EZCAM®
CAD/CAM

CNC Programming Simplified

EZ-Mill 3D Machining Solids Tutorial

www.ezcam.com

CHAPTER 3.
EZ-MILL 3D TUTORIAL

OVERVIEW

This tutorial begins with a simple surface geometry representative of a part model that might be imported from one of the popular solid modeling packages. This tutorial can be performed with either EZ-MILL 3D or EZ-MILL Pro. It is important to remember however, that EZ-MILL 3D machining capabilities are limited to a maximum of 20 surfaces per work step, so it may be necessary to refine your selection to just the surfaces needed for a particular operation when applying these strategies to other parts. Whenever practical it is best to select all surfaces, then localize the machining with boundary curves or other limiting techniques. This method will minimize tedious surface picking and ensure there are no small sliver surfaces excluded, which could lead to an incorrect result or gouging toolpath.

The focus of this tutorial is on how to apply the primary surface machining strategies, not on finding the most efficient method to machine the part. After walking through these exercises using the suggested parameters, you are encouraged to alter the parameters and try different options and techniques. It is best to understand how the various combinations work when applied to a well-defined simple part, before moving on to more complex applications.

CAVITY MACHINING

This tutorial will machine a part containing a drafted wall pocket with a convex spherical floor, and 4 though holes. While this part can be machined with many different strategies, this example focuses on introducing the most common surfacing methods, Constant Z and Vertical Projection. We will also introduce the technique of creating 2D Curve geometry from 3D surface data. We will machine the part using 5 work steps, two for surface roughing, two for surface finishing, and the last one for drilling.

BASIC PROGRAMMING STEPS

Before beginning the exercise let us first summarize the steps that will be followed.

STEP 1. **Load (Import) the Surface Geometry of the Part to be machined**
The best method for exporting geometry from a solid modeling CAD system for use in EZ-CAM is to create an IGES trimmed surface model. This exercise will begin with an IGES file saved from Rhino, our recommended low cost CAD modeling companion product.

STEP 2. **Check Surface Directions**
Confirm that the surface normal directions (machining sides) are set correctly, and if necessary, reorient them.

STEP 3. **Create auxiliary Curve Geometry from the 3D Surface Geometry**
We will use the XY intersection function to create several 2D curves to be used for localizing the 3D surface milling operations, and defining the 2D drilling path.

STEP 4. **Create Machining Work Steps**
Create machining work steps, specify tool and operation parameters, select surfaces and/or Path Curves, and verify the toolpaths. We will create rough surface milling operations leaving stock on all surfaces, then finish surface milling operations removing the remaining material leaving minimal cusps, and lastly, a simple drilling operation for the holes.

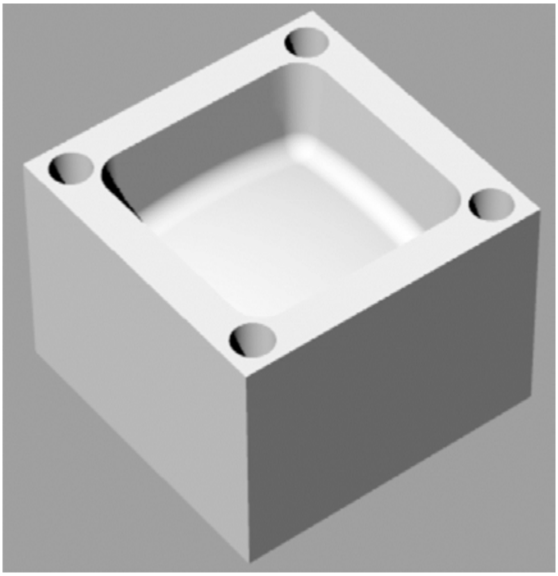
STEP 5. **Check Material Removal and Surface Finish**
Check to make sure all material is removed and that the surface finish smoothness is acceptable. We will define the machining stock then visualize the machining process using the Preview function.

STEP 6. **Post Processing**
Post process the work steps to create the final machine code file for a specific machine/control.

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

THE PART

The following image shows the part we will machine. It is a 4”x4” block that is 3” tall. There is a 3”x3” pocket with 5 degree drafted walls, and .375” radii corner fillets. The floor of the pocket has a convex 8” spherical radius, and the depth from the top of the part to the top of the sphere is 1”. There are .25" radii fillets between the walls and the floor, and there are four .5” diameter thru holes equally spaced at 3.25” between centers.

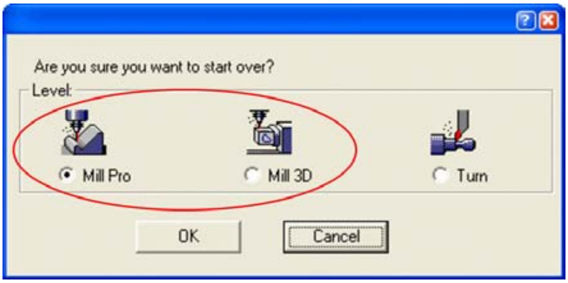


Picture 3-1

SETTING THE PREFERENCES

This tutorial uses geometry defined in inch units, and all parameters and output will based on these units. Before beginning we must first make sure the system settings are compatible with the exercise.

1. Select ”New” command from the “File” menu to restart EZ-Mill and to clear the memory before continuing with the tutorial. Make sure that one of the EZ-Mill levels is active and press OK to start over.



The “New” dialog is also used to switch between the EZ-Mill and EZ-Turn module. 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 ”Setup” command from the “View” menu

3. Type “0” for “X Center”, “0” for “Y Center” and “5” for “Size”.
This sets the window size from the edge of the window to the center of the window, allowing enough room to see all of the part as it is created.

4. Select “Inch” option button as the parts input dimension system.

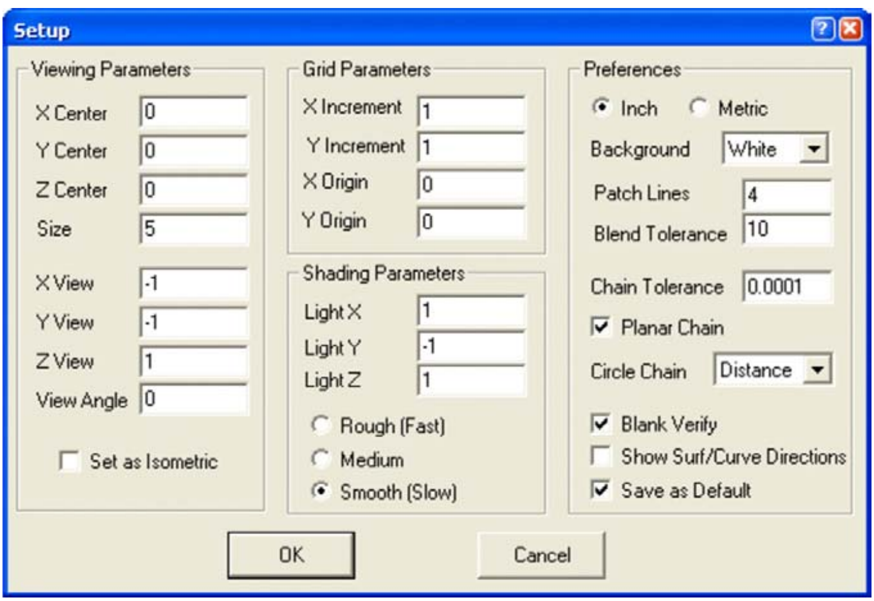
5. Click the “Background” list box and select “White”.

6. Check the box "Blank Verify" on the right. This will cause verified tool paths to be blanked every time the view is changing or the screen is redrawn.

7. Un-Check "Show Surf/Curve Direction". This will hide the small arrows indicating the surface normals and curve directions. It can later be activated at any time.

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

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



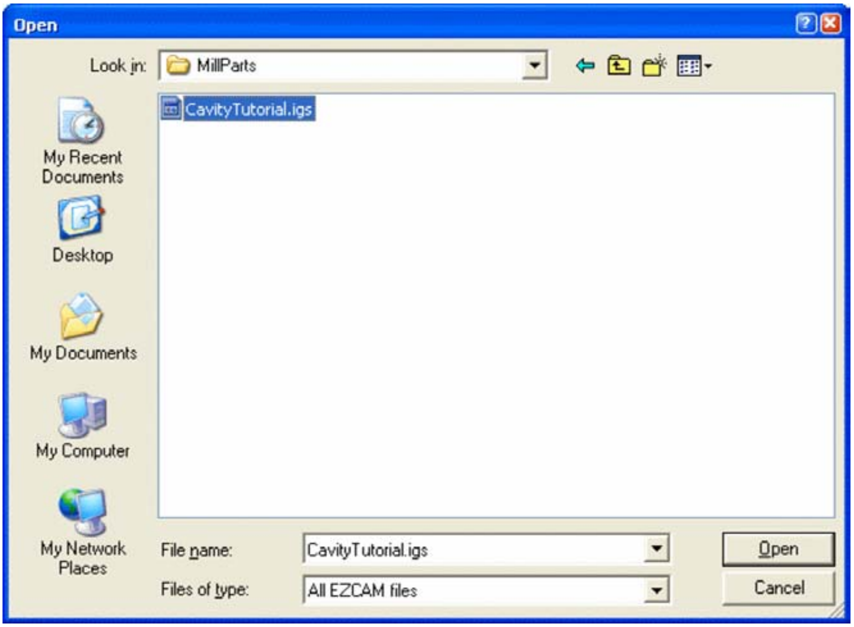
The initial setup for the EZ-MILL 3D Cavity tutorial is now complete. Continue with the next section to create the geometry necessary for this part.

LOADING THE SURFACE GEOMETRY

We will begin the exercise by loading the surface geometry that represents our part. There are 2 different procedures that can be followed, you can either load a trimmed surface IGES file directly into EZ-MILL, or if you have installed the Rhino evaluation program, you can load the Rhino *.3dm sample file into Rhino, then copy and paste it into EZ-MILL using the clipboard. The following section will explain both methods.

LOADING THE SURFACE DATA VIA IGES IMPORT

The IGES file containing the trimmed surface geometry has already been copied into the "EZCAMW\MILLPARTS" folder by the EZ-CAM setup. Follow the steps below to load the data.



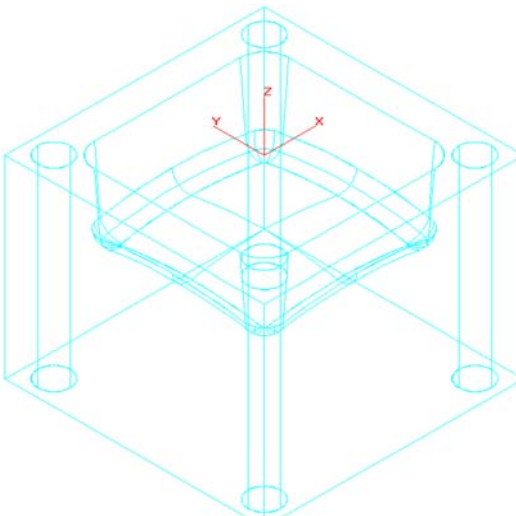
Picture 3-2

1. Select "Open" command from the "File" menu to open the file dialog. In **Picture 3-2** you can see the dialog displayed on a Windows XP professional 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 “IGES (*.igs)”.

4. Select the file “CavityTutorial.igs” and click the “Open” button. The imported surface geometry should then appear as shown in **Picture 3-3**.



Picture 3-3

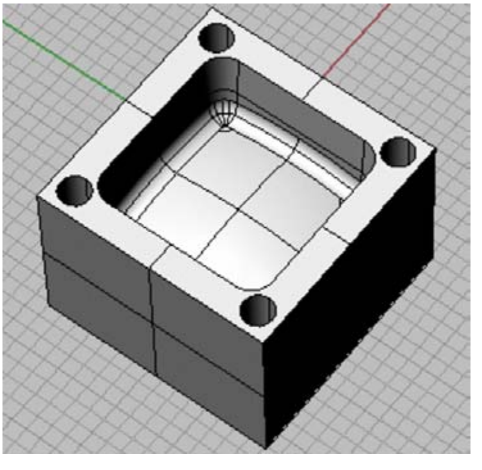
COPY AND PASTING THE MODEL FROM RHINO

The Rhino package is an extensive nurbs-modeling tool designed for the Microsoft Windows platform. You can find the setup files for the Rhino evaluation version on the EZCAM setup CD. The file with the Rhino model used in this tutorial has already been copied into the "EZCAMW\MILLPARTS" folder by the setup.

1. Start the Rhino application and select "Open" command from the "File" menu to open the file dialog.

2. Navigate to the "EZCAMW \ MILLPARTS" folder on the drive where you installed the EZCAM software and pick the file "CavityTutorial.3dm". Click the "Open" button to load the file. With a left-mouse click on the "Render Viewport" command icon you can get a shaded view of the part. The model should now appear as shown in **Picture 3-4**.

Render Viewport

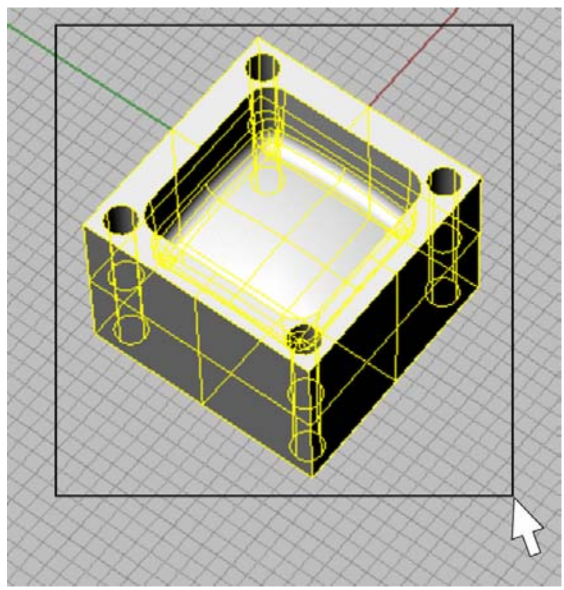


Picture 3-4

3. Next step is to copy all rhino entities to the clipboard. Therefore click the “Cancel” command to ensure no other command is active and select the model by dragging a frame around it using the cursor. As you can see in **Picture 3-5**, the selected entities are then highlighted in yellow. Finally use the “Copy to Clipboard” command to copy everything to the windows clipboard.


Cancel


Copy to Clipboard (Ctrl+C)



Picture 3-5

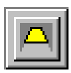
4. Switch back to the EZ-MILL application and choose the “Paste from Rhino” command (or Ctrl+V) from the “Edit” menu to paste the surface geometry from the clipboard into the current EZ-MILL session window. Note that the coordinate system used in Rhino is matched to the World coordinate system of EZ-MILL.

5. You can now switch back to Rhino and exit the application by choosing File – Exit.

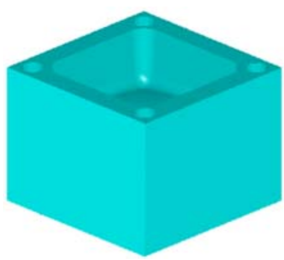
CHECKING THE SURFACE NORMALS

Each surface of a model contains a direction "normal" which points to the outside (machining side) of the surface. Some machining cycles of EZ-MILL (Constant Z and Surface) use the surface normal when calculating the toolpath, so it is best to ensure they are set correctly before proceeding. In most cases when loading surfaces from a solid, the surface normals will already be set correctly to the outside. However, in some cases they may not all be correct, and it will be necessary to set them manually. Following are the steps to confirm and correct the surface machining side.

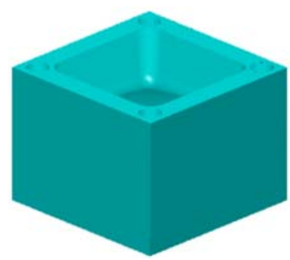
1. The best method to determine the machining side orientation is to shade the model. Surfaces that are correctly oriented will display in a light blue color, and those incorrectly oriented will display in a darker blue. Choose the "Shade" command from the "View / View Control" menu to shade the model.



Shade



Machining Sides set correctly for all Surfaces



Machining Sides set Incorrectly for Outer Side of Block

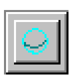
2. If the machining sides are not set correctly, EZ-MILL has a quick and effective method for setting them when the surfaces form a closed volume, as would be the case for a solid model. To use this method, choose the "Solid Surfaces " command from the "Surfaces / Machining Side" menu. When working with surfaces that do not form a closed volume, one of the other Machining Side options such as "Toggle Surface" must be used to set them manually. This step is not necessary for our tutorial geometry since the machining sides are already set correctly.

> When setting individual surface machining sides it is sometimes useful to turn off surfaces shading and activate display of the direction indicator arrows by checking the "Show Surface/Curve Direction" option found on the "Setup" dialog (see "View" menu).

DEFINING AUXILIARY CURVE GEOMETRY

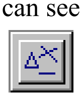
Since there is no wire frame geometry included with the surface geometry, we will use the "XY Intersection" function to create the curves we will need for our machining boundaries and drilling path. A curve around the top of the pocket will be used for Pocketing and Zig-Zag projection type cycles to rough and finish the spherical floor of the cavity. This same curve will also be used to restrict the "Constant Z" Contouring finishing cycle to prevent cuts from occurring outside this curve along the outer walls of the block. A second curve consisting of the four holes will finally be used to define our drilling path.

1. To create the intersection curves choose the "XY Intersection" command from the "Curves" menu and accept the default name "Crv1" for the new curve



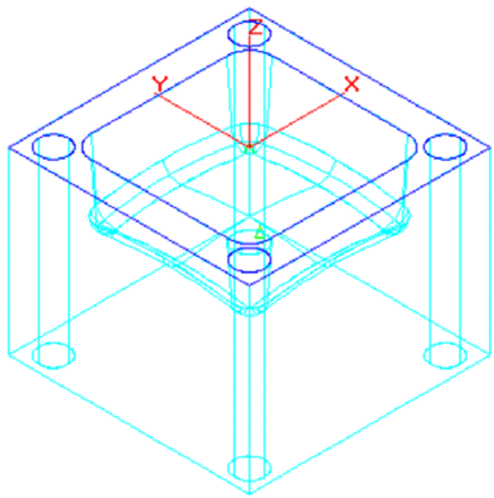
XY Intersection

2. Now there are two ways to define the Z-axis level at which the intersection profiles will be calculated. We can use the cursor to right-click on one of the top corners of the block using the "Snap All" selection option, or simply enter "0" into the "Z" field of the "Value Entry Box". Finally press ENTER to start the calculation process. You can see the result in **Picture 3-6**.



Snap All

Right-Click on Corner to select it's current Z Level (0)



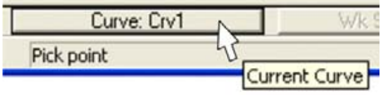
Picture 3-6

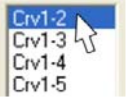
Using the left mouse button when selecting one of the top corners of the block, starts the calculation process immediately without the need to press the ENTER button.

CONNECTING THE DRILLING PATH CURVE

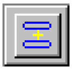
The next step will be to combine the 4 separate curves created for each hole into a single curve connecting them with rapid links.

1. First we need to select one of the circular curves as the "Current Curve". The other curves representing the three remaining holes will later be appended to this profile. Select the "Current Curve" status button from the bottom of the EZCAM window. Then use the cursor to pick the "Crv1-2" from the "Selection List Box" on the right side of the screen. Now the status button should display "Curve: Crv1-2".

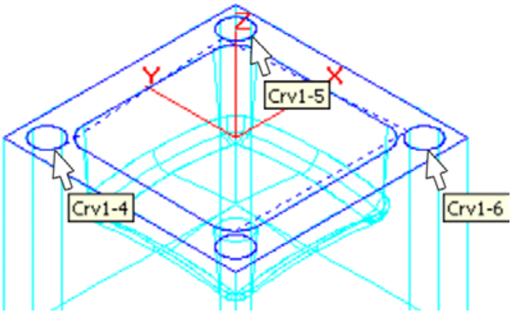




2. Next, select the "Copy/Append" command from the "Curves" menu and pick the curves Crv1-6, Crv1-5, and Crv1-4, either directly from the viewport or the "Selection List Box". Note how the selected curves are removed from the list and a dashed line (rapid link) now connects them to our original curve Crv1-2. See **Picture 3-7**.



Copy/Append



Picture 3-7

CREATING THE PART PROGRAM

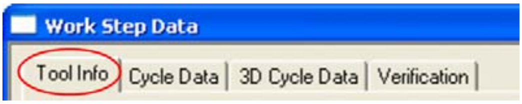
Now that we have created the auxiliary curve geometry needed for our machining cycles, we are ready to begin defining the work steps to machine the part. Each work step uses a specific cycle (Constant-Z, Pocketing, etc.) along with its associated tool settings and parameters to create a toolpath that machines the assigned curves and/or surfaces. Our part program will consist of the following work steps.

Work Step ID	Purpose
Rough1	Rough the cavity down to a depth of .975 (.025 above the top of the spherical floor) leaving a 2D Stock Allowance of .025, using a Constant-Z cycle with a Pocketing toolpath pattern generated inside of each surface slice boundary.
Rough2	Rough the floor surfaces of the cavity leaving a 3D Surface Allowance of .025, using a Pocketing cycle with the toolpath pattern generated inside curve Crv1 and projected onto the floor surfaces.
Finish1	Finish the drafted walls of the cavity inside Crv1 to a depth of 1.0 using a Constant-Z cycle with a Contour toolpath generated along each surface slice boundary.
Finish2	Finish the floor surfaces of the cavity using a Zig-Zag cycle with the toolpath pattern generated inside of curve Crv1 and projected onto the floor surfaces. An Upper Check plane set to Z-1.0 will be used to prevent the toolpath from machining the previously finished wall surfaces.
Drill1	Drill the 4 holes to a depth of 3.15 using curve Crv1-2 as the path.

CREATING "ROUGH1" – CONSTANT Z POCKETING

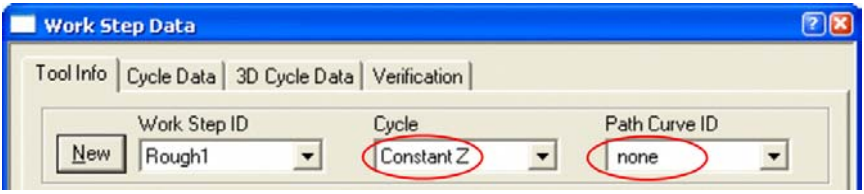
Our first work step will use the "Constant-Z" cycle to rough out the pocket cavity in .25 depth levels with a Pocketing toolpath pattern. Since the walls of this pocket have a uniform draft angle it would also be possible to machine it using just the standard "Pocketing" cycle, machining curve "Crv1" with a specified draft angle of 5 degrees. But for the purpose of introducing the more generalized surfaced based approach, we will use the "Constant-Z" cycle using only surface data.

1. Select the "Work Step Data" command in the "Machining" menu to open the "Work Step Data" dialog. Once it is open switch to the "Tool Info" tab.



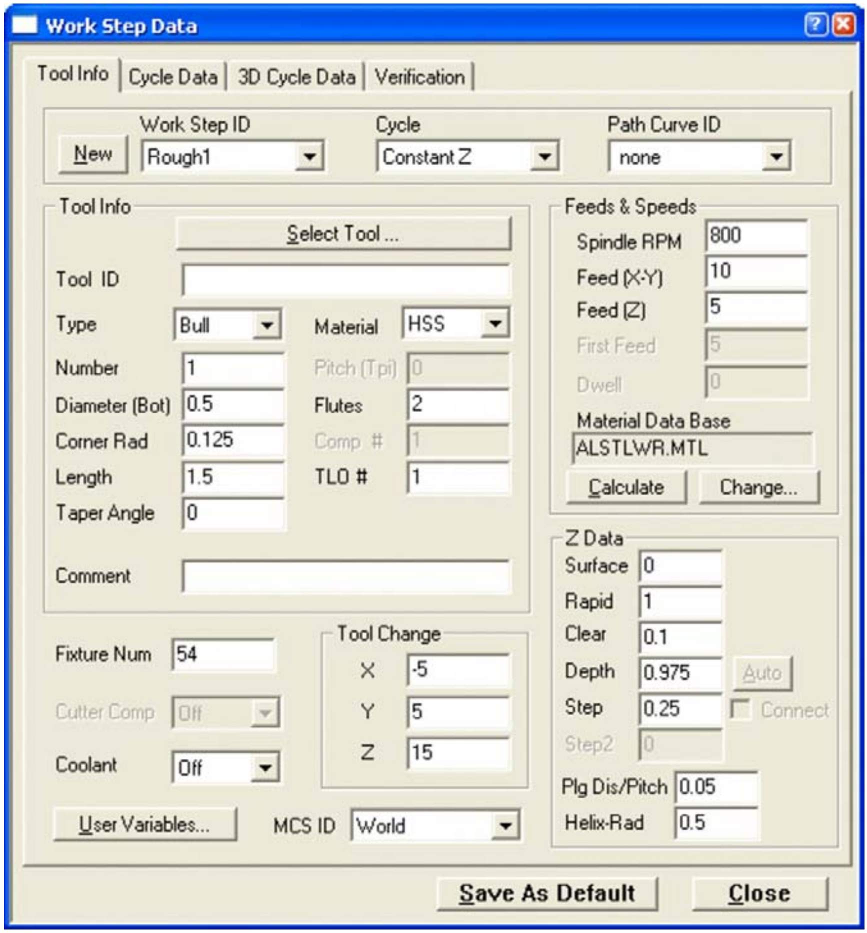
2. Press the "New" button and input "Rough1" as the new Work Step ID. Confirm with OK.

3. Select "Constant Z" from the cycle list. As the pocket boundaries will be automatically calculated by the cycle, set the path curve list box to "none".



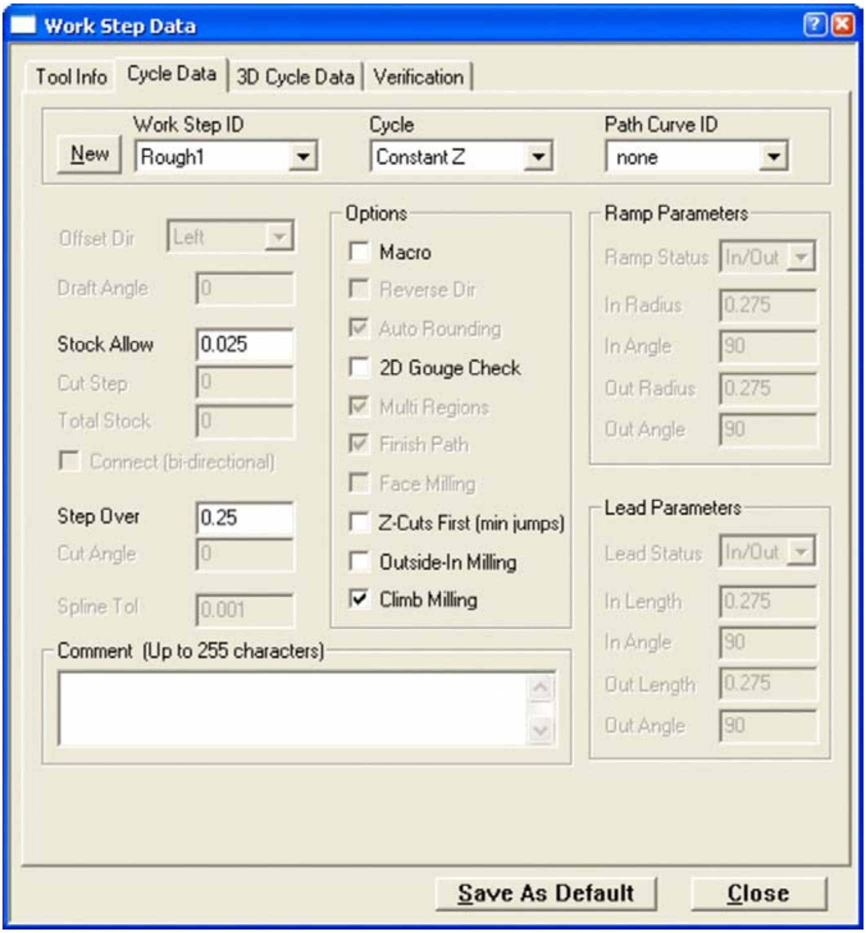
4. On the "**Tool Info**" tab, change the settings according to the table below and ensure that all parameters are set as shown in **Picture 3-8**.

<table>
<tr><th>Dialog Field</th><th>Value</th><th>Comment</th></tr>
<tr><td>Type</td><td>Bull</td><td>Bull type endmill</td></tr>
<tr><td>Number</td><td>1</td><td>Tool number in tool magazine.</td></tr>
<tr><td>Diameter (Bot.)</td><td>0.5</td><td>Defines the bottom diameter of the tool</td></tr>
<tr><td>Corner Radius</td><td>0.125</td><td>Corner radius at tooltip</td></tr>
<tr><td>Length</td><td>1.5</td><td>Tool length used for preview simulation</td></tr>
<tr><td>Z Surface</td><td>0</td><td>Set our Z surface to the top of the block</td></tr>
<tr><td>Z Rapid</td><td>1</td><td>Rapid positioning plane</td></tr>
<tr><td>Z Clear</td><td>.1</td><td>Plunge plane (Rapid to Feedrate)</td></tr>
<tr><td>Z Depth</td><td>0.975</td><td>Set final depth to 0.025 above the spherical floor</td></tr>
<tr><td>Z Step</td><td>0.25</td><td>Incremental depth per Z-level pass</td></tr>
<tr><td>Plg.Dis./Pitch</td><td>0.05</td><td rowspan="2">In order to gently enter into each Depth Level without plunging, we define a helical entry by specifying a Pitch and Helix Radius.</td></tr>
<tr><td>Helix-Rad.</td><td>0.5</td></tr>
</table>



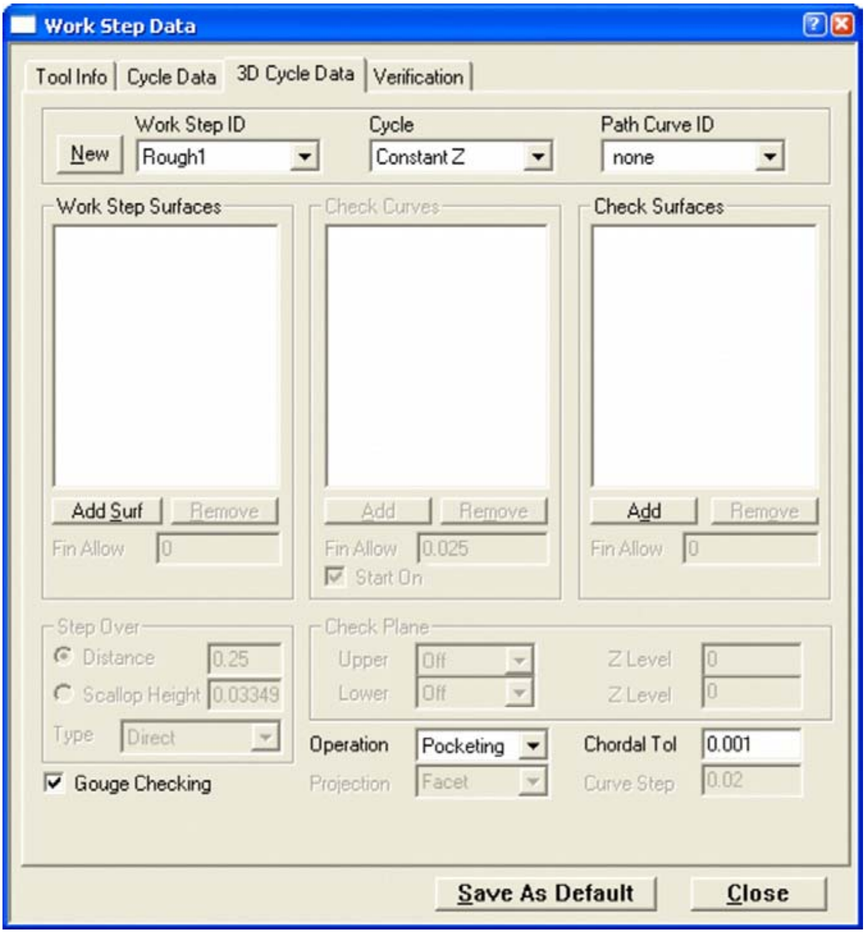
Picture 3-8

5. On the "**Cycle Data**" tab we set our "Stock Allowance" to "0.025" to ensure material remains along the walls to be removed later with our finishing operation. It is important to remember that this allowance is in 2D and is applied to the calculated slice boundaries only. The "Step Over" between passes is set to "0.25" (the diameter of the flat on the bottom of our Bull endmill). The remaining parameters should be set as shown in **Picture 3-9**.



Picture 3-9

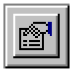
6. Next, select the "**3D Cycle Data**" tab and set the "Operation" to Pocketing. Turn ON the "Gouge Check" option that tells the system to determine the machining side of the slice boundaries based on the surface machining side, rather than a nested loop method, which starts pocketing inside the outermost loop and alternating thereafter. Check that the remaining parameters are set as shown in **Picture 3-10** and close the dialog.

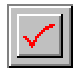


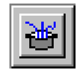
Picture 3-10

7. Now we have to select the surfaces that will be machined. Normally this is done using the "Add Surf" button below the "Work Step Surfaces" table on the "3D Cycle Data" tab. As this only allows selection from a list it is not very flexible, especially when trying to select a specific surface from a huge list. Therefore it's often easier to pick the surfaces on the screen. In the next section we'll show you how to do this.

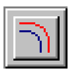
First select the "Select Cut Surfaces" command from the "Machining" menu (or Icon) and activate the "Verify" mode. As in many cases, there is no need to refine the selection, so we'll simply select all part surfaces using the "Select All" command. Finally press ENTER to complete selection and let the system add the entities to the "Work Step Surfaces" table.

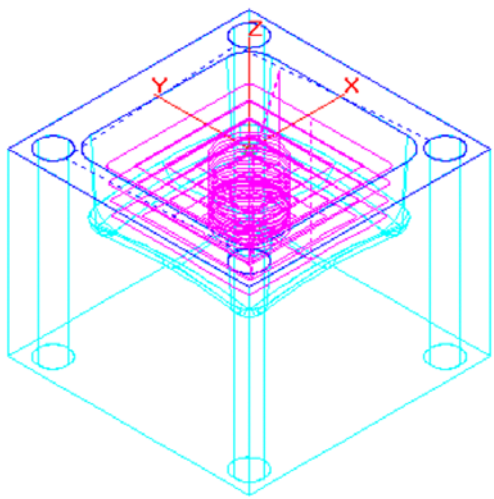

Select Cut Surfaces


Verify Mode

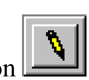

Select All

8. Click the "Verify" button and the system starts calculating the toolpath. See **Picture 3-11**.


Verify



Picture 3-11

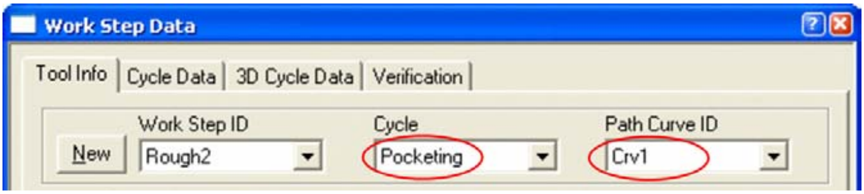
The Work Step #1 is now complete. Hit the "Redraw" button  to refresh the screen and remove the verified tool path display.

CREATING WORK STEP "ROUGH2" – PROJECTED POCKETING

Our second machining work step will rough the spherical floor of the pocket cavity using a Pocketing cycle and projecting the toolpath onto the floor. The pocketing pattern will allow our tool to enter the material near the top of the sphere were there is the least material, then gradually step outward removing more and more material.

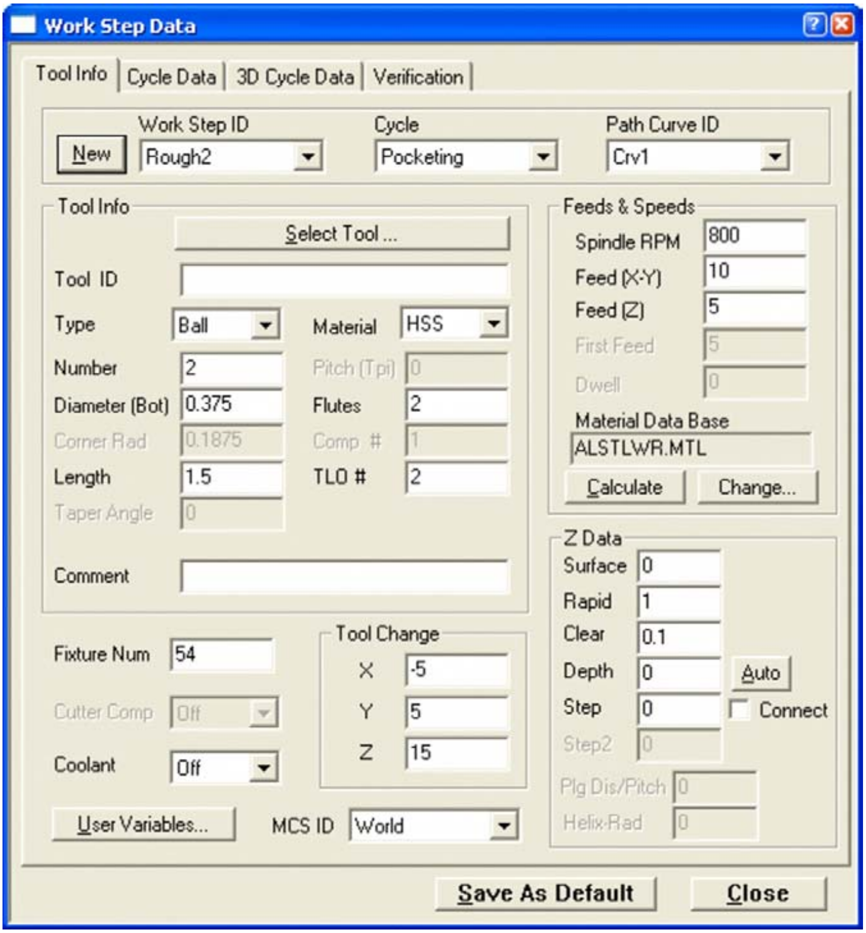
1. Select the "Work Step Data" command in the "Machining" menu to open the "Work Step Data" dialog. Once it is open switch to the "Tool Info" tab, press the "New" button and input "Rough2" as the new Work Step ID. Confirm with OK.

2. Select "Pocketing" from the cycle list and "Crv1" as the path curve.



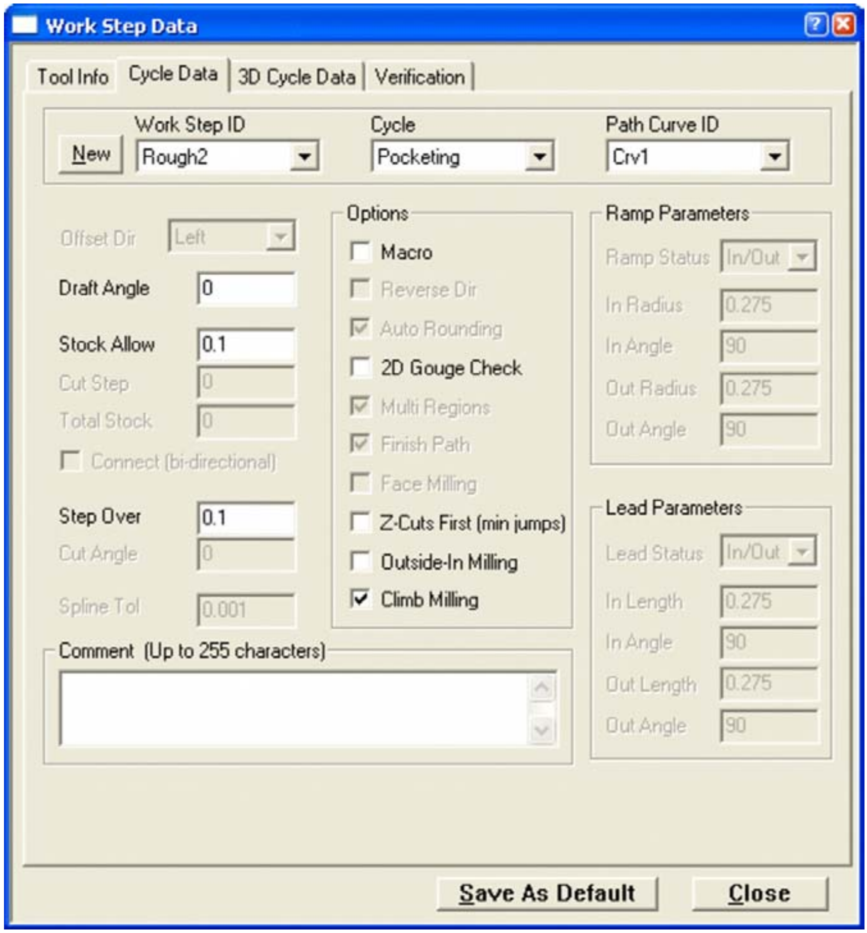
3. On the "**Tool Info**" tab, change the settings according to the table below. All other parameters are identical to the previous work step. Check and ensure that everything is set as shown in **Picture 3-12.**

Dialog Field	Value	Comment
Type	Ball	Ball type endmill
Number	2	Tool number in tool magazine.
Diameter (Bot.)	0.375	Defines the full diameter of the tool
Z Depth	0	Pocketing depth defined by surfaces
Z Step	0	For finishing no step is required



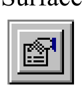
Picture 3-12

4. On the "**Cycle Data**" tab we will set our "Stock Allowance" to "0.1" to ensure the pocket pattern that is to be projected to the floor is far enough away from the walls to prevent it from climbing them. Since we are roughing and are not too concerned with the size of the cusps, we will set our "Step Over" between passes to"0.1". The remaining parameters should be set as shown in **Picture 3-13**.

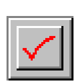


Picture 3-13

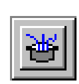
5. Next, go to the "**3D Cycle Data**" tab to define the machining surfaces as in the previous work step, but this time with the dialog still open. First select the "Select Cut Surfaces" command and activate the "Verify" mode. Then use the "Select All" command to mark all part surfaces and press ENTER to add them to the "Work Step Surfaces" table.



Select Cut Surfaces

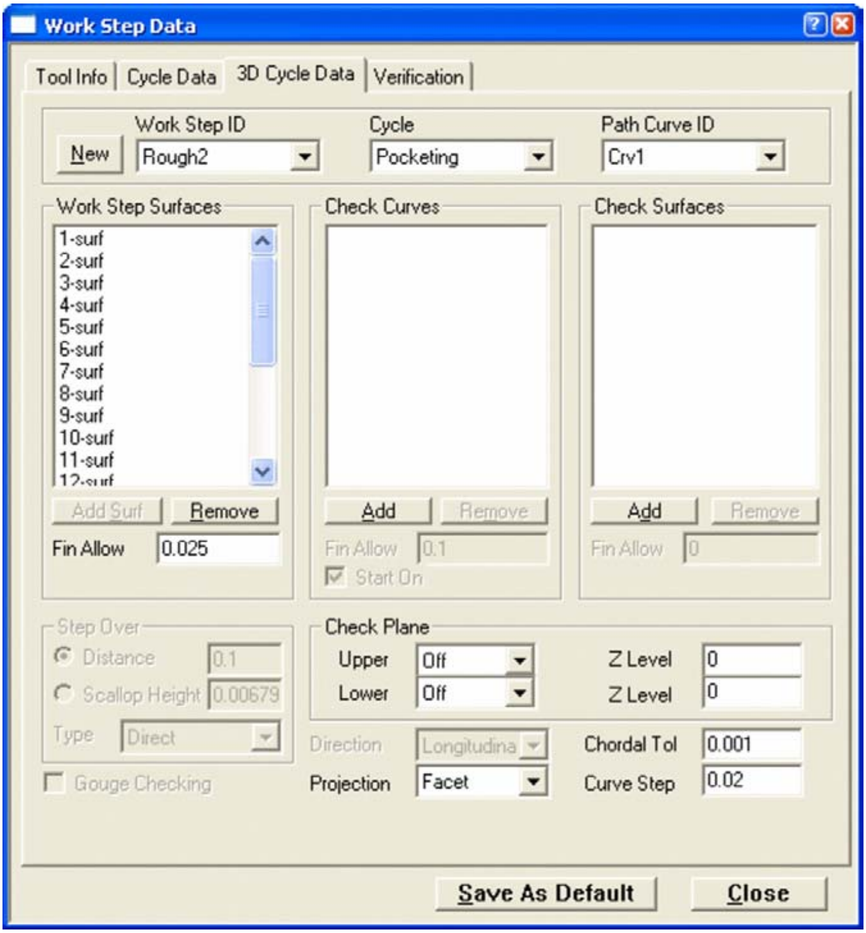


Verify Mode



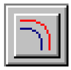
Select All

6. Once the surfaces are selected we can set their "Fin Allow" to "0.025" to ensure there is stock remaining on the floor of the pocket for our subsequent finishing operation. Ensure that everything else is set as shown in **Picture 3-14** and close the dialog.

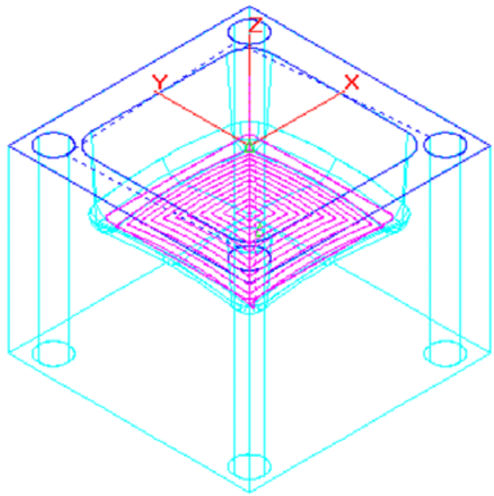


Picture 3-14

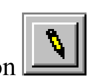
7. Click the "Verify" button and the system starts calculating the toolpath as shown in **Picture 3-15**.



Verify



Picture 3-15

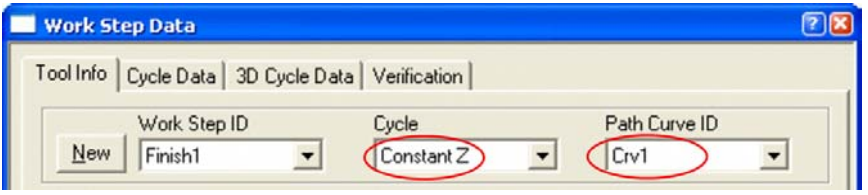
The Work Step #2 is now complete. Hit the "Redraw" button  to refresh the screen and remove the verified tool path display.

CREATING WORK STEP "FINISH1" – CONSTANT Z CONTOURING

Our third machining work step will finish the drafted walls using a contouring operation with the "Constant-Z" cycle. We will machine the walls in .025 depth steps to the depth of 1.0 (the top of the sphere) to prevent undesirable contour passes from occurring on the spherical floor below this level.

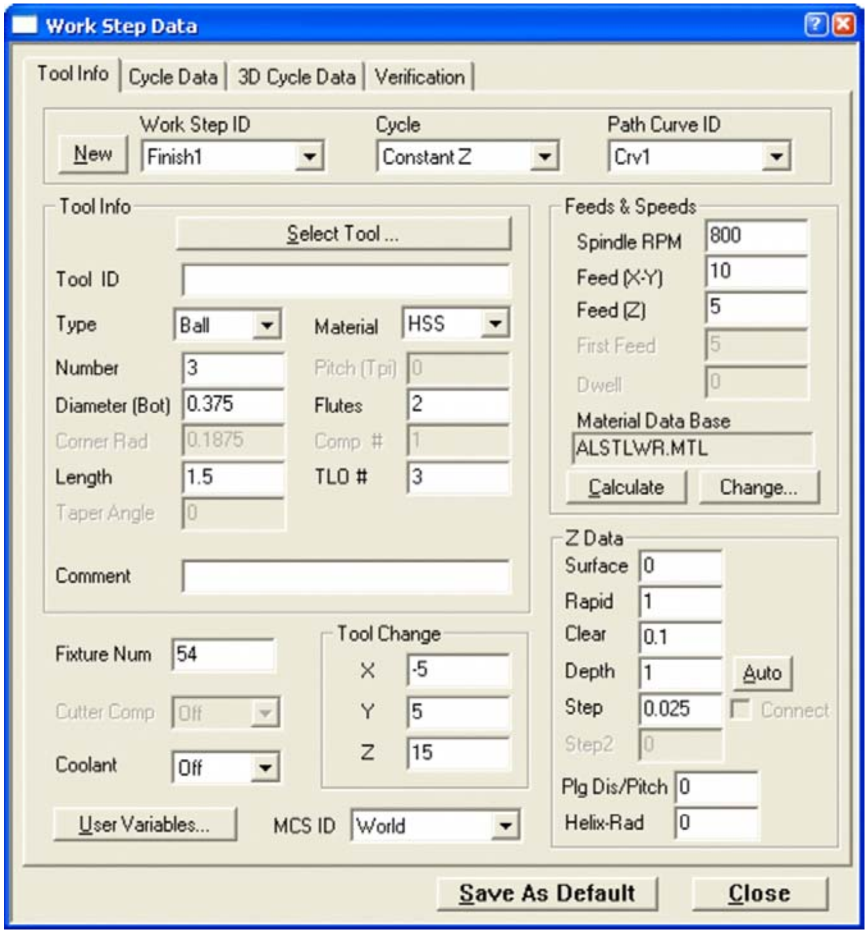
1. Select the "Work Step Data" command in the "Machining" menu to open the "Work Step Data" dialog. Once it is open switch to the "Tool Info" tab, press the "New" button and input "Finish1" as the new Work Step ID. Confirm with OK.

2. Select "Constant Z" from the cycle list and "Crv1" as the path curve in order to restrict the contouring cuts to those inside the cavity, excluding those that would otherwise be generated along the outer walls of the block.



3. On the "**Tool Info**" tab, change the settings according to the table below and ensure that all parameters are set as shown in **Picture 3-16.**

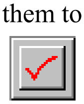
Dialog Field	Value	Comment
Type	Ball	Ball type endmill
Number	3	New tool number for finishing operation
Diameter (Bot.)	0.375	Defines the full diameter of the tool
Z Depth	1	Max depth of computed toolpath
Z Step	0.025	Incremental depth per Z-level pass



Picture 3-16

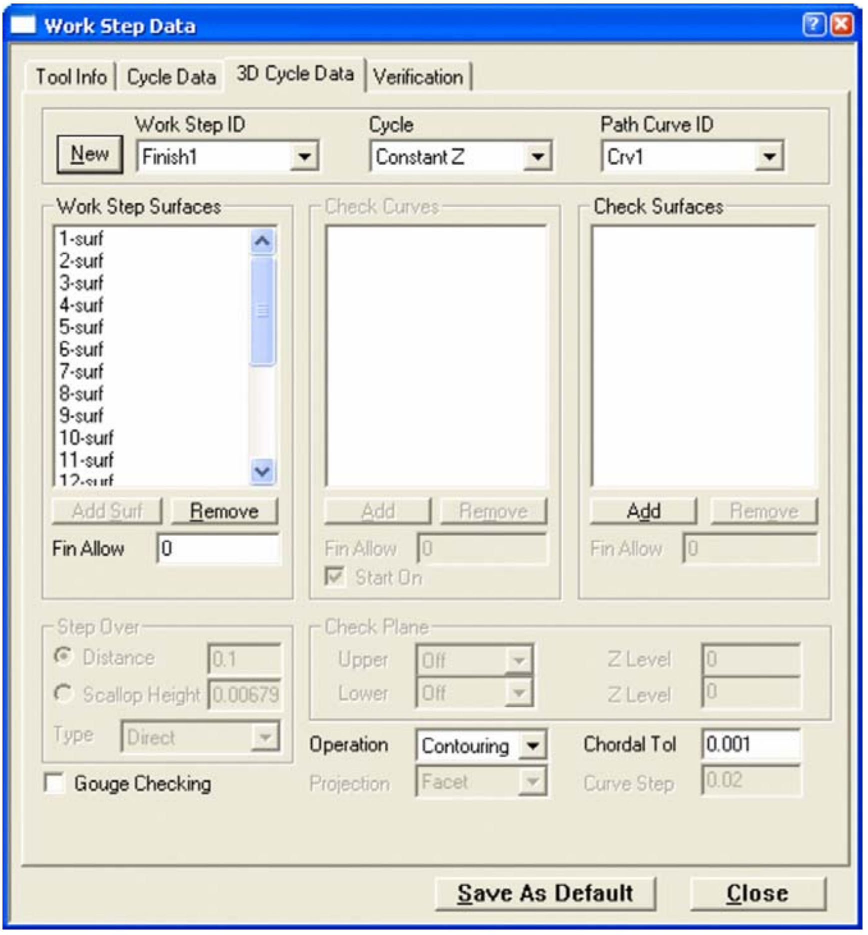
4. Next, go to the "**3D Cycle Data**" tab to define the machining surfaces as in the previous work step. First select the "Select Cut Surfaces" command and activate the "Verify" mode. Then use the "Select All" command to mark all part surfaces and press ENTER to add them to the "Work Step Surfaces" table.


Select Cut Surfaces


Verify Mode

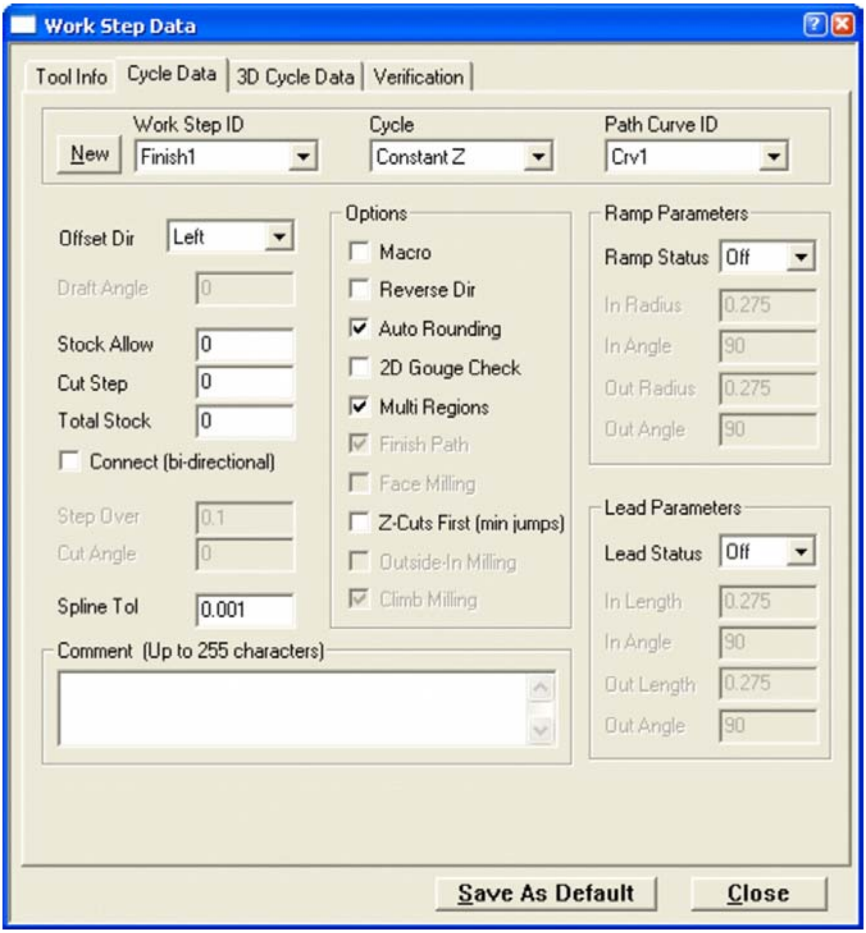

Select All

5. Once the surfaces are selected, their "Fin Allow" is set to "0" since we are finishing the walls and do not want to apply a 3D allowance to the surfaces. Also turn OFF the "Gouge Check" option since it would tell the system to remove any contour passes resulting from undercut regions. Since our part does not contain any undercuts there is no need to enable this function, and doing so would just add further to the computation time. Finally check that "Operation" is set to "Contouring" to compute contour passes only. Everything else should be set as shown in **Picture 3-17**.



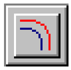
Picture 3-17

6. On the "**Cycle Data**" tab we will set our "Stock Allowance" to "0" since we are finishing the walls and do not want to apply a 2D allowance to the calculated contour slices. "Total Stock" and "Cut Step" are also set to "0" since there is no need to apply additional side passes to each contour. The remaining parameters should be set as shown in **Picture 3-18**. Finally close the dialog.

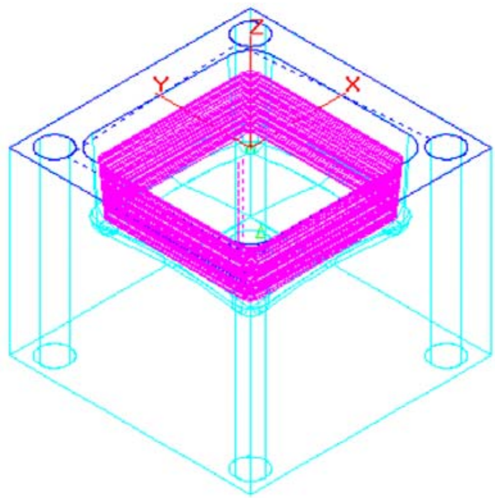


Picture 3-18

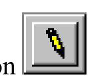
7. Click the "Verify" button and the system starts calculating the toolpath as shown in **Picture 3-19**.



Verify



Picture 3-19

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

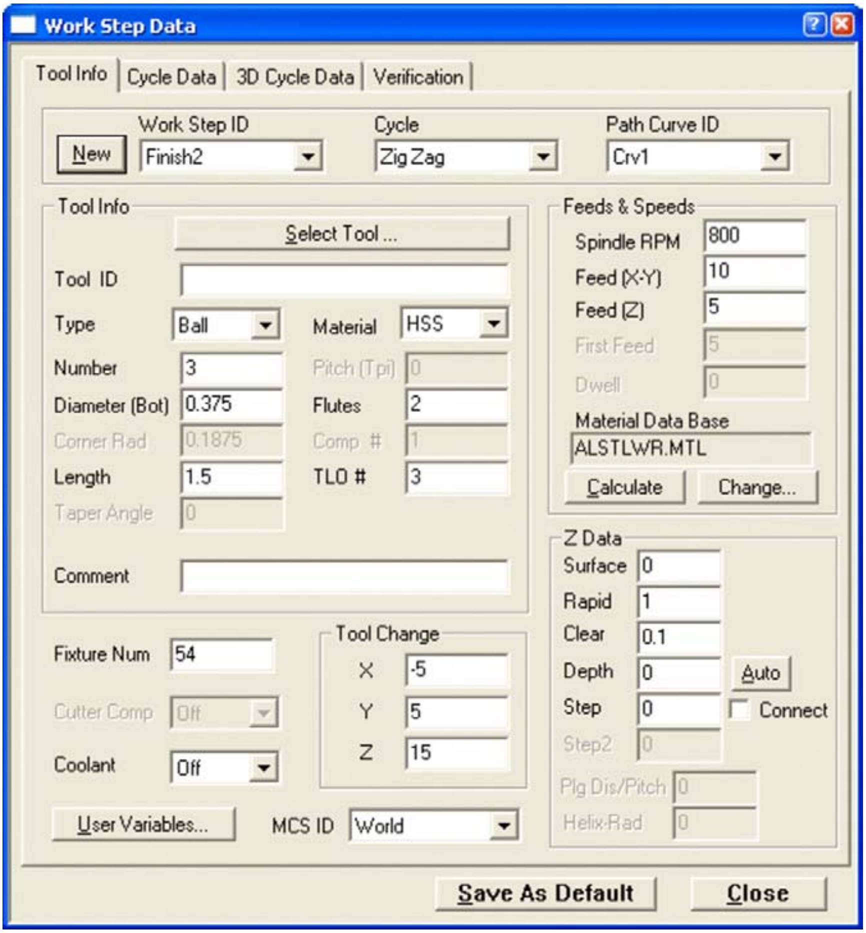
CREATING WORK STEP "FINISH2" – PROJECTED ZIG-ZAG

The fourth machining work step will finish the spherical floor of the pocket cavity using the Zig-Zag cycle by projecting it's toolpath onto the floor. The zigzag pattern will allow our tool to step over leaving minimal cusps, and we can use the "Variable Step Over" option to automatically reduce the step over for cuts occurring along the walls.

1. Select the "Work Step Data" command in the "Machining" menu to open the "Work Step Data" dialog. Once it is open switch to the "Tool Info" tab, press the "New" button and input "Finish2" as the new Work Step ID. Confirm with OK.

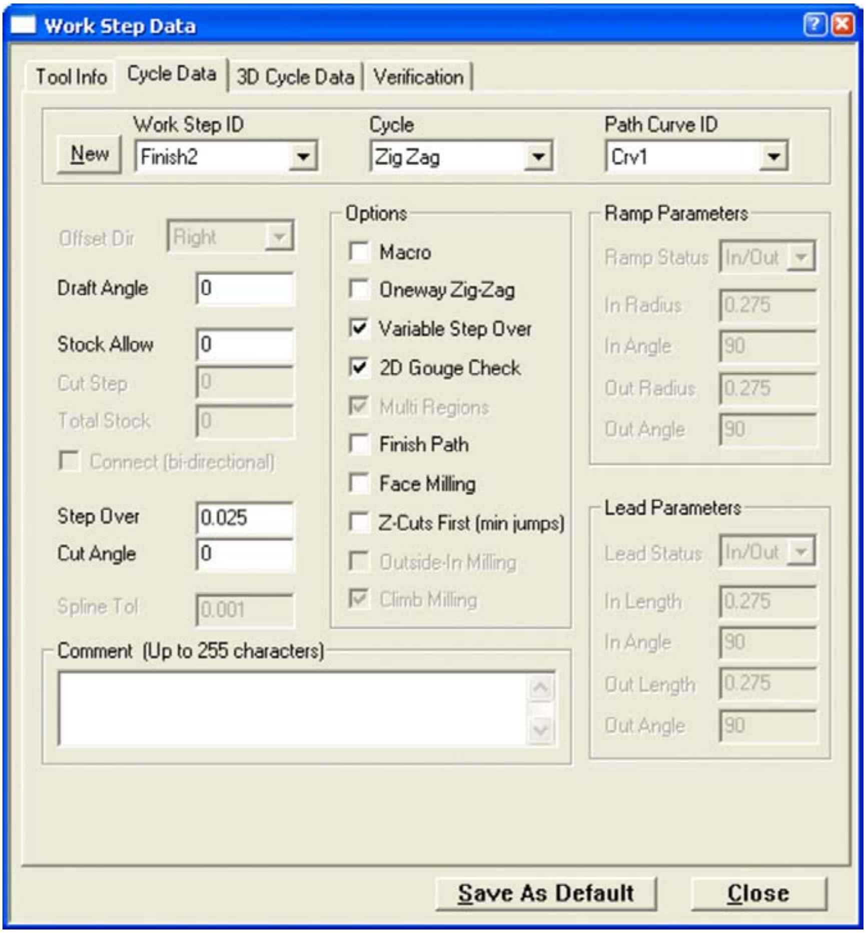
2. Select "Zig-Zag" from the cycle list and "Crv1" as the pocket boundary from the path curve list box.

3. On the "**Tool Info**" tab, we'll change the "Depth" and "Step" to "0". This is because we are finishing and there is no need to adjust the Depth of the projected toolpath. All other parameters are identical to the previous work step. Check and ensure that everything is set as shown in **Picture 3-20.**



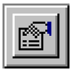
Picture 3-20

4. On the "**Cycle Data**" tab, "Stock Allowance" is set to "0". This ensures the zigzag pattern that is to be projected to the floor covers the entire area within our curve Crv1 boundary. Since we want to leave minimal cusps on the floor, our "Step Over" distance between passes is set to "0.025". We will also turn ON the "Variable Step Over" option to automatically reduce the distance between cuts occurring along the walls, where the surface slope is steepest. The remaining parameters should be set as shown in **Picture 3-21**.

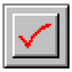


Picture 3-21

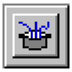
5. Next, go to the "**3D Cycle Data**" tab to define the machining surfaces as in the previous work step. First select the "Select Cut Surfaces" command and activate the "Verify" mode. Then use the "Select All" command to mark all part surfaces and press ENTER to add them to the "Work Step Surfaces" table.



Select Cut Surfaces

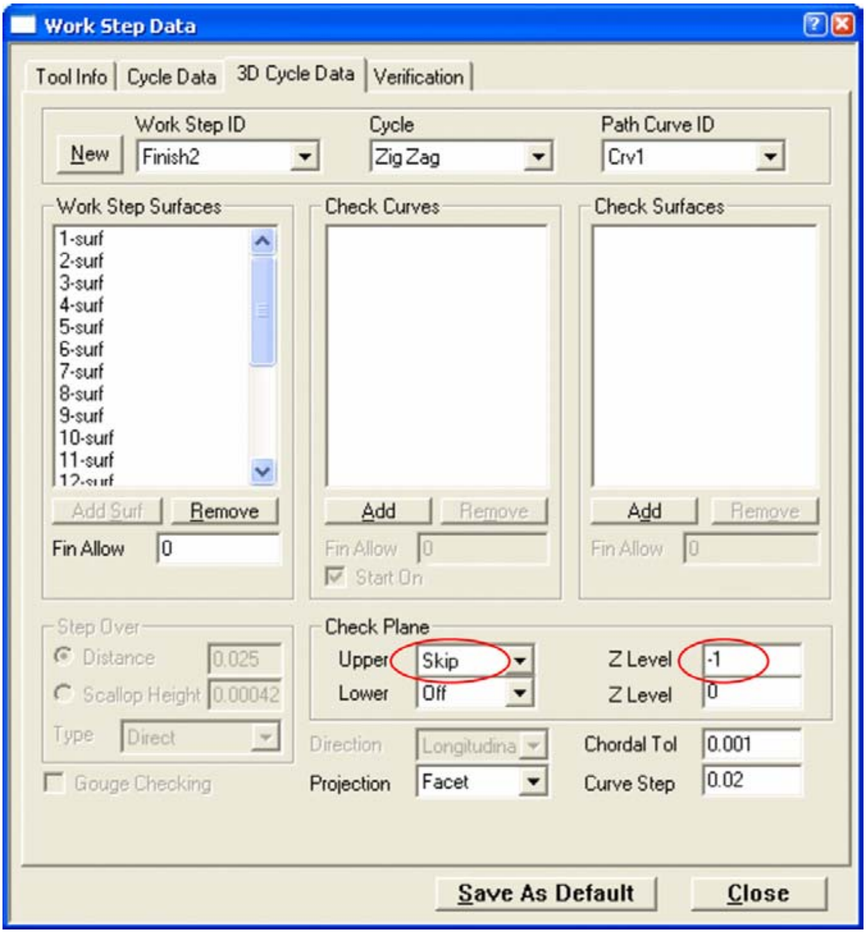


Verify Mode



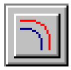
Select All

6. Once the surfaces are selected, check that their "Fin Allow" is "0" to ensure there is no stock remaining on the floor of the pocket after our finishing operation is complete. In order to prevent our projected zigzag toolpath from climbing the cavity walls and re-cutting areas that were finished with our previous "Finish1" work step, we will specify an "Upper Check Plane" to "Skip" everything above the "Z Level" of "-1.0" (the absolute Depth to which we finished in the previous work step). Ensure that everything else is set as shown in **Picture 3-22** and close the dialog.

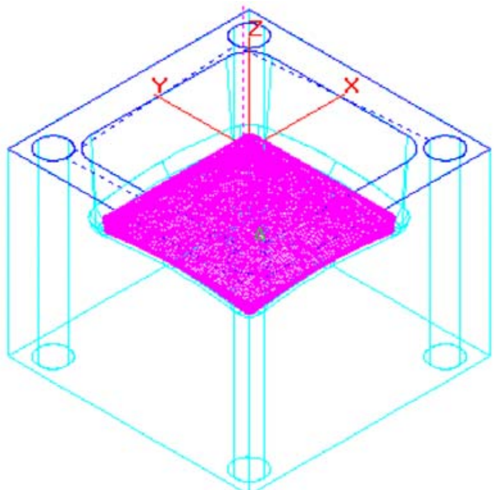


Picture 3-22

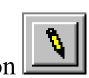
7. Click the "Verify" button and the system starts calculating the toolpath. See **Picture 3-23**.



Verify



Picture 3-23

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

CREATING WORK STEP "DRILL1" – DRILLING

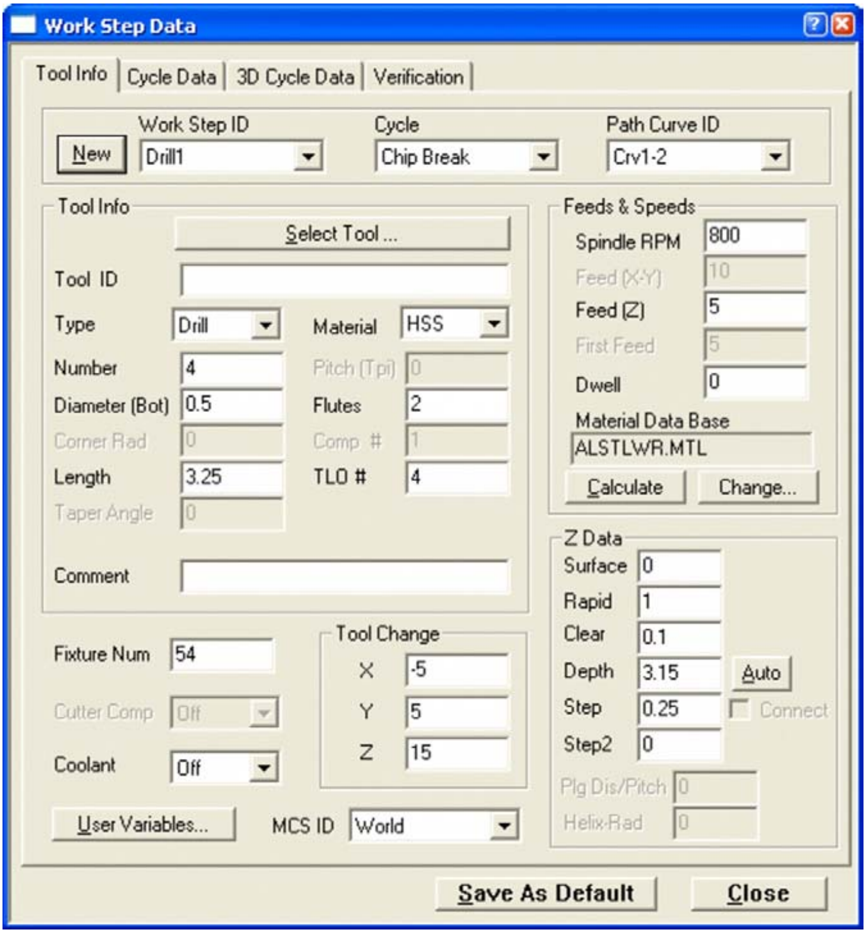
Our last machining work step will drill the .5 diameter though holes using the "Chip Breaking" drill cycle.

1. Select the "Work Step Data" command in the "Machining" menu to open the "Work Step Data" dialog. Once it is open switch to the "Tool Info" tab, press the "New" button and input "Drill1" as the new Work Step ID. Confirm with OK.

2. Select "Chip Break" from the cycle list and "Crv1-2" as the path curve.

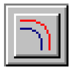
3. On the "**Tool Info**" tab, we define a new tool and adjust the depth settings as shown in the table below. Ensure that everything else is set as shown in **Picture 3-24** and close the dialog.

Dialog Field	Value	Comment
Type	Drill	Ball type endmill
Number	4	Tool number in tool magazine.
Diameter (Bot.)	0.5	Defines the full diameter of the tool
Length	3.25	Tool length used for preview simulation
Z Surface	0	Set absolute reference plane for operation
Z Depth	3.15	Drill depth (tooltip)
Z Step	0.25	Required step for chip breaking

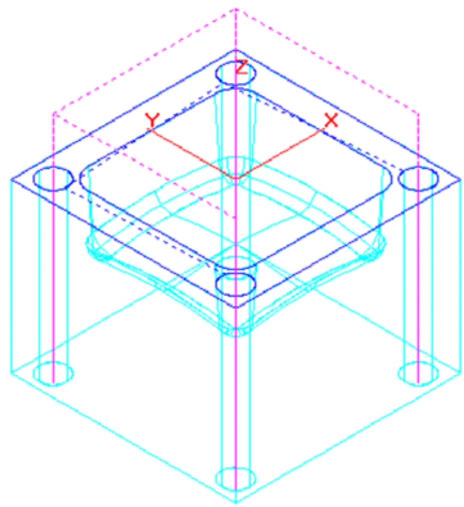

Work Step Data
Tool Info
Cycle Data
3D Cycle Data
Verification
New
Work Step ID
Drill1
Cycle
Chip Break
Path Curve ID
Crv1-2
Tool Info
Select Tool ...
Tool ID
Type
Drill
Material
HSS
Number
4
Pitch (Tpi)
0
Diameter (Bot)
0.5
Flutes
2
Corner Rad
0
Comp #
1
Length
3.25
TLO #
4
Taper Angle
0
Comment
Feeds & Speeds
Spindle RPM
800
Feed (X-Y)
10
Feed (Z)
5
First Feed
5
Dwell
0
Material Data Base
ALSTLWR.MTL
Calculate
Change...
Z Data
Surface
0
Rapid
1
Clear
0.1
Depth
3.15
Auto
Step
0.25
Connect
Step2
0
Plg Dis/Pitch
0
Helix Rad
0
Fixture Num
54
Cutter Comp
Off
Coolant
Off
Tool Change
X
-5
Y
5
Z
15
User Variables...
MCS ID
World
Save As Default
Close

Picture 3-24

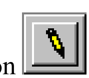
4. Click the "Verify" button and the system starts calculating the toolpath as shown in **Picture 3-25**.



Verify



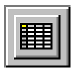
Picture 3-25

The Work Step #5 is now complete. Hit the "Redraw" button  to refresh the screen and remove the verified tool path display.

VIEW ESTIMATED MACHINING TIMES

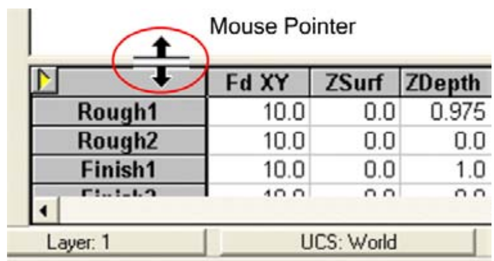
Now that all work steps have been defined and verified we can view the estimated machining times (not including rapid traverse or tool change times) displayed in the "Time" column of the integrated spreadsheet.

1. Open the spreadsheet by selecting the "Show Spreadsheet" command from the "Machining" menu or the toolbar.

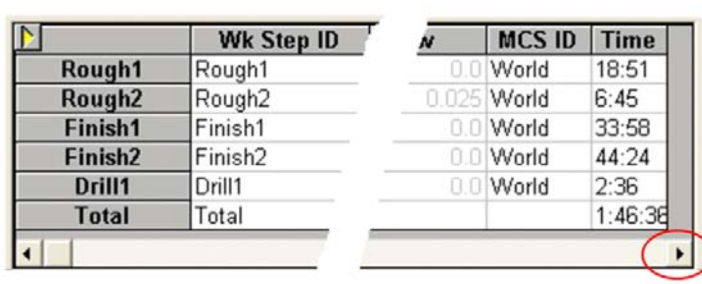


Show Spreadsheet

In case not all work steps are displayed, move the cursor to the upper border of the spreadsheet until the mouse pointer changes to the double–arrow. Then keep the left mouse button pressed while dragging the cursor up to the size you want.

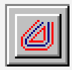


2. Now click anywhere into the spreadsheet to display the bottom slider with it's left/right arrows and use the right arrow until the "Time" column gets visible on the screen. Please note that the displayed time is based on what was previously verified. If any toolpath verification process was terminated, the estimated time in the corresponding column section may not reflect the correct value.

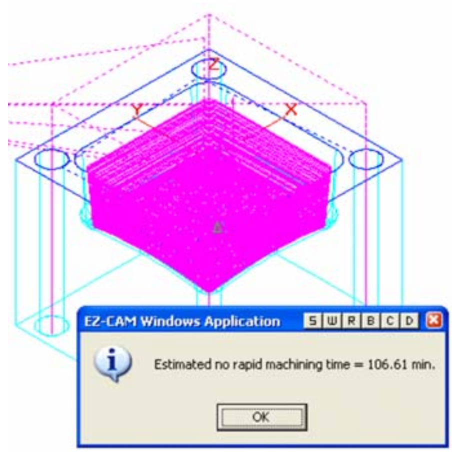


Click here to scroll the Columns to the right

If you are not sure that all work steps have been verified correctly, or have started a new session and loaded your previously saved work, use the “Verify All” command from the “Machining” menu. It performs an on-screen verification of all work steps in memory, in the machining order. The total machining time (not including rapid traverse or tool change time) is displayed in a dialog box at the end of the verification process as shown in **Picture 3-26**.



Verify All



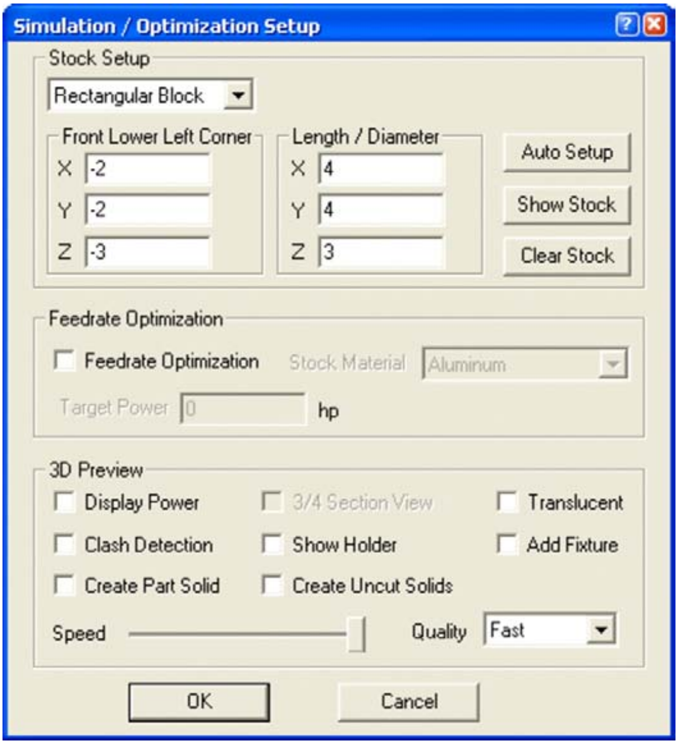
Picture 3-26

It is also very important for the “3D Preview” solid simulation that is explained later to have all toolpaths verified correctly. Only tool moves that have been verified will be simulated.

3D SOLID MODEL PREVIEW

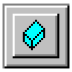
One of the most powerful EZ-CAM features is the 3D solid preview function. This function shows an animated tool cutting a solid model of the programmed part. After previewing our program we are left with an accurate solid model representation, which allows us to closely examine the surface finish and resulting part details. Once the simulation is finished or interrupted by the user pressing "Esc" key, all dynamic view commands to rotate, zoom or move the simulated model on the screen are available. If no stock was defined prior to calling the "3D Preview" command, the system automatically calculates the stock size, according to the maximum calculated tool movements. For the tutorial we will manually assign the stock size using the "Stock & Optimization Setup" dialog that can be opened from the "Machining" menu.

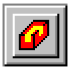
1. Select the "Stock & Optimization Setup" command from the "Machining" menu and input the values as shown in **Picture 3-27**. Close the dialog with OK.

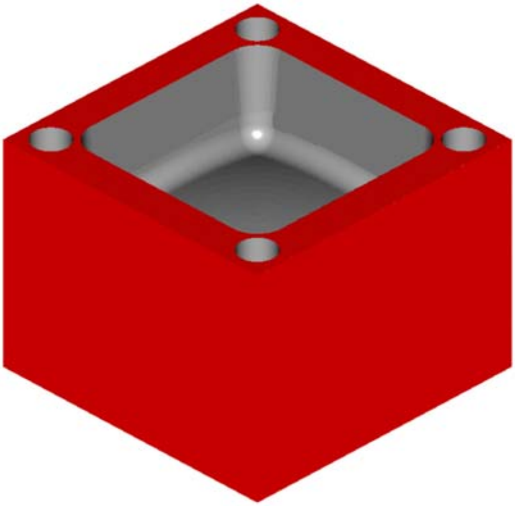


Picture 3-27

2. Before starting the preview select the “Isometric View” command. Then 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, ranging from 1 (slowest) to 9 (fastest). See **Picture 3-28**.

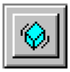

View Isometric


3D Preview

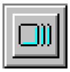


Picture 3-28

3. Once the simulation stopped you can change the on-screen view by using the dynamic view commands (Rotate, Pan, Zoom) from the “View / Dynamic Viewing” menu.


Dynamic Rotate


Dynamic Zoom


Dynamic Pan

SAVING THE PART PROGRAM

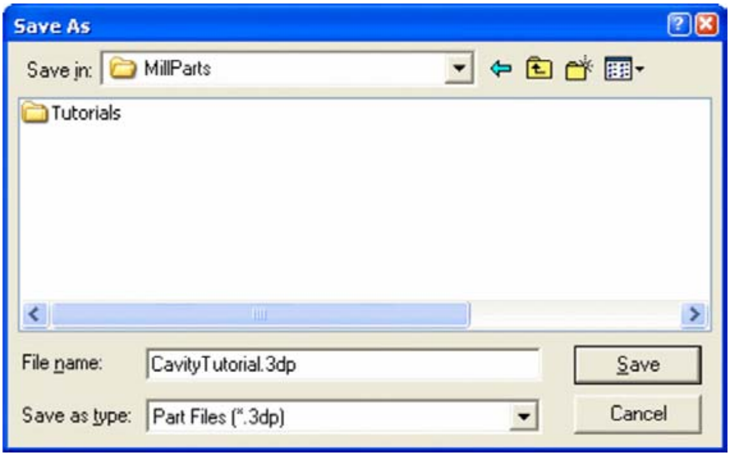
It is very important to save the newly created or edited part from memory to disk periodically during a session, as well as at the end to ensure that no information is lost. The “Save” and “Save as” commands under the “File” menu transfer files from system memory to hard disk or other media. In EZ-MILL, the part information is stored in three different file types. The “Part” file using the extension "3DP", the associated “Geometry” file with extension "GEO" and the surface file with the extension “3GX”.

File Type : **GEOMETRY**
Extension : **GEO**
Data : Geometry Elements (lines, arcs, etc.), Curves,
Surface Boundary Curves (hidden entities)
User Coordinate Systems (UCS)

File Type : **PART Files**
Extension : **3DP**
Data : Work Step Data (Technology & Machining Information)

File Type : **SURFACE Files**
Extension : **3GX**
Data : Surface Data

There is no specific rule what should be saved first. Of course, if there is only one kind of data in memory (Work Steps or Geometry) the “Save as” dialog will automatically be set to the correct file type.



Picture 3-29

1. Select the "Save as" command from the "File" menu.

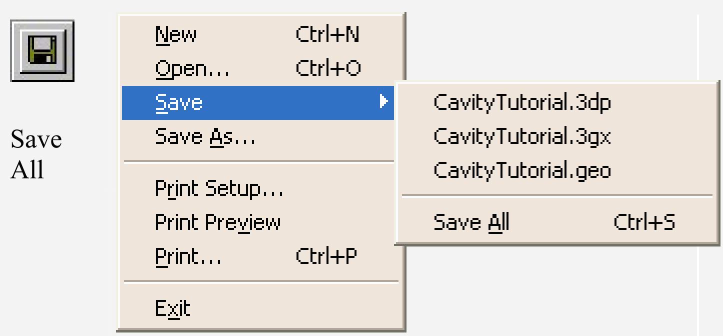
2. Select the appropriate drive and folder where the geometry and part files should be stored. You can use the "EZCAMW \ MILLPARTS" folder that was automatically created by the setup routine.

3. Select "Part Files (*.3dp)" from the "Save as type" list box to save the machining data first.

4. Type the new filename "CavityTutorial" in the "File Name" box and click the "Save" button. The file extension is added automatically. See **Picture 3-29**.

5. Repeat steps 1 to 4 to save the geometry and surface information as well. Select the appropriate file type from the "Save as type" list box. This automatically changes current file extension, but keeps same filename as used previously when storing the geometry data.

If you have already saved the geometry, the software automatically inserts a part file with the same name but different extension (*.3DP) in the "Save" menu when the first Work Step is created. All you have to do is to select "Save All" option from the "File" menu or the corresponding toolbar button.



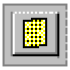
The software will save and <u>overwrite the existing files without any screen prompt</u>. You can use this command anytime for fast saving of your work.

It is not possible to save data when the software is running in evaluation mode. The “Save”, “Save as” and “Print” commands are disabled.

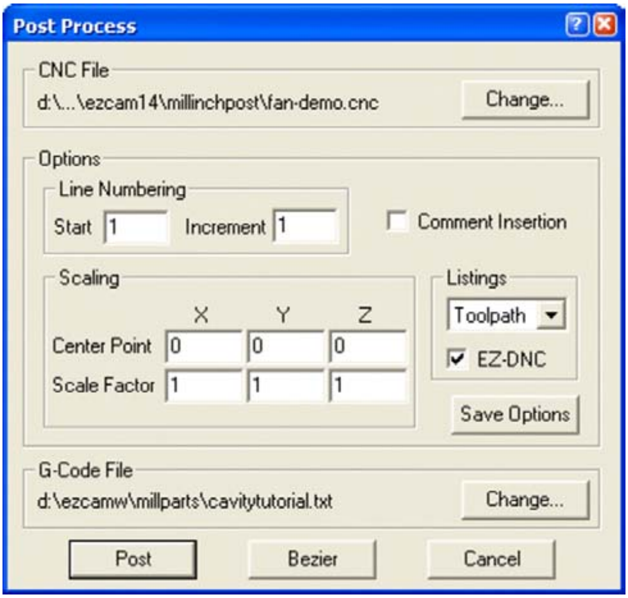
CREATING CNC CODE

Now that the part program has been created, it must be converted to run on a NC control by running the “Post” command with the appropriate “Post-Processor” for your machine.

1. Select “Post” command in the “Machining” menu to open the “Post Process” dialog.



Post



Picture 3-30

2. First you need to select the postprocessor. If the one desired is already loaded and displayed in the section "CNC-File", continue to the next step. Otherwise use the "Change" button to browse your system for a different one. For this tutorial you may use the "FAN-DEMO.CNC" post (standard inch post that creates Fanuc style code).

> Standard postprocessor folders created by the EZ-CAM v14 setup:
>
> INCH
> <DRIVE>: \ EZCAMW \ EZCAM14 \ MILLINCHPOST
>
> METRIC
> <DRIVE>: \ EZCAMW \ EZCAM14 \ MILLMETRICPOST

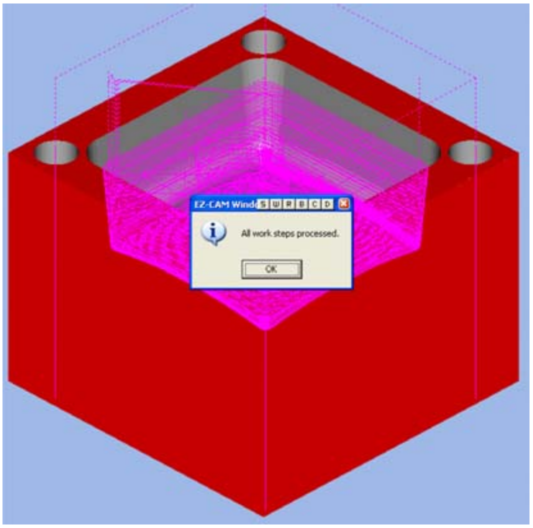
3. Select the "Toolpath" option from the "Listings" list box to watch the process by viewing the toolpath that is created parallel to the computed NC code. Displaying the G-Code as in the other tutorials makes no sense, since in 3D programming the text display would increase overall computing time drastically.

4. Activate (check) the "EZ-DNC" option. This will automatically start the "EZ-DNC" application when posting of the part file is finished and load the newly created file for sending it top the machine using the serial port. See Chapter 6 "Communication with the Control" for more information about EZ-DNC.

5. Next is the "G-Code File" section. Here, the default name and folder for the computed program file is displayed. The name is taken from the part file that was previously saved. The default directory is "EZCAMW\MILLGCODE".

> Ensure that part file and postprocessor share the same dimension unit ("Inch" for this tutorial). The system will generate a "Dimension Unit Conflict" message, but then automatically scale the NC-Code according to the dimension specified in the postprocessor.
>
> View the online help for more information about the "Setup" dialog located in the "View" menu.

6. Click the “Post” button to start posting. Once all Work Steps have been processed, a final message as shown in **Picture 3-31** is displayed.



Picture 3-31

7. Finally click the OK button to close the message dialog box

Congratulations!

You've completed the EZ-MILL 3D Cavity Tutorial !