smoothing set to **50**.



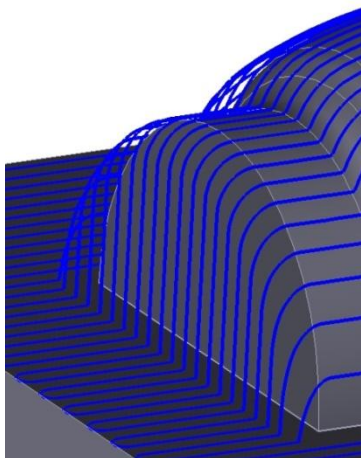
Z Corner Radius

This parameter adds a radius in the Z direction on this type of tool path. A small radius can be added to smooth out high speed toolpaths. In the following example however a relatively large radius will be added which will make it easier for us to see the result. Try the following and then set the parameter back to 0 (zero)

Z Corner Radius = 5mm.



Z Corner Radius = 0mm.



Select the “**Limiting**” option.

Limiting is used to keep the toolpath confined within some type of containment. Limiting is split into seven categories: Boundaries, Reference Tool, Check, filters. You can mix and match between different types of limiting.

Limiting Parameters :Boundaries (including XY and Z)

XY:

This limits the toolpath in XY by using containment curves. The curves are either specified by the user or are obtained automatically from existing geometry.

Type Contain: Refers to the pattern used by the containment curves. As follows:

Simple Box: This uses containment curves based on the implied bounding box of the “Part” (not the stock) geometry. Use this type if the part **is** rectangular.

Silhouettes: This type uses containment curves based the implied silhouette of the “Part” geometry. Use this type if the part **is not** rectangular in shape.

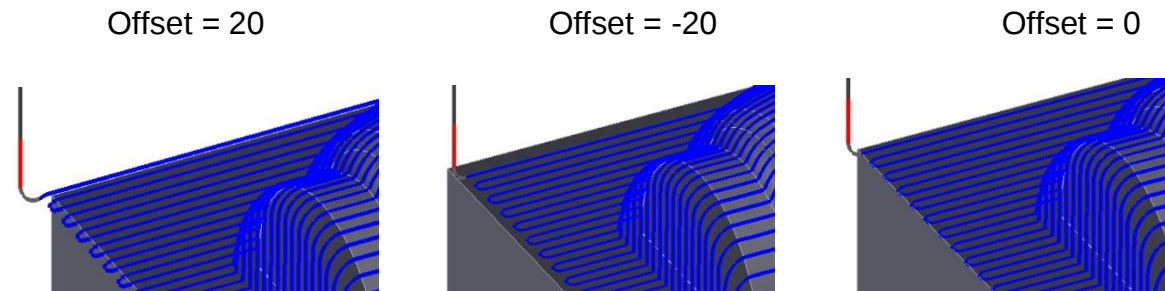
Containment: This type uses actual containment geometry that is specified by the user. You will need to select the containment geometry (Cam feature type “Contain”). This can be any class of Cam features, however cam closed profiles are recommended.

More information on creating “Containment” profiles can be found in the **HELP Manual**.

% Offset

An offset to the containment can also be applied and is based on the % of the current Tool Diameter. As shown in the following examples.

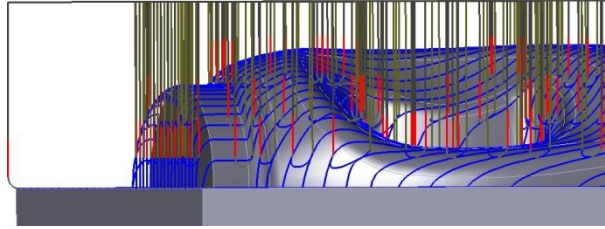
Try them out and then set the % Offset back to 0 (zero).



Z:

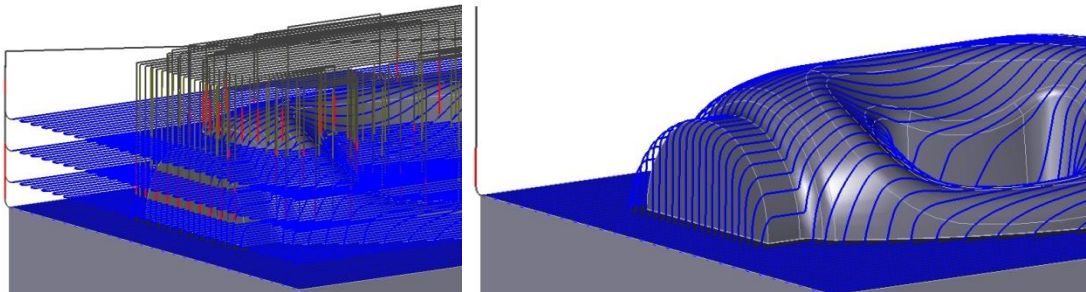
The **Top** and **Bottom** fields allow you to specify Z clipping planes that will limit the toolpath in Z. You can enter a value or select a point for each of the fields. The top and the bottom are detected automatically if the fields are left blank (no value entered).

The **Max Cut Depth** Field sets a maximum cut depth that will be applied. It forces the tool to Engage and retract in many places limiting the cut depth to whatever is set in the field as shown below. The tool retracts all the way up to the Z Clear value in this example, more about retract options later.



The **Rough Behaviour** capability takes the max depth option one step further. It creates an area clearance pattern at each Z level specified by the Max Cut Depth value. This removes material out to the current XY containment and then automatically connects, or links the toolpath at each of the Z levels. Use this option to turn a finishing operation into a combination roughing/finishing operation.

Try the options out and then remove the value from the Max Cut Depth field and set the Rough behaviour back to “no”, then re-calculate the toolpath.



Limiting Parameters: Reference Tool

Reference Tool –A reference tool is used to determine which regions could not be cut (using that tool and the current type of operation). It is used to determine regions where further machining would be required. The reference tool must exist in the current CamPlan but it is not necessary for an operation using that tool to exist.

Min Rest Height – This parameter sets the minimum height of detected rest material. The default is 0.3mm. A low value will increase sensitivity but will also increase the possibility of false detection that can occur when coarse tessellation tolerance is used.

Ref Shoulder - When set to "Ignore", tool shoulder information will not be exported/considered for tool path calculation.

When the Ref Shoulder parameter is set to **Consider (default)**, ZW3D CAM will try to determine the minimum tool hanging location (from the tip of the tool) to achieve a **gouge free tool path**. If this height is less than the tool height, a warning Message is issued. See below. This information along with the tool information is saved with the tool path.

WARNING: Minimum tool hanging height should be 100.00 to avoid gouging.

Anti Witness- This is a coefficient for fine tuning the rest finishing operations.

Convert 3D -> 2D – If this parameter is enabled, the projective cycles will be better chained (topological linking can be applied).

Rest Rough – Options Yes/No. If this parameter is enabled rest roughing will be applied providing a “**Ref Op**” (Reference Operation) is defined. The Ref Op is located in the operation section of the CamPlan manager tree. We used the Ref Op parameter on previous examples when we covered roughing earlier in this tutorial. If enabled, the reference operation is used to simulate the tool path’s affect on the stock. The results of the simulation define the “cut shape” of the work piece and then a rest finishing operation of the selected type is calculated. If disabled, a finishing operation is calculated. If Rest Rough is enabled on finishing operations then you must specify a “Stock” feature in the finishing operation.

% Anti Fragment - This specifies the anti fragmentation distance for 3D limiting.

We will be doing examples of rest machining with finishing operations later in this tutorial.

Limiting Parameters: Filters(including Filters and Angular)

Filters:

% Small Area – This will filter tiny regions in angular limiting. The areas detected during the toolpath generation that are less than the % Small Area Filter value times the total area of the surface are ignored. It is used to stop the tool from detecting and cutting really small unnecessary areas.

Angular:Angular limiting can be used to clip the toolpath with respect to the angle of the part in relation to the tool.

Angle Range – Sets the minimum clipping angle (default = 0).Sets the maximum clipping angle (default = 90).

Prev Cut Dir – This will cut only those regions that tilt between the Min angle and the Max Angle values **and** also perform additional analysis to determine and cut any uncut material left over from a previous lace cut operation. The value in this field should be the same as the Cut Dir value of the previous lace cut operation. If a previous lace cut operation is not present, this field should be left blank.

% Antiskate Offset – This value is used to offset limiting curves with a safe distance. This avoids potential skating near steep walls due grid inaccuracies.

Let's try out some of the **Angular limiting** options and also a new **Cutting parameterCut Dir.**

Cut Direction – You can specify an angular value in degrees or pick the button to select edge geometry to define a vector.

In our next example we are going to specify a cutting direction and also apply **angular limiting** to force the tool to cut only flat regions.

In the operation Parameters/Path Setting enter the following options as shown.

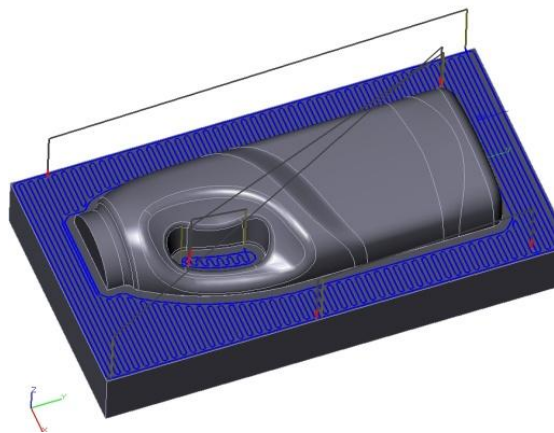
- **Add Finish Pass** = Yes.
- **% Finish Distance** = 50.
- **Cut Angle** = 0 (This is equal to the X direction).

To limit the toolpath to cut only flat regions we will apply an angular limit range using the Angle Range options in the operation Parameters/Limiting form.

- Min Angle = 0
- Max Angle = 0

Calculate the toolpath.

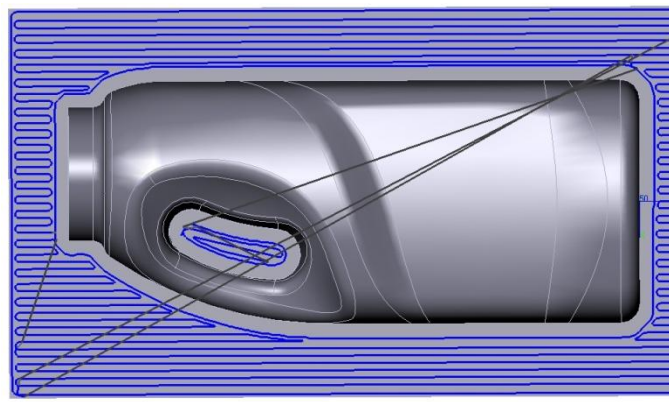
Only the “Flatlands” on the part have now been cut, because we have specified an angular limiting range of 0 to 0 degrees. Also notice the affect of specifying Add Finish Pass and % Finishing distance.



The Toolpath is also cutting the part in an X direction because we specified this in the Cut Angle field, Cut Angle = 0. This is perhaps not the best way of cutting this particular part. A more efficient toolpath would be achieved if the toolpath was in the Y direction, Cut Angle = 90. However we can utilize a very powerful feature with QUICK MILL which automatically finds the longest span when determining toolpath direction.

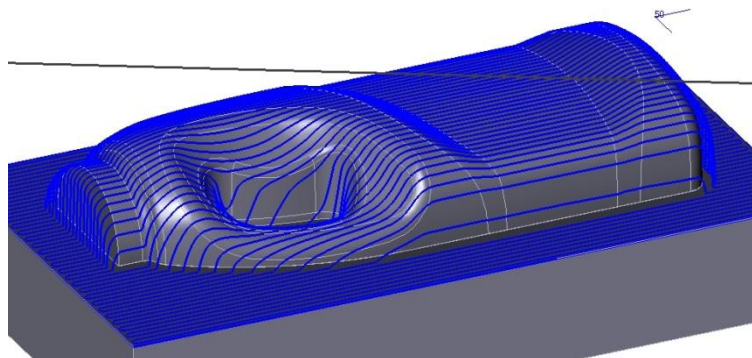
Simply leave the Cut Angle field blank and then recalculate the toolpath.

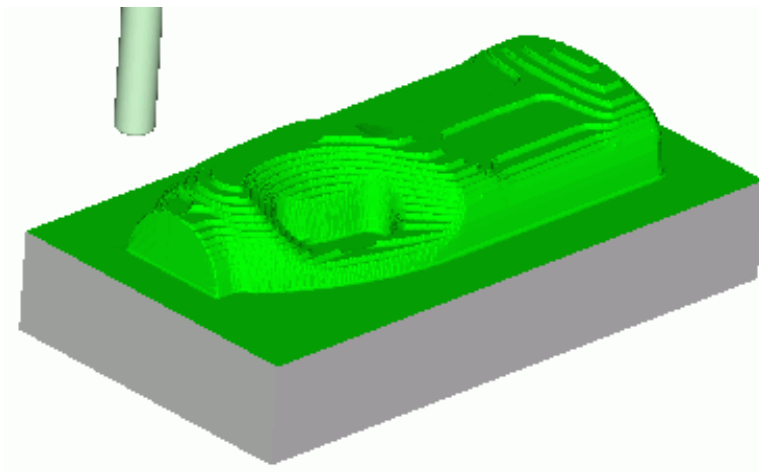
The bulk of the toolpath now cuts in the Y direction, but notice the area in the handle shape, this cuts the region utilizing the “Longest Span” feature.



Combination strategies

We can use angular limiting on a combination of different QUICK MILL toolpaths let's try this on our part, but first set the angular range Maximum and Minimum options back to their respective default values of Min = 0 and Max = 90 and change the “Cut Angle” to be blank (remove the “0”). Now recalculate the toolpath.





The HSM Lace operation cuts the part nicely. However there are regions of the part, shown clearly , where the part requires further machining. We can use a combination strategy to perform another operation cutting only those regions that require further machining.

We are going to make a copy of our existing HSM Lace operation and modify the new operation to machine the required regions on our part.

First let's save our part.

Save the Part.

Making a copy of an existing operation.

Make a copy of the existing HSM Lace operation by placing the mouse pointer over the operation name and pressing the right mouse button.

Select the **“Duplicate”** option and a new identical operation “HSM Lace 2” will be created.

Modify the new operation and change the **Cut Angle** parameter in the Path Setting to force the tool to cut in the X direction. Calculate the toolpath.

This would clean up the part nicely but there are unnecessary cuts over the top of the part where the previous operation has already cut the part successfully.

Use the Angular limiting option to limit the toolpath on our new operation to be within a specific range.

Edit the new operation as follows:

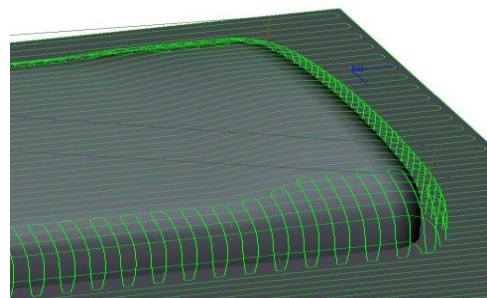
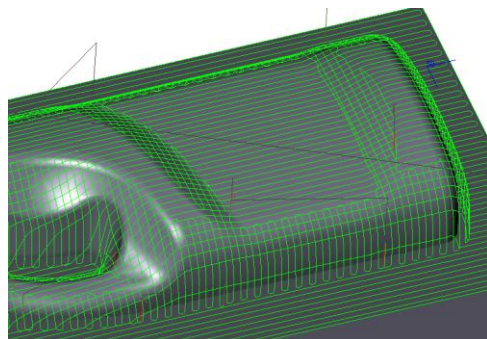
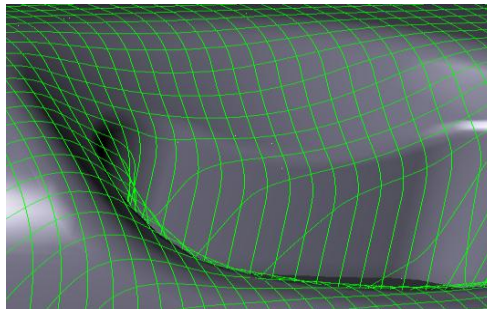
Path Setting parameters

- **Add Finish Pass** = No
- **% Finish Distance** = 0

Limiting/Filters/Angle Range parameters

- **Min Angle** = 20
- **Max Angle** = 90

Calculate the toolpath.



The image above still shows that there are areas on the part that are unnecessarily cut. The top right hand area shows this clearly.

The previous operation cut this correctly; we should not have to cut this again with our new operation. The region is within the specified angle limit range of 20 to 90 degrees so it has been cut with our new operation. We can use another limiting parameter to improve the new toolpath further.

Set the Limiting-Filters-Angular Parameter – Previous Cut Direction to 90 and recalculate the new toolpath.

Save the File.

Chapter 9 Finishing Leads and Links

In this chapter we will cover:

- Leads and Links.
- Long Links.
- Long Links types.
- Short Links.
- HSM Lace and Lace differences.
- At a Glance toolpath Analysis.
- % Short Link Limit.
- Toolpath Link Smooth Distance.
- Max Plunge Length.
- Lead In/Out using a Z Level operation.
- Short Link types.
- Forcing Lead in/out with Short Links.
- Lead In/Out types.

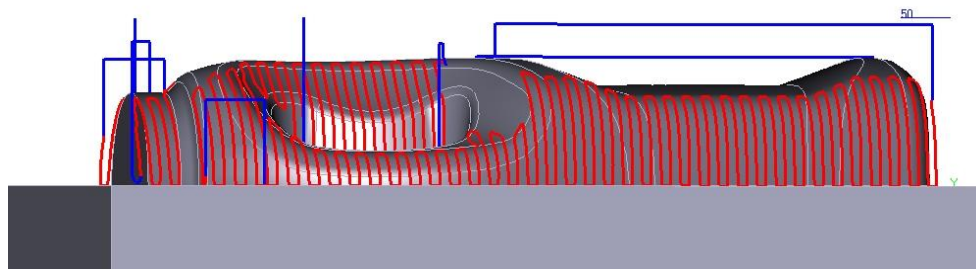
Load the file “Mill_Bottle.Z3” and then load the CAM Plan “Model_Cam_plan”

Link and Lead.

All discreet regions of any QUICK MILL toolpath are connected together by leads and links. Leads are split into two categories Lead in and Lead out. A lead in occurs when the tool moves in to engage material and a lead out occurs when the tool leaves material. The connecting move between the leads is called a link. Links themselves are also split into two categories short links and long links. Normally a long link is required when the connecting move between the leads has to move up to clear obstacles such as high walls or clamps and so on. Short links are used when the tool remains or stays close to the part when moving over to the next cutting region. Let's take a more detailed look at leads and links.

Long links

Hide all the toolpaths in the CamPlan with the exception of the second HSM Lace.

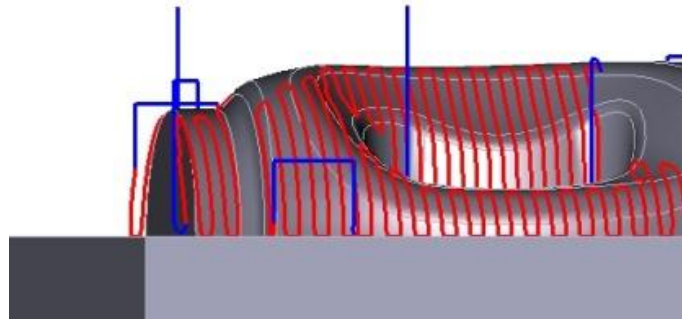


The clearance moves shown in blue in are **Long Links**. In this example the Long Link moves up to a user definable safe distance. There are other types of Long Links available to us and these are defined in the operation **Parameters/Link and Lead** form.

The **Long Link type** in that form is a **Straight, Optimized** long link. This moves the tool up to a safe distance as specified by the **Safe Distance** parameter. Change this value from 10 to 1 and calculate the toolpath.

The clearance moves are now much closer to the part.

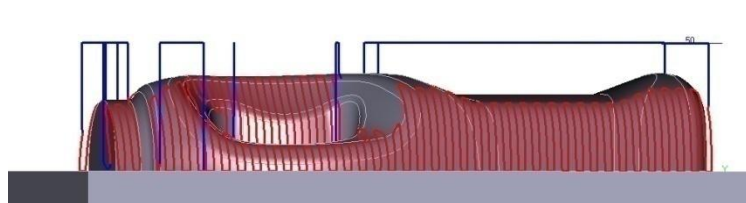
Change the Safe distance value **back** to **10mm** and recalculate the toolpath.



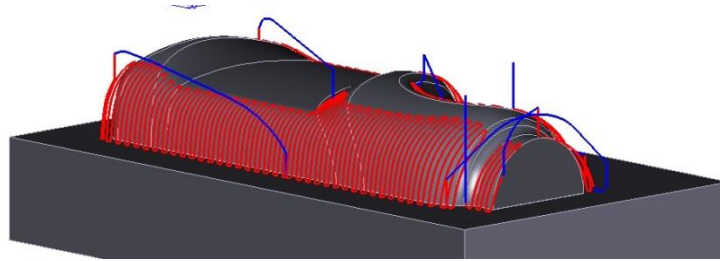
Further examples of Long links

Try the following **Long Link types** and then set the long link back to **Straight, Optimized**.

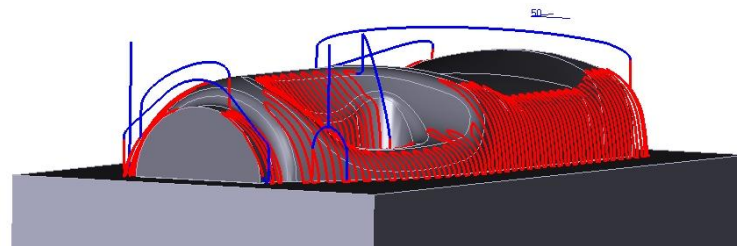
Clearance - The tool moves up to the value specified By the Clear Z parameter in the clearance form.



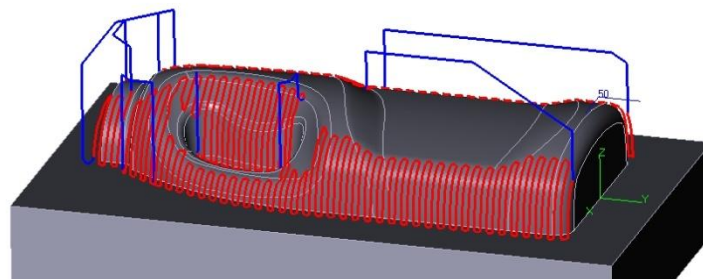
Curved Elastic Band– The tool clears all objects by the shortest distance using the safe distance value.



Spline – This is similar to Convex Hull but features a spline move. This is useful for high speed machines. All moves are smooth with no sudden change in direction.



Rapid Clearance/Optimized – These two options are used if the milling machine cannot move at rapid in all three axes at the same time. If the machine is allowed to generate the move itself then no collision checking will be done. If the machine has to move in this way, use this Long Link option to ensure that correct collision checking is performed.



Short Links

Long links are used to move the tool at rapid speed up to the clearance plane when positioning the tool to the next machining region. What are Short Links and when are they used?

Let's create a new operation type **Lace**. We will take this opportunity to learn the important differences between Lace and HSM Lace operation types.

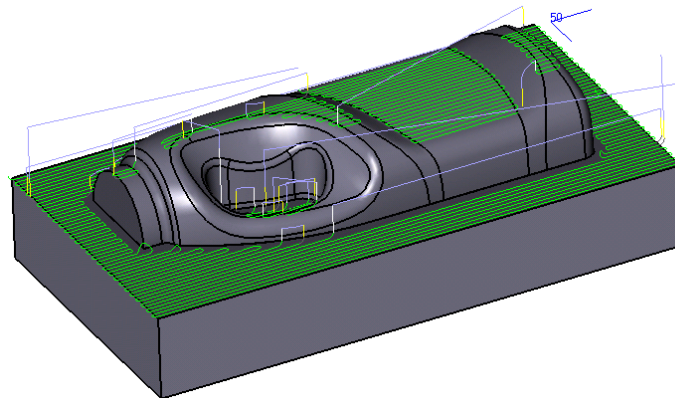
Use the following:

Lace cut **not** HSM Lace cut.



- Tool = 10mm Diameter Ball Endmill (Ball Endmill D10)
- Path Tol = 0.1
- Stepover = Absolute
- Stepover Size = 3
- Angle Range Min Angle = 0
- Angle Range Max angle =10
- Long Link Type = Curved, Optimized
- Features = Part Mode
- Features = Stock

The new Lace operation toolpath.



This Lace operation cuts the part within the Angular Limiting range of 0 to 10 degrees. The clearance moves are **Long Links**.

Initially there does not seem to be any difference between this Lace cut and a similar HSM Lace Cut. In order to more clearly explain these differences and to discuss short links we will first cover “At a Glance Toolpath Analysis.”

At a Glance Display parameters

Please refer to the **HELP Manual** and “search” for the topic “Displaying Tool Paths” for a more complete description of “Tool Path Analysis and Appearance.”

This feature allows you to control the display properties of toolpaths. Each toolpath operation contains an appearance tab that provides analysis and appearance options for individual toolpaths. You can set style, width and display modes, and many more options.

The options that will be most useful to our current discussion topic, Lace/HSM, Lace and short links are described below, but once again please refer to the online help for a more complete description.

Some of the options:

Options/Display Mode

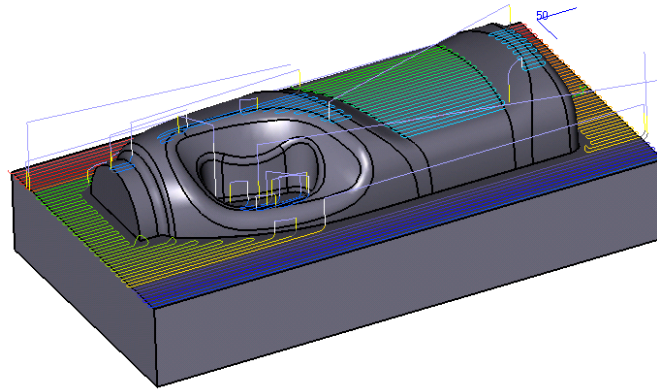
- **Standard** – The toolpath reverts to the standard ZW3D colors for toolpaths.
- **User Color** – This option allows you to define the toolpath color.
- **Distance per Span** – This mode displays each toolpath span with a color spectrum which starts with blue and ends with red.
- **Distance All** – This mode is similar the previous one but the color spectrum applies to the toolpath across the entire part, blue at the start and red at the end.
- **Z Value** – This mode is an analysis of the Z depth across the entire part. The lowest Z depth is blue and the highest is red.
- **Feedrate** – This displays an analysis of the feed rate across the entire tool path.
- **Lead In Lead out, Plunge and Link(Filters tab)** – Check each appropriate box to display the lead and links.

Options/

- **Redraw after Save** – Check this box to redraw the display after the operation form is updated (i.e. after you pick **OK**).

Set the Appearance options as follows:

- Pick **Distance all**.
- Check the **Redraw after save** box.



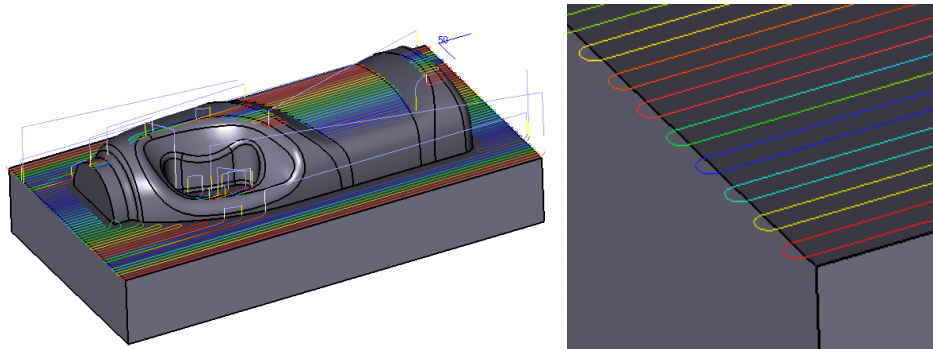
This displays a color spectrum of the entire part.

The Long Links can be clearly seen.

Notice how the part has been cut in discrete regions due to our angular limiting option. If this was a HSM lace operation a similar toolpath would be created. Remember we said earlier that Lace type operations are projected toolpaths. The Lace toolpath pattern is first calculated in XY and then projected onto the part. If this were a HSM operation each of the discrete toolpath region patterns would be calculated in XY and then this region pattern is projected onto the part. Lace is different. Each Lace toolpath span, as opposed to a toolpath region, is calculated in XY and then projected onto the part.

What's a toolpath span?

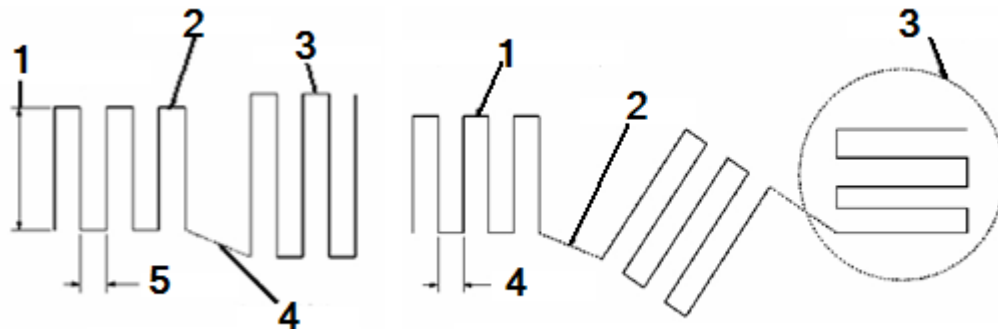
Pick the Distance per Span option from the Display form.



This also helps with the description of a toolpath span. Each individual zigzag move is a single toolpath span. Take a closer look at the end of one of the spans. Notice at the ends of each span there is an "Arc" like move. In an HSM toolpath we learned that this was achieved by XY smoothing. Each region pattern in a HSM cut is calculated first in XY and then projected onto the part. So the step over smoothing is done before the pattern is projected onto the part and is controlled by XY smoothing.

With lace each span is projected individually onto the part so no XY smoothing is possible and is not available in the Lace form. However the ends of each zigzag move have to be connected. This is achieved in a **Lace** cut with a Link. In a Lace cut the clearance moves are **Long Links** and the stepover moves are **Short Links**.

The short link in our Lace operation is a **Spline** Short Link, producing in our example an arc like move. Other Short Link types are: **Straight** and **Step**, but more about short link types later.



Lace

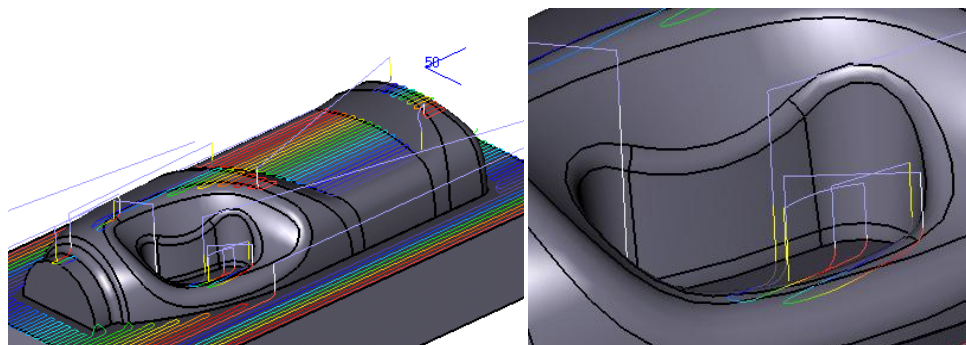
Cut - Stepovers use Short Links
HSM Lace cut - Stepovers use XY Smoothing
 (1: Single toolpath pass 2& 3: Short Link 4: Stepover 5: Long Link) (1: Step over smoothing 2: Long Link 3: Toolpath Region 4: Stepover)

Note: Because HSM cut **regions** are projected onto the part, the stopover cannot be variable.

Lace cuts are not projected as a region. Each span is projected so the stepover can be variable.

% Short Link Limit (Link and Lead parameters)

We have seen that with the Lace operation clearance moves are Long Links and stepover moves are Short Links. Short Links are not limited to Lace cuts and appear elsewhere in different operation types as we will see later. We need the ability to influence the way in which Long Links and Short Links are applied. Let's take a closer look at our current Lace example.



Looking closer we can see that there are some areas of the toolpath where a Long Link has been used when a Short Link would be a better choice. We can use a toolpath link parameter called % Short Limit to decide on whether a Short or Long Link is used.

The default value of the **% Short Link Limit** is 300%. A simple rule is used to decide on whether a Short or Long Link is used based on the % Short Link Limit parameter and the stepover.

The **% Short Link Limit** is applied to the stepover value .The current value of the stepover in our example is 3mm. The Short Link Limit is 300% so:

300% of 3mm = 9mm.

The rule applies to the distance from the end of one toolpath span (or region in HSM Lace for example) to the start of the next span. In our **current** example:

Any distance value that is **smaller** than 9 will be output as **Short Link**.

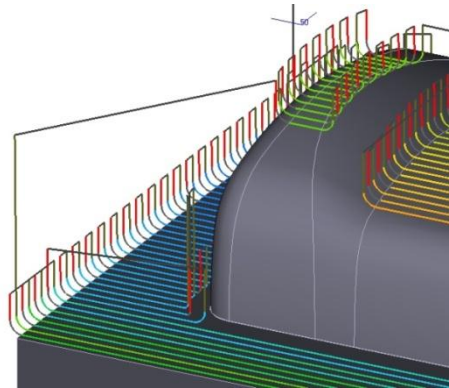
Any distance value that is **larger** than 9 will be output as **Long Link**.

The stepover in our example is 3 and outputs a short link. However in the area shown, the distance at the end of one tool path span and the beginning of the next is approximately 9, so we get long links here where short links would be clearly better suited. Let's change the % Short Link limit to force short links.

What value do you think we should use?

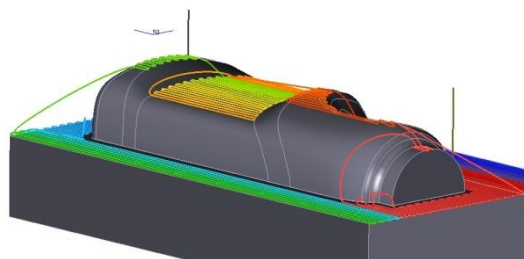
Getting the % Short Link Limit right.

- **% Short Link Limit = 0**
- 0% of the stepover of 3 = 0
- All distance moves are larger than this value so only **Long Links** are output.

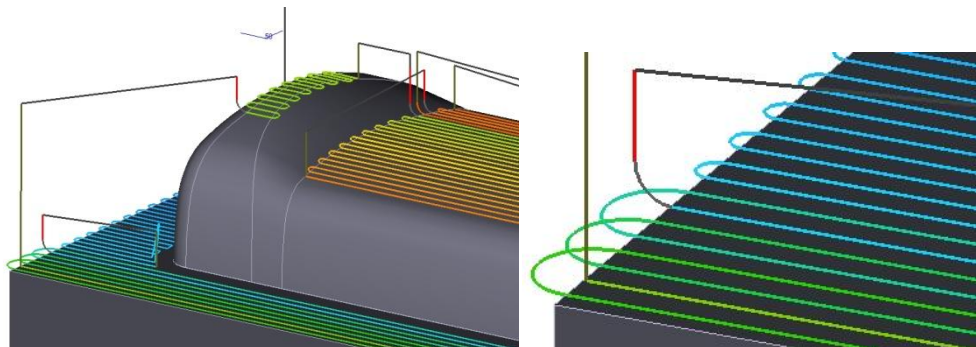


- % Short Link Limit = 10000
- 10000% of the stepover of 3 = 300

All distance moves are smaller than this value so only **short links** are output.



- % Short Link Limit = 500
- 500% of the stepover of 3 = 15
- Distance moves **larger** than 15 will be output as a **Long Link**.
- Distance moves **smaller** than 15 will be output as a **Short Link**.



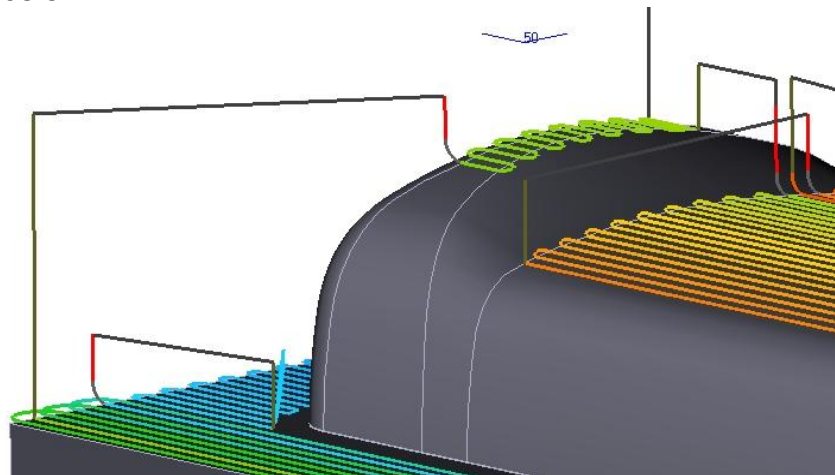
Tool path Link Smooth Distance

High speed milling machines perform better when there are no sudden changes in direction. If you take a close look at the previous example you can see that the clearance moves each have a sharp corner. We can force a smooth move at these sharp corners with the Toolpath Link “**Smooth Distance**” parameter.

Smooth Distance = 0.

Toolpath does not have a Smooth Distance.

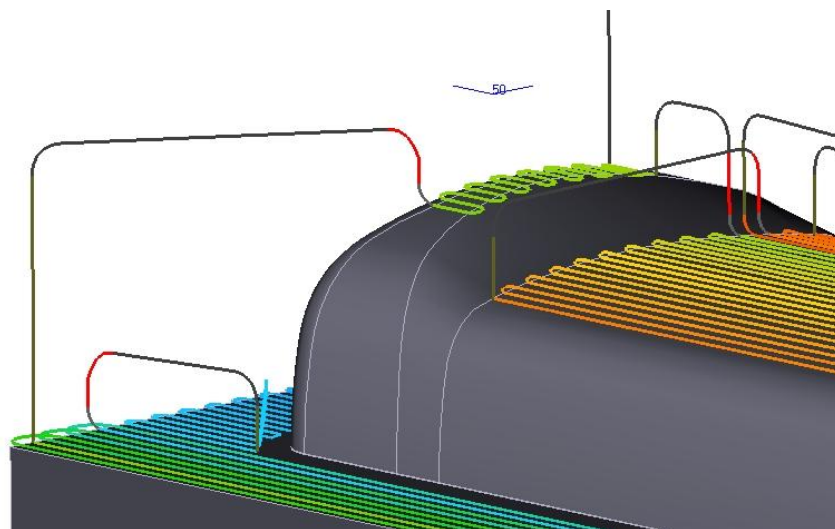
See the result below.



Smooth Distance = 5.

Toolpath has a Smooth Distance of 5mm (radius) applied.

See the result below.



Max Plunge Len

The **Max Plunge Len** option allows us to control the length of plunge moves. Let's take a look at the default option

For the following example change the "**Safe Distance**" parameter to **50mm** and "**% Short Link Limit**" to 500. We are using this large clearance move to highlight the plunge move clearly.

Now open the **Feed parameters** form and hit "**Speeds, Feeds**" button, change some of the Feed rates parameters as shown below.

- Rough = 500
- Finish = 400
- Plunge(%) = 500
- Retract(%) = rapid
- Traversal(%) = rapid

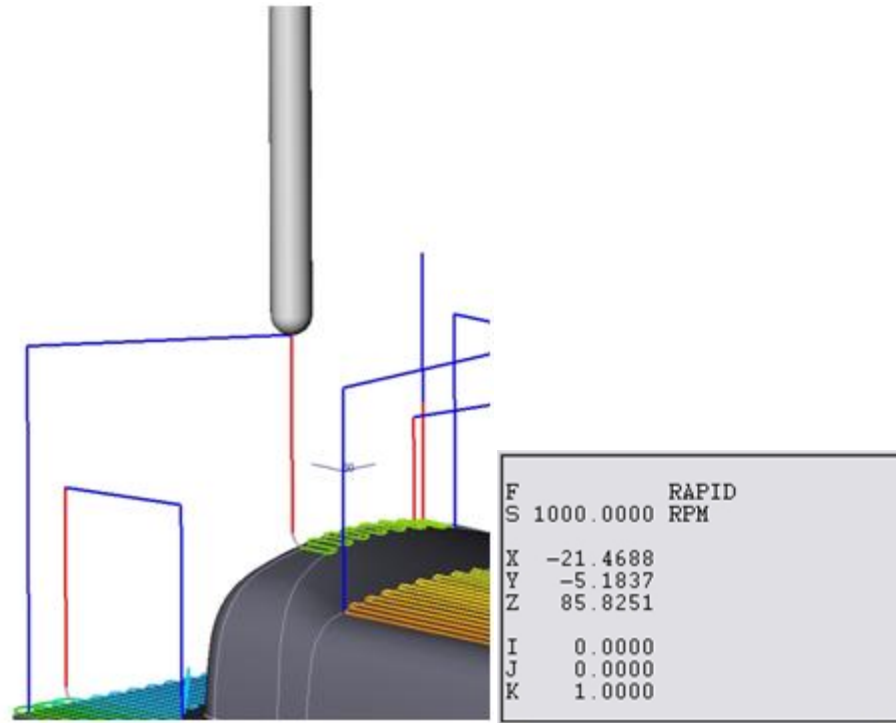
Now **calculate** the tool path.

The blue colored moves shown are rapid moves.

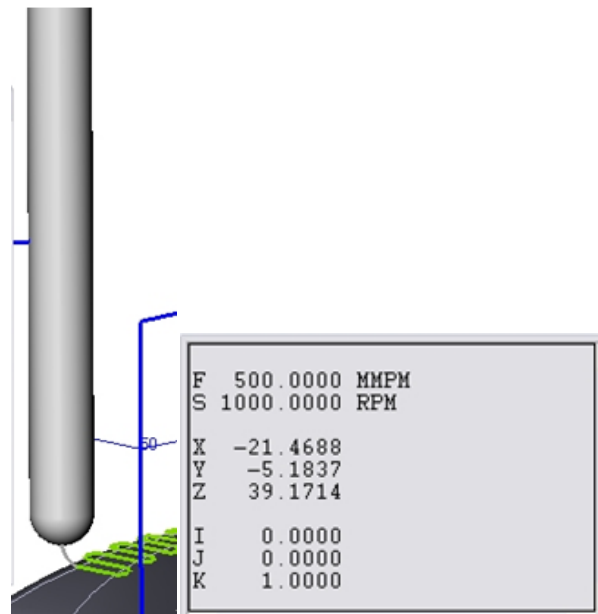
The red colored moves are plunge moves.

The default plunge moves down to the start of a "Lead in" where it will eventually start to cut material.

The plunge move feedrate in this example is 500mm, so there is a relatively large move performed at plunge feedrate.



Toolpath verify



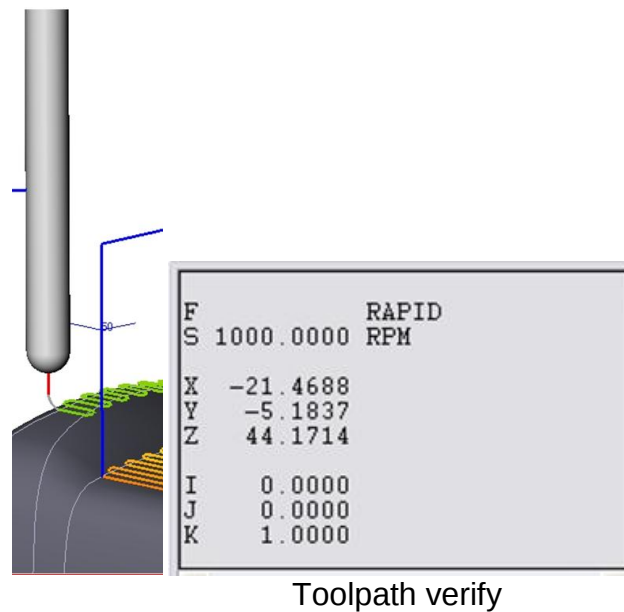
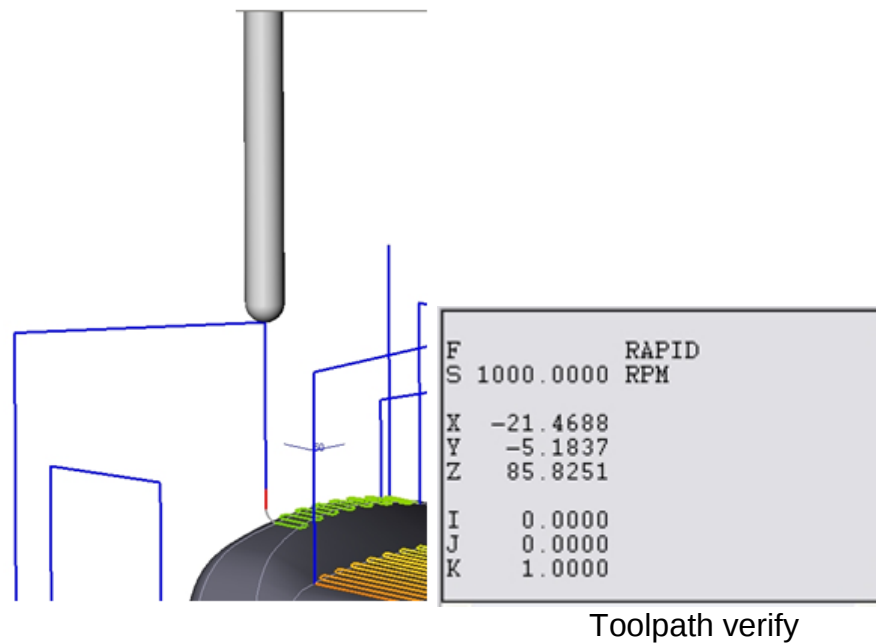
Toolpath verify

Changing the Max Plunge Length

We can change the maximum length of the plunge move, using the **Max Plunge Length** parameter.

Change the **Max Plunge Length** value to 5mm.

Re-calculate the toolpath.

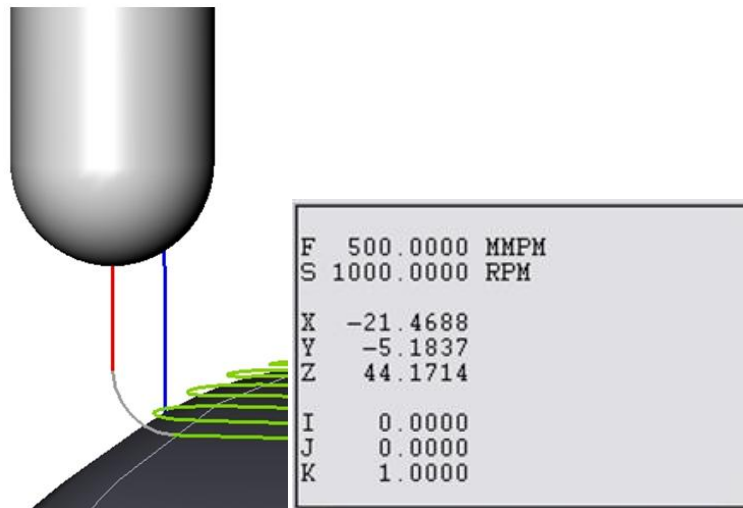


Once again rapid moves are shown in **blue** and plunge moves are shown in **red**.

The tool moves at rapid to a position above the part as before.

However, this time, see above, the tool moves down first in rapid and then shown left, down at plunge feedrate.

The length of the plunge move is the length specified by the **Max Plunge Length** parameter.



Toolpath verify

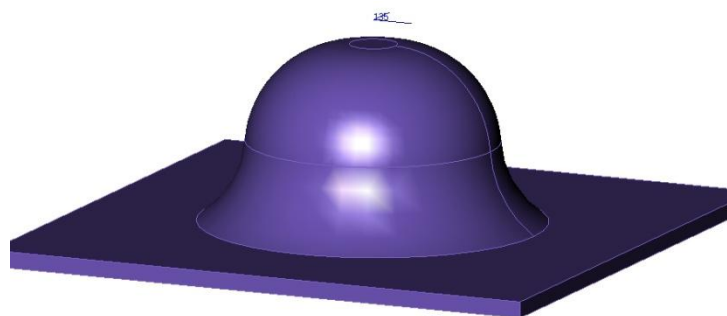
Lead in/out and the Z level operation



To more easily gain an understanding of Lead in/out we will use a new operation type called Z Level.

Save your existing file, clear session.

Load the file "Bell_Shape.Z3" and the CamPlan "Bell_Cam_Plan".

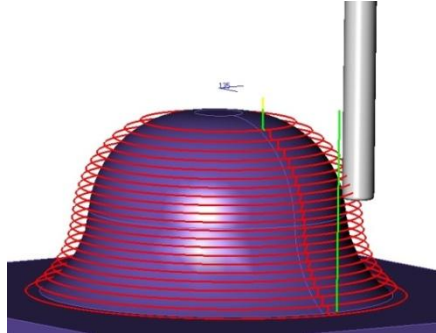


Create a new operation to the following specification.

- Use **Zlevel** operation.
- Tool = Endmill D20/ 2.
- Set **Path Tolerance** to 0.1mm in the Cutting parameters form.

- Set **Stepdown** to **Absolute** in the Cutting parameters form.
- Stepdown value = 5.
- Pick the part **Bell_Shape** shown above as **Features**.

Calculate the toolpath.



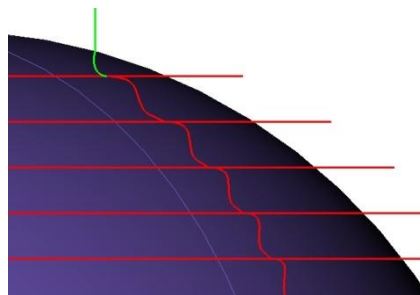
The Z level operation is **not** a projected toolpath. The tool cuts the part using constant Z steps in a side cut manner.

Short Link Types

The movement between each of the Z levels is a Short Link.

The default Short Link type is “**Spline**”

The tool moves down to the next Z level in a “Smooth” cut staying in contact with the part.

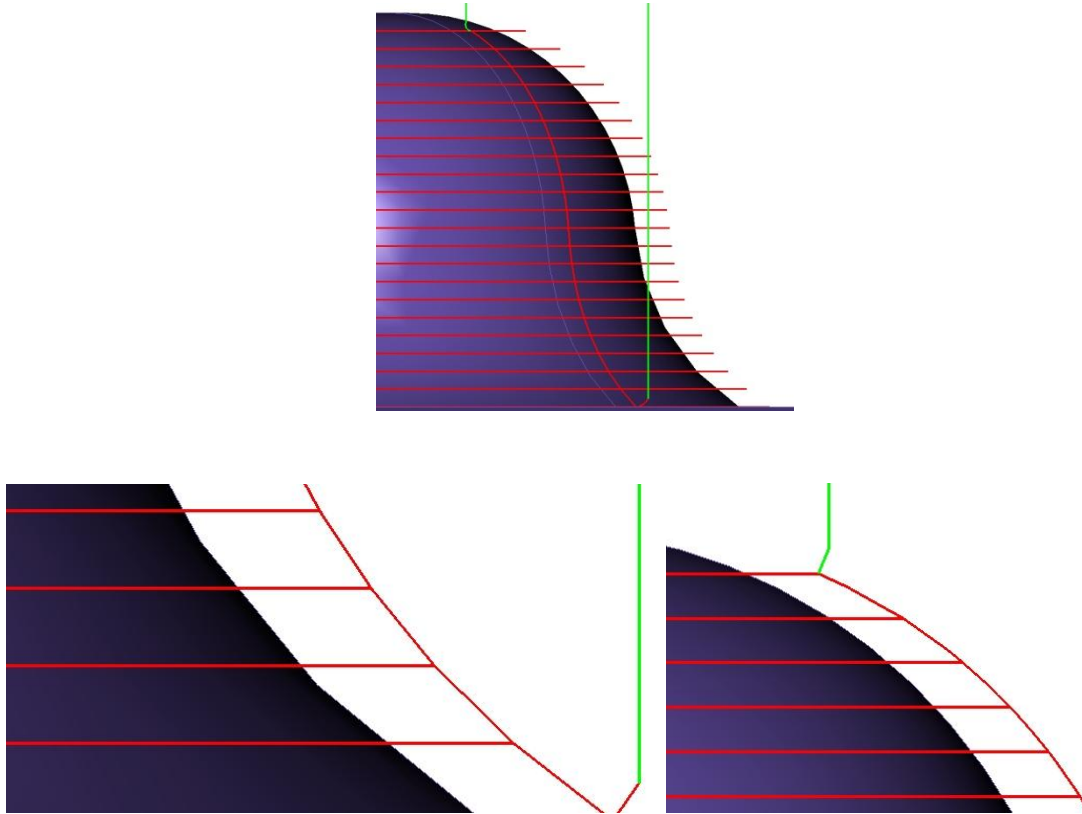


Another Short Link Type is “**Straight**”

The tool moves down to each Z level in a straight cut without gouging the part, in one of two ways:

At the lower levels of the Z level cuts, the part is concave in shape and the tool move down in a straight line as shown below left.

At the top of our bell shape the part is convex. This time the tool move is “straight” but on the surface of the part producing a curve motion which is the shortest distance between Z levels without gouging the faces. See below.



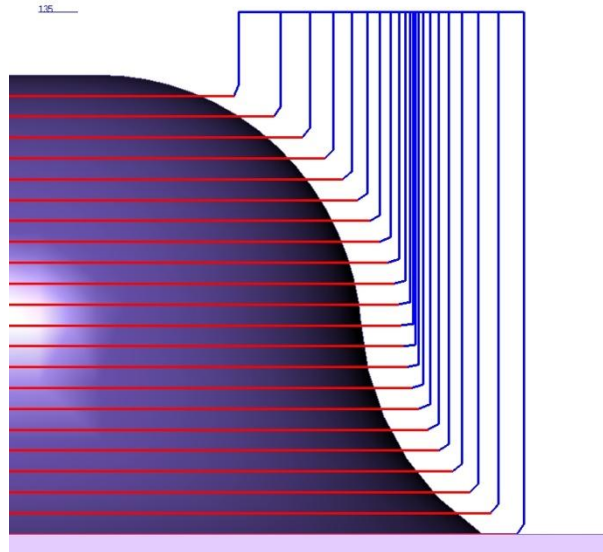
The moves down to each Z level on the previous example were cut with a Smooth Short Link.

This keeps the tool in contact with the material. This can be an advantage with some types of materials while with other types a “witness” mark can be left on the part. You can force the tool to move away from the part (**Lead out**), move to the next Z level and then move in to the part (**Lead in**) in a number of different ways.

Option 1 :clearance moves between each level by using a Lead in/out and a Long Link.(Link tab, Lead and Link parameters form)

Change the“% **Short Link Limit**” to **0** which will force Long Links.

The Long Link type shown is “Clearance” you may of course specify other types of Long Link.

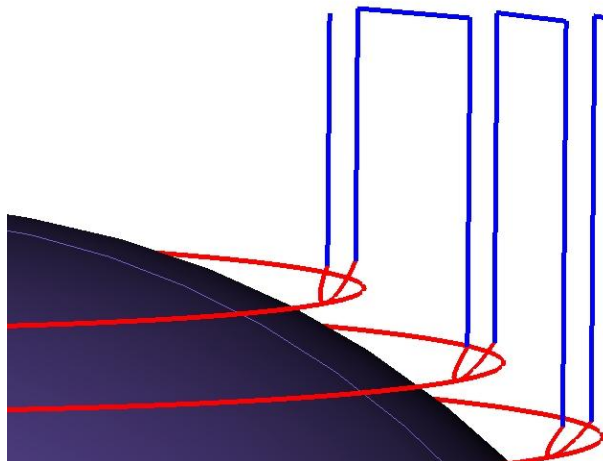


Notice as shown below there is a Lead in and Lead out at each Z level as well as the Long Link (up to clearance in this example).

The Lead in/out moves, are the small arc moves.

Forcing a Long Link at Lead in and Lead out is perhaps not the most efficient way of cutting this type of part.

What we need is a **Short Link** combined with the Lead in/out instead of a **LongLink** with Lead in and out combination.



Set the “% Short Link Limit” back to 300% and recalculate the toolpath.

Option 2 Using Lead in/out with a short Link.(Lead in/Lead out tab, Lead and Link parameters form)

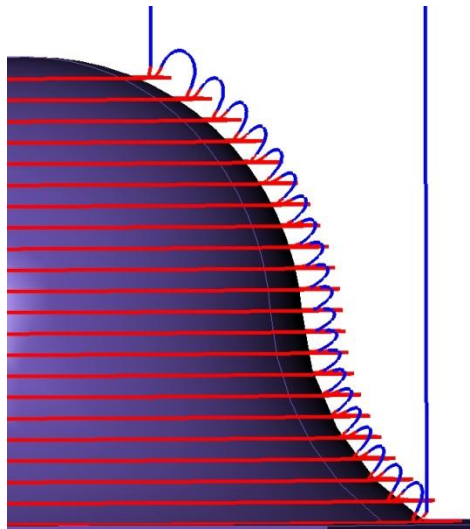
The default moves between each Z step in a Z Level cut are, as previously shown, Short Links without any lead in/out. We changed the toolpath to move away from the part between each Z step using a combination of Lead in/out and a Long Link. The next example is a much nicer way of cutting Z level toolpaths. We will use a lead in/out between Z steps as before, but this time we will use a more efficient **Short Link** instead of a **Long Link**.

The image below left shows the default Lead in/out Tabs. There is a field in the Lead In section called “After Short Link” and the default is “No”. This means that there will be no Lead In **after a Short Link**(Lead in tab), and similarly in the Lead Out section there will be no Lead Out **before a Short Link**(Lead out tab).

So by default Lead in/out is only applied with Long Links. We need Short Links **with** a Lead in/out.

Change both fields “**After Short Link**” and “**Before Short Link**” to “Yes”. This will force a Lead in/out with Short Links.

Calculate the toolpath



The toolpath shown has a Lead in/out and a Smooth Short Link between each Z Step.

Lead In/Out Types

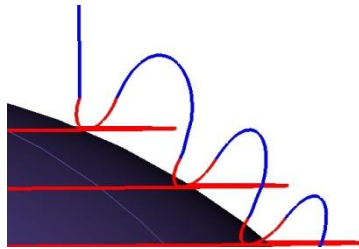
Types Lead In/Out options are:

- **Normal** - Produces 3-D Lead in/out moves perpendicular to the part.
- **Vertical** - Produces a vertical Lead in/out.

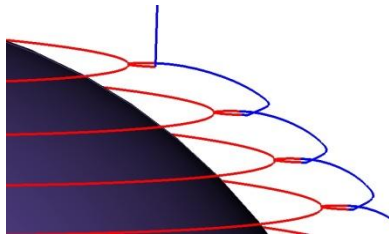
- **Horizontal** - Produces a horizontal Lead in/out move, particularly useful for Z level toolpaths.

Try the following options:

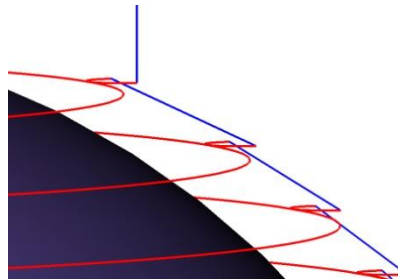
Lead in/out is set to **Normal**, the moves in and out of the part are normal to the faces.



Lead in/out is set to **Horizontal**, the moves in and out of the part are Parallel to the cutting plane.



Lead in/out is set to **Horizontal** and the Short Link Type is set to **Straight**.



For more information on Lead In/Out options, please refer to the **HELP Manual**.

Chapter 10 More Finishing Options

In this chapter we will cover,

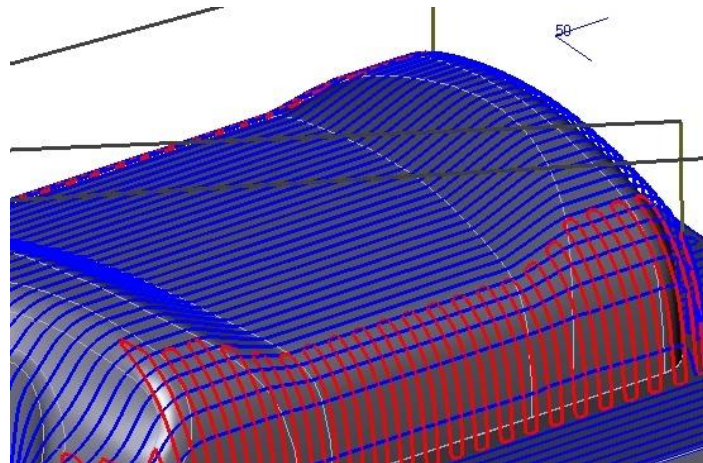
- Combination milling strategies (Combos).
- Offset 2D.
- Offset 3D.
- Z Level with flat place detection.
- Adding Uniform Cuts to produce constant scallop height and constant tool load.
- Tool path flow Climb, ZigZag, Top to Bottom, etc.
- Converting “Classic” milling tool paths to High Speed tool paths with Spline Linking.

Combination milling

Load the file “Mill_Bottle.Z3” and then load the CAM Plan “Model_Cam_plan”.

Combination milling strategies (Combos) enable you to combine different milling strategies to achieve the optimum cutting efficiency. You can fine tune the milling process by combining multiple cutting technologies into one unified solution.

We have already used milling combos (see Chapter 7) with the HSM Lace cut as shown below.

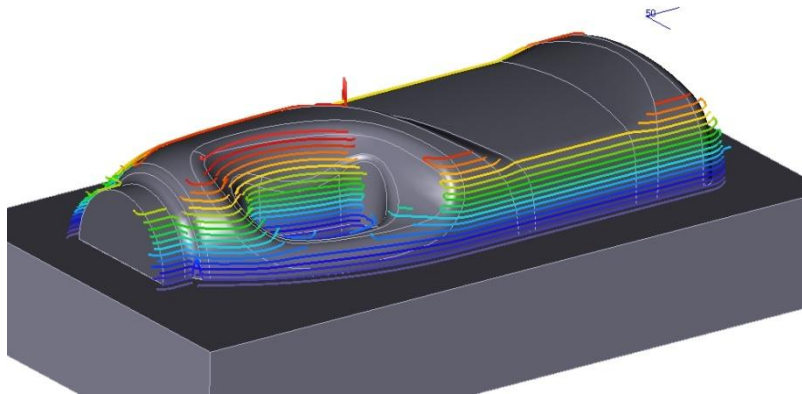


This image shows a combo toolpath based on two HSM Lace operations. Let's try a different combo based on HSM Lace and Z level.

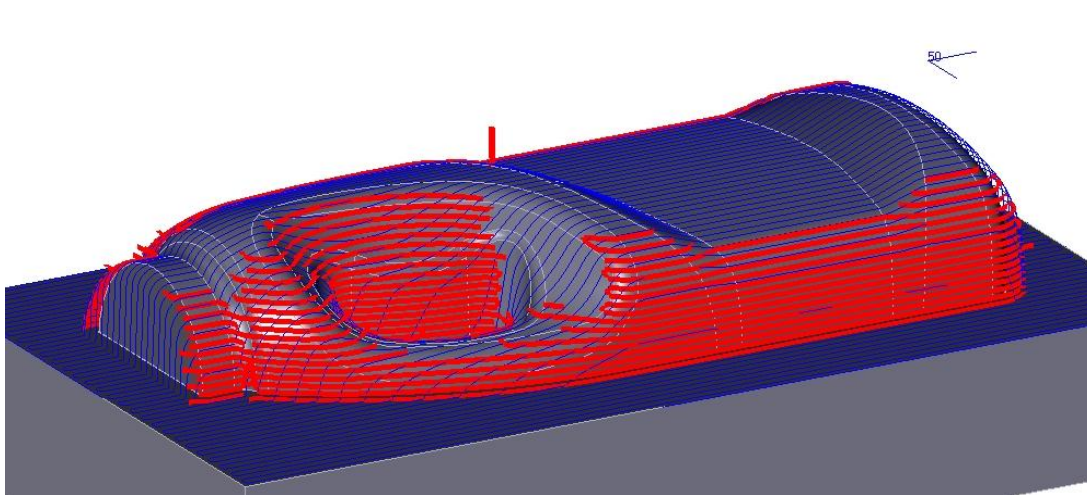
Create a Z level operation to cut the part in a similar way to the Red HSM Lace cut shown above. Use a 10mm Ball end tool with **2 mm** step size producing the toolpath shown below.

The Leads and Links have been removed for clarity.

Try this exercise on your own. The operation parameters are shown on the next page.



HSM Lace (blue) Z Level (red) Combo



The parameters used to create the Combo Z Level cut shown here are shown below.

The Primary parameters.

- **Path Tolerance** = 0.1.
- **Stepdown**= Absolute.
- **Stepdown Size** = 2.

The Limiting parameters/Filters/Angular

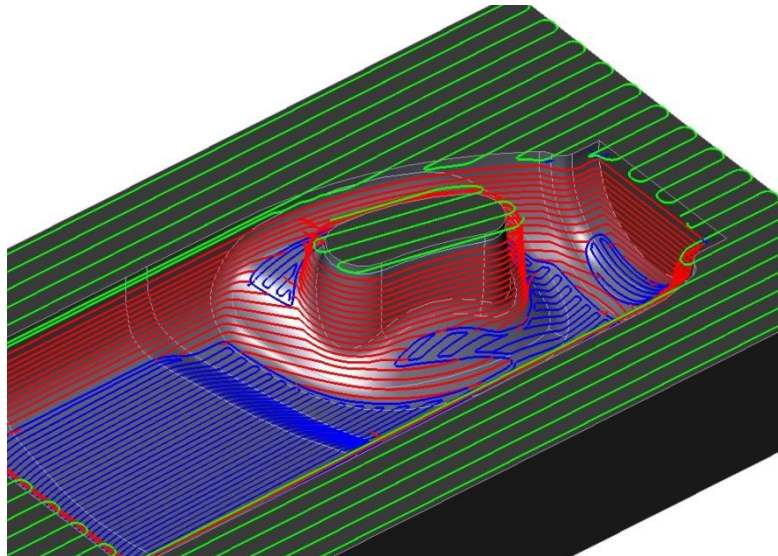
- **Min Angle** = 20.
- **Max Angle** = 90.
- **Prev Cut Dir.** = 90.

Save the file.

Load the CAM Plan “Blowmold_Cam_plan”.

More Combo Milling Strategies

In the next exercise we will combine three separate operations to produce the toolpaths shown below.



Create three separate operations as shown right, as follows:

The green toolpath.

- HSM Lace operation .
- Tool 12mm diameter square corner End Mill(Endmill D12/0).
- **Stepover** = Absolute.
- **Stepover size** = 6.

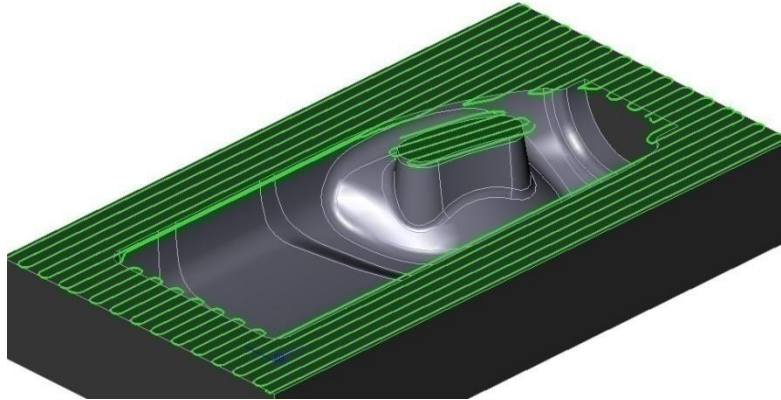
The red toolpath.

- Z Level operation. .
- Tool 10mm diameter ball End Mill(Ball Endmill D10).
- **Stepdown** = Absolute.
- **Stepdown size** = 2.

The blue toolpath.

- HSM Lace operation .
- Tool 10mm diameter ball End Mill.(Ball Endmill D10)
- Stepover = Absolute(Cutting parameters form).
- Stepover size = 2.

The **HSM Lace, Z Level, HSM Lace Combo** operation parameters used in the above operations is shown in the next pages.



The green tool path mills the flat areas on the top of the part. This is achieved by using angular limiting. Notice the toolpath automatically calculates the longest span.

Primary parameters:

- **Path Tolerance** = 0.1
- **Stepover** = Absolute.
- **Stepover size** = 6

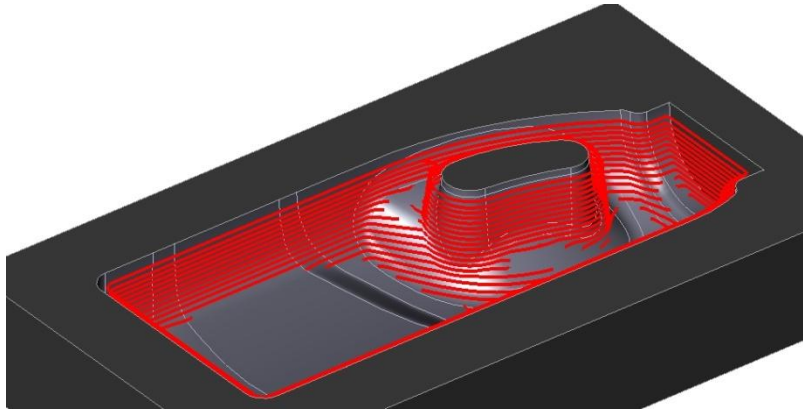
Path Setting parameters:

- **Add Finish Pass** = No
- **% Finish Distance** = 0
- **Cut Angle** = "Blank" Longest Span

Limiting parameters/Filters/Angular:

- **Min Angle** = 0
- **Max Angle** = 0

The next toolpath (red) is a Z level side cut operation which also uses angular limiting to mill the steep regions.



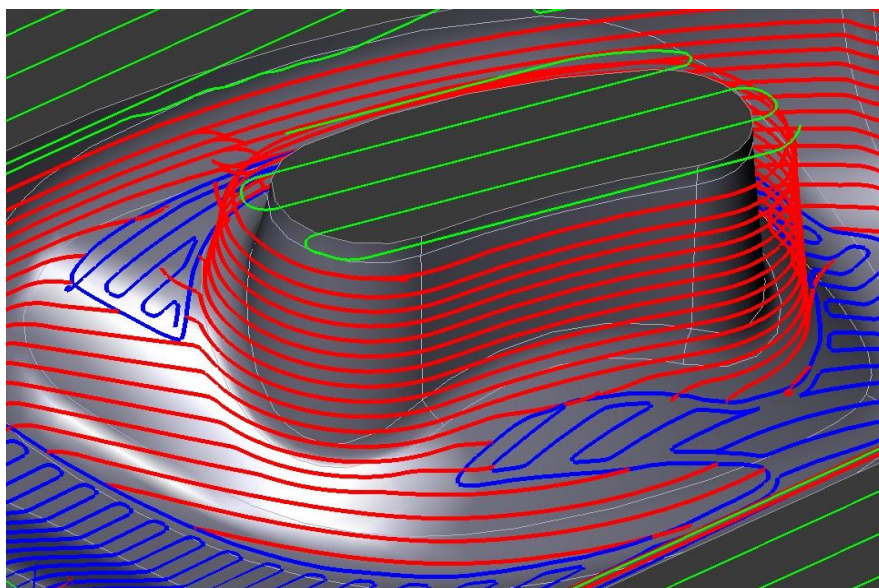
Primary Parameters:

- **Path Tolerance** = 0.1
- **Stepdown** = Absolute.
- **Stepdown Size** = 2.

Limiting parameters/Filters/Angular:

- **Min Angle** = 45.
- **Max Angle** = 90.

The last toolpath (blue) is a longest span HSM Lace cut also using angular limiting, this time to mill the shallow areas. Note the clean up pass on each region and also how nicely the toolpath cuts all the regions not cut by the Z level operation.



Primary parameters:

- **Path Tolerance** = 0.1
- **Stepover Type** = Absolute.
- **Stepover Size** = 2.

Path Setting parameters:

- **Add Finish Pass** = Yes.
- **% Finish Distance** = 50

Z Limiting, to stop the tool milling the top planar faces of the part.

- **Top** = 44 (the top of the part is 45mm)

Limiting parameters/Filters/Angular:

- **Min Angle** = 0
- **Max Angle** = 45

HSM Offset 2D and Offset 3D finishing operations

HSM Offset 2D and Offset 3D toolpaths use a similar spiral type pattern to finish the part. However there are two very important differences between the two types. HSM Offset 2d is a projected toolpath so the pattern is calculated first and then projected onto the part. This toolpath is fast and is particularly well-matched to parts that have shallow features. Parts that have steep features are better suited to the Offset 3D operation. Offset 3D takes longer to calculate but it is a full 3D toolpath and is not projected onto the part. With Offset 3D the user can mill the part with a constant cusp, or scallop height, to produce an even finish and uniform tool loading as the step over can be variable.

HSM Offset 2D

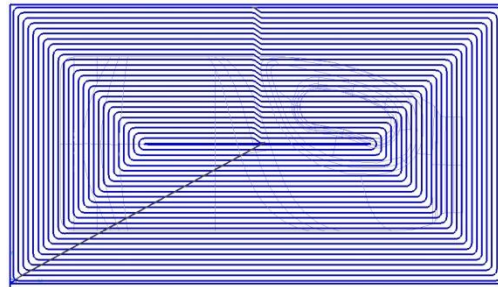


Take this opportunity to save your part and clear the session.

Now re-load the file “Mill_Bottle.Z3” and then load the CAM Plan “Blowmold_Cam_plan”.

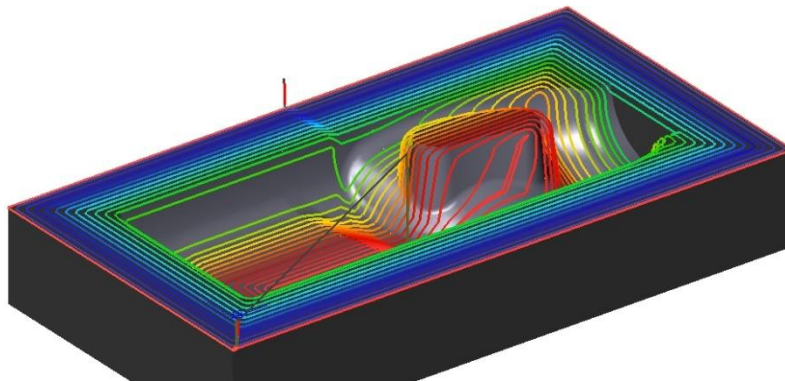
Hide all the current toolpaths. Next create a new HSM Offset 2D operation, create the toolpath to the following specifications.

- Tool = 10mm Ball End(Ball Endmill D10).
- **Path Tolerance**=0.1mm
- **Stepover** = Absolute
- **Stepover Size** = 3mm
- **Path Setting/Path Pattern** = From Boundaries.



Below left shows the calculated HSM Offset 2D toolpath from above. The spiral pattern is clearly seen and the Stepover of 3mm is constant when viewed in this orientation.

However the actual toolpath does not have a constant step over in XY and Z
Below shows the same toolpath viewed at an angle. This shows that the spiral type toolpath is projected onto the part and that the steep regions are not finished.

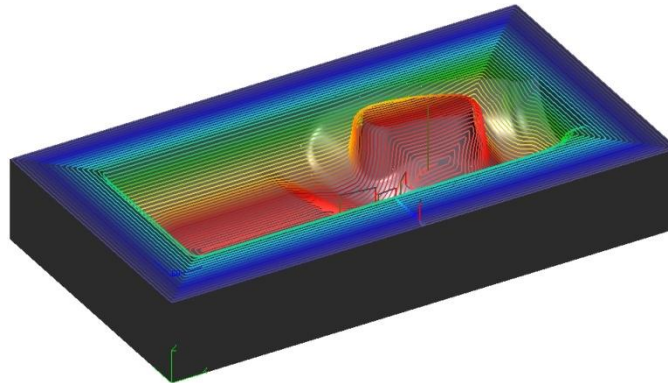


Offset 3D

Let's try the Offset 3D operation. Create and calculate the toolpath for an Offset 3D operation to the following specifications.

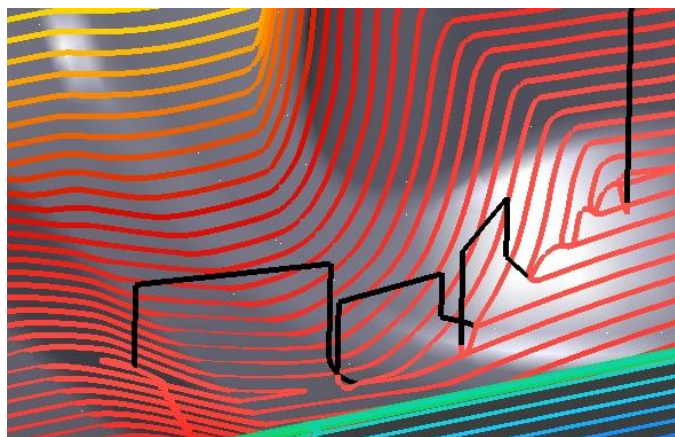
- Tool =10mm Ball End(Ball Endmill D10).

- Path Tolerance = 0.1
- Stepover = Scallop
- Scallop Size= 0.1.
- Path Setting/Path Pattern = From Boundaries.



The Offset 3D cut shows a constant scallop height toolpath machined on the detailed undulating surfaces of the part. This type of cut also produces a uniform tool loading.

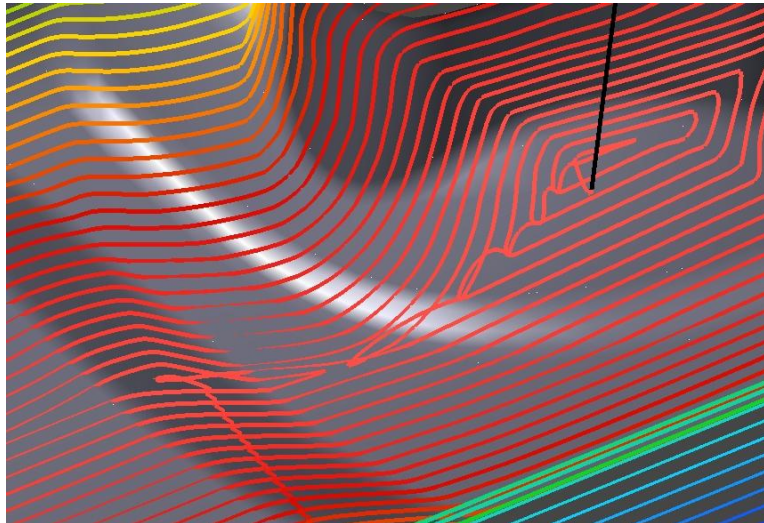
Notice the optimised clearance moves around the “Boss” region, shown on the part in black. If we want the tool to stay in contact with the part instead of producing these clearance moves, how could we accomplish this?



Did you get it? The answer's in the linking.

The black moves in the previous image are Long Links. We need to modify them to Short Links as shown below.

Change the “% **Short Link Limit**” to “2000” to force short links as shown below.

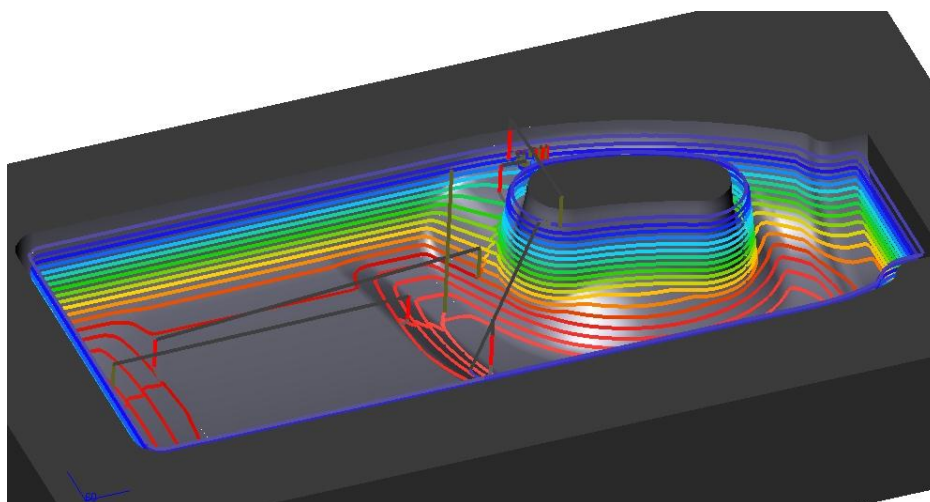


Z Level with Flat Place Detection



This type of operation offers an efficient toolpath with excellent continuity and is especially useful for parts that have long shallow features. Start by creating a Z level operation with the following specification.

- Tool = 10mm Ball End(Ball Endmill D10).
- Path Tolerance = 0.1.
- Stepdown Type = Scallop.
- Scallop Size = 0.1.



The Z Level cut shown above has a cusp height option applied, but this type of operation is unable to cut all the part with a cusp height option **and** keep a constant Z step. Z level operations have a constant Z step and the flat areas on the part have not been machined. There is however an option available which will allow us to add additional cuts to certain operations, including our current Z Level operation. This is achieved with the “**Add Uniform Cuts**” option. The option is shown on the next page.

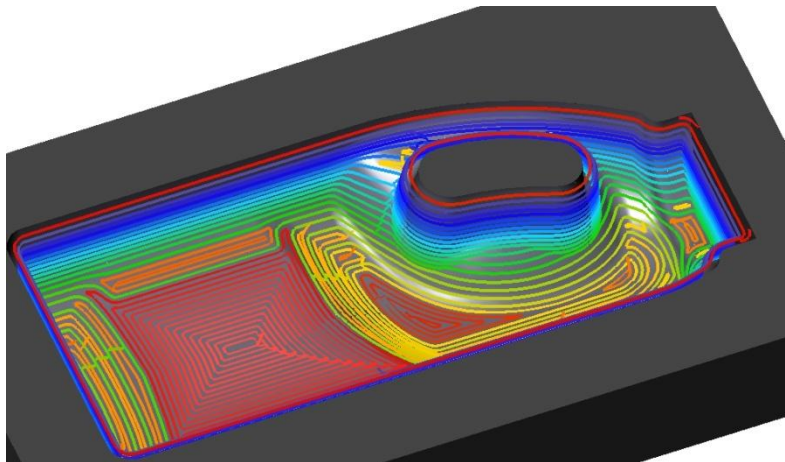
Adding Uniform Cuts.

Set the Z Level operation **Advanced/Path Pattern parameters** form.

- **Add Uniform Cuts** = Yes.

Set the Z Level operation Limiting Parameters form.

- Z Limiting option **Top** = 44.9(Just below the top face of the part to avoid cutting the top flat faces).
- The finished toolpath with the additional uniform cuts is shown below, please note the leads and links have been removed from the image for clarity.



Save the Part.

Parallel Lace – Top to Bottom (or Bottom to Top)

Load the file “**Mill_Bottle.Z3**” and then load the CAM Plan “**Model_Cam_plan**”.

Hide all the toolpaths.

You have complete control of tool path direction and flow (Climb, Conventional, ZigZag, Top to Bottom, etc.) The following example highlights the automatic Top to Bottom Lace milling functionality.

Create and calculate a new **Lace** operation to the following specification.



Tool = 10mm Ball End(Ball Endmill D10).

Primary parameters:

- **Path Tolerance** =0.1.
- **Stepover Type** = Absolute.
- **Stepover Size** = 8.

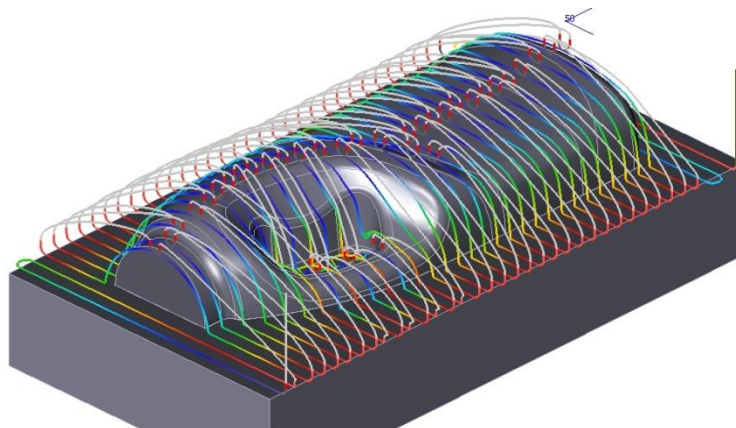
Path Setting parameters:

- **Cut Angle** = 0 (Forces cuts to be in X).
- **Cutting Direction** = Top to Bottom.

Link and Lead Parameters:

- **Long Link Type** = Spline.
- **Safe Distance** = 3

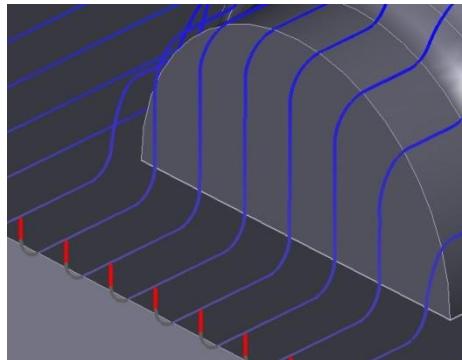
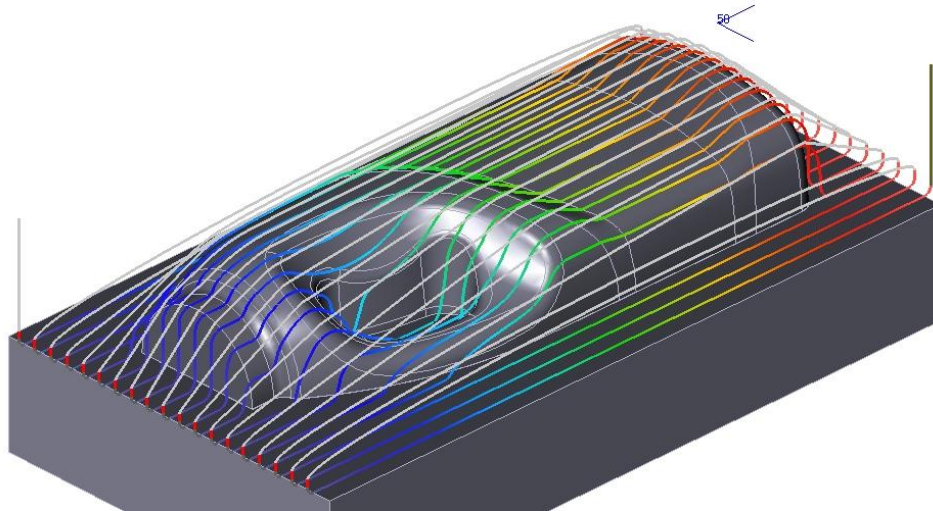
The Display parameter **Distance per Span** has been used to show how the tool moves from Top to Bottom at each span (Blue to Red)



Spline Linking and Z corner radius.

The last example used Spline Linking to convert the “Classic” Lace cycle to a high speed cycle. In fact any classic type milling cycle can be converted into a high speed tool path in this way.

Change the last Lace operation to cut the part in the Y direction and with cutting “Flow” to be Climb. The operation parameters that you need to change are shown on the bottom of this page, but try it out yourself first. Note the addition of 4mm Z corner radii (the Spline Links have been removed from this figure for clarity).



Change the Lace operation cutting parameters as follows.

- **Cut Angle** = 90 (cuts in Y direction).
- **Cutting Direction** = Climb.
- **Z Corner Radius** = 4.

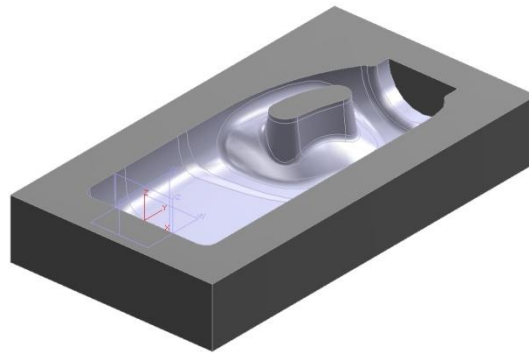
Save the File.

Chapter 11 Special Finishing Methods

In this chapter we will cover,

- Rest Finishing Using a Reference tool.
- Z Level Rest Finishing.
- Pencil Cut Milling
- Combo Pencil Cut rest milling.
- Progressive tool holder.

Rest Finishing using a reference tool.



Load the file “Mill_Bottle.Z3” and then load the CAM Plan “Blowmold_Cam_plan”.

Hide all the current toolpaths.

QUICK MILL has the ability to create toolpaths that efficiently remove material left by a previous cutting tool. This is achieved by using a “Reference Tool”.

The reference tool is used to determine which regions could not be cut by using that tool, in other words it is used to determine where further machining would be required. For our next exercise let's assume that our part has been machined with a 30mm Ball Endmill. Clearly this size of tool will not be able to cut all of the part and there will be regions where further machining will be needed. These regions we will cut with a 6mm Ball Endmill.

It is not necessary for the 30mm Ball End mill operation to exist in the Cam Plan, but it is necessary for the tool to exist in the Cam Plan. The example tool library supplied does not contain a 30mm Ball so let's create one add it to the Cam Plan and also export it to our example tool library.

- Place the mouse pointer in the graphics area and Right-click.
- Select the **Tool Manager** option from the menu.

- Create the “Ball Endmill D30” tool(Name= Ball Endmill D30, Cutter Dia= 30, Radius= 15, Angle= 0, Flute Len= 50, Flutes= 4, Tool len= 50).
- Press the “**Add to Library**” button to add the new tool to our example tool library.
- Press “**OK**” to add the tool to the current Cam Plan.

Next we need to create the 6mm Ball Endmill rest finishing operation. All finishing operation types have an option to use a “reference tool”. 3D non-projective toolpaths are superior to the 2D projective types such as Lace and offset 2D. Probably the best choice of operation type when using a reference tool is the **Offset 3D** operation. Create and calculate an Offset 3D operation as shown below to the following specification.

- **Tool** = 6mm Ball Endmill(Ball Endmill D6).
- **Features** = blowmold.

Primary parameters:

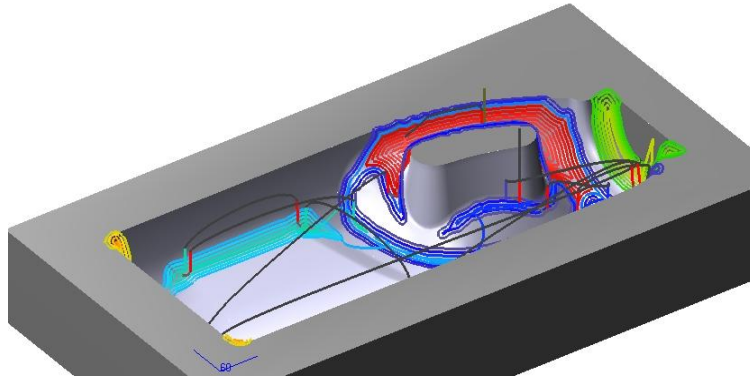
- **Path Tolerance** = 0.1.
- **Stepover** = Scallop
- **Scallop Height** = 0.1.

Link and Lead parameters:

- Long Link type = Spline.
- % Short Link Limit = 2000 (Cuts down on tool lifts).

Limiting parameters:

- **Reference tool** = Ball Endmill D30 (Pick the button and choose from the pull down list).
- **Min Rest Height** = 0.4. This parameter sets the minimum height of detected rest material. The default value is **0.3**. A low value will increase sensitivity but will also increase the possibility of false detection that can occur when a coarse Tessellation tolerance is used.



Z Level Rest Finishing

Although as previously stated the Offset 3d operation is the preferred choice when using a reference tool. Other finishing operation types can also be used. Create and calculate a Z Level rest finishing operation as shown below using similar parameters as the previous Offset 3D exercise. Note the Links have been removed from the image for clarity and the rainbow appearance has been set to “Z Value” (Red highest Blue lowest).

- **Tool** = 6mm Ball Endmill (Ball Endmill D6).

Primary parameters:

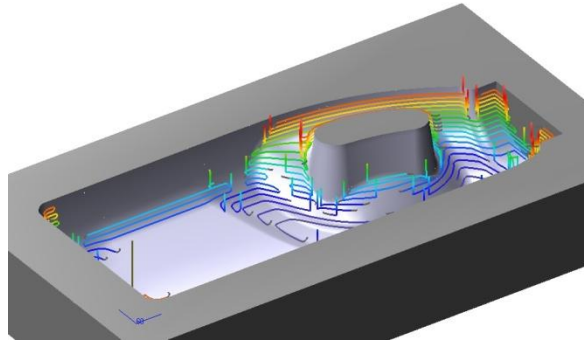
- **Path Tolerance** = 0.1.
- **Stepdown** = Absolute.
- **Stepdown Size** = 2.

Link and Lead parameters:

- **Long Link type** = Spline.
- **% Short Link Limit** = 300.

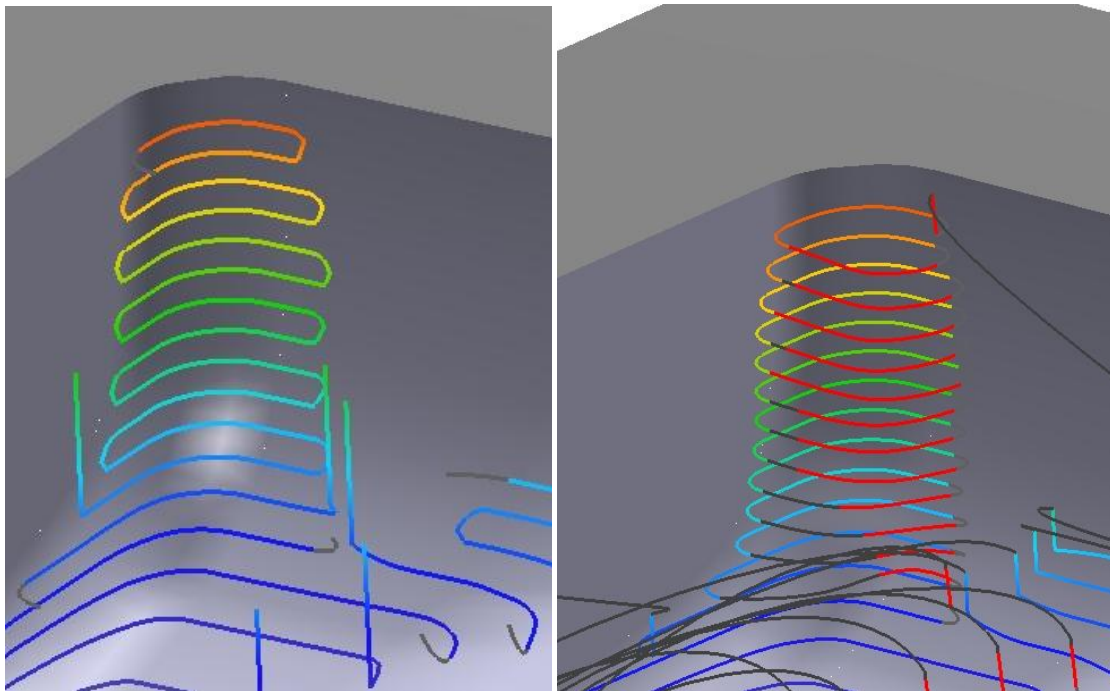
Limiting parameters:

- **Reference tool** = Ball Endmill D30 (Pick the button and choose from the pull down list).
- **Min Rest Height** = 0.1.



The previous exercise cut the part using a zigzag flow which alternated between climb and conventional milling. Also the tool stayed in contact with the part when moving down to each Z level as can be seen in the image below left (links not shown).

The image below right shows a modified toolpath which climb mills and also moves out from the part between Z levels. Change the relevant parameters in your operation and recalculate the toolpath. Try it yourself first, but if you have problems, suggested parameters are shown at the bottom of this page.



Path Setting parameter/Cutting Control:

- **Cut Direction** = Climb.

Link and Lead Parameters:

- **% Short Link Limit** = 3000.

- **Safe Distance** = 3.
- **After Short Link** = Yes.
- **Before Short Link** = Yes.

Pencil Milling

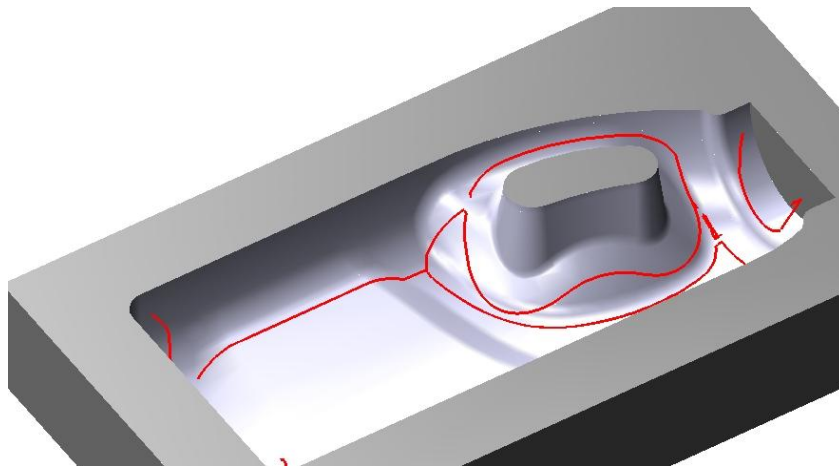
Pencil trace cuts are used to clean up the cusps that can occur at the intersections (or valleys) of intersecting faces. Better results will be obtained if there are either, no fillets present between the intersecting faces, or if the tool is larger than the fillet radius. If the tool is smaller than the fillet radius some drifting of the toolpath (within tolerance of course) can occur. There are options available within QUICK MILL to help cut down on tool drifting which we will be shortly taking a closer look at. But first let's create a Pencil cut operation and apply it to our current Cam Plan.

Create and calculate a **Pencil Cut** operation to the following specification.

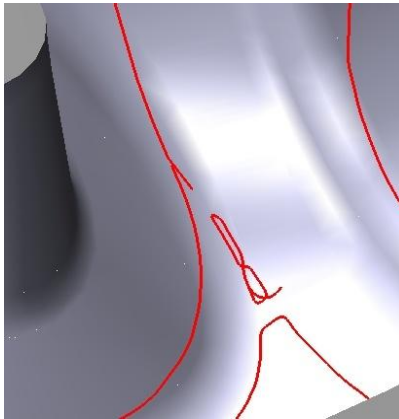
- Create a new 25 Ball Endmill(Name= Ball Endmill D25, Cutter Dia= 25, Radius= 15, Angle= 0, Flute Len= 50, Flutes= 4, Tool len= 50).

Primary parameters:

- Set the **Path Tolerance** to 0.1.
- **Calculate** the toolpath.
- The Leads and Links have been removed from the image.

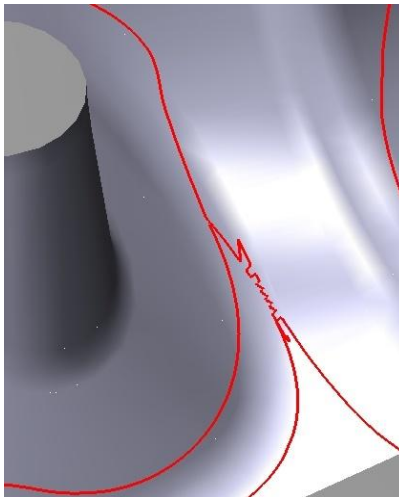


Some tool drifting occurs.



- **Path Tolerance**= 0.01

Tightening up the path tolerance can improve the tool path, but some drifting still occurs.

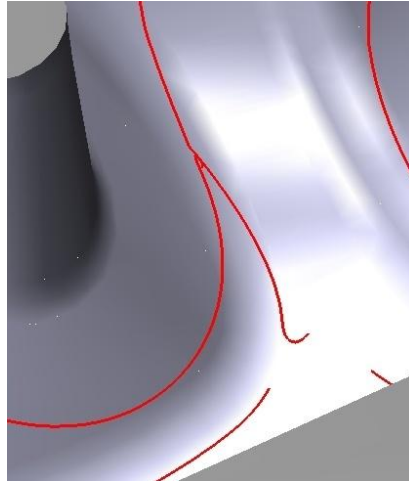


Operation Path Setting/Cutting Control Parameter

- **% Sensitivity** = 30 (Default = 50).

This parameter tells QUICK MILL how “Sensitive” to be when detecting areas to cut with the pencil trace.

Lower the value to 30% which when combined with the tighter **Path Tolerance** of 0.01 produces the tool path as shown left.



Pencil Cut Drive (Combo Pencil Cut rest milling)

In the next exercise we will create a Combo cut, combining the previous Pencil cut with another Pencil (drive) cut to produce the toolpath shown below. The previous operation is in **red** and the new operation is shown in **blue**. The resulting toolpath is smooth and continuous, producing fewer tool lifts than other rest milling techniques.

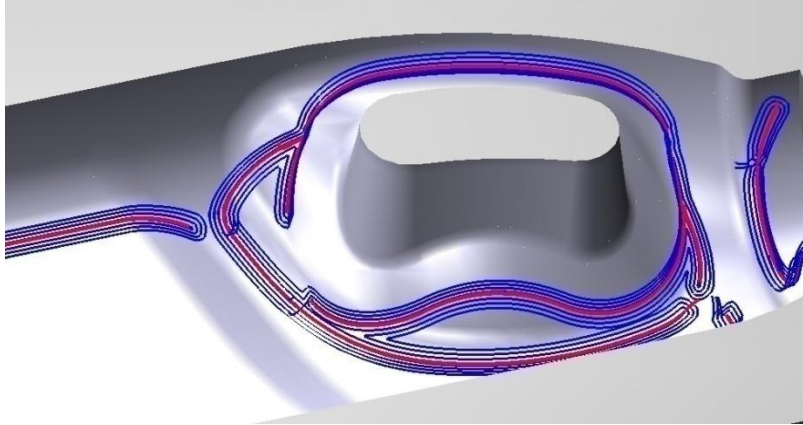
Since the new Pencil cut operation is almost identical with the previous one we can copy the existing operation and then simply modify the relevant parameters in the new one.

- Right-click on the “Pencil Cut 1” operation.
- Select “**Duplicate**”.

This creates a new identical operation to the previous one.(Pencil cut 2)

Change the operation **Primary Parameters** as shown.

- **Number of cuts** = 4 (Four cuts each side of the normal Pencil trace).
- **Stepover** = Scallop.
- **Stepover Size** = 0.01.

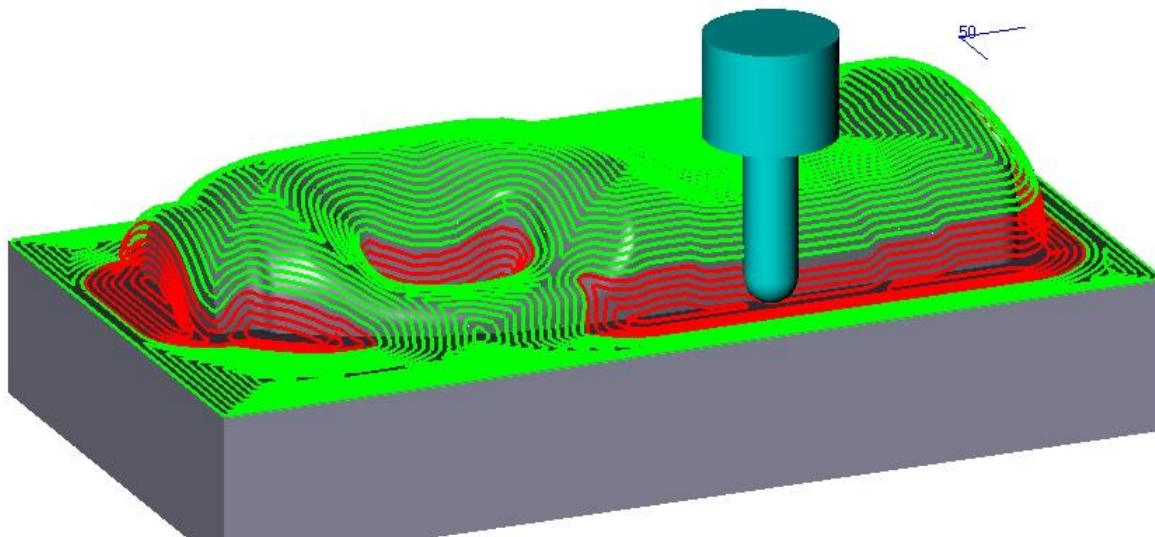


Save your file now.

Progressive Holder

**Load the file “Mill_Bottle.Z3” and then load the CAM Plan “Model_Cam_plan.”
Hide all toolpaths.**

This capability is especially useful for calculating tool paths with short, rigid tools. QUICK MILL calculates the material left by avoiding holder collisions as shown below (green) and then produces an efficient rest milling path using a longer tool (red).



Start by creating the Offset 3D green tool path to the following specification.
First we need a suitable short tool.

Create a 16 Ball end mill.

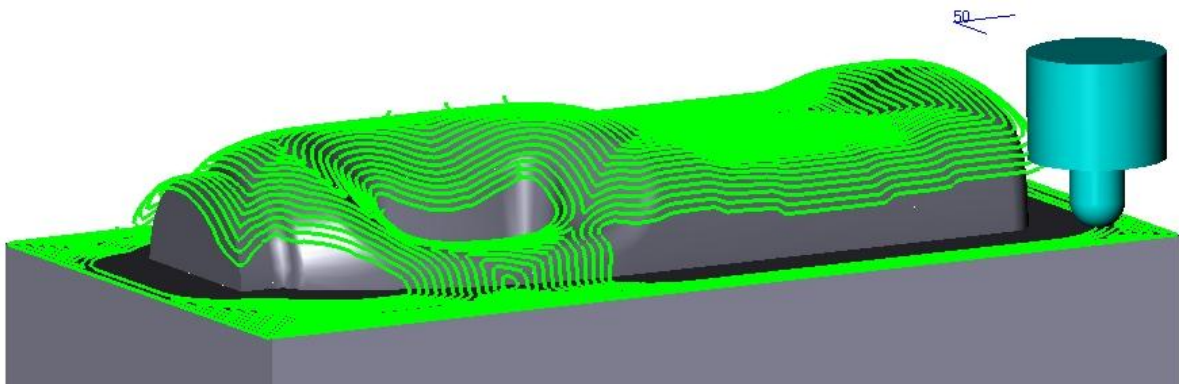
Note this time we also need to define the shoulder diameter and shoulder length.

- **Cutter Dia** = 16.
- **Radius** = 8.
- **Flute Len** = 12.
- **Tool len** = 20.
- **Shoulder Dia** = 40.
- **Shoulder Len** = 30.

The Offset 3D cutting parameters are as follows

- **Path Tolerance** = 0.1
- **Stepover** = Scallop
- **Stepover Size** = 0.1

Calculate the tool path.



Next let's create the offset 3D operation that uses the extended (longer) tool.

Copy the existing previous operation using the right click >duplicate command on the "Offset 3d 1" operation.

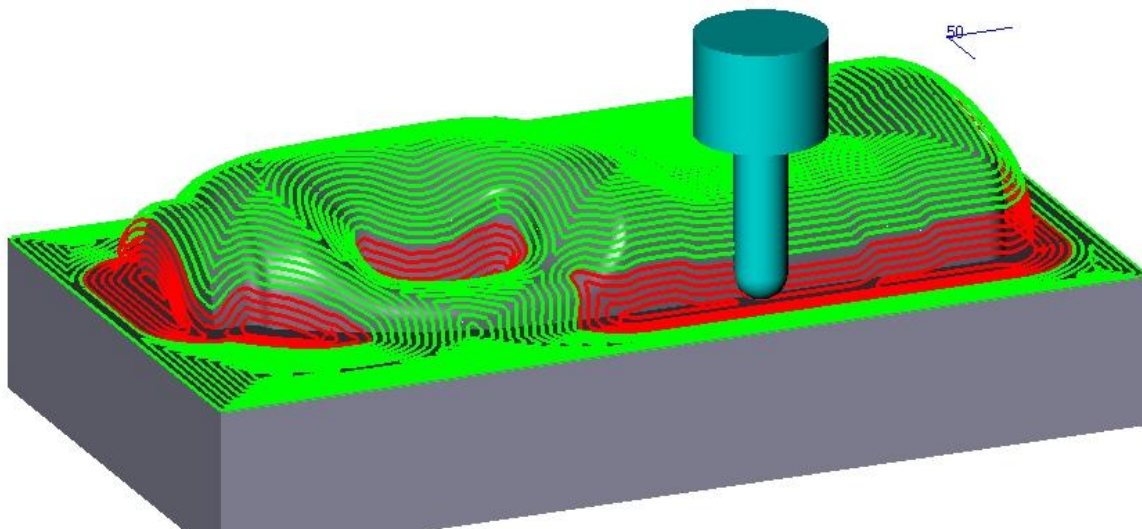
Create a new tool to the following specification.

- **Cutter Dia** = 16
- **Radius** = 8

- **Flute Len** = 50
- **Tool Len** = 50
- **Shoulder Dia** = 40 (optional)
- **Shoulder Len** = 30 (optional)

The cutting parameters are the same as the previous operation, but we do need to specify the previous (reference) tool "16 Ball Endmill" in 3D limiting.

Calculate the tool path



Save the File.

Next we need to specify a machine tool description.

Place the mouse pointer over the "**Machine**" icon of the Can tree. Hit the right mouse button and select "**Manage**" option.

The options shown above refer in this case to the "Machine Manager Form". However, the same options are used in other forms "Tools" for example. The following is a general description which also applies when these options are used in other forms.

- **Select**
This displays the Machine Manager form, (or Tool Manager for example).

This is used to select a machine to assign to the active setup, providing the machine already exists in the current CAM Plan.

- **Edit**
This displays the Machine Manager form and activates the current machine for editing.
- **Rename**
Renames the current machine assigned to the active setup. Enter a new name for editing.
- **Remove**
Removes the current machine assigned to the active setup. But does not remove it from the CAM Plan.
- **Manage**
Displays the Machine Manager form for managing machines in the active CAM Plan and other machine libraries.
- **Delete**
Deletes a machine definition from the active CAM Plan.

The Machine Manager form is used to define and manage Cam Machines for use in the active CAM Plan. Machines can be defined or selected from libraries of previously defined machines. A machine library is any ZW3D Cam Plan which contains machine definitions.

The Cam machine contains information necessary to format an NC Program for a particular machine tool. It also defines the configuration of the tool changer.

Further information on the machine manager and other forms such as Tools, Speeds and Feeds and so on can be found in the on line Help.

After selecting “**Manage**” as shown above, the Machine Manager Form is displayed.

Enter a name for your machine in the “**Name**” field.

Select the “**Apply**” button and your machine will be added to the current CAM Plan.

Next press “**OK**”.

The Cam Manager form will be closed.

The Machine will be added to the active setup and the name will be displayed in the Cam tree.

Appendix A 3 Axis Machining

3Axis CAM First Tool Path

The 3 Axis toolpaths are hidden by default. Check the “Enable 3 Axis Direct Nurbs” option on the CAM tab in the **Configuration** form.

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

Let’s get started on our first toolpath by starting ZW3D and opening a file.

Select the **Open File** icon. 

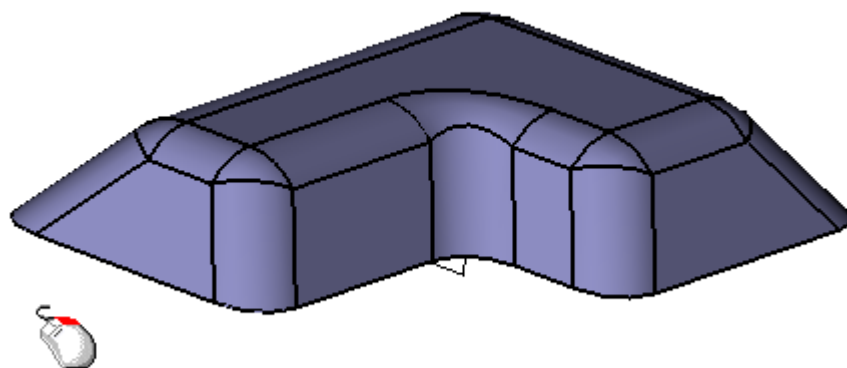
Choose the file “CAM_FIRST_LOOK.Z3” from the program directory\training folder.

Select the “**CAM_FIRST_PART**” from the object preview window.

Select **Open**.

In the first chapter we created a new file to start our CAM Plan. Here we are going to just add a new CAM object to an existing File.

Notice there is already a “**CAM_FIRST_PLAN**” in the objects list on the right. You can look at this object later for reference if needed.



Right-click and select **CAM Plan**.

This is a new shortcut method for starting a CAM Plan. This brings up ZW3D CAM and the CAM Plan Manager. The part is already defined as the geometry("CAM_FIRST_PART" has been added to the **Geometry** already).

The name of the CAM Plan is also automatically set to PART NAME_CAM.

(You will also notice in the image above, another shortcut to create a 2D Sheet. This creates a 4 view layout of the part in a sheet named PART NAME_2D.)

As we work through this exercise, you will begin to see that the ZW3D CAM Plan Manager is "Hierarchical" and looks similar to a popular Operating System file tree.

ZW3D CAM Plan Manager

Since we are not using an assembly this time, you will have to add STOCK to the plan

Double left click on **Geometry**.

Find the path from the Shape Browser.

Select "**CAM_FIRST_STOCK**".

Click "Yes" to insert the stock.

The next order of business will be to create our Operations.

Double Left-click **Operations** and select **Zigzag** Rough under the **3x Nurbs** tab. 

A new operation has been created called Zigzag Rough 1.

Under the Zigzag Rough 1 operation in the CAM Plan Manager, double left-click **Tool (undefined)**.

Choose a 16mm Flat End Mill (Endmill D16/0) from the MM library supplied.

Click **OK** to finalize tool parameters. ("Endmill D16/0" is added to the **Tool** option)

The CAM Manager should indicate you have selected 16MM Flat EndMill as the tool for the Zigzag Roughing operation. ("Endmill D16/0" is added to the **Tool** option)

Next we will establish the parameters that will determine the ZigZag Rough operation.

Double left-click on **Parameters**.

Take a minute to review all of the available parameters on this menu. The Parameters item in the ZW3D CAM Manger condenses many Operation definitions into one form.

Change the **Max Depth** on the **Cutting** Tab to 3 and the pick the **OK** button.

This variable will determine the maximum Z-level cut depth.

Now we are going to set up our tool path.

Double left-click **Features (undefined)** and choose **CAM_FIRST_PART** from the Geometry section of the CAM Plan Manager.

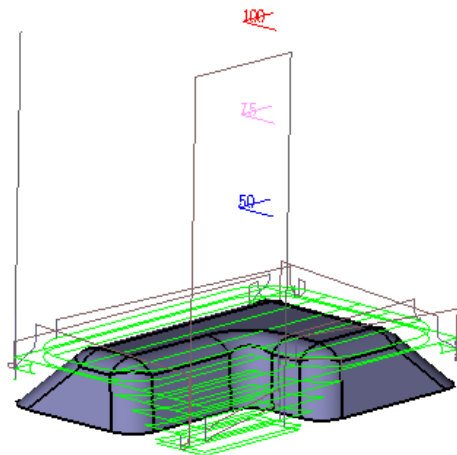
Also choose **CAM_FIRST_Stock** from Geometry section before ending this command.

the part **CAM_FIRST_PART** and the stock **CAM_FIRST_Stock** are added under the **Features** option.

We have defined all the necessary parameters and are ready to calculate the toolpath.

Right-click on zigzag rough and select **Calculate**.

You can now see that ZW3D has created your tool path.



Let's save the part by clicking the **Save** icon or using the pulldown menu.



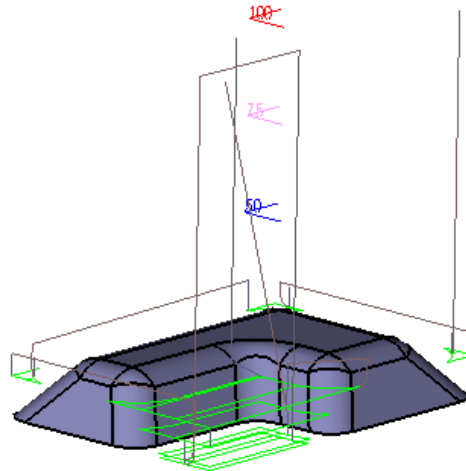
Let's change our **Max Depth** and recalculate the tool path.

Open **Parameters**, change **MAX Depth** to 5 and click **Calculate**.

ZW3D creates a new toolpath as shown below.

You can see that the distance between cuts has increased.

Pick **OK**.



Now we will create a Spiral Cut.

Double left click on **Operations** and pick the Finishing **-3X spiral cut**  under the **3x Nurbstab**.

(Make sure the 3X NurbsTab has been displayed in the Toolbar.

If it is not, right click your mouse on the Tool bar and select the option of Ribbon Tabs, and then choose the 3X Nurbs from the context menu of it.

Next we will need to select a tool for the new operation.

Pick the 16mm Ball End Mill (Ball Endmill D16) from the Library and pick **OK**.

We will want to make some adjustments to the default 3X Spiral Cutting parameters to optimize our toolpath.

Open **Parameters Form**

From the Cutting tab, left-click on the **Step Type** button and select **Scallop Height** from pop-up list.

We also need to change the **Step Value**.

Change the Step Value to 0.02 and pick **OK** to close the form

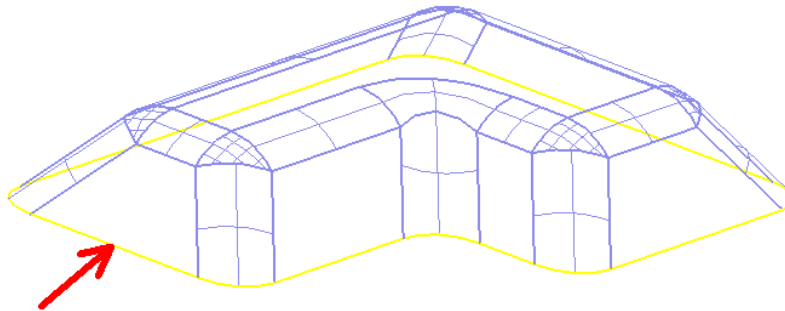
Double Left-click on **Features** (under 3X Spiral Cut 1).

From, Select feature for operation form pick **Part: CAM_FIRST_PART**.

After inserting the Feature Part, return to the CAM ManagerTree and right-click on **CAM_FIRST_PART** at the **Geometry** level, select **AddFeature**, then select **Profile**.

ZW3D prompts you to “Select geometry (by Filter).”

<Shift+Left-click> on the lower part edge (as indicated below), Middle-click to accept.



ZW3D should prompt that 12 edges have been picked.

The **Profile Feature** form will appear with the default Name profile1.

Select **OK** on the form.

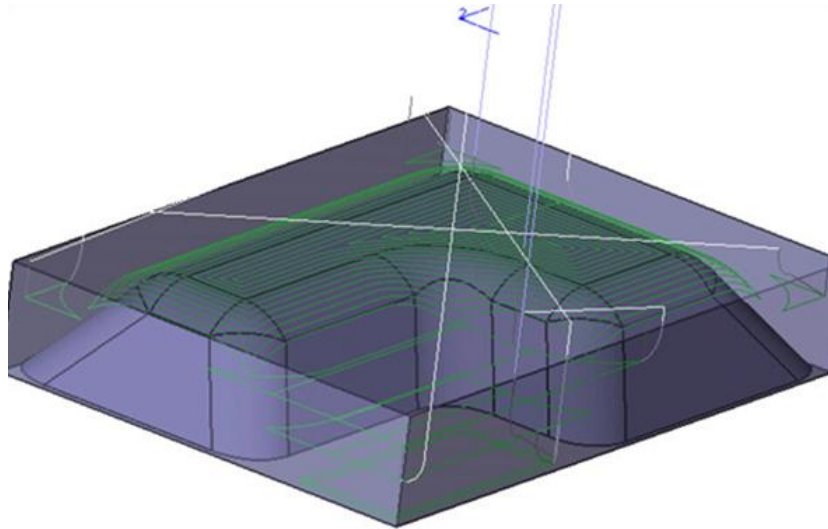
The profile1 will appear under the **Part: CAM_FIRST_PART**.

Double left-click on Features (under 3X Spiral Cut 1).

Select **Profile 1** and **CAM_FIRST_Stock** from the Select features for operation form. You should see both added to the list of **Features** under the 3X spiral cut1 operation.

Now, right-click on 3X spiral cut 1 and select **Calculate**.

Congratulations! You have successfully completed a set of toolpaths.

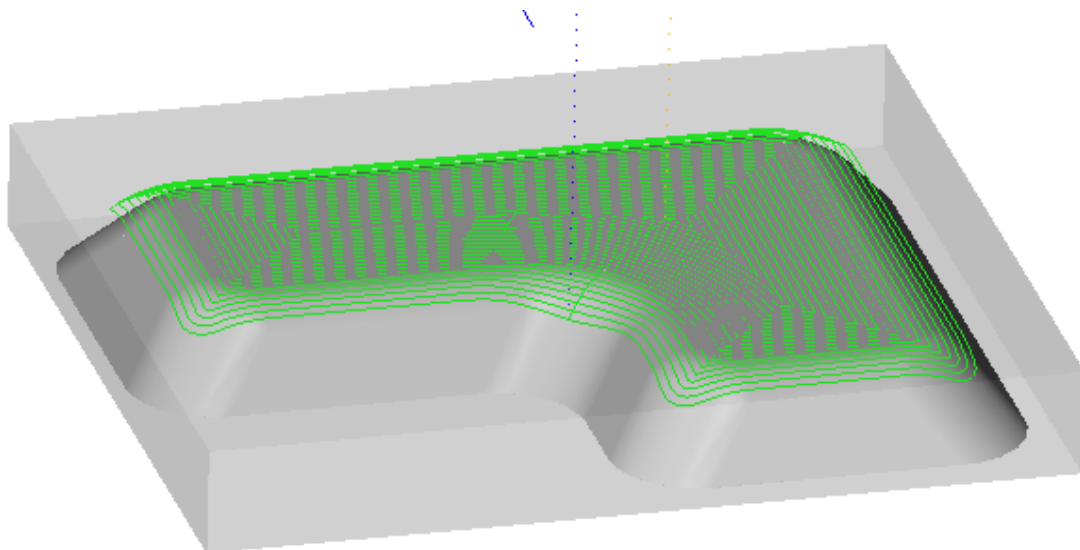


Save the part by clicking the **Save** icon or using the pulldown menu.

Let's make the display easier to manage by hiding the Zigzag Roughing tool path and only look at the 3X spiral cut.

Right-click on zigzag rough 1, and select Show/Hide.

Double click on the operation's icon will also show/hide the tool paths.

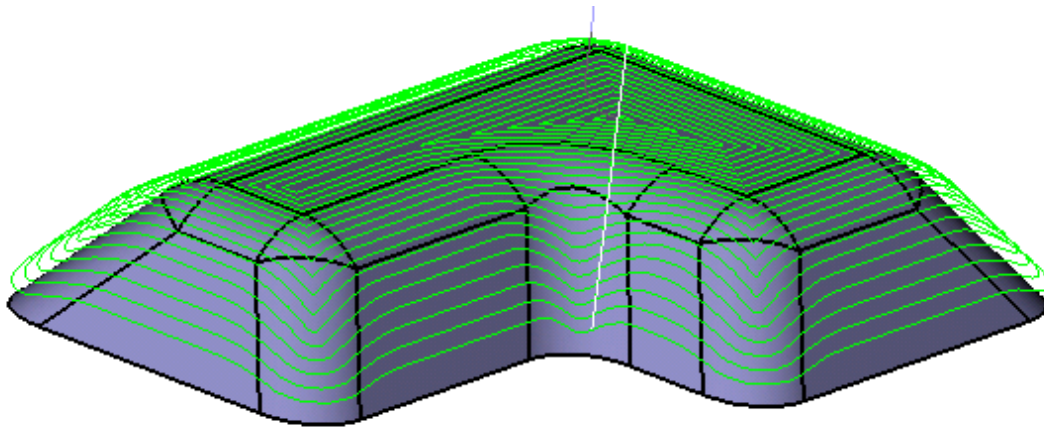


Notice that the tool path needs to extend past the edge of part.

Double left-click 3x spiral cut 1 **Parameters**.

Click the **Tool Location** button and select “**On Boundary**” on the cutting form, then click **Calculate**.

The result should look like this:

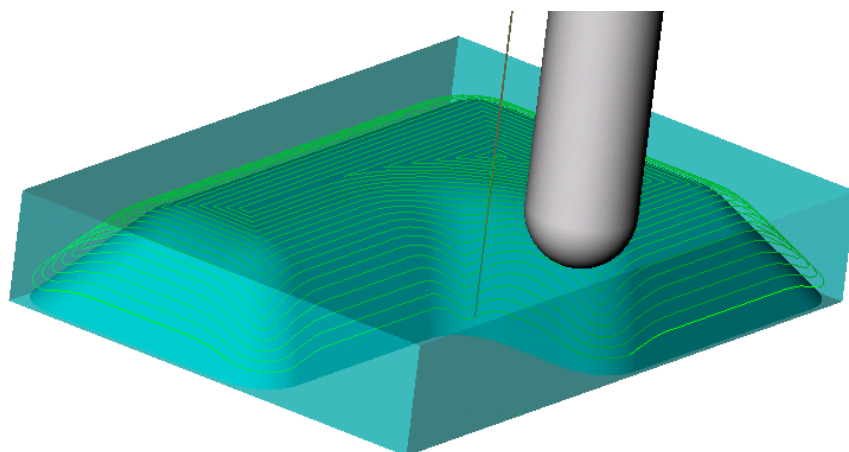


We will now walk through a Wireframe tool path verification.

Right-click on 3X spiral cut 1 and select **Verify**.

Use the > or >> buttons to see the tool verify.

Close Verify by clicking the black X in upper right corner of the Toolpath Verify form.

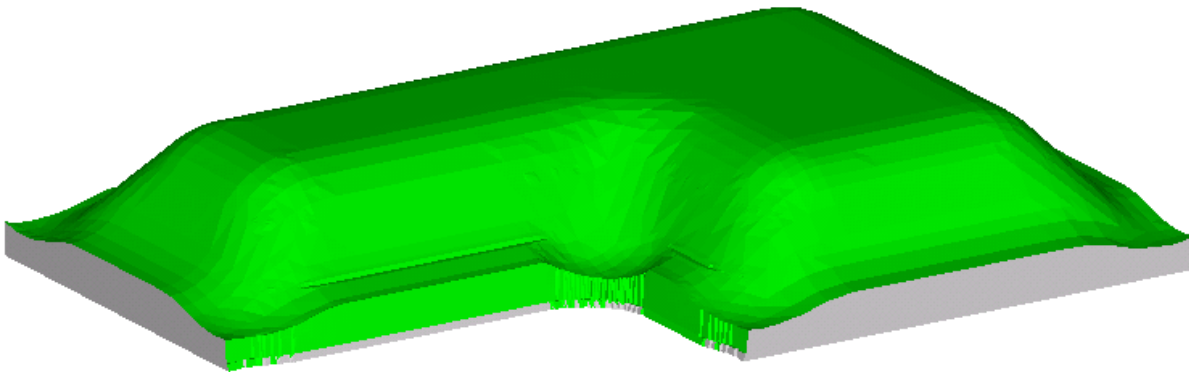


Let's confirm the Wireframe Verification with a Solid Verification.

Right click on 3X spiral cut and pick Solid Verification

Make sure **Verify active operation only** is unchecked to see the result of both tool paths.

Click the **Run simulation** button on the Solid Verify Session form.



Once you are satisfied with the tool paths generated we need to define the machine and add all the tools to the tool changer.

Select **Machine** from the CAM Plan Manager.

Give your machine a name and select **Apply**.

Select **Tool Changer** and select **Add All in Machine Tool Changer** form.

Select **OK** to close the Machine form.

Once the Machine is created we can send the toolpath to the Flexpost processor and generate our G and M codes. Let's check our output!

Right-click on **Output**, and select CL/NC Output setting , display the Output setting form

Enter a Part ID, Programmer and a File Name.

Then, right click on the folder of NC , select the options "output NC" and "Output CL" code file separately to output the NC code file and CL file.

Output **Tooling List**.

Scroll and review Tools in the displaying box.

Output **Operation List**.

Scroll and review Operations.

Let's save the part by clicking the **Save** icon or using the pulldown menu.

Congratulations ! You have created and verified your tool path!

2-1/2 Axis Milling

How to Define 2-1/2 Axis Milling Operations Using the CAM Plan Manager

This is just an outline:

1. In the CAM Plan Manager, double left-click on Operations, then select any 2x Mill operation from the form provided. The operation will be added to the manager tree.
2. In the manager tree, select Parameters under the operation you just added, and set them using the definition form provided. Pick **OK** when you are done. Each 2-1/2 axis operation uses a unique definition form. They are shown below under each operation type. See 2-1/2 Axis Milling Operation Form Parameters for a definition of each parameter.

Notice that (undefined) is shown next to Features under the 2-1/2 axis operation you just added. This means that no features are yet defined for the operation.

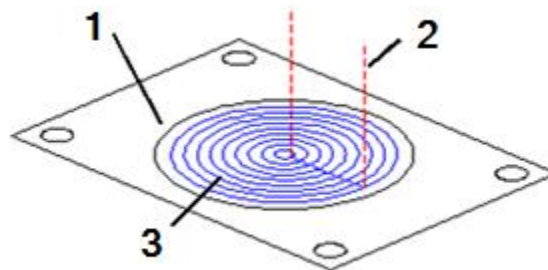
3. Your CAM features (once you define them) are listed under the Geometry section of the manager tree. Firstly, hit **Features**, and then Under Geometry, find the CAM component (i.e., part : name) and expand it (i.e., select  to the left of the part) to list the CAM features currently defined for that component and select one. It will now be listed under Features for the 2-1/2 axis operation you just created. The (undefined) flag will be removed.

To create a new CAM profile feature: Again, under Geometry right-click on the CAM component, select Add Feature and select **profile**. You will be prompted to select geometry from the graphics window.

Spiral Cut Operation (2-1/2 Axis)



The spiral cut operation is a facing (area clearance) or pocketing technique, which advances the tool at each depth by proceeding toward or away from the part boundaries.

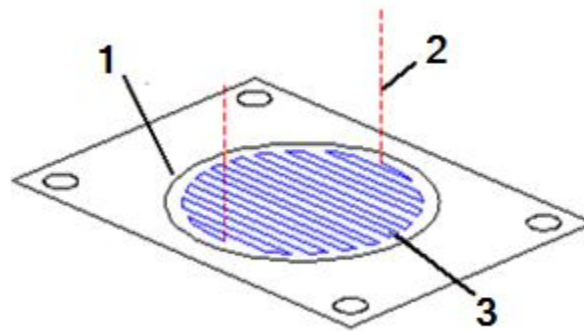


Sample Spiral Cut Operation(2-1/2 Axis)

(1:Geometry 2:Tool Motion 3:Tool path)

ZigZag Cut Operation (2-1/2 Axis)

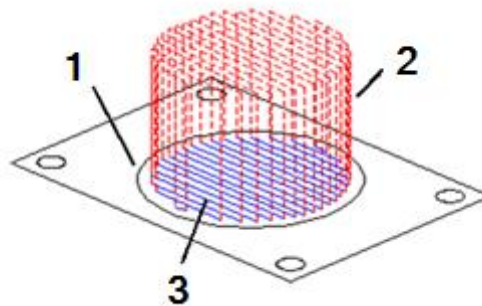
The zigzag cut operation is a facing (area clearance) or pocketing technique. It advances the tool at each depth through a sequence of straight parallel cuts, reversing the tool direction at the end of each cut.



Sample ZigZag Cut Operation (2-1/2 Axis)
(1:Geometry 2:Tool Motion 3:Tool path)

Box Cut Operation (2-1/2 Axis)

The box cut operation is a facing (area clearance) or pocketing technique, similar to the zigzag cut except the cuts are all in the same direction. The tool is lifted between each cut.



Sample Box Cut Operation (2-1/2 Axis)
(1:Geometry 2:Tool Motion 3:Tool path)

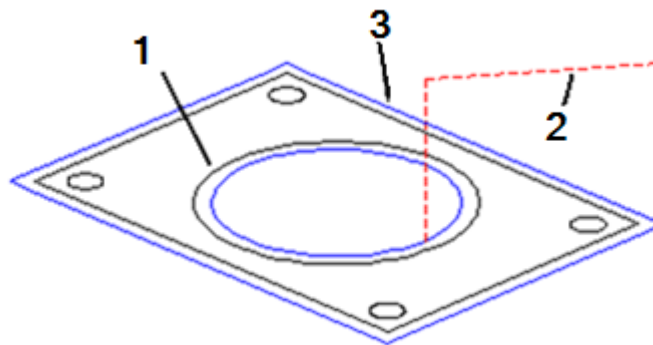
Contour Cut Operation (2-1/2 Axis)

With the contour cut operation, a medial or spine curve is calculated for each cutting zone. Tool movement proceeds in cuts generated parallel or perpendicular to that curve.

Profile Cut Operation (2-1/2 Axis)



The profile cut operation cuts any number of open or closed curve boundaries (CAM Profile features) or CAM Components containing geometry profiles. Self-intersecting profiles are also supported as long as the **Tool Location** parameter is set to "**on boundary**." See 2-1/2 Axis Milling Operation Form Parameters for a definition of each parameter. This allows engraving operations or complex patterns to be machined without error.



Sample Profile Cut Operation (2-1/2 Axis)
(1:Geometry 2:Tool Motion 3:Tool path)

Chamfer Cut Operation (2-1/2 Axis)



For the 2-1/2 Axis Chamfer Cut set the Feature class to "Chamfer." You can pick profiles, curves, surfaces, or features (CAD chamfer feature). Tip Control is also provided on the Cutting tab.

Depth of Cut Parameters (2-1/2 Axis)

- **Type**

Uniform Depth - This uses the maximum depth of cut for all depths, measured from the top of the region (see Stock Data below).

Non-Uniform - This uses First Zcut (see below) for the first cut, Last Zcut above the lowest point in the region and then divides the region between evenly for the remaining depths of cut.

Base Only - One pass is made at the lowest depth for the region.

Base & Island Top - Like Base Only, but it also creates a cut at the top of each island.

Island Top - This makes a cut at the depth of each island top.

- **Max** - This is the standard depth of cut used for determining the Z height of cuts.
- **Min** - Cuts will not be made closer than this depth apart, unless cuts are being placed at the depths of island tops.
- **First Zcut** - This is for non-uniform depths only, at top of the region.
- **Last Zcut** - This is for non-uniform depths only, at bottom of the region.

Stock Data Parameters (2-1/2 Axis)

Use the Stock Data tab of each 2-1/2 Axis Opdef (Operation Definition) form to define cutting depth for each operation. If the cam feature has its own depth (e.g., step, slot or pocket), and stock parameters are also defined, the top from the stock data will be used. The bottom will be the higher value between the stock bottom and the feature's actual bottom.

- **Rough Cut Offset** - This is a thickness to leave after the last rough cut is made. This is ignored if the boundary condition is on.
- **Finish Cut Offset** - This is a thickness to leave after a finish (optional) side clean cut is made. This is ignored if the boundary condition is on.
- **Vertical Offset** - This is a thickness of material to leave above the bottom of the region.
- **Top Point** - Use this to set the top of the part. This is not used when the top is already known from an existing CAM feature. Enter a z value, an absolute coordinate value (x,y,z) or select a point from the graphics window. Point input options are available by right-clicking the mouse. This Top Point parameter and the Vertical Clear parameter (see Lead and Link parameters Tab - Auto Engage/Retract) together define the "safety plane."
- **Bottom Point** - Use this to set the bottom of the part. Like Top Point above, this is not used when the bottom is already known from the CAM feature. Again, enter a z value, an absolute coordinate value (x,y,z) or select a point from the graphics window. Point input options are available by right-clicking the mouse.

Notes

- Note that any offset defined in the CAM Features for this operation will remain after all machining is completed. It will also be used in the ON boundary condition,

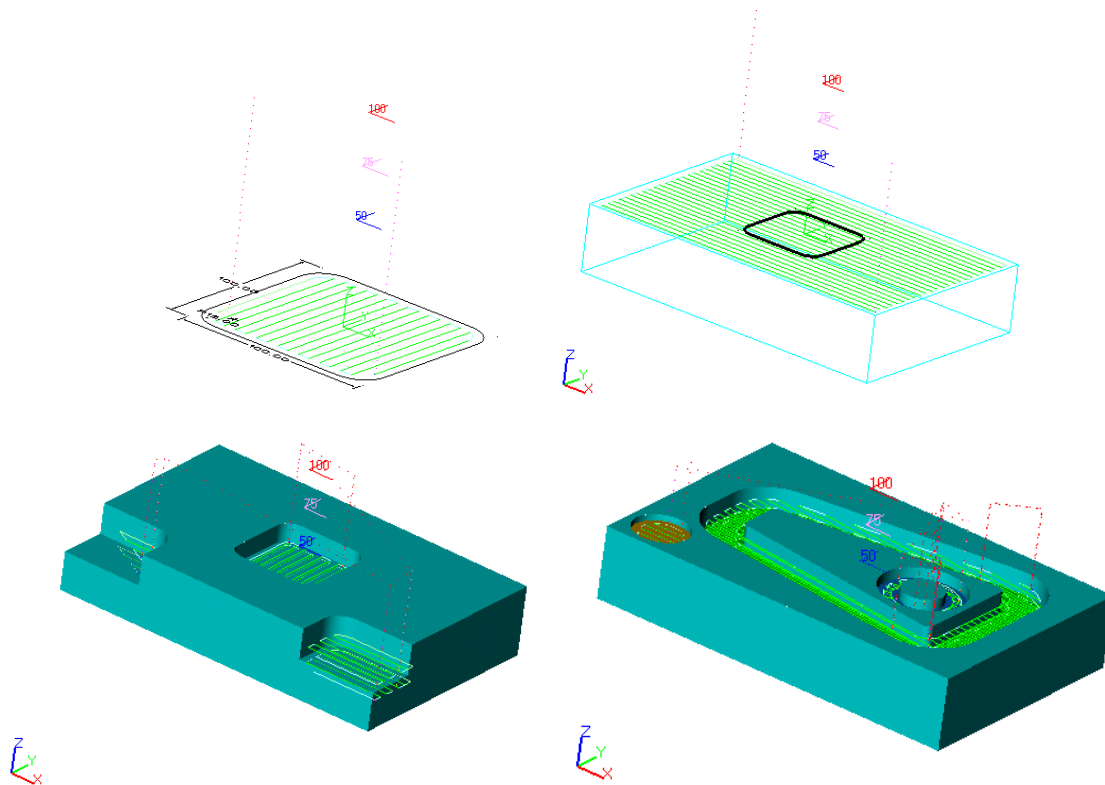
in the TANTO, or PAST conditions. The following offsets would be added to the offset defined in the Feature.

- All 2-1/2 axis tool paths allow the subtypes "rest mill" and "clamp cleanup." In those cases a reference tool path should be selected (ref oprn). If the subtype is set to "standard" the reference tool path will be ignored. In the case of "rest mill" the tool should be smaller than that used in the reference operation. For "clamp cleanup" the tool may be the same size or smaller than that used in the reference operation.
- This note may help to define cam features correctly and consistently for 2-1/2 axis milling operations with boundaries having islands. When machining 2D pockets with multiple cutting layers, islands defined from Cam Profiles assume infinite heights, so that no tool path can ignore them. Islands defined by any other cam feature remember their geometric heights. They will be ignored to obtain better and efficient cutting patterns if the current cutting z level is higher.

CAM 2-1/2 Axis Pocketing

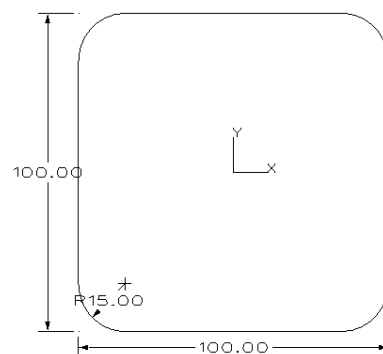
The object of this lesson is to explore 2-1/2 axis pocket machining with ZW3D. We will use a variety of sketches, surfaces and features to define our geometry and we will examine the operation definitions in more depth.

We will be showing you the best techniques and methods in order to help you optimize your CAM programming time. We have created ZW3D cam with the goal of providing you with the tools to do more work in less time.

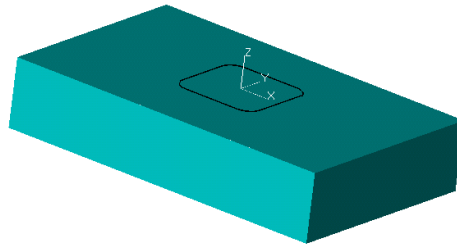


Start this lesson by loading the file **2D_Pocket.Z3** - Click **OK** to go to the Object level.

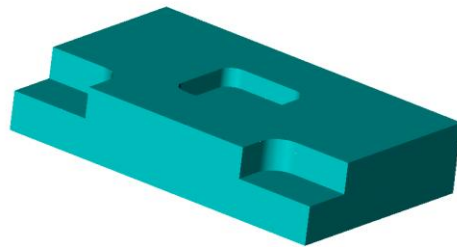
Here are the objects contained in this file.



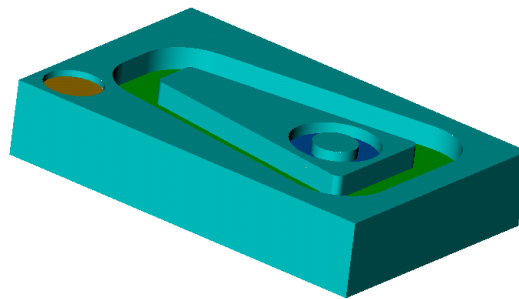
Pocket1 geometry



Pocket2 geometry



Pocket3 geometry



Pocket4 geometry

We will create a CAM plan for each of the objects containing the following settings:

Tool – 10mm flat end mill(Endmill D10/0).

Machine – ZW3D will create a default Machine 1 with tool added to changer.

Operation –2 ½ ZigZag with 10mm tool.

Select **Insert > New Cam Plan** from the pull down menu, and name the new CAM object: **Base CAM Plan**.

Then, when in the CAM level, select **Edit > Preferences** and set the default Units to MM.

First we will create a new cutting tool.

Use the Right-click in the ZW3D work space, and select the **Tool Manager** option, for the new operation.

Select the **Endmill D10/0**, from the TOOLS METRIC- Legacy library.

Select the **Apply** button and review the **Shape** settings.

Select OK

Adding machine tools

1. Use the **Right-click** in the work space, and select the **MachineManager** option,

- Enter **Hurco** for the Machine **Name**.

Adding the cutting tool to the machine's tool changer.

- Select the **Tool Changer** Button.
- **Yes** to confirm Machine name change.
- Select the **Add All** Button in the Machine Tool Changer form.
- Select **OK** on the Machine Tool Changer form.
- **Yes** to confirm the Tool Changer modification.
- Select **OK** on the Machine form.

Adding the machine tool to the CAM plan

Double Left-click on the **Machine** item in the tree and pick the new Hurco machine or Right-click>**Select**, pick Hurco from the list of available machine tools.

Selecting the operation

Select the **Operations** item.

Select **ZigZag** cut from the 2 Axis milling list. 

Double left-click on the **Tool** item.

Pick **Endmill D10/0** from the list of available cutting tools.

Pocket Exercise 1:

Now that we have a CAM plan created, we can make copies of it for each exercise.

Use the **Exit** CAM Plan command to return to the Root Object menu.

Select the **Base CAM Plan** object. Select the **Copy** command button. 
(This puts your generic CAM plan in a clipboard or buffer.)

Now, select the **Paste** command button 4 times. 

In this exercise we will:

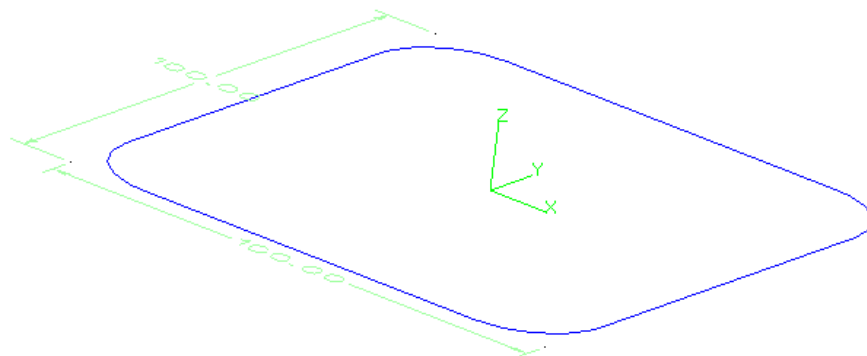
Use the Component manager to reference the Pocket1 object; use the Feature manager to select the pocket geometry and finish the 2 axis Zigzag toolpath.

Activate **Base CAM Plan_1**.

Select **Geometry** from the CAM Plan Manager.

Select the **Pocket1** object from the Shapes Browser.

Select the **Yes** button to add the geometry to the plan.



Describing the Geometry Features to cut

Right-click on the Part : Pocket1 (1) object.

Select the **Add Feature** option.

Select the **Pocket** option.

Select the 8 curves shown above (<shift> for Chain pick).

Use the Middle-click to accept the list.

Pocket Feature form is displayed.

Select the **OK** button on the Pocket Feature form.

The **pocket 1** feature is added under the part **Pocket**.

The default for Pocket **Depth** is 10mm. For this exercise we want to use a Depth of 0.

Right-click on the pocket 1 feature

Select the **Edit** option.

Select **p0** from the pocket list, and change the **Depth** to 0.

Select the **Apply Attributes** button to accept the changes.

Select the **OK** button.

Adding the Geometry Feature to the Operation Features

Right-click on the **Features** of ZigZag cut 1 operation.

Use the **Add** option button.

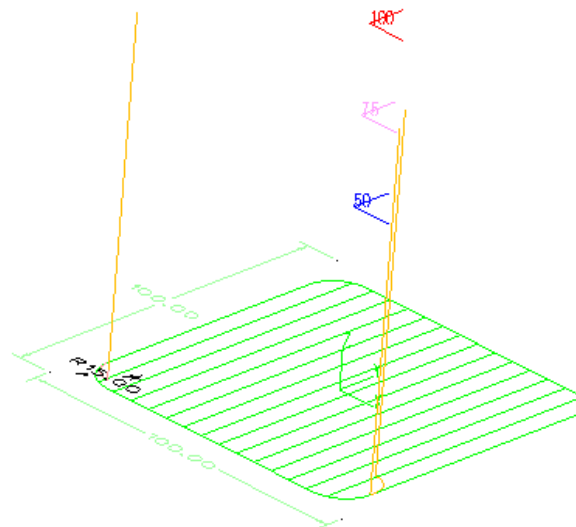
Select the **pocket 1** Geometry Feature.

Calculating the toolpath

Right-click on the ZigZag cut 1 operation.

Select the **Calculate** option, and the toolpath shown at the beginning of this exercise will be displayed.

See the result below.



Pocket Exercise 2:

Machining Multiple Pockets using Curves.

Now let's cut the Pocket2 part object.

Exit the CAM Plan.

Activate the **Base CAM Plan_2** object.

In this exercise we will reference the Pocket2 part object and create 2 separate Features for 2 sets of pocket geometry. We will create 2 separate Zigzag Operations and then mill the top face of our solid at a depth of 1mm and cut the inner pocket to a depth of 6mm.

Select **Geometry** from the Plan Manager Window.

Select the **Pocket2** object from the Shapes browser window.

Select the **Yes** button to add the geometry to the plan.

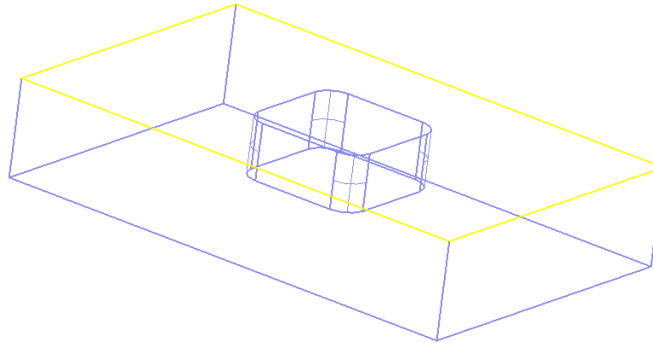
Describing the outer pocket Features

Right-click on the **Part : Pocket2 (1)** object.

Select the **Add Feature** option button.

Select the **Pocket** option.

Select the 4 outer curves (curves in yellow in the figure below).



Use the Middle-click to accept the list.

Change the pocket **Depth** to 1 in the Pocket Feature form.

Select the **Apply Attributes** button.

Change the feature name to **Outer pocket**.

Select the **OK** button from the Pocket Feature form.

The **pocket** feature is added under the part **Pocket**.

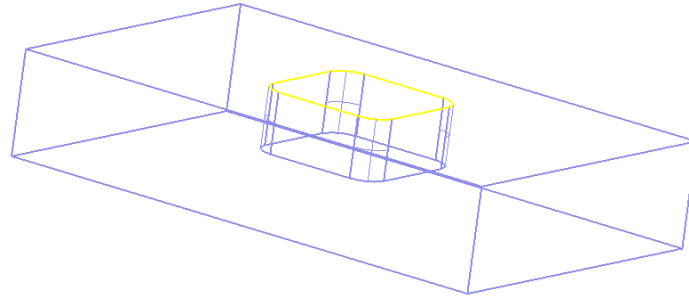
Describing the inner pocket Features

Right-click on the Part :**Pocket2 (1)** object.

Select the **Add Feature** option button.

Select the **Pocket** option button.

Select the 8 inner curves (curves in yellow in the figure below).



Use the Middle-click to accept the list.

Change the feature name to **Inner pocket**.

Change the pocket **Depth** to 6

Select the **Apply Attributes** button.

Select the **OK** button from the Pocket Feature form.

Adding the Outer Geometry Feature to the Operation Features

Right-click on the Operation's **Features**.

Use the **Add** option button.

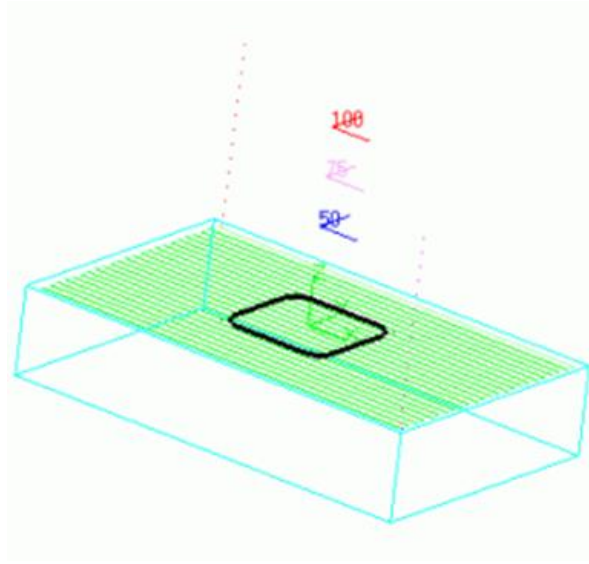
Select the **OuterpocketGeometry** Feature.

Calculating the toolpath

Right-click on the ZigZag cut 1 operation.

Select the **Calculate** option button.

See the result below.



In our Base CAM Plan we only created one cutting operation. In this exercise we need two operations, so let's create the second one.

Double left-click on Operations in the CAM Tree.

Select **ZigZag** cut from the 2 Axis milling list.

Selecting the tool for the second operation

Select **Endmill D10/0** from the list of available cutting tools.

Adding the Inner Geometry Feature to the second Operation Feature

Use the right-click on the zigzagcut2 **Features** option.

Select the **Inner** pocket **Geometry** Feature.

Hiding the toolpath

Use the Right-click on the zigzagcut 1 Operation. Select the **Show/Hide** option button to blank the toolpath.

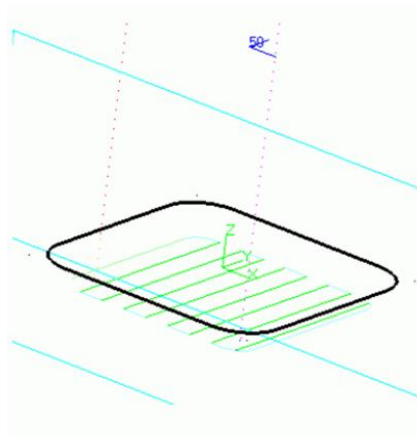
Calculating the toolpath

Use the Right-click on the Zigzagcut 2 Operation. Select the **Calculate** option.

See the result below.

To rotate the toolpath as shown below, click on **Parameters**.

In the Cutting parameters form, set **Auto Cut Angle** to No, and **Cut Angle** to 90.

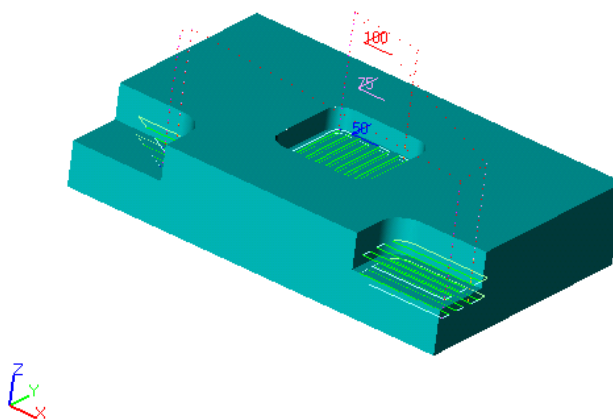


The Pocket 2 exercise complete.

Pocket Exercise 3:

Machining Pockets using surfaces and features:

Now let's look at the Pocket3 part object.



Exit the CAM Plan and then activate the **Base CAM Plan_3** object.

In this exercise, we will reference the Pocket3 part object. We will create 1 feature for 3 sets of independent pocket geometry and create 1 Zigzag Operation to cut all 3 pockets.

Cutting depths will be determined by the surface geometry.

Select **Geometry** from the Plan Manager Window.

Select the **Pocket3** object from the Shapes browser window.

Select the **Yes** button to add the geometry to the plan.

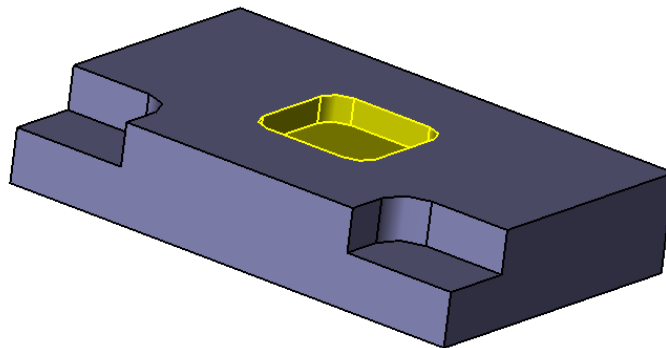
Describing the center pocket Features

Use the Right-click on the Part : Pocket3 (1) object.

Select the **Add Feature** option button and then select **Pocket** from the list.

Change the **pick filter** to **Feature**.

Select the center pocket (shown in yellow in the figure below).

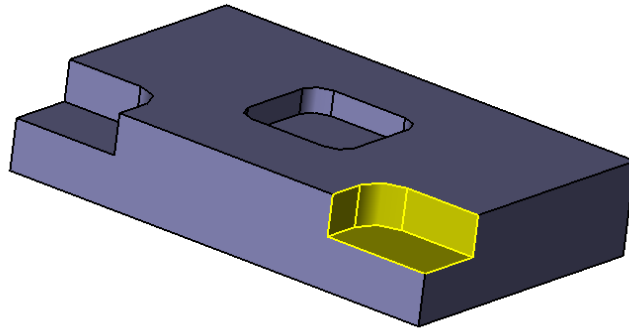


Use the Middle-click to accept the list.

Select the **Add Pockets** option button from the **Pocket Feature** form.

Change the **Pick filter** to **Surface**.

Select the 4 surfaces in the right pocket (shown in yellow in the figure below).

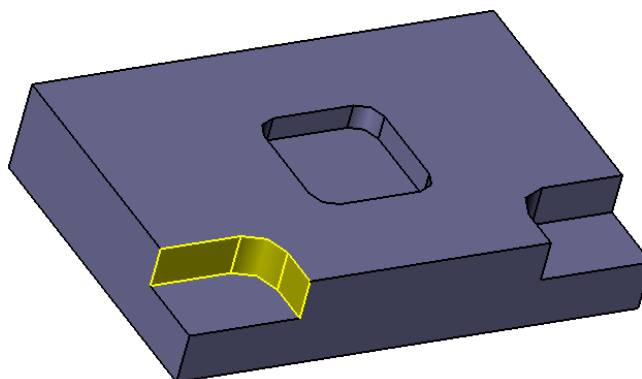


Use the Middle-click to accept the list.

Select the **Add Pockets** option button from the **Pocket Feature** form.

Change the **Pick filter** to **Surface**.

Select only the 3 vertical surfaces on the left (shown in yellow in the figure below).



Use the Middle-click to accept the list.

Set the **Tolerance** to 0.1 in the **Pocket Feature** form.

Select **Apply Attributes** button.

Select **p0** from the list of pockets. Notice that there is no depth associated with it.

By using surfaces or features as feature geometry, ZW3D will automatically determine the depth of cut for each pocket.

Adding the Geometry Feature to the Operation Feature

Use the Right-click on the **Features of** Zigzagcutoperation.

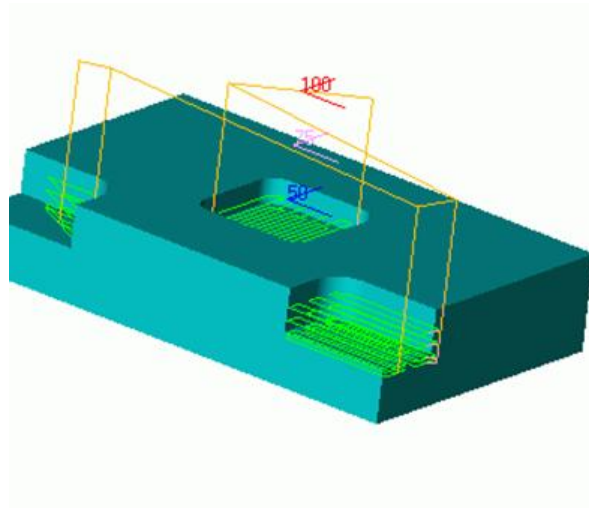
Select the **pocket** Feature from the **Geometry**.

Calculating the toolpath

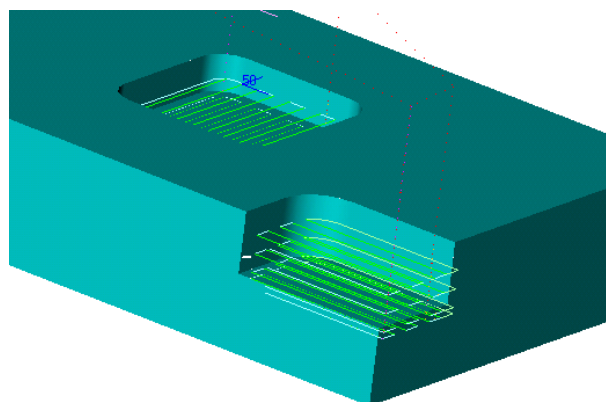
Use the Right-click on the Zigzagcut 1 operation.

Select the **Calculate** option button.

Save your part.



Now let's examine the tool paths, and answer a few questions. Notice that each of the pockets has multiple Z level passes. This is a default setting found under the Zigzag Operation Parameters form.



From the Zigzagcut 1 Operation tree, select **Parameters**.

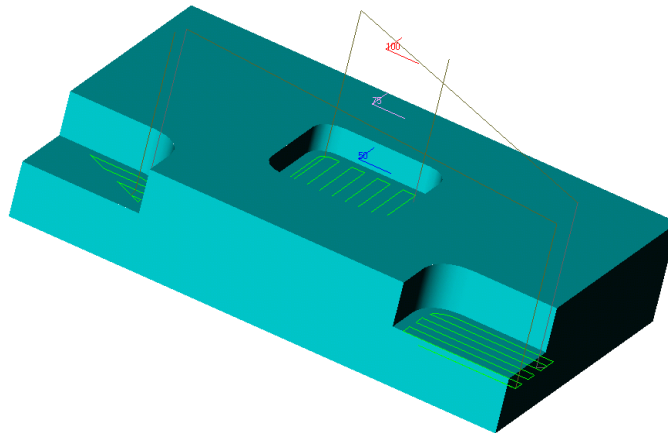
From the Operation definition Form, select the **Stock and Depth of Cut** icon tab.

Change those values to adjust the depth for each level of cut.

To cut only the bottom of the pockets, change the Cut Depth **Type** to **Base Only**.

Select the **Calculate** option button.

Now look at the front pockets.



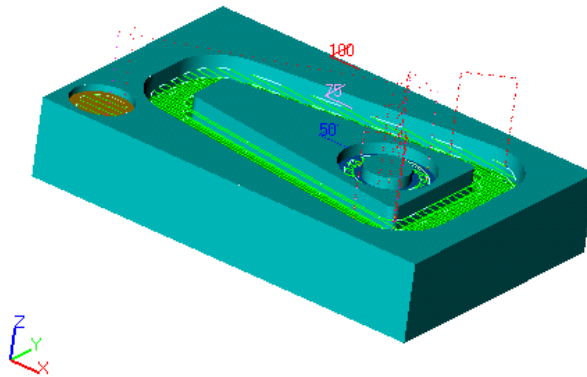
Notice that right pocket is cut completely, but the left pocket looks incomplete. This is the result of our feature geometry selection.

For the right pocket, we selected the 3 side surfaces and the bottom surface. ZW3D uses the surface edges to determine the boundary for the tool, and the surface data for the tool depth. Without the bottom surface, the left pocket did not have enough surface edges to form a closed boundary, so an implied straight line between the open boundaries was used.

Pocket Exercise 4:

Machining Nested Pockets.

Now let's cut the Pocket4 part object.



Exit the CAM Plan and then activate the **Base CAM Plan_4** object.

In this exercise we will reference the Pocket4 part object. We will create 1 feature for 3 sets of nested pocket geometry and create 1 Zigzag Operation to cut all 3 pockets.

Select **Geometry** from the Plan Manager Window.

Select the **Pocket4** object from the Shapes browser window.

Select the **Yes** button to add the geometry to the plan.

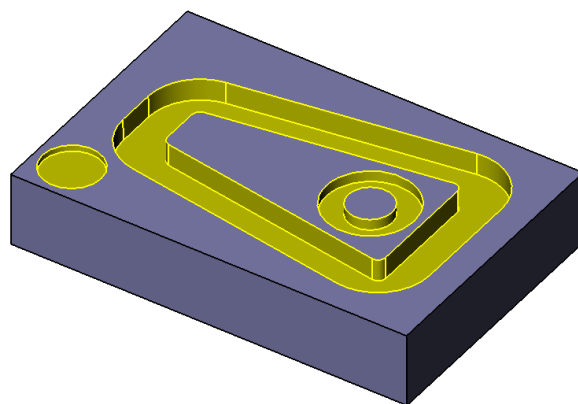
Describing the pocket Features

Right-click on the Pocket4 item and select **Add Feature**.

Select the **Pocket** option.

Change the pick filter to **Feature**.

Select the 3 pockets (shown in yellow in the figure below).



Use the Middle-click to accept the list.

Select the **OK** button from the Pocket Feature form.

Adding the Geometry Feature to the Operation Feature

Use the Right-click on the **Features of** Zigzagcut operation.

Use the **Add** option button.

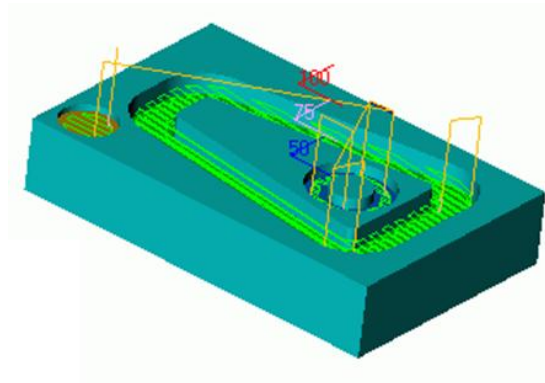
Select the **pocket 1** from the Geometry section of the tree.

Calculating the toolpath

Use the Right-click, on the Zigzagcut 1 Operation.

Select the **Calculate** option button.

See the result below and save your part.



3 Axis Milling

3 Axis Milling Operations

The 3 Axis milling operations are a broad family of tool paths appropriate for numerous disparate milling situations. Therefore, each has a number of strengths and limitations related to the tool motion technique upon which each is based. All of the operations use frames, speeds, containment, approach & retract macros, and operator control of curve and surface tolerances and thicknesses.

Geometry for these tool paths can be specified through either the use of complete CAM Components or through surface or solid CAM features. Stock can be defined explicitly by using a component within the assembly whose CAM Class Attribute is set to stock. Stock can also define implicitly using profile, surface or solid CAM features. Also refer to the Notes below.

How to define 3 axis milling operations ?

Using the CAM Plan Manager

In the CAM Plan Manager, right-click on Operations, select Insert and then select the 3 axis milling operation from the form provided. The operation will be added to the manager tree.

Select Parameters under the operation you just added and set the parameters using the definition form provided. Pick OK when you are done. Each 3 axis operation uses a unique definition form. They are shown below under each operation type. See 3 Axis Milling Operation Form Parameters for a definition of each parameter.

Notice that (undefined) is shown next to Features under the 3 axis operation you just added. This means that no features are yet defined for the operation.

1. Your CAM features (once you define them) are listed under the **Geometry** section of the manager tree. Under **Geometry**, find the CAM component (i.e., **part :name**) and expand it (i.e., select  to the left of the part) to list the CAM features currently defined for that component and select one. It will now be listed under Features for the 3 axis operation you just created. The **(undefined)** flag will be removed.

2. To create a new CAM feature: Again, under **Geometry** right-click on the CAM component, select **Add Feature** and follow the forms and prompts. You will be prompted to select geometry from the graphics window.

Map Cut Operation (3 Axis)



The map cut operation cuts any number of trimmed surfaces or solids. The cutting motion proceeds in a series of parallel cuts in one or both directions. Cutting motion is limited by the extent of the part geometry.

No boundary is explicitly defined for this operation type. When there is need for a part boundary the X, Y extent box of the part surfaces and/or solids are used. See also 3 Axis Milling Operation Form Parameters for the "Top," "Bottom" and "Max Depth" parameters.

Iso Cut Operation (3 Axis)

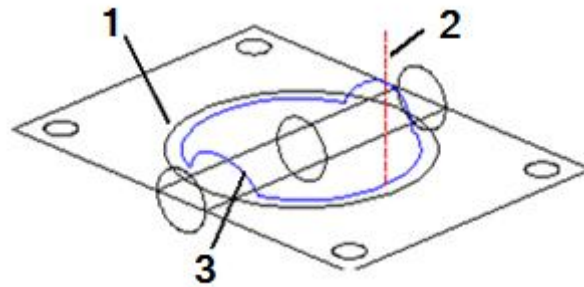


The Iso cut operation uses the isolines (lines of constant U or V parameter value) of individual surfaces to guide the tool contact point. It is a 3 axis tool path which operates on one surface at a time. It can produce a very nice surface finish because it follows the surface "**flow lines**." However, it is **NOT** a gouge-free tool path.

Profile Cut Operation (3 Axis)



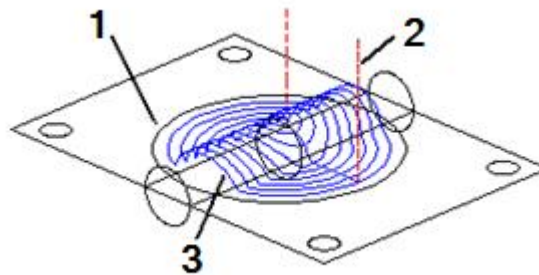
The 3 axis profile cut operation cuts any number of trimmed surfaces or solids using any number of open or closed curve boundaries to guide the tool centerline.



Sample Profile Cut Operation (3-Axis)
(1:Geometry 2:Tool Motion 3:Tool path)

Spiral Cut Operation (3 Axis)

The 3 axis spiral cut operation is an area clearance technique which cuts any number of trimmed surfaces or solids using any number of closed boundaries to enclose the cutting motion. The tool advances at each depth by proceeding toward or away from part boundaries.



Sample Spiral Cut Operation
(1:Geometry 2:Tool Motion 3:Tool path)

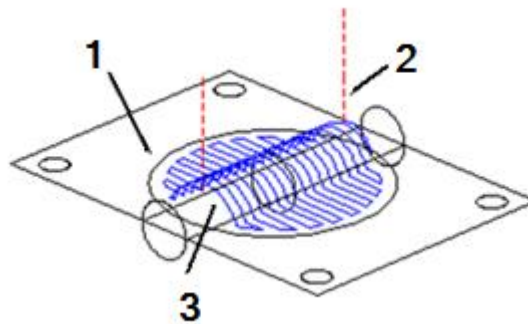
Box Cut Operation (3 Axis)

The 3 axis box cut operation is an area clearance technique, similar to 3-axis zigzag cut except the cuts are all in the same direction. The tool is lifted between each cut.

Zigzag Cut Operation (3 Axis)

The 3 axis zigzag cut operation is an area clearance technique which cuts any number of trimmed surfaces or solids using any number of closed boundaries to enclose the

cutting motion. The tool advances at each depth through a sequence of parallel cuts, reversing the tool direction at the end of each cut.



Sample Zigzag Cut Operation (3- Axis)
(1:Geometry 2:Tool Motion 3:Tool path)

Contour Cut Operation (3 Axis)



With the 3 axis contour cut operation, a medial or spine curve is calculated for each cutting zone. Tool movement proceeds in cuts generated parallel or perpendicular to that curve.

Rest Cut Operation (3 Axis)



The rest cut operation removes material remaining after previous milling operations. This can be convenient and quicker (relying on a reference tool and surface analysis) or slower and more precise (relying on a reference operation and solid verification). The precise technique allows rest milling to account for remaining stock after any number of milling operations by utilizing the work piece modeling capabilities obtained from verification.

Notes:

This operation utilizes the same tool motion projector as other 3-axis operations. The rest mill path can use either a reference tool for cleanup, or a reference operation (after solid verification) to determine the regions with remaining material.

The "Subtype" setting of the Mill Tactic Operations Form has no effect on this tool path and the "Ref Oprn" setting may or may not be specified.

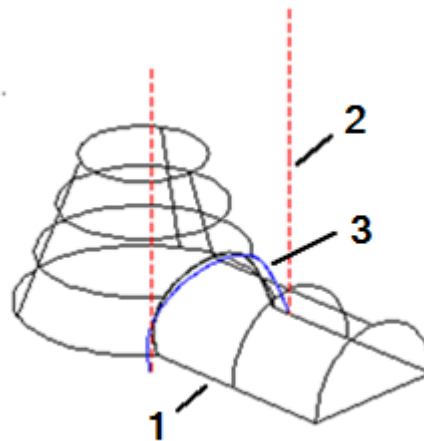
Pencil Trace Operation (3 Axis)



The pencil trace operation removes material in creases and tight curvature regions on the part. This is useful for both cleanup and from relieving material near steep surfaces in preparation for other milling operations. It utilizes the same tool motion projector as the other 3 axis operations do.

Note

This operation utilizes the same tool motion projector as other 3-axis operations. The rest mill path can use either a reference tool for cleanup, or a reference operation (after solid verification) to determine the regions with remaining material.

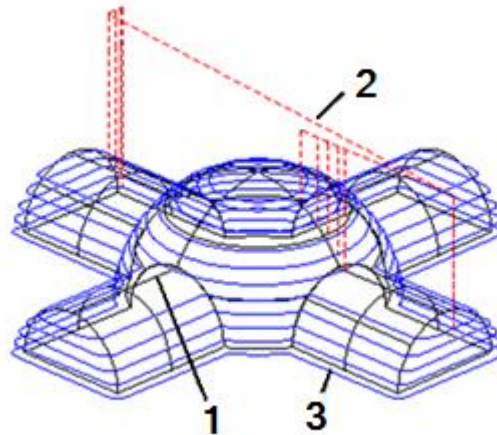


Sample Pencil Trace Operation
(1:Geometry 2:Tool Motion 3:Tool path)

Side Cut Operation (3 Axis)



The side cut operation is a 2-1/2 axis tool motion controlled by depth of cut and surface or solid geometry. The tool can ramp from one depth down to the next and the cuts can be sorted based on cutting region or depth. This operation type relies, to some extent, on the same 2-1/2 axis milling techniques used in rough milling as well as the tool motion projection utility.



Sample Side Cut Operation
(1:Geometry 2:Tool Motion 3:Tool path)

Scallop Removal Operation (3 Axis)



The scallop removal operation is a 3 axis tool path that requires the input of either a map cut or a side cut reference operation. The reference operation is specified by selecting Ref Oprn from the Mill Tactic Operations Form and selecting the appropriate map cut or side cut operation.

The direction of the reference operation's cuts and surface steepness are used to determine regions which will need more machining. The regions and then the new tool path are computed. This operation can use a different tool than the reference operation. This operation relies, to some extent, on the same 2-1/2 axis milling techniques used in rough milling as well as the tool motion projection utility.

Peeling Cut Operation (3 Axis)



The peeling cut operation combines 2-1/2 axis motion in steep areas on the part and 3-axis motion in less steep regions. This improves machining efficiency and surface finish. You have complete control over the cutting pattern for the 3 axis motion.

The cutting can be limited to just the steep areas, the less steep areas, or both in one tool path. This operation relies, to some extent, on the same 2-1/2 axis milling techniques used in rough milling as well as the tool motion projection utility.

Surface Engrave Operation (3 Axis)



This is a 3 axis surface engraving operation (i.e., characters on parts). The input requirements for this operation are the surface part and the engraving characters. All of the input characters are considered as tool ON conditions.

Notes:

As with all operations involving surfaces, care should be taken to ensure the correct side of the surface to machine is either set in modeling (test by shading) or in the CAM surface feature set. Since solids always have the normal directed outward, this need not be set for CAM components containing solids or in CAM solid feature solid sets.

3 axis operations rely on controlling the tool center by computing a motion sequence and then projecting those motions onto the part geometry. This utility is powerful and relatively quick. It is especially useful for semi-finishing and finishing operations. It is not as effective working parallel to steep regions on the part.

3 axis operations rely on a CAM profile feature or stock geometry to help limit and direct the cutter motion over the part geometry. The mapcut operation uses the part extents to determine the region to cut. If it is not possible for the tool to contact the part while following the profile, no tool motion is generated through that region (unless a "bottom" Z is specified in the operation's definition form). See 3 Axis Milling Operation Form Parameters for more about the parameters for each operation.

3 Axis Milling Exercise

The following lessons will illustrate some of the power of ZW3D CAM.

Let's begin by reviewing the methodology. The CAM tree is designed to walk you through the order of operations from top to bottom. The first step is under Geometry and is used to add the part and the stock. ZW3D allows you to navigate between the CAM plan and the part or stock by using a Right-click on the part or stock in the CAM tree and by selecting the **Edit Component** option.

To return to the CAM Plan, select the **Exit** part button.

Don't forget that a **Right-click** in the ZW3D workspace is a convenient way to access the CAM commands and an alternative to selecting items on the tree.

- After the geometry has been added to the plan, and all toolpaths calculated, we add the **Machine**. We would **Right-click** and select **Machine Manager** and then select the **Name** button to display a list of machines.
- We would enter **SNK Vertical** as the machine name and **OK** the form.

You will notice in the CAM Plan Manager that Clearances and Frames are listed under Setup1. These are already assigned default values and we will not change them here. You will see that a set of default clearance planes were automatically created when the CAM assembly was inserted into the plan.

Setup1 will also automatically use the world XY datum plane as the default Frame. We can use these clearances and frames for now. You can customize them in your future CAM planes.

Let's continue down the tree and take a look at some of the Operations. Appendix A of this manual contains matrix tables of operations called Operation Form Parameters. I recommend that you pull these out or copy them so that they can be used for quick reference.

We will begin by opening the file "CAM_Works.Z3" (program directory \ training). This part contains multiple pre-defined operations that are already for you to explore along with the CAM Fundamentals Training Guide.

Three objects will appear in the ZW3D Objects Preview window: The **Process_Part** (CAM Plan), **VX_2_3_Part** (Part) and the **VX_Stock** (Part).

Open the **Process Part CAM Plan**. You can see the part and the stock. You will also see that many operations have been added to the setup of the CAM PLAN.

Mapcut and Spiralcut Lesson

We are going to start this exercise by creating a Mapcut operation. The Mapcut operation is a simple and reliable cut pattern.

Right-click in the ZW3D work area and select the **Insert Operation** option.

Pick the **Mapcut** option in the **3 Axis** Milling section of the Operation Type form. You can see that the operation immediately appears at the bottom of the CAM tree.

Pick the **Tool** object under the mapcut 1 operation to open the **ToolList** form.

Select the **Ball Endmill D6** tool from the list.

Right-click on **Features** and select **Add** to add the Surface 1 geometry from the **Geometry** section of the tree (unfold the Part: VX_2_3_Part) to the operation.

Pay close attention to the command prompt:

Select Part/Surface (Optional: Contain)

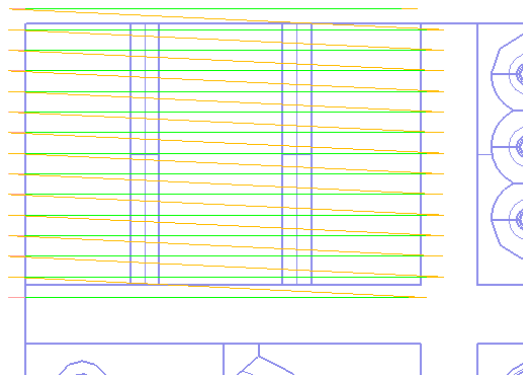
Each toolpath has different feature requirements. As you can see there is no stock requirement.

Remember, once you are familiar with the menus, Double left-click on the tree will activate the first command on the menu.

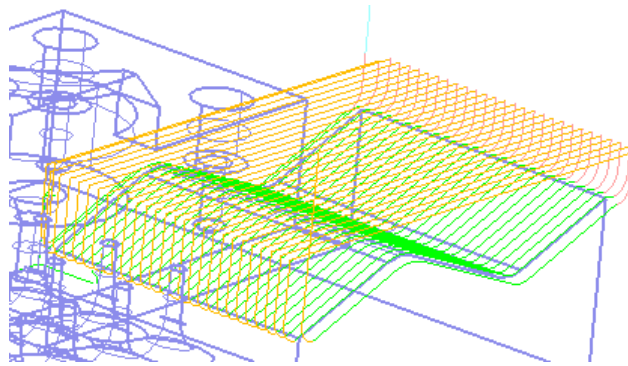
Stock is not used in finish tool paths so let's turn it off for now.

Double click on the Stock or Right-click and select **Show/Hide** to Hide the stock.

The next step is to calculate the tool path by selecting the **mapcut 1** operation with the right mouse button and then picking the **Calculate** option. You can see the green cut lines and the orange tool movement lines on the part below.



Top View



Iso View (from the back)

Now we need to move the operation up the tree. Select **Output** at the bottom of the tree and the **Output form** is displayed. Highlight the Mapcut 1 operation under the **Organize** tab.

Move the **mapcut 1** operation to just below the two **profilecut** operations by using the **Move Up** button

Click **Dismiss** to close the form.

For the sake of this exercise, we have decided that we are not happy with the operation so we are going to make some changes.

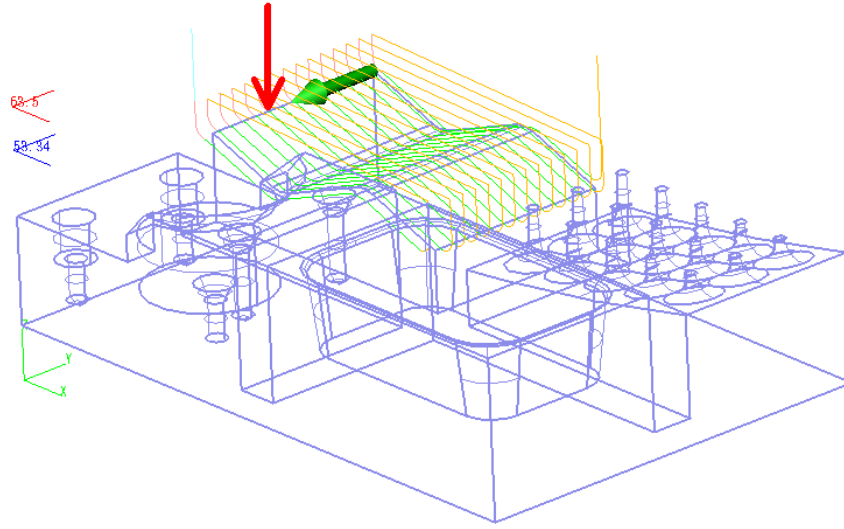
11. Right-click on the **Parameters** item listed below the Mapcut 1 operation and pick the **Edit** button. The **Cutting Parameters** elements are displayed.

Change the **Surface Thick** to .3 and change the **Step Value** to 30. The step value changes the distance between tool passes.

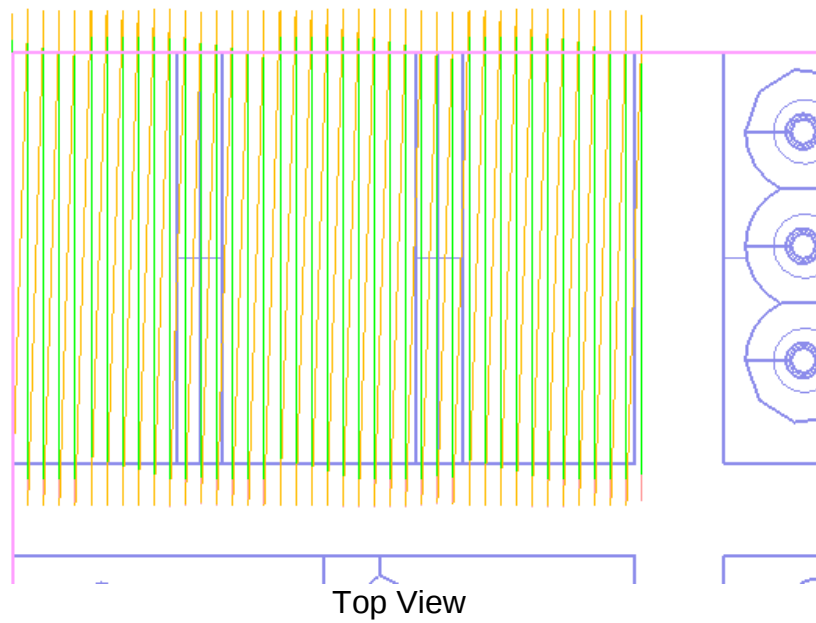
We are also going to change the cut direction by selecting the **Cut Dir** button and picking the edge of the part as indicated in red arrow below. ZW3D will display a large green arrow to indicate which direction the toolpath will follow.

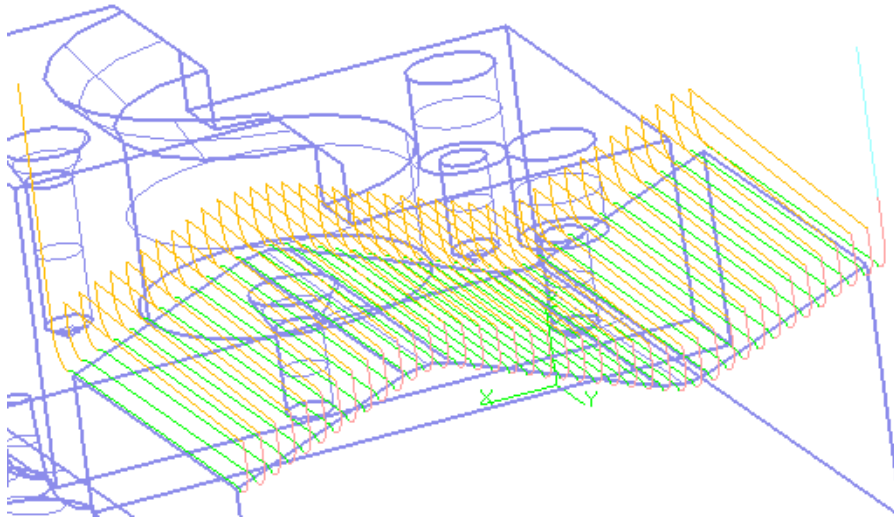
Click **Calculate** at the bottom of the form.

See the result below.



Your part should now look like the part shown below. Notice that the tool is now cutting across the width of the surface

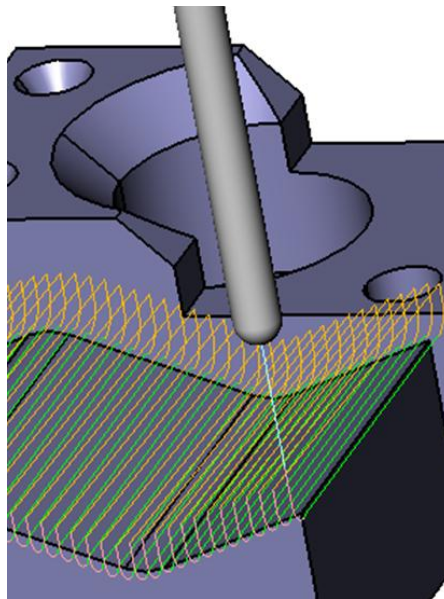




Iso View

Now we will verify the toolpath by a **Right-click** on **mapcut 1** and selecting the **Verify** option. The **Toolpath Verify** form will appear as well as a graphic of the tool.

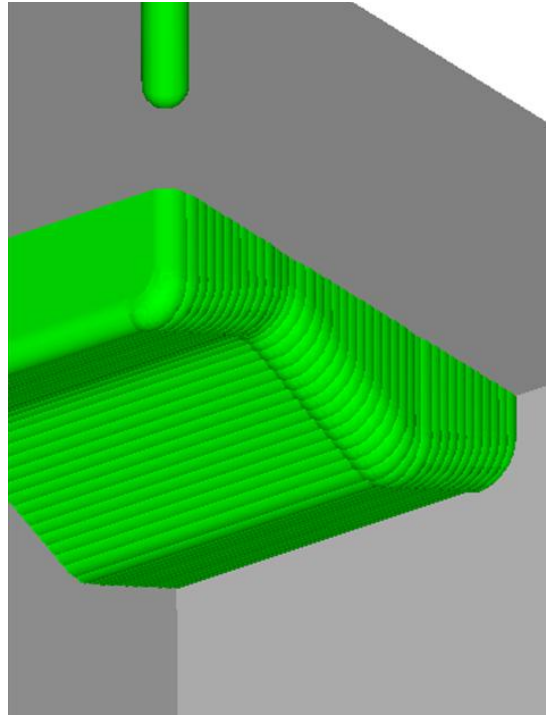
Select the “>” button on the form to watch the tool follow the toolpath.



We can also perform a solid verification of our toolpath by right-click. Pick the **SolidVerification** option to see the Solid Verify Session form.

The Solid Verify session form is displayed.

Click on **Run simulation**.



Notice the colors have changed on our Stock and Part. These are default colors that can be found under **Options** button on the Solid Verify session form. Also, the only operation performed was the mapcut 1 operation because we told ZW3D to skip suppressed operations and Verify active operation only. To suppress an operation, go back to the **Organize** tab after selecting Output and highlight the operation, pick the **Suppress** button. Parentheses will appear around the suppressed operation. Highlight and pick the **Unsuppress** button to remove them.

Otherwise, all of the operations in this CAM plan would have been verified if **Verify active operation only** option is unchecked.

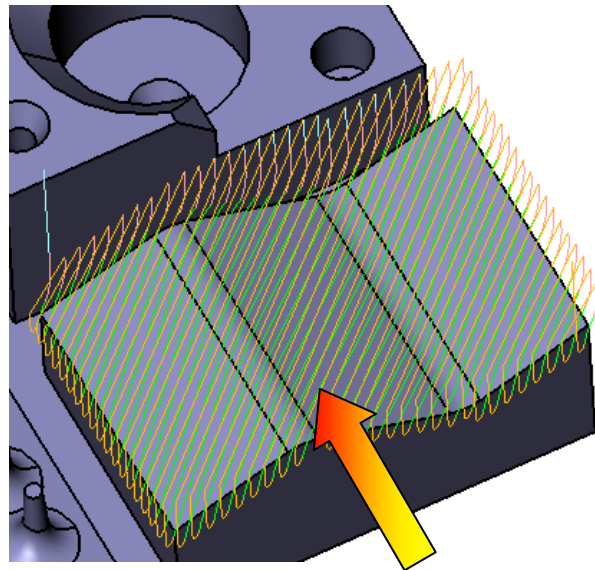
Run the simulation again, this time try to **Pause**, then click on the **Show Tpath Data** tab to see the tool path data, and later **Step forward run simulation**.

We are going to make one more edit to this part before we move on.

Return to the **Parameters** item under the mapcut 1 operation and Right-click to choose the **Edit** button. Select the **Cut Dir** button again and type in 45. This will cause the toolpath to cross the part at a 45 degree angle as shown below.

Also try -45.

Click on the **Calculate** button and ZW3D will recalculate the toolpath. You will need to do this every time that you make changes to the operation Parameters. The part should look like the picture below.

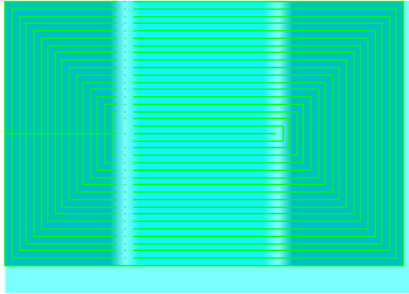


Toolpath at 45 Degrees

Return to the Output item on the tree and solid verify just the new toolpath.

Let's take a look at another operation. We are not going to review all of the operations in this CAM Plan but they are there for you to examine and experiment with. Remember, you can always close without saving if you have created something undesirable. You can also delete and re-create operations in the tree if you wish.

1. Double click on the icon of **Mapcut 1** to **Hide** it.
2. Double click on the icon of **3X Spiral Cut 1** to **Show** it. The Figure below shows the **spiral cut1** operation over the upper left-hand surface of the model.
3. Taking a look at the features, you might ask, How do I know what **surface 1** is? **Left-click** on it and it will highlight.
4. Now that it is yellow, how do I return it to its original color? (See bottom of page) **Right-click** on the **Part: ZW3D_2_3_Part (1)** and select the **Edit Component** option. ZW3D will send you to the Part level for modeling.
5. **Right-click** on the geometry in the tree and choose **Unhighlight** as shown below.



Top view of 3X spiral cut 1 over surface 1

Open the 3-Axis Spiral Cut Operation Definition form under the 3X Spiral Cut 1 operation. (Remember how? **Parameters** option)

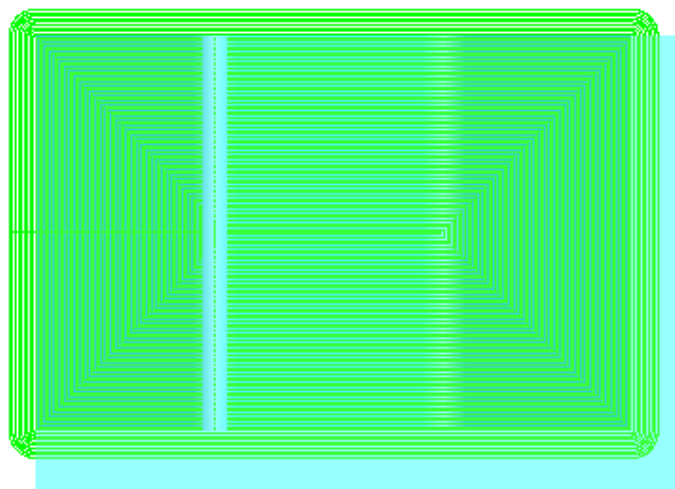
3x spiral cut 1 cutting parameters form is displayed. We are going to change some of the cutting parameters and then recalculate the toolpath to see how the changes have manipulated the operation.

We are going to change the **Step Value**, the **Curve Tol**, and the **Surface Tol** values. Change the **Tool Location** and see what happens to the toolpath on the edge of the part. The current tool location –on boundary, places the center of the tool on the edge of the geometry.

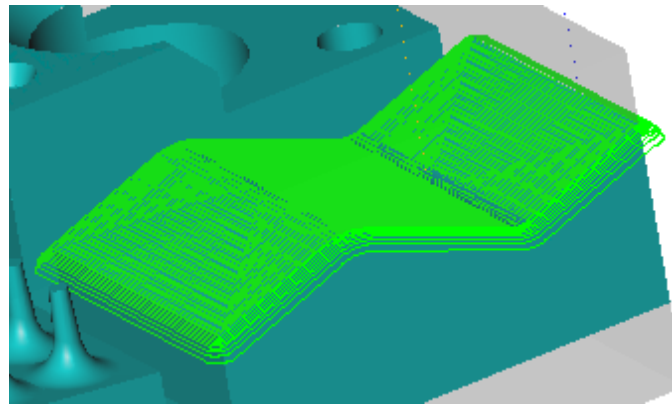
Here is a best practice hint: The Curve Tolerance and Surface Tolerance should be no larger than the scallop height. The scallop height is a numerical value and not a percentage value.

Change the **Curve Tol** and **Surface Tol** to 0.005. Set the **Step Value** to 0.005. Change the Tool Location to **past boundary**.

Calculate!



Top View



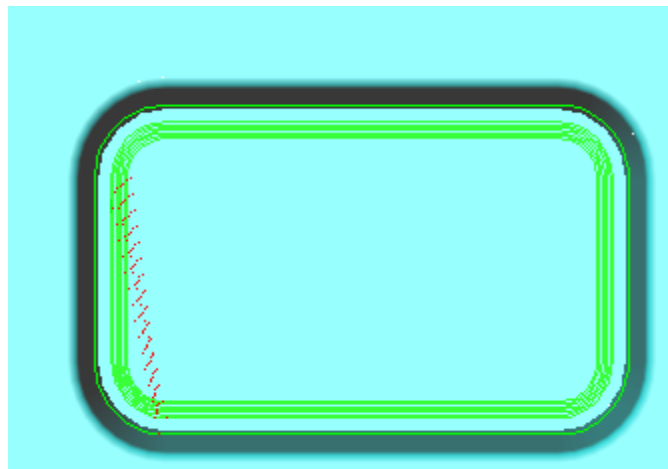
Auxiliary View

Your toolpath should look like the pictures above.

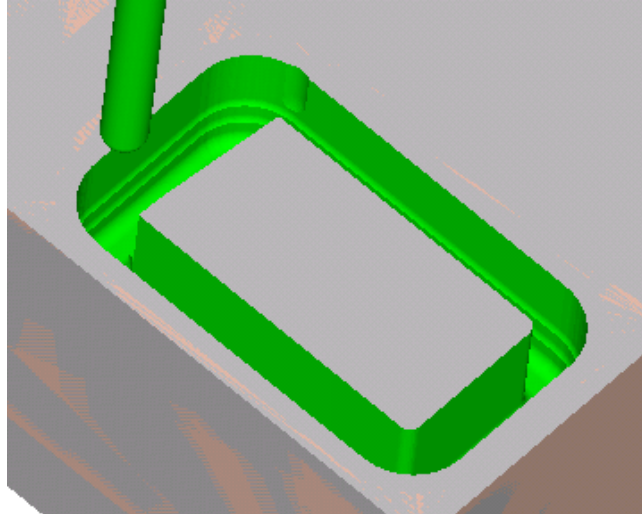
Sidecut and Boxcut Lesson

We are going to try another operation and then invite you to explore the remaining 3 Axis operation types on your own. If you have questions, please do not hesitate to contact your local ZW3D support Applications Engineer.

From the CAM Tree, select the **Sidecut 1** operation and with a **Right-click** and select **Show** the toolpath.



Top View



Auxiliary View (Solid Verification)

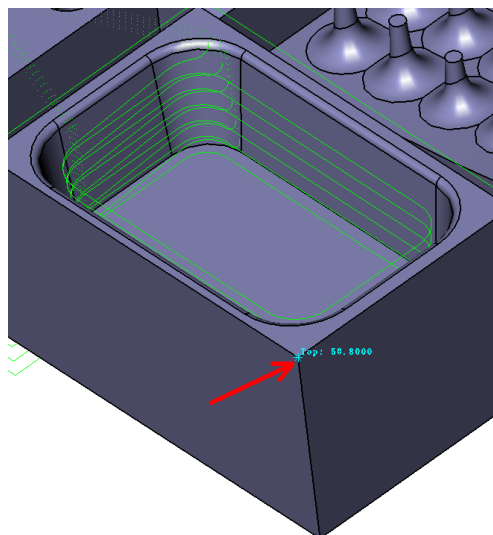
Observe the toolpath and the approach in the figures on the right. This operation, by nature, leaves the center of the feature still intact (Solid Verify) so we would need to come back with a different operation to finish creating this pocket.

Pick the Parameters item under the Sidecut 1 operation to display the Operation Definition form for the Sidecut operation that we just verified. Take note of the **Top**, **Bottom** and **Max Depth** values.

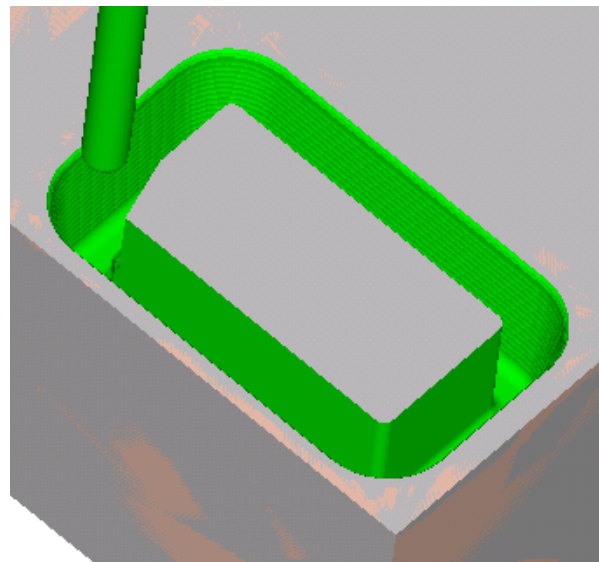
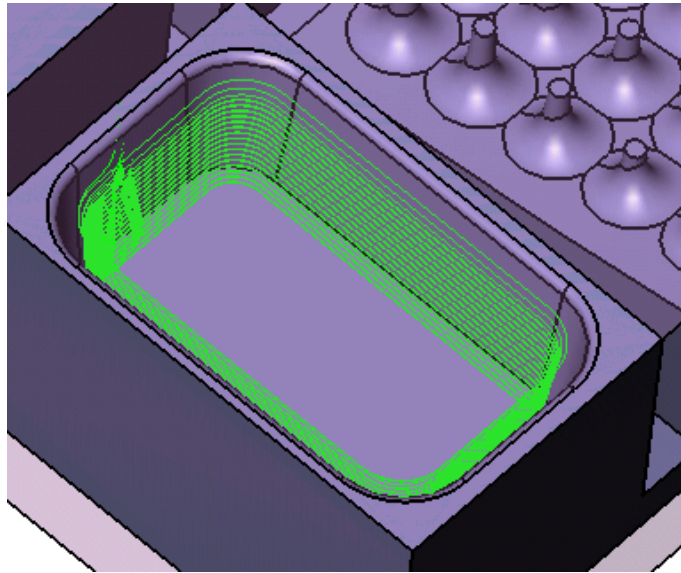
Select the **Top** button and then the top corner of the part as indicated in the picture below. Set **Max Depth** to 3. As for the **Bottom**, **we will leave that value blanked** so that the bottom surface of the geometry drives the toolpath.

Pick **Calculate** button.

See the toolpath below.



Next, change the **Max Depth** value to **1.0**. The **Max Depth** value is the depth that the tool cuts into the stock in a single pass. After making these changes to the **Cutting parameters**, **Accept** the form. To view the way that these changes affect the toolpath, **Right-click** on the operation and select **Calculate**. Your part should look like the figures below.



Compare these two figures to the figures of the toolpath before we made the changes to the Tool Parameters.

As stated previously, the sidecut 1 operation leaves the center of the geometry still intact and that we need to use yet another operation to finish the part. Using the sidecut 1 operation first allows us to use the box cut 1 operation to clear the center of the pocket without breaking the tool.

The box cut 1 operation has already been defined in this CAM Plan and we will not change the parameters for this lesson. Pay attention to the Output command instead. Remember that we can have multiple operations execute in series and, of course this entire CAM Plan will execute from the first operation through the last.

We are going to do a Solid Verify operation on just the sidecut 1 operation and the box cut 1 operation together. The purpose of this is to show you how to combine operations. The operations do not have to be back-to-back in the CAM Plan as these two are.

Pick the **Output** item at the bottom of the tree. From the **Organize** tab, pick an operation and then pick the **Suppress** button (one at a time). Do this to all of the operations in the list except the sidecut 1 and 3X box cut 1 operations.

Select the **Verify** Tab, select the **Start Verify** button, check the **Skip suppressed operations** box and DO NOT check the **Verify active operation only** box this time.

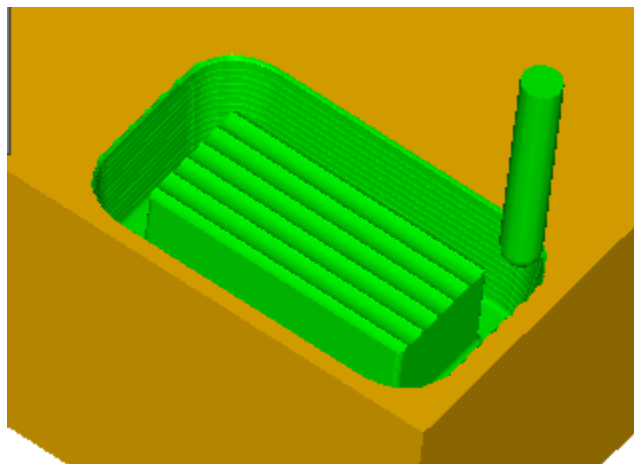
Run the simulation.

The tool will move through the sidecut 1 operation and then move immediately through the box cut 1 operation as shown below.

Great Job!

That's it for 3 Axis Milling for now.

Try creating more Multiple Operations on your own!



Appendix B Operations Form Finishing Parameters

Appendix B

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

Operations Form Parameters

Quick Mill Operation Form Parameters

The following tables list all of the parameters located on the Quick Mill finishing operation definition forms. Many are common to all operations. Others are unique to specific operations. Refer to each Quick Mill operation topic for additional and specific instructions

Quick Mill Operations Form Parameters verses Operation Type												
Form Parameters	Offset 3D	Lace	Drive Curve	Z Level	Pencil Cut	Flow 3D	Engrave 2D	Bulge	HSM Smooth Flow	HSM Offset 2D	HSM Lace	HSM Flow
Primary Parameters												
Frame	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
Path Tolerance	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
Surface Tolerance	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
Surface Thick	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
Z Surface Thick	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
Stepover Type	✓	✓				✓			✓	✓	✓	✓
Scallop	✓	✓				✓			✓	✓	✓	✓
Absolute		✓				✓			✓	✓	✓	✓
Min Stepover Value		✓										
% Tool Diameter		✓		✓		✓			✓	✓	✓	✓
Stepdown				✓								

Quick Mill Operations Form Parameters verses Operation Type												
Form Parameters	Offset 3D	Lace	Drive Curve	Z Level	Pencil Cut	Flow 3D	Engrave 2D	Bulge	HSM Smooth Flow	HSM Offset 2D	HSM Lace	HSM Flow
Primary parameters continued...												
Scallop				✓								
Stepdown Size				✓								
Z Min Step Size				✓								
Directrix Step								✓				
Number of Cuts			✓		✓							
Stepover			✓		✓							
Scallop			✓		✓							
Stepover Size			✓		✓							
% Tool Diameter			✓		✓							
Z Step Value			✓		✓							
% First Step			✓		✓							

Quick Mill Operations Form Parameters verses Operation Type												
Form Parameters	Offset 3D	Lace	Drive Curve	Z Level	Pencil Cut	Flow 3D	Engrave 2D	Bulge	HSM Smooth Flow	HSM Offset 2D	HSM Lace	HSM Flow
Path Setting Parameters												
Cut Angle		✓									✓	
Cut Direction	✓	✓	✓	✓	✓	✓		✓	✓	✓	✓	✓
Path Pattern	✓		✓							✓		
Z Progress				✓								
% Smoothing	✓		✓		✓				✓	✓	✓	✓
XY Corner Radius				✓								
Z Corner Radius	✓	✓	✓		✓	✓			✓	✓	✓	✓
Enhance Corners	✓		✓	✓		✓			✓	✓		
Pencil XY Range			✓									
Synchro Z Level				✓								
Start Points				✓								
Cut Order				✓								
% Sensitivity					✓							
Path						✓						✓

Pattern												
Quick Mill Operations Form Parameters verses Operation Type												
Form Parameters	 Offset 3D	 Lace	 Drive Curve	 Z Level	 Pencil Cut	 Flow 3D	 Engrave 2D	 Bulge	 HSM Smooth Flow	 HSM Offset 2D	 HSM Lace	 HSM Flow
Path Setting Parameters continue...												
% Max Distance							✓					
% Bulge								✓				
Noise X								✓				
Noise Y								✓				
Noise Z								✓				
Add Finish Pass									✓	✓	✓	
%Finish Distance									✓	✓	✓	
Path Prelink										✓	✓	✓
Advanced Cut Direction			✓		✓							
Path Pattern					✓							

Quick Mill Operations Form Parameters verses Operation Type												
Form Parameters	Offset 3D	Lace	Drive Curve	Z Level	Pencil Cut	Flow 3D	Engrave 2D	Bulge	HSM Smooth Flow	HSM Offset 2D	HSM Lace	HSM Flow
Advanced Parameters												
Spiralization												
Min Curve count	✓	✓	✓	✓	✓	✓				✓	✓	
% Aggressivity	✓	✓	✓	✓	✓	✓				✓	✓	
Overmill	✓	✓	✓		✓	✓				✓	✓	
Uniform Cut												
XY Step Value UC		✓		✓					✓	✓	✓	
Z Step Value UC		✓		✓					✓	✓	✓	
% First Step UC		✓		✓					✓	✓	✓	
% Smoothing UC		✓		✓					✓	✓	✓	
Path Pattern UC		✓		✓					✓	✓	✓	
Enhance Corners UC		✓		✓					✓	✓	✓	
Uniform Cuts First				✓								
Advanced Setting												
% Interference				✓								
% Overstep									✓			

Operations Form Parameters

2-1/2 Axis Operation Form Parameters

The following tables list all of the parameters located on the rough milling operation definition forms. Many are common to all operations. Others are unique to specific operations. The first table is a matrix of form parameters verses the operation types that support them. The second table lists a description of each form parameter and their options (if any). Refer to each 2-1/2 axis milling operation topic for additional and specific instructions.

2-1/2 Axis Milling Operation Form Parameters verses Operation Type									
Form Parameter	 Spiral	 Box	 ZigZag	 Contour	 Profile	 Chamfer	 Helical	 Corner	 Ramp
Frames, Speeds and Feeds (all operations)									
Path Setting parameters									
Air Access		✓	✓						
Corner Control	✓		✓			✓		✓	✓
Cut Angle	✓			✓	✓	✓		✓	✓
Cut Direction				✓					
Cut Order	✓	✓	✓	✓	✓	✓	✓		
Z Progress							✓		
HSM Mode		✓							
Pitch							✓		
Profile Side					✓				✓
Ramp Angle									✓
Allow Tool Lift	✓				✓	✓			
Slowdown Distance	✓	✓	✓	✓	✓	✓		✓	
Start Angle							✓		
Path Pattern	✓								
Lift in Region	✓		✓		✓				
Tip Depth						✓			
Primary parameters									
Number of Cuts					✓	✓	✓		
Bottom Thick							✓		
Side Thick							✓		
Stepover	✓	✓	✓	✓	✓	✓			
Stepover value	✓	✓	✓	✓	✓	✓	✓		

Path Tolerance	✓	✓	✓	✓	✓	✓	✓	✓	✓
Tool Compensation	✓				✓	✓	✓	✓	✓
Island Cleanup	✓	✓	✓	✓	✓				
Island Top Clean	✓	✓	✓	✓	✓				
Pre-Drill Holes	✓	✓	✓	✓	✓	✓			
Side Finish Type	✓	✓	✓	✓	✓				✓
Start Points	✓				✓			✓	✓
Tool Home Start	✓	✓	✓	✓	✓	✓	✓	✓	✓
Tool Home End	✓	✓	✓	✓	✓	✓	✓	✓	✓
Limiting parameters									
Tool Location	✓	✓	✓	✓	✓				✓

2-1/2 Axis Milling Operation Form Parameters verses Operation Type									
Form Parameter	 Spiral	 Box	 ZigZag	 Contour	 Profile	 Chamfer	 Helical	 Corner	 Ramp
Link and Lead parameters-Others									
Approach Distance							✓		
Check all of Part	✓	✓	✓	✓	✓	✓	✓	✓	✓
Engage Arc Radius %							✓		
Retract Arc Radius %							✓		
Return Level							✓		
Auto engage/retract		(see options and icons below)							
Linear 	✓	✓	✓	✓	✓	✓		✓	✓
Linear, Linear 					✓	✓		✓	✓
Linear, Circular 					✓	✓		✓	✓
Circular 	✓	✓	✓	✓	✓	✓		✓	✓
Circular, Linear 	✓	✓	✓	✓	✓	✓		✓	✓
 Circular, Linear, Linear					✓	✓		✓	✓

Circular, Slope					✓	✓		✓		✓
None	✓	✓	✓	✓	✓	✓		✓		✓

2-1/2 Axis Milling Operation Form Parameters verses Operation Type									
Form Parameter	 Spiral	 Box	 ZigZag	 Contour	 Profile	 Chamfer	 Helical	 Corner	 Ramp
Link and Lead Tab – Auto Engage/Retract									
Safe Distance	✓	✓	✓	✓	✓	✓		✓	
Engage Arc Radius	✓	✓	✓		✓	✓			✓
Engage Overlap	✓	✓	✓	✓	✓	✓		✓	✓
Activation Range	✓				✓	✓			✓
Link and Lead Tab – Link									
Horizontal Clear	✓	✓	✓	✓	✓	✓			✓
Traverse Type	✓	✓	✓	✓	✓	✓			
Vertical Clear	✓	✓	✓	✓	✓	✓			✓
Slope Angle					✓	✓			✓
Plunge Type	✓			✓					

3 Axis Milling Operation Form Parameters

The following tables list all of the parameters located on the 3 axis milling operation definition forms. Many are common to all operations. Others are unique to specific operations. The first table is a matrix of form parameters verses the operation types that support them. The second table lists a description of each form parameter along with their options (if any). Refer to each 3 axis milling operation topic for additional and specific instructions.

3 Axis Milling Operations Form Parameters verses Operation Type														
Form Parameters	 Map	 Iso	 Profile	 Spiral	 Box	 ZigZag	 Contour	 Rest	 Pencil	 Side	 Scallop	 Peel	 Surf Engr	 3X Iso
Cutting - Tab														
Frames, Speeds and Feeds (all operations)														
Containment	✓		✓	✓	✓	✓	✓	✓	✓	✓		✓	✓	
Auto Cut Direction					✓	✓					✓	✓		
Bi-Direction					✓	✓		✓						
Bi-Dir Slope								✓						
Bottom	✓	✓	✓	✓	✓	✓	✓	✓		✓		✓	✓	
Bottom Limit %														✓
Corner Loop				✓				✓				✓		
Curve Thickness	✓	✓	✓	✓	✓	✓	✓	✓		✓	✓	✓	✓	
Curve Tolerance	✓	✓	✓	✓	✓	✓	✓			✓	✓	✓	✓	
Cut Direction	✓				✓	✓		✓			✓	✓		
Cut Order									✓				✓	✓

3 Axis Milling Operations Form Parameters verses Operation Type														
Form Parameters	Map	Iso	Profile	Spiral	Box	ZigZag	Contour	Rest	Pencil	Side	Scallop	Peel	Surf Engr	3X Iso
Cut Overlap								✓		✓		✓		
Cut Pattern								✓			✓	✓		✓
Cut Region								✓				✓		
Cutting Mode		✓	✓	✓			✓	✓	✓	✓	✓	✓		✓
Horizontal Thickness										✓				
Max Depth	✓	✓	✓	✓	✓	✓	✓			✓			✓	
Min Flat Length									✓					
Min Steep Length									✓					
Number of Cuts			✓											
Pre-drill Points			✓	✓	✓	✓	✓							
Profile Side			✓											
Ramp Angle								✓		✓		✓		
Reference Tool								✓						
Region Slope								✓						
Rest Type								✓						
Side Cleanup				✓	✓	✓	✓							
Side Region										✓		✓		
Slope Limit					✓	✓					✓	✓		
Spiral Progress				✓										
Start Point													✓	
Steep Angle									✓					
Step Type	✓	✓	✓	✓	✓	✓	✓	✓			✓	✓		✓

Step Value	✓	✓	✓	✓	✓	✓	✓	✓			✓	✓		✓
Surface Thickness	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
Surface Tolerance	✓	✓	✓	✓	✓	✓	✓			✓	✓	✓	✓	✓
Tolerance								✓	✓					
Tool Location			✓	✓	✓	✓	✓						✓	
Top	✓	✓	✓	✓	✓	✓	✓	✓		✓		✓	✓	
Top Limit %														✓
Under Cut								✓		✓		✓		
Vertical Thickness										✓			✓	
Leads and Links - Tab														
Full Approach	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	
Partial Approach	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	
Full Retract	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	
Partial Retract	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	

Appendix C Operations Type - Icon Maps

QuickMill Operations Type Icon Map

QuickMill Operations

Roughing



Finishing



High Speed Fishing



2 & 3 Axis Milling Operations Type Icon Maps

2X Mill Operations

Holes



Center



Drill



Peck Drill



Chip Drill



Ream



Bore



Fine Bore



Counter Bore



Countersink



Tap

Milling



Spiral Cut



Zigzag cut



Box cut



Contour cut



Profile Cut



Chamfer Cut



Corner Round



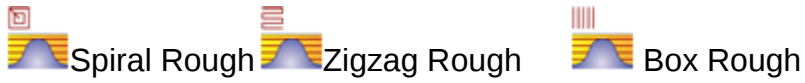
Helical Cut



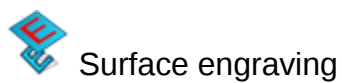
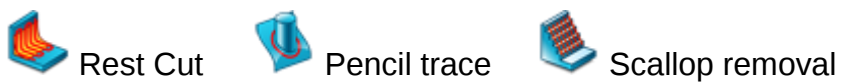
Ramp Cut

3X Mill Operations

Roughing



Finishing



5 Axis Mill Operations Map



Included here for reference only.

5 Axis milling is covered in the “***CAM Advanced training manual.***”

Directive Operations Map

