# 1. TABLE OF CONTENTS

# CHAPTER 1.
# EZ-MILL PRO / 3D MACHINING WIZARD TUTORIAL

## OVERVIEW

This tutorial is intended to present to you the 3D Machining Wizard, which makes 3D part programming easier than the regular Work Step Data window. The 3D Machining Wizard collects information about the most important machining parameters from the user, in a brief window, avoiding confusing settings. Some parameters are pre-defined to the most appropriate values for the current machining method; however, the user always has the option to alter them as to his needs.

The solid model can be imported into EZ-MILL Pro in several file formats, like sldprt, igs, step, sat, x_t or x_b. Beginning from version 19, SolidWorks users can transfer the 3D model directly from the SolidWorks working window to EZ-CAM in one click. The 3D Machining Wizard automatically detects and assigns all the surfaces to the recently created Work Step; however, the user can choose to deselect any of them.

Throughout the tutorial you will find important notes , tips or references to the online help where additional information on the commands and functions is provided.

## CAVITY MACHINING

In this tutorial we will machine a detergent bottle mold. It is a cavity of a bottle divided from the centerline. It has a boss corresponding to the handle portion, which creates an island in the middle of the cavity. The mold cavity has both steep and flat sections which we need to consider at the finishing stage. Surface transitions have fillets in several dimensions.

The first operation will be roughing with a large tool to get rid of the bulk material inside the cavity. Secondly we will apply a re-roughing workstep to machine the uncut material with a relatively small tool. The finishing operation will be carried out by a 3D Equidistant machining method providing even surface roughness throughout the mold. Finally we will introduce a re-finishing technique and cut the remaining material left by the relatively lager tool of the previous finishing operation.

## BASIC PROGRAMMING STEPS

Before we continue with the tutorial let us explain the basic steps needed to create the part program.

**STEP 1**. **Load (Import) the 3D Solid Model of the Part to be machined**
For our sample part we will use a 3D model created in Rhino and saved in ACIS format "sat". Other common CAD translation formats can also be used to import external data to EZ-Mill Pro, such as igs, 3dm, step, x_t, x_b, etc. SolidWorks users can directly transfer the solid model to the EZ-CAM window using the "Update Solidworks Model" command.

**STEP 2.** **Define World/UCS origin on the 3D Solid Model**
We will change the World/UCS origin point to a reference location on the real part stock.

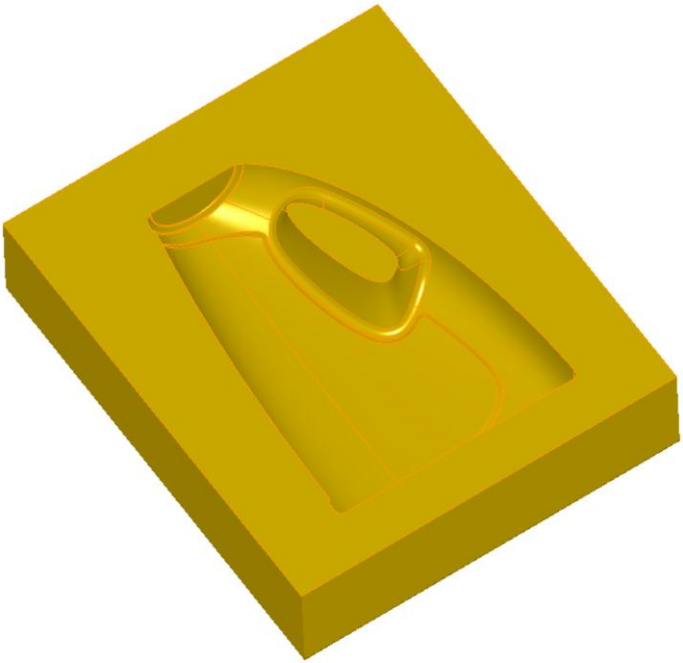
**STEP 3.** **Create Work Steps and set all Machining Parameters**
We will create machining work steps, specify tool and operation parameters, create path Curves (if required), and verify the toolpaths. Also we will create rough and re-rough surface milling operations, leaving stock on all surfaces then finish and re-finish surface milling operations; removing the remaining material, leaving minimal cusps.

**STEP 4.** **Check Material Removal and Surface Finish**
We will check to make sure all material is removed and the surface finish smoothness is acceptable.

The EZ-MILL Pro 3D Machining Wizard Tutorial is set up in Inch with all Inputs and Dimensions in Inch !

## THE PART

Below is the image of the sample part we will machine in this tutorial. It is a detergent bottle mold with an approximate size of 13.5”x11.5”. The cavity of the mold has steep walls near the top surface, and a boss with steep walls corresponding to the handle of the bottle. There are fillets with radii changing between .079” and .2” in surface transition sections.

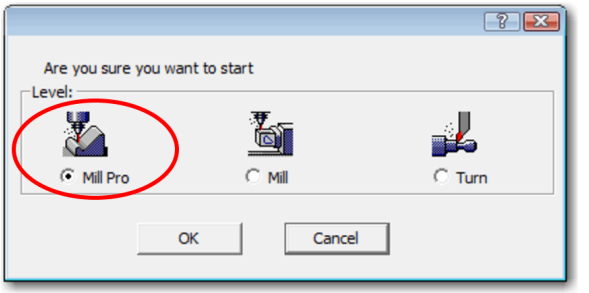


**Picture 1-1**

## SETTING THE PREFERENCES

Let's first set some important parameters such as units and other viewing preferences to ensure the compatibility of your system with the tutorial.

1. Select the "New" command from the "File" menu to restart EZ-Mill Pro and to clear the memory before continuing with the tutorial. Make sure the "EZ-Mill Pro" level is active and press OK to start over.




The "New" dialog is also used to switch between the EZ-Mill and EZ-Turn modules. Before the dialog opens the system checks the software protection key for activated modules. Modules or levels that are not activated will be marked by appended "DEMO" text. When working in "Demo" (evaluation) mode, it is not possible to print or save data. The corresponding "Save", "Save as" and "Print" commands are disabled.

When closing the EZCAM application, the system automatically stores the last used level as default for the next session.

2. Select the "Setup" command from the "View" menu

3. Select the "Inch" option button as the input dimension system.

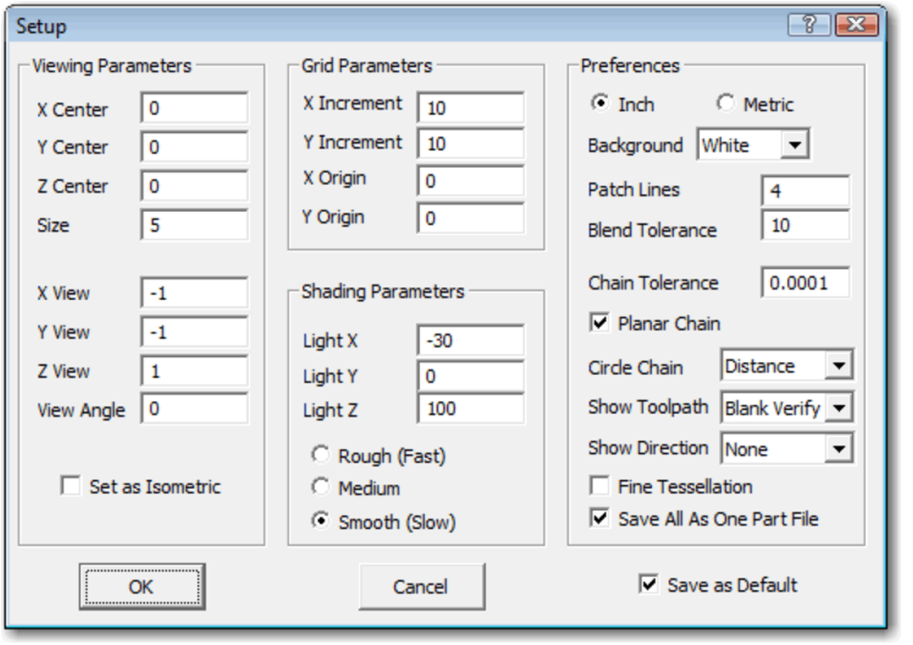
4. Click the "Background" list box and select "White".

5. Click the "Show Toolpath" list box and select "Blank Verify". This will cause verified tool paths to be blanked when the screen is redrawn.

6. Click the “Show Direction” list box and select "None". This will hide the small arrows indicating the surface normals and curve directions. It can later be activated at any time.

7. Check "Save as Default". The system will store all dialog settings as defaults for future sessions.

8. After the preferences have been correctly set, click OK.




**Picture 1-2**

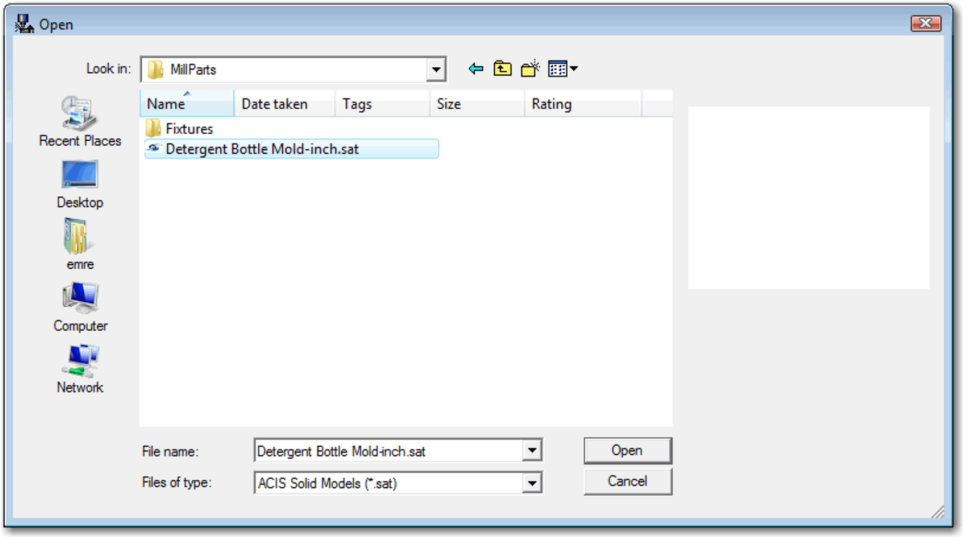
The initial setup for the EZ-MILL Pro 3D Machining Wizard Tutorial is now complete. Continue with the next section to create the geometry necessary for this part.

## LOADING THE SOLID PART

We will begin the exercise by loading the surface geometry that represents our part. There are several methods to accomplish this process. The first method is to import a solid/surface type of model into EZ-CAM in one of the common CAD file formats, such as iges, sat, step, sldprt, sldasm, x_t, x_b, dwg, vda or 3dm. Secondly, you can open the model in our companion CAD product, Rhino, then copy/paste to EZ-Mill using the special command "Paste from Rhino" found under the "Edit" menu. The last method is for SolidWorks users; they can load the model into SolidWorks and transfer the file to EZ-CAM by using the new "Update Solidworks Model" command. In this tutorial we choose the first method and load a SAT file representing the 3D model.

You may refer to "EZ-Mill 3D Machining Solids Tutorial" to get information about "Copy and Pasting the Model from Rhino"

The "SAT" file containing the 3D model has already been copied into the "EZCAMW\MILLPARTS" folder by the EZ-CAM setup. Follow the steps below to load the data.




**Picture 1-3**

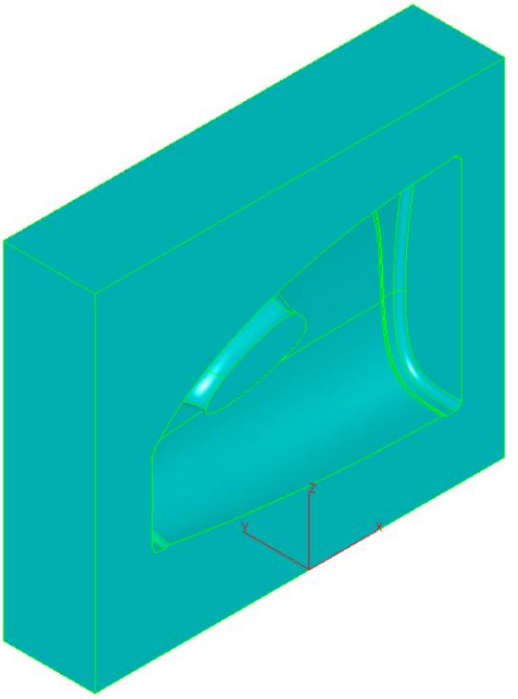
1. Select the "Open" command from the "File" menu to open the file dialog. In **Picture 1-3** you can see the dialog displayed on a Windows Vista workstation system. This

dialog may vary according to the version of the Windows™ operating system running on your machine.

2. Select the folder “EZCAMW \ MILLPARTS” on the drive where you installed the software

3. In the “Files of Type” list select “ACIS Solid Models (*.sat)”.

4. Select the file “Detergent Bottle Mold-inch.sat” and click the “Open” button. The imported surface geometry should then appear as shown in **Picture 1-4**.



**Picture 1-4**

## DEFINING WORLD/UCS ORIGIN OF THE PART

We will define the part origin point as the zero position that will be assigned to the stock before machining on the CNC machine. In this tutorial the front left corner of the part block will be selected as the origin point--using the default coordinate system "World". The top plane of the part will be designated as Z=0 position.

1. To define the new "World" origin point, select the "World/UCS, Three Points" command from the "View" menu or click the corresponding button.

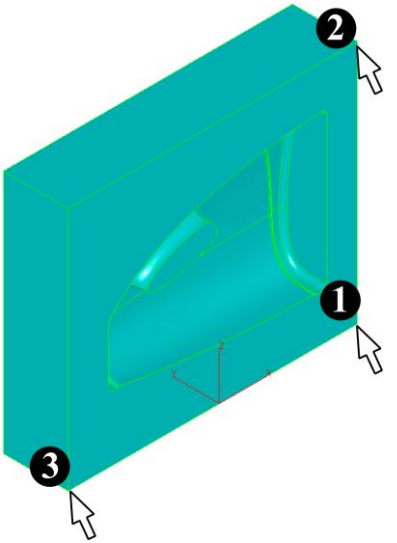
   World/UCS, Three Points

2. Select the "Snap All" command from the "Edit > Point Picking" menu or click the corresponding button.

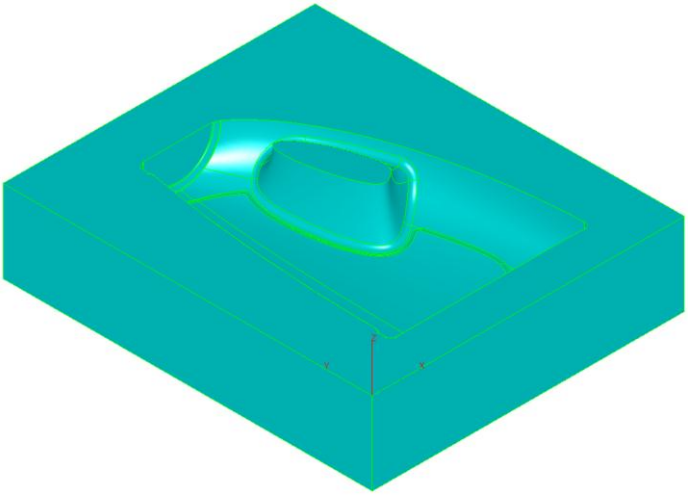
   Snap All

3. Click the corner points 1, 2 and 3 on the part; See **Picture 1-5**. These points designate new origin point, X-axis direction and Y-axis direction respectively.




**Picture 1-5**

4. The part will be oriented as to the new position of “World” coordinate system that is appropriate for machining as shown in **Picture 1-6**.



**Picture 1-6**

## CREATING THE PART PROGRAM

Now that we have completed setting the origin point of the part we are ready to begin defining the work steps to machine it. Each work step uses a specific method (Roughing, Finishing, etc.) and a Toolpath Type (Pocketing, Parallel, etc.) along with its associated tool settings and parameters to create a toolpath that machines the assigned curves and surfaces. Our part program will consist of the following work steps.

| Work Step ID | Purpose |
|---|---|
| Roughing | Rough the cavity leaving a stock allowance of .005" with a .625" flat tool. We will use a Pocketing toolpath pattern generated within the boundaries of each surface slice computed using a Z-step value of .04". |
| Re-Roughing | Machine the uncut material sections where the previous tool of the Roughing operation could not fit. The system will automatically detect each uncut boundary and create an auxiliary curve. A pocketing type of toolpath will be used with a relatively smaller flat tool with a diameter of .1875". |
| Finishing | Finish the cavity surfaces, limited by the curves defining the outer and inner boundaries. The 3D Equidistant type of toolpath will generate evenly distributed toolpath passes all around the finished surfaces. A ball nose tool with a diameter of .375" will be used. |
| Re-Finishing | Re-Finish the rest material remaining from the Finishing operation, using a relatively smaller ball nose tool with a diameter of .125". The system will automatically detect the sections that have smaller indentation and concavity than the Finishing tool can cut and generate a 3D toolpath around them. |

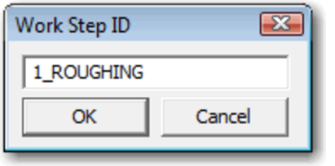
## CREATING WORK STEP "1_ROUGHING"

The first work step will machine the bulk material inside the cavity in order to prepare it for the finishing operation. We will use the "3D Machining Wizard" for defining the machining parameters. The wizard automatically selects all the visible surfaces as "cut surfaces" and sets the "surface" parameter in the Z Data section to the highest Z point of the surfaces. Besides it sets several parameters to the most appropriate values for the active operation. We will not specify any boundary curves for the roughing operation and let the system detect the outer boundary (silhouette) of the cut surfaces.

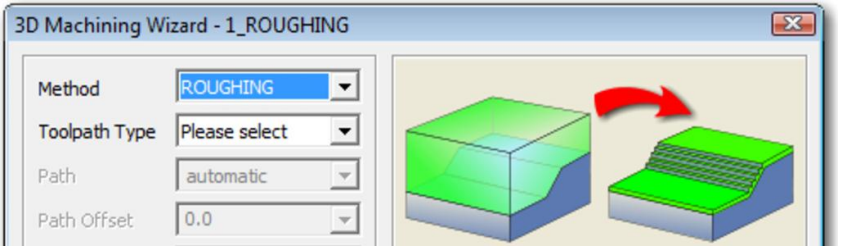
1. Select the "3D Machining Wizard" command in the "Machining" menu or click the corresponding button.

3D Machining Wizard

2. Input "1_ROUGHING" as the new Work Step ID. Confirm with OK.



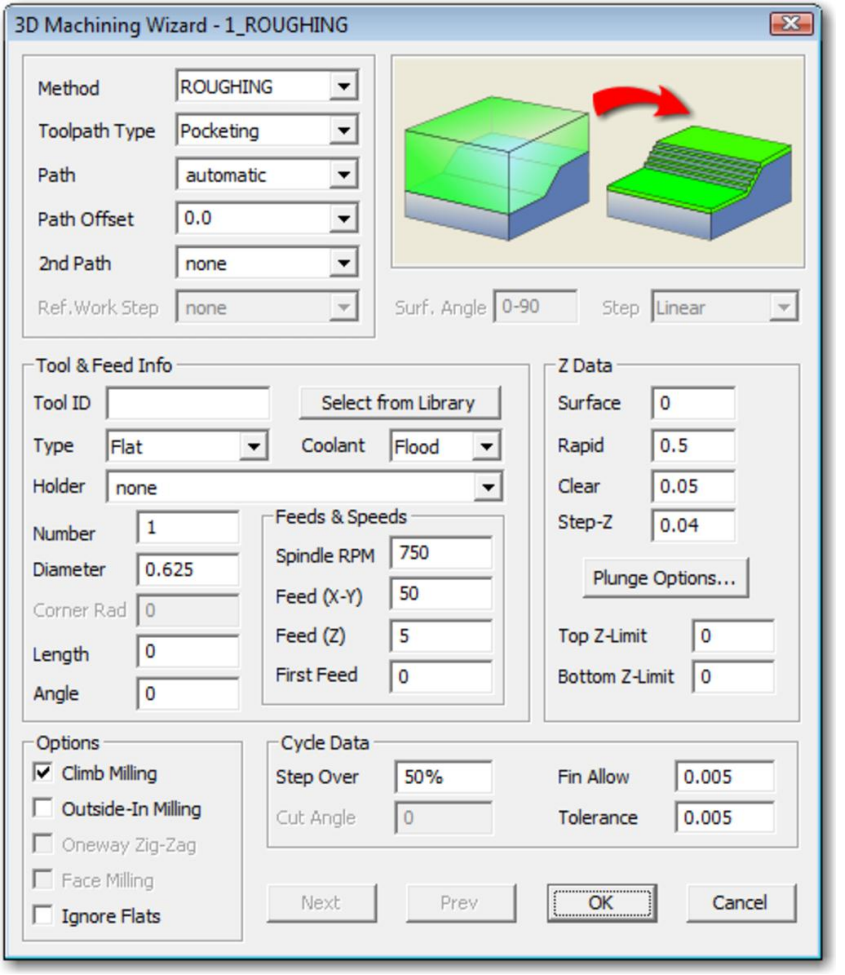

3. Select "ROUGHING" from the Method list box.




4. Select "Pocketing" from the Toolpath Type list box. Until a toolpath type is selected all other input boxes appear as disabled.

5. On the "**3D Machining Wizard**" window, change the settings according to the table below and ensure that all parameters are set as shown in **Picture 1-7**.

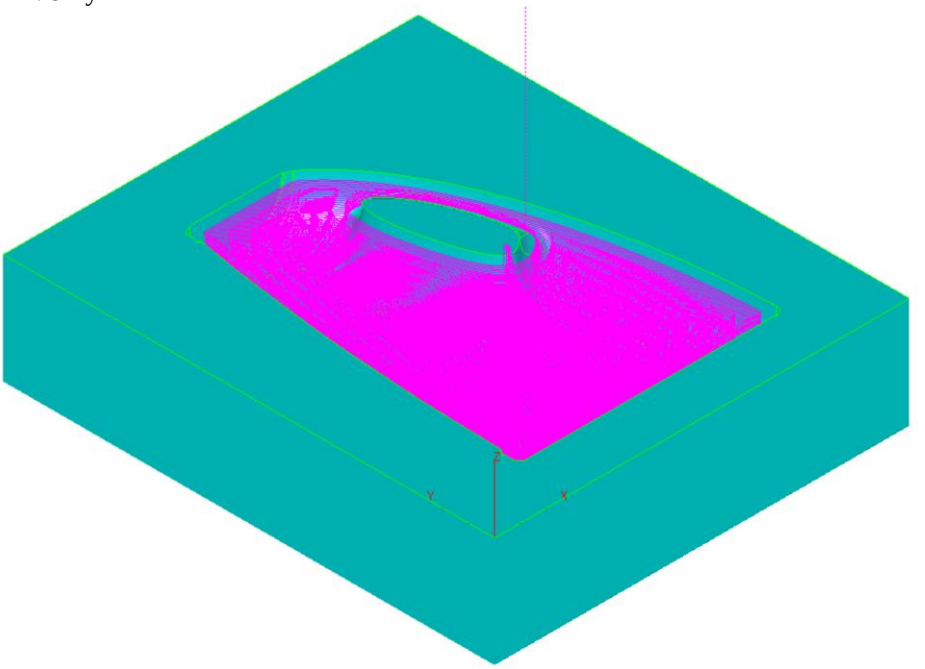
| Dialog Field | Value | Comment |
|---|---|---|
| Type | Flat | Flat type endmill |
| Number | 1 | Tool number in tool magazine |
| Diameter (Bot.) | 0.625 | Defines the diameter of the tool |
| Spindle RPM | 750 | Sets spindle RPM to 750 |
| Feed (X-Y) | 50 | Cutting feed rate in XY plane (inches/minute) |
| Feed (Z) | 5 | Cutting feed rate for Z depth moves |
| Surface | 0 | Set our Z surface to the top of the block. This value is automatically detected by the system as the top point of the solid body. |
| Rapid | 0.5 | Rapid positioning plane over "Z Surface" plane during transversal rapid moves |
| Clear | 0.05 | Plunge plane over "Z Surface" plane (Rapid to Feedrate) |
| Step-Z | 0.04 | Incremental depth per Z-level pass |
| Fin Allow | 0.005 | Finishing allowance value. Material that remains along the surfaces to be removed later with the finishing operation |
| Step Over | 50% | The distance between the tool passes on the machining plane. The value is given as the percentage of the tool diameter. (you may opt to use an absolute value instead) |


3D Machining Wizard - 1_ROUGHING
Method
ROUGHING
Toolpath Type
Pocketing
Path
automatic
Path Offset
0.0
2nd Path
none
Ref.Work Step
none
Surf. Angle
0-90
Step
Linear
Tool & Feed Info
Tool ID
Select from Library
Type
Flat
Coolant
Flood
Holder
none
Number
1
Diameter
0.625
Corner Rad
0
Length
0
Angle
0
Feeds & Speeds
Spindle RPM
750
Feed (X-Y)
50
Feed (Z)
5
First Feed
0
Z Data
Surface
0
Rapid
0.5
Clear
0.05
Step-Z
0.04
Plunge Options...
Top Z-Limit
0
Bottom Z-Limit
0
Options
Climb Milling
Outside-In Milling
Oneway Zig-Zag
Face Milling
Ignore Flats
Cycle Data
Step Over
50%
Cut Angle
0
Fin Allow
0.005
Tolerance
0.005
Next
Prev
OK
Cancel

**Picture 1-7**

6. Click the “Verify” button and the system starts calculating the toolpath. See **Picture 1-8.**

Verify



**Picture 1-8**

The Work Step #1 is now complete. Hit the “Redraw” button  to refresh the screen and remove the verified tool path display.

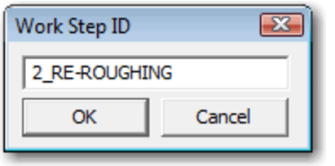
## CREATING WORK STEP "2_RE-ROUGHING"

The second machining work step will remove material only in the sections where the roughing tool could not fit. We will use a tool with a relatively smaller diameter. EZ-CAM system will detect the uncut boundaries at every roughing slice of the reference Work Step, and create an auxiliary curve that will be machined according to the selected re-roughing toolpath type.

1. Select the "3D Machining Wizard" command in the "Machining" menu or click the corresponding button.

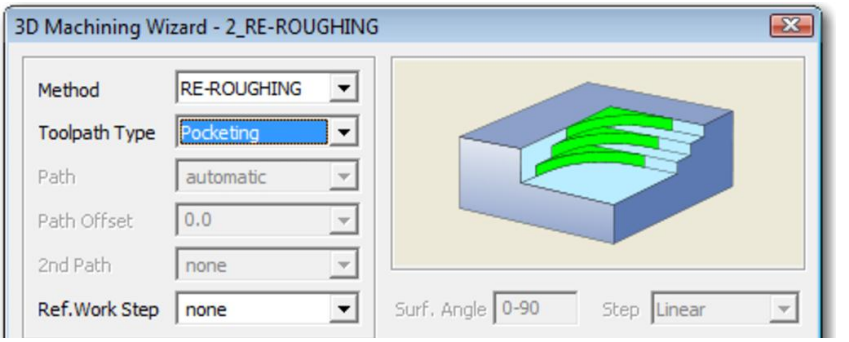
3D Machining Wizard

2. Input "2_RE-ROUGHING" as the new Work Step ID. Confirm with OK.




3. Select "RE-ROUGHING" from the Method list box.

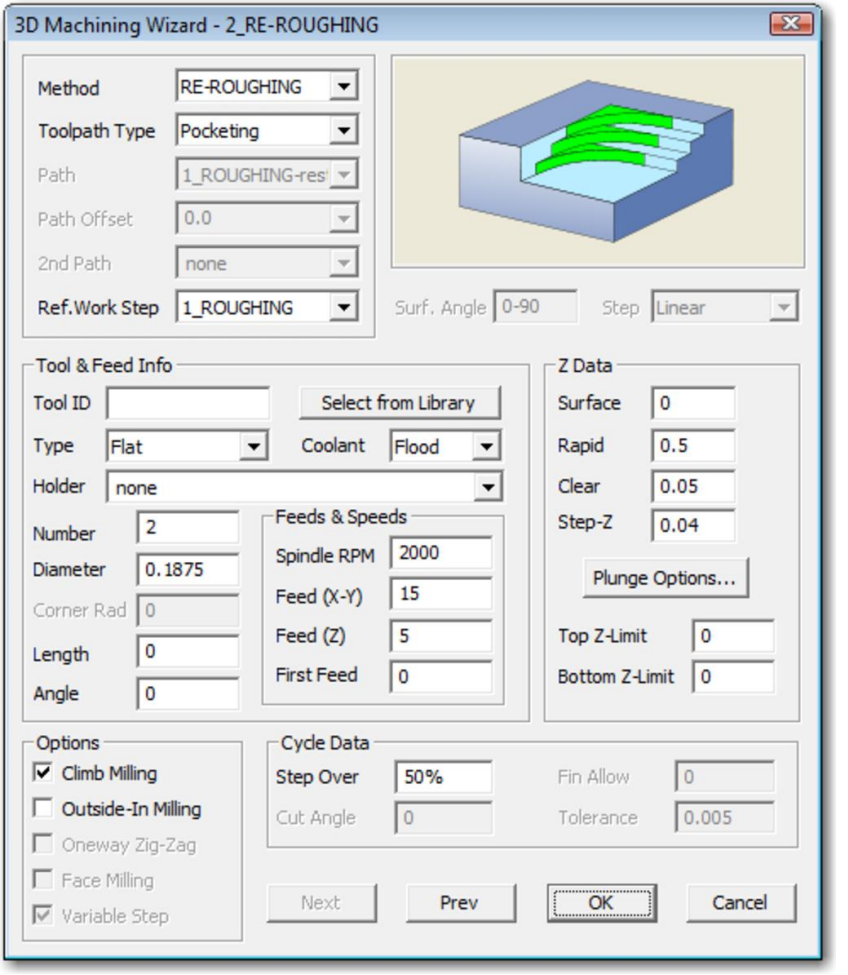
4. Select "Pocketing" from the Toolpath Type list box.




5. Select "1_ROUGHING" from the Ref.Work Step list box.

6. On the "**3D Machining Wizard**" window, change the settings according to the table below and ensure that all parameters are set as shown in **Picture 1-9**.

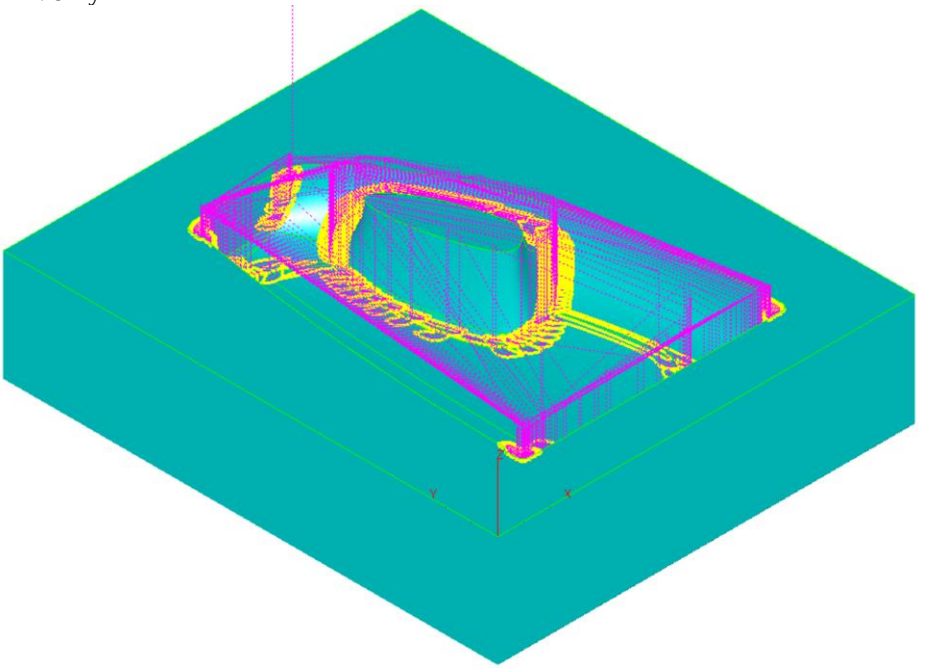
| Dialog Field | Value | Comment |
|---|---|---|
| Type | Flat | Flat type endmill |
| Number | 2 | Tool number in tool magazine |
| Diameter (Bot.) | 0.1875 | Defines the diameter of the tool |
| Spindle RPM | 2000 | Sets spindle RPM to 2000 |
| Feed (X-Y) | 15 | Cutting feed rate in XY plane (inches/minute) |
| Feed (Z) | 5 | Cutting feed rate for Z depth moves |
| Surface | 0 | Set our Z surface to the top of the block. This value is automatically detected by the system as the top point of the solid body. |
| Rapid | 0.5 | Rapid positioning plane over "Z Surface" plane during transversal rapid moves |
| Clear | 0.05 | Plunge plane over "Z Surface" plane (Rapid to Feedrate) |
| Step-Z | 0.04 | Incremental depth per Z-level pass is automatically set to the same value as the reference work step, however can be reduced to a lower value in order to decrease the tool load |
| Step Over | 50% | The distance between the tool passes on the machining plane. The value is given as the percentage of the tool diameter. (you may opt to use a decimal value instead) |


3D Machining Wizard - 2_RE-ROUGHING
Method
RE-ROUGHING
Toolpath Type
Pocketing
Path
1_ROUGHING-res
Path Offset
0.0
2nd Path
none
Ref.Work Step
1_ROUGHING
Surf. Angle
0-90
Step
Linear
Tool & Feed Info
Tool ID
Select from Library
Type
Flat
Coolant
Flood
Holder
none
Number
2
Diameter
0.1875
Corner Rad
0
Length
0
Angle
0
Feeds & Speeds
Spindle RPM
2000
Feed (X-Y)
15
Feed (Z)
5
First Feed
0
Z Data
Surface
0
Rapid
0.5
Clear
0.05
Step-Z
0.04
Plunge Options...
Top Z-Limit
0
Bottom Z-Limit
0
Options
Climb Milling
Outside-In Milling
Oneway Zig-Zag
Face Milling
Variable Step
Cycle Data
Step Over
50%
Cut Angle
0
Fin Allow
0
Tolerance
0.005
Next
Prev
OK
Cancel

**Picture 1-9**

7. Click the “Verify” button and the system starts calculating the toolpath. See **Picture 1-10**.

Verify

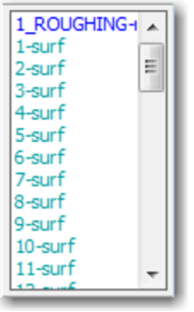


**Picture 1-10**

8. An auxiliary curve in yellow representing the machining area around the uncut sections appears after generating the toolpath. In order to hide it, select the “Blank” command from the “Edit” menu or click the corresponding button.

Blank

9. Select the curve “1_ROUGHING-rest” in the object list box. The yellow uncut boundary curve and the toolpath disappear.




10. Click the “Escape” button in the toolbar or hit “Esc” button on the keyboard to abort Blank command and to prevent accidental selection of an object on the screen.

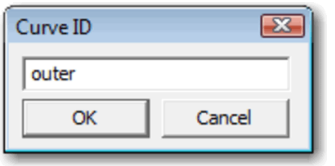
Escape

## DEFINING THE BOUNDARY CURVES FOR FINISHING OPERATION

We need to create curves to define machining borders for our finishing operation by using the solid model representation of the 3D part. We have few choices for curve creation from the 3D model such as "Face Boundary", "X-Y Intersection" or "Silhouette Boundary" commands. Here we use Face Boundary command to create the outer boundary of the mold cavity and inner boundary from the flat surface around the handle section.

1. To create the outer finishing boundary curve choose the "Face Boundary" command from the "Curves" menu, type "outer" as the Curve ID and click OK.

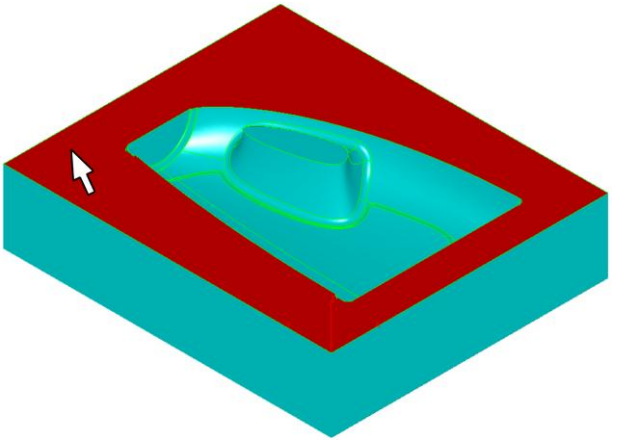
Face Boundary




2. Select the "Verify Mode" command in the "Edit" menu or click the corresponding button. This mode displays the selected object in different color and waits for our confirmation allowing us to abort the process or continue selecting multiple objects.

Verify Mode

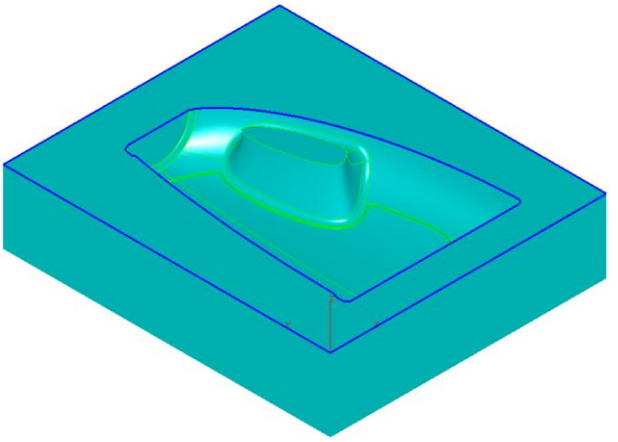
3. Now we should select the surface on the part from which the face boundary curve will be created. Click anywhere on the outermost top surface. After clicking, the color of the surface changes to inform you that it's selected. See **Picture 1-11**.



**Picture 1-11**

4. Click the “Enter” button in the toolbar or hit “ENTER” button on the keyboard to confirm. See **Picture 1-12.**

Enter



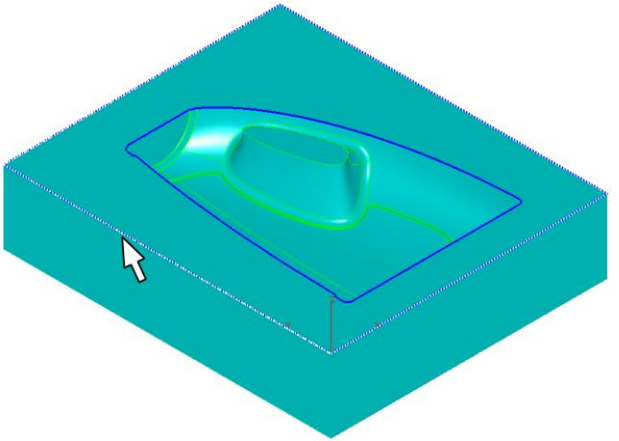
**Picture 1-12**

5. We only need the curve created at the intersection of the top plane and the cavity. So it is better to delete the outermost one to prevent confusion. Select the “Delete” command from the “Edit” menu or click the corresponding button. Then select the “Curve…” command from the “Edit > Discrimination” menu or click the corresponding button.


Delete


Discrim. Curves

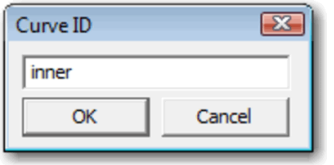
6. Click the blue curve on the outer edge of the part and hit ENTER. See **Picture 1-13**.



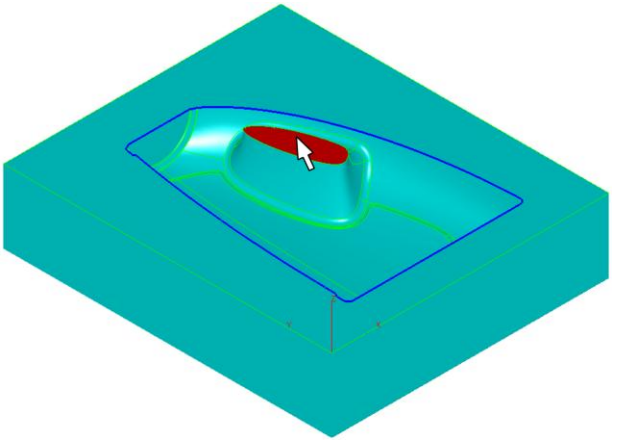
**Picture 1-13**

7. For creating the inner finishing boundary curve choose the “Face Boundary” command from the “Curves“ menu, type “inner” as the Curve ID and click OK.


Face Boundary

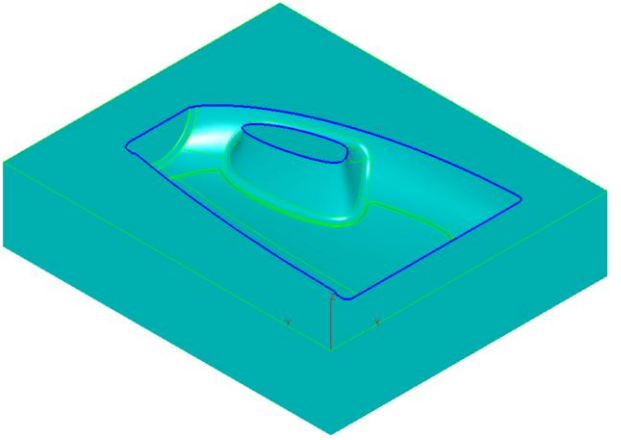


8. Click the planar surface on top of the boss in the cavity corresponding to the handle of the bottle. After clicking, the color of the surface changes to inform you that it's selected. See **Picture 1-14**.



**Picture 1-14**

9. Click the "Enter" button in the toolbar or hit "ENTER" button on the keyboard to confirm. You can see the result in **Picture 1-15.**



**Picture 1-15**

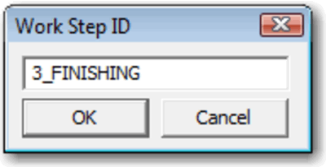
## CREATING WORK STEP "3_FINISHING"

In our third work step we will finish the cavity surface using our most efficient finishing technique "3D Equidistant". This technique ensures an equal 3D distance between adjacent toolpath passes resulting in an even surface roughness all over the part eliminating the need for another finishing operation if the step over value is adequate.

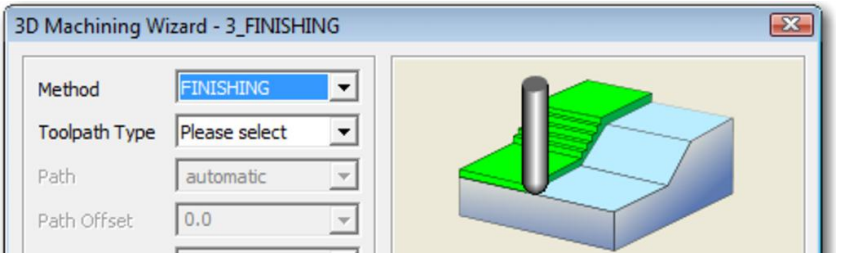
1. Select the "3D Machining Wizard" command in the "Machining" menu or click the corresponding button.

3D Machining Wizard

2. Input "3_FINISHING" as the new Work Step ID. Confirm with OK.




3. Select "FINISHING" from the Method list box.



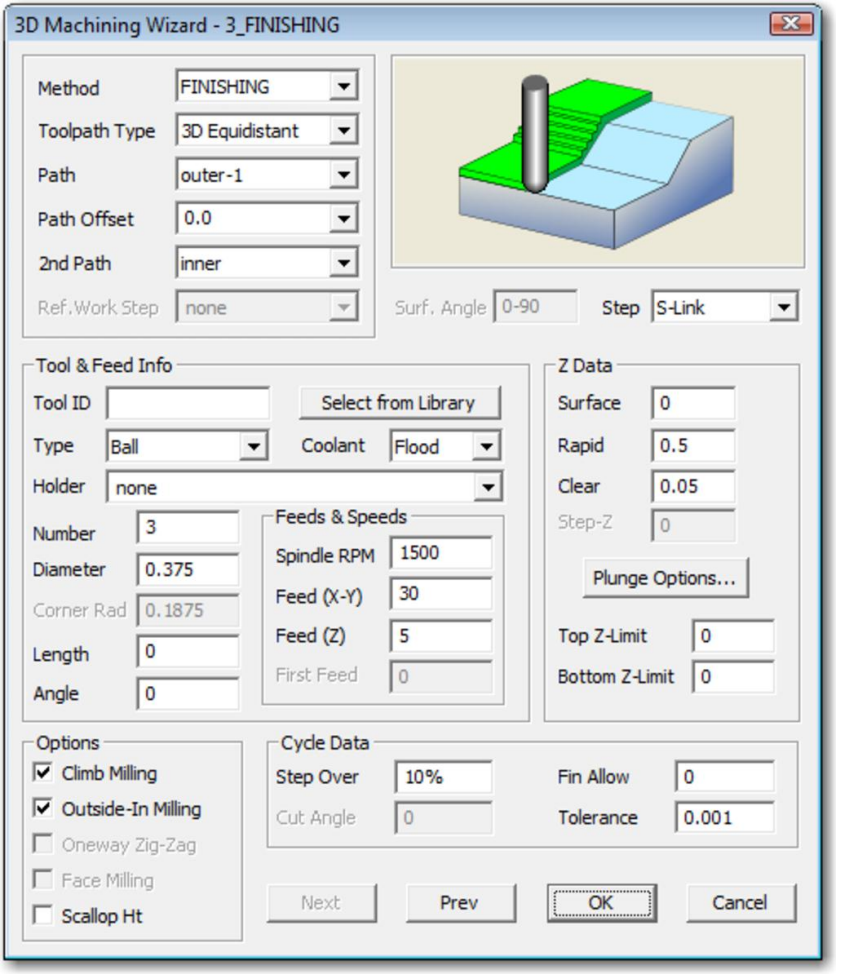

4. Select "3D Equidistant" from the Toolpath Type list box.

5. Select "outer-1" from the Path list box to define outermost machining boundary.

6. Select "inner" from the 2nd Path list box to define the island region that should not be machined by the finishing operation.

7. On the "**3D Machining Wizard**" window, change the settings according to the table below and ensure that all parameters are set as shown in **Picture 1-16**.

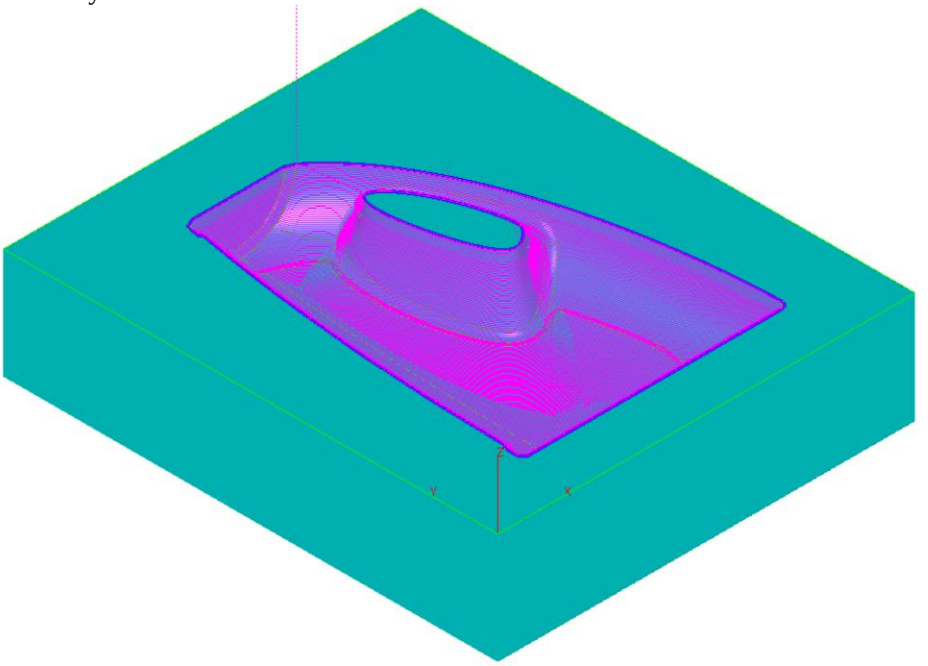
| Dialog Field | Value | Comment |
|---|---|---|
| Type | Ball | Ball type endmill |
| Number | 3 | New tool number for finishing operation |
| Diameter (Bot.) | 0.375 | Defines the full diameter of the tool |
| Spindle RPM | 1500 | Sets spindle RPM to 1500 |
| Feed (X-Y) | 30 | Cutting feed rate in XY plane (inches/minute) |
| Feed (Z) | 5 | Cutting feed rate for Z depth moves |
| Step | S-Link | This parameter controls the step over moves between toolpath passes. S-Link eliminates any possible corner on the step over move eliminating knocking effect and ensuring smoother operation of the machine |
| Fin Allow | 0 | Finishing allowance value is set to 0 to reach the final surface after performing this operation |
| Step Over | 10% | For 3D Equidistant type of toolpath Step Over is the 3D distance between the tool passes. The value is given as the percentage of the tool diameter. Since we want to leave minimal cusps on the surface "Step Over" distance is set as 10% of the tool diameter. However, lower values can be preferred to obtain smoother surface finish. |
| Outside-In Milling | ON | Click this checkbox to generate the toolpath starting from the outermost loop and collapsing to the center. This ensures toolpath loops arranged from top to bottom |


3D Machining Wizard - 3_FINISHING
Method
FINISHING
Toolpath Type
3D Equidistant
Path
outer-1
Path Offset
0.0
2nd Path
inner
Ref.Work Step
none
Surf. Angle
0-90
Step
S-Link
Tool & Feed Info
Tool ID
Select from Library
Type
Ball
Coolant
Flood
Holder
none
Number
3
Diameter
0.375
Corner Rad
0.1875
Length
0
Angle
0
Feeds & Speeds
Spindle RPM
1500
Feed (X-Y)
30
Feed (Z)
5
First Feed
0
Z Data
Surface
0
Rapid
0.5
Clear
0.05
Step-Z
0
Plunge Options...
Top Z-Limit
0
Bottom Z-Limit
0
Options
Climb Milling
Outside-In Milling
Oneway Zig-Zag
Face Milling
Scallop Ht
Cycle Data
Step Over
10%
Cut Angle
0
Fin Allow
0
Tolerance
0.001
Next
Prev
OK
Cancel

**Picture 1-16**

7. Click the “Verify” button and the system starts calculating the toolpath. See **Picture 1-17**.

Verify



**Picture 1-17**

The Work Step #3 is now complete. Hit the “Redraw” button  to refresh the screen and remove the verified tool path display.

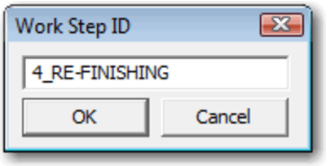
## CREATING WORK STEP "4_RE-FINISHING"

In our last machining work step we will remove the material left in the areas with smaller indentation and concavity than the Finishing tool could complete. We will select the last finishing operation as the Reference Work Step and pick a relatively smaller tool. Then the system will automatically detect the uncut boundaries and generate a 3D Equidistant finishing toolpath.

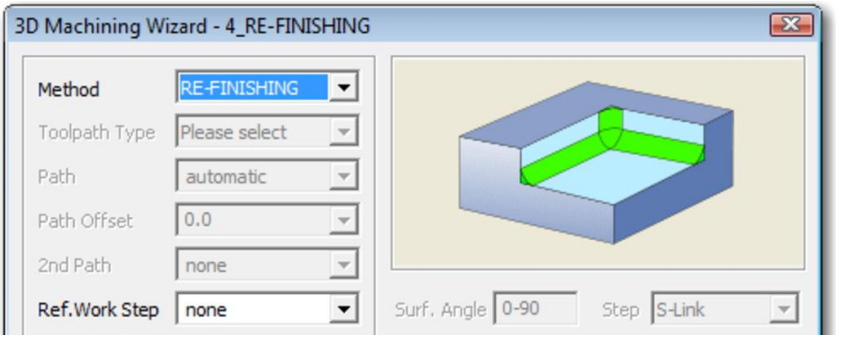
1. Select the "3D Machining Wizard" command in the "Machining" menu or click the corresponding button.

3D Machining Wizard

2. Input "4_RE-FINISHING" as the new Work Step ID. Confirm with OK.

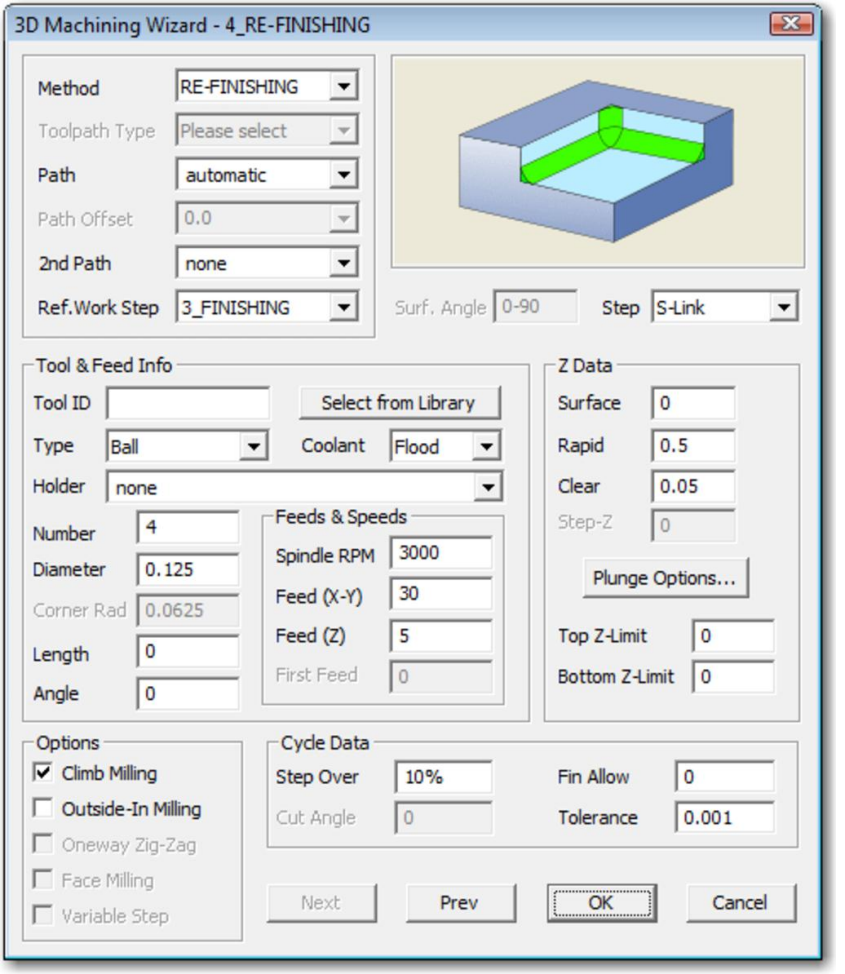



3. Select "RE-FINISHING" from the Method list box.




4. Select "3_FINISHING" from the Ref.Work Step list box.

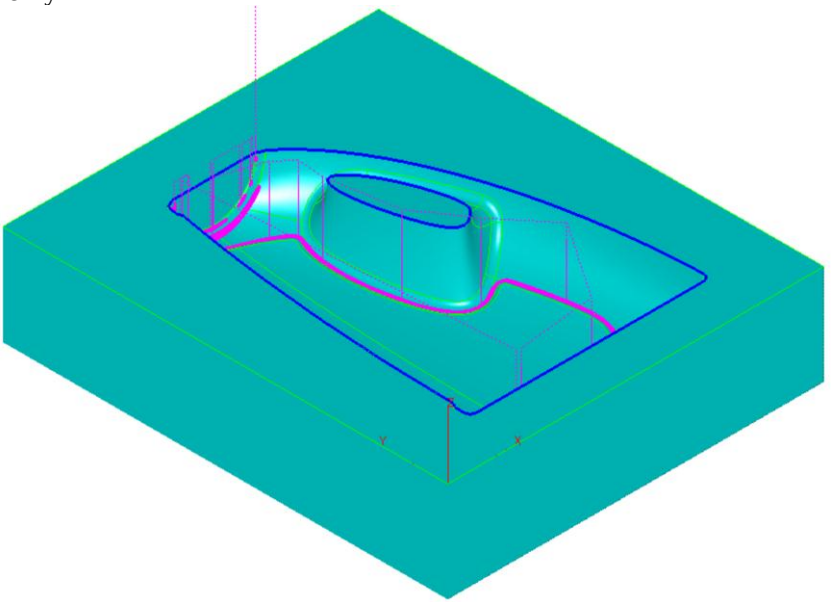
5. On the "**3D Machining Wizard**" window, change the settings according to the **Picture 1-18** below and ensure that all parameters are the same.




**Picture 1-18**

6. Click the “Verify” button and the system starts calculating the toolpath. See **Picture 1-19**.

Verify



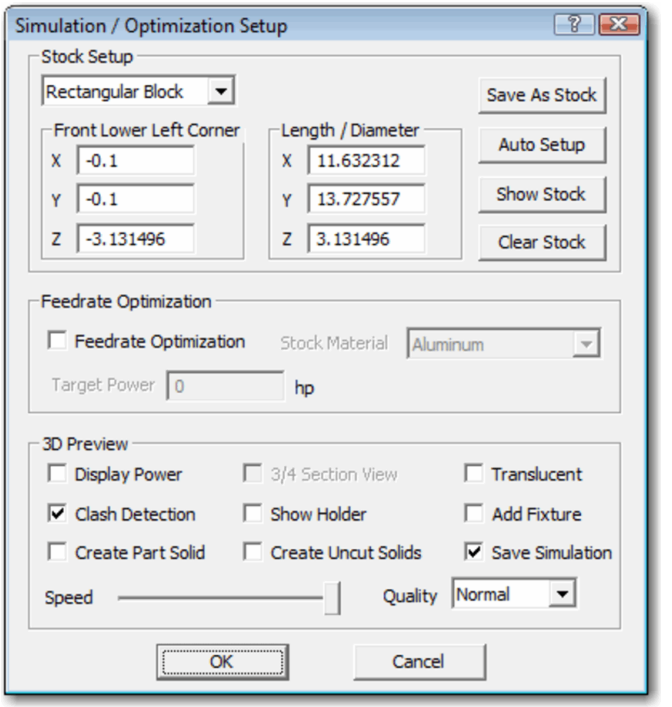
**Picture 1-19**

The Work Step #4 is now complete. Hit the “Redraw” button  to refresh the screen and remove the verified tool path display.

## 3D SOLID MODEL PREVIEW

One of the most powerful EZ-CAM features is the 3D solid preview function. This function shows an animated tool cutting a solid model of the programmed part. After previewing our program we are left with an accurate solid model representation, which allows us to closely examine the surface finish and resulting part details. If no stock was defined prior to calling the "3D Preview" or "RapidCut" commands, the system automatically calculates the stock size by adding a small margin to the sides and to the bottom of the visible solid model on the screen.

1. Select the "Stock & Optimization Setup" command from the "Machining" menu and click "Auto Setup", values should appear as shown in **Picture 1-20**. You may also enter your own values if the stock is different from the solid model boundaries. Clicking and activating "Clash Detection" makes the system detect and warn you about the collisions between the tool and the stock. Also click "Save Simulation" to enable browsing through simulation results of each work step (see Paragraph – 4). Close the dialog with OK.




**Picture 1-20**

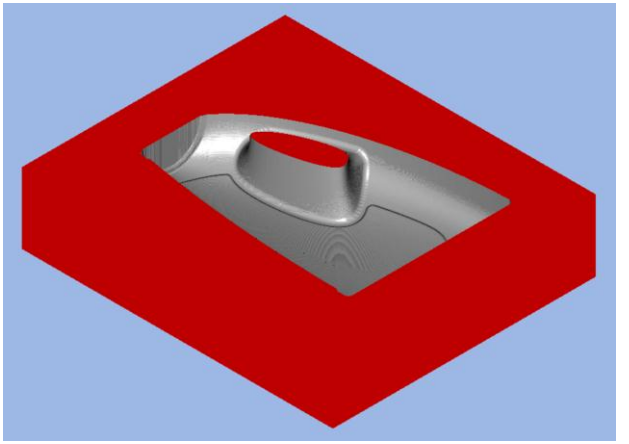
It is not necessary to open "Stock & Optimization Setup" window and run Auto Setup if you are executing the solid simulation for the "first time", system calculates the stock boundaries automatically. But if somehow the stock size or position is improper at any time, you may need to run Auto Setup or input your own values that describe the real stock that will be mounted on the CNC machine.

If you need information about using custom stock for the solid simulation please refer to the Indexing section of the "EZ-MILL 4th Axis Tutorial".

2. Start the simulation using "3D Preview" command from the "Machining" menu or the corresponding button. The simulation speed can be controlled any time by pressing one of the numeric keyboard buttons (not the ones on the num-pad side), ranging from 1 (slowest) to 9 (fastest). Pressing 0 will activate Step Mode. If you need to see the simulation result immediately, you can use "RapidCut" command by executing it from the "Machining" menu or the corresponding button. The simulation result is shown in **Picture 1-21**.


3D Preview


Rapid Cut



**Picture 1-21**

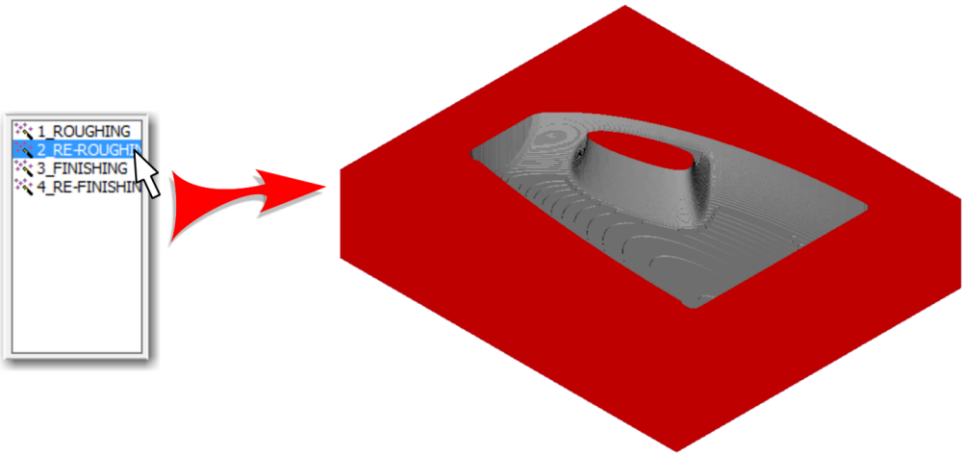
3. During 3D simulation you can change the on-screen view by using the dynamic view commands (Rotate, Pan, Zoom) with their corresponding buttons. Alternatively you can activate rotate by holding down right button and pan by holding down middle button of the mouse while dragging it on the screen. As usual mouse wheel executes zoom command during the simulation as well.


Dynamic Rotate


Dynamic Zoom

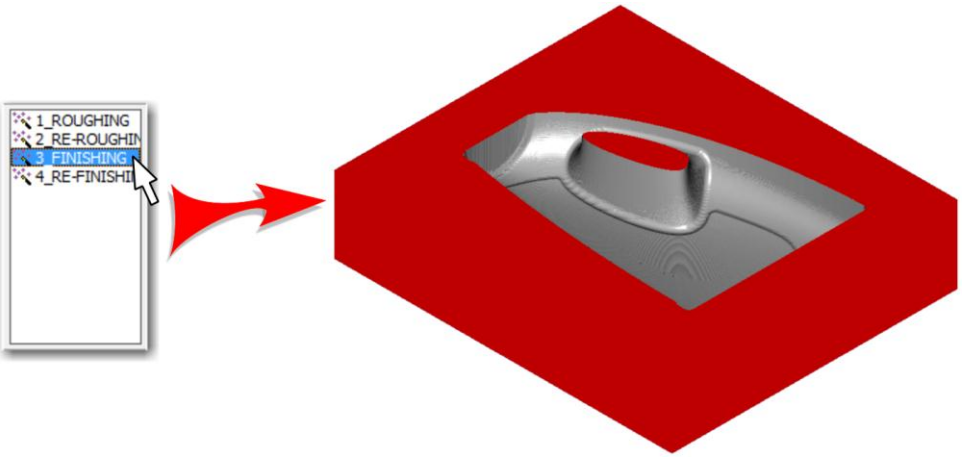

Dynamic Pan

4. After completing the 3D simulation it is possible to browse through the simulation results of each work step. Simply click the desired works step from the list box at the right size of the screen and EZ-CAM instantly shows the situation at the end of the selected work step. Click “2_RE-ROUGHING” in the Work Step list box, you will see the simulation result at the end of second operation “re-roughing”. See **Picture 1-22**.




**Picture 1-22**

5. Click “3_FINISHING” in the Work Step list box, you will see the simulation result of the third operation “finishing”. See **Picture 1-23**.




**Picture 1-23**

Alternatively, after first clicking on one of the work steps in the list box you can use “up” and “down” arrow keys on the keyboard in order to browse through the simulation stages.

If you have returned to the CAD view and now want to see the simulation model at a specific work step (or to start simulation from the corresponding stage), use Recall Part command from “Machining” menu to view the model at the end of the active work step (that is selected in the list box) and then start simulation.

**Congratulations!**

**You've completed the EZ-MILL Pro 3D Machining Wizard Tutorial !**