EZ-MILL - PRO

3D Surface Tutorial

Release 13.0

Copyright Notice

1. TABLE OF CONTENTS

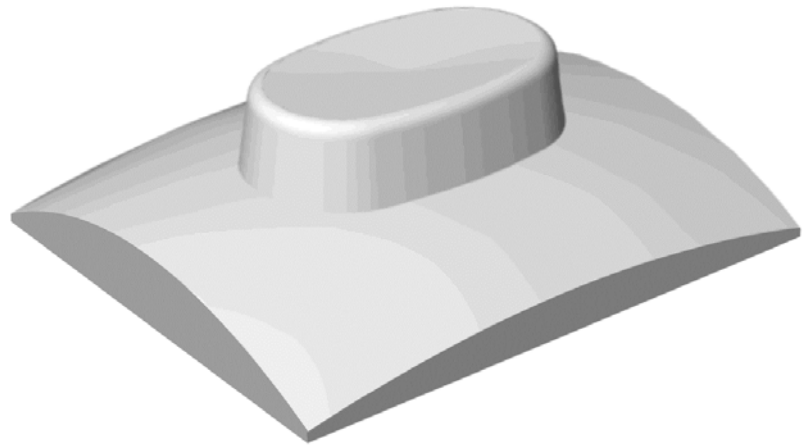
CHAPTER 1.
EZ-MILL 3D TUTORIAL

OVERVIEW

This 3D tutorial is intended for users who purchased the MILL-PRO option with enhanced surface creation and machining capability. It is recommended to step through the Mill 2D tutorials first. This will help you to gain considerable experience in 2D operations like geometry and curve creation, copying and transforming entities, as well as general machining strategies before continuing to the 3D tutorial.

The 3D tutorial includes step-by-step instructions that describe the complete process of creating the NC program for the 3D part shown in **Picture 1-1**and **Picture 1-2**. We will load a previously created geometry file to show how to define the curves that are needed to create the various surface types for the part. Once surface creation is done we continue to define work steps for roughing and finishing the part.

Please remind that there may be several possible ways to machine that particular 3D part depending on tooling, part material and machine. To keep this tutorial simple we decided to stay with only four work steps and focus on basic machining principles rather than trying to find the most efficient way to machine the part.



Picture 1-1

BASIC PROGRAMMING STEPS

Before we continue with the tutorial let us explain the steps that will be explained.

STEP 1. **Geometry / Curve Creation / Surface Creation**

Surface geometry is created from two and three-dimensional curves that are created by chaining or linking together geometry elements. Because the tutorials focus is surface creation and machining we will skip the geometry creation and load a geometry file from disk instead. The geometry data is then used to define the curves needed for surface creation. In addition we also have to create a boundary curve that is later used in the roughing operation.

To improve and train your geometry skills it may be a good idea to use the existing geometry file as an aid to try it yourself. In that case use the drawing provided in **Picture 1-2** for the parts dimensions.

Use the previously defined curves to create the parts surfaces. Finally trim some of the surfaces to get the correct shape of the part.

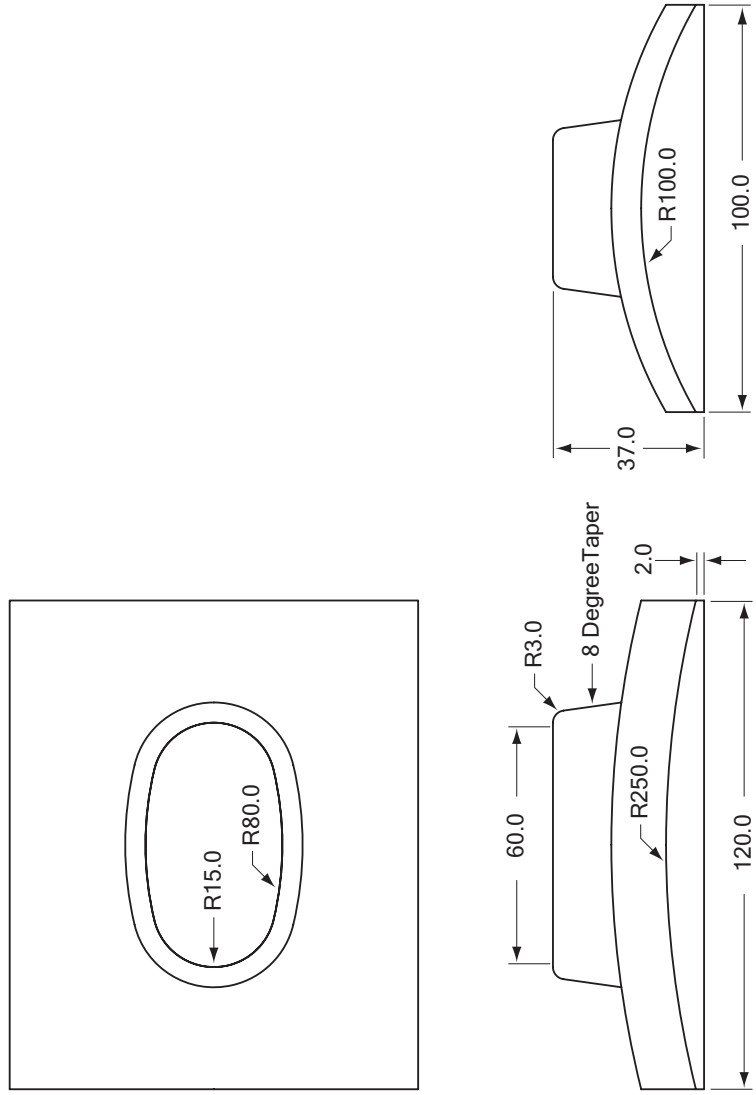
STEP 2. **Create Work Steps and set Machining Parameters**

Define Work Steps for roughing and finishing operations. Apply the parameters as required by type of operation and tool that is used. Visualize the computed tool path to assure correct tool operation and proper setting of machining parameters.

STEP 3. **Post G-Code**

Select the "Postprocessor" related to the type of control and let the software create the G-Code file.

This tutorial is set up in metric dimensions !



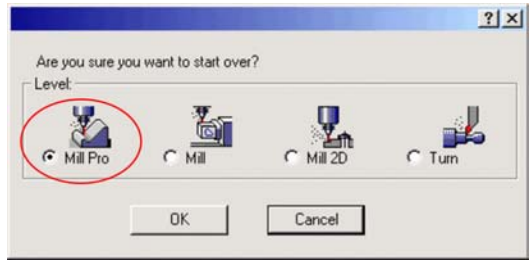
Picture 1-2

Blueprint of EZ-Mill 3D Tutorial

SETTING PREFERENCES

Before continuing with the construction of the sample part, several parameters should be set so that the system is compatible with the instructions in this tutorial. Also the size of the workspace should be set. The sample part is about 125mm in the X-axis and 105mm in the Y-axis. Because of the size of the part, it is not convenient to work in the default window; therefore the window and some default settings have to be changed.

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 "**Mill-Pro**" level is active and press **OK** to start over.



The "**New**" dialog is also used to switch between EZ-Mill levels and the EZ-Turn module. Before the dialog opens, the system check's the software protection key for activated modules. Modules or levels that are not activated will be marked by appended "DEMO" text. The system automatically saves the current level as default for future sessions.

When starting the system in "Demo" (for evaluation) mode, it is not possible to print or save data. The "Save" and "Save as" commands are disabled. Continue with the next step in the tutorial.

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

3. Type "**0**" for "**X Center**", "**0**" for "**Y Center**", "**10**" for "**Z Center**" and "**100**" for "**Size**". This sets the viewport and window allowing enough room to see all of the part as it is created.

4. Select "**Metric**" option button as the parts input dimension system.

5. Click the "**Background**" list box and select "**Black**".

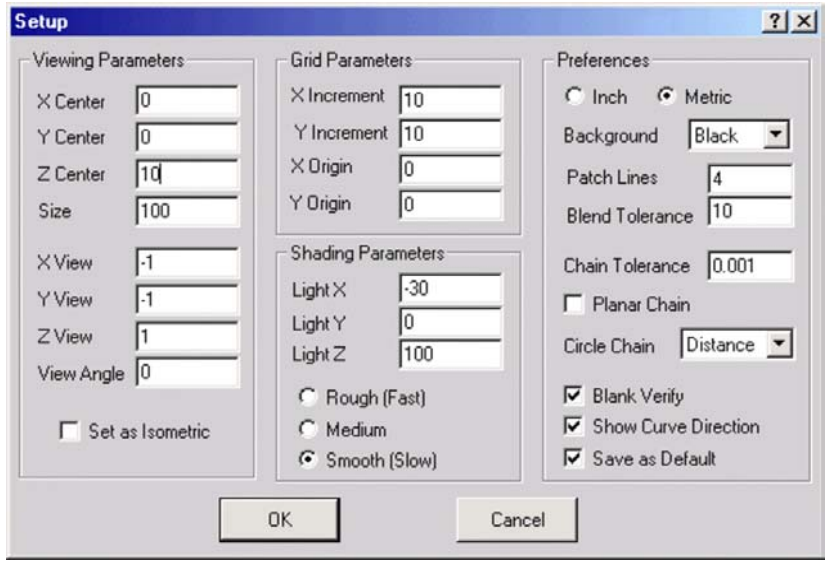
6. Ensure that the "**Planar Chain**" checkbox is **not checked**.

7. 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.

8. Check the box "**Show Curve Direction**" option. The system then displays a small arrow at the first curve point to indicate start and direction of the curve.

9. Check the box "**Save as Default**". The system will store all dialog settings as defaults for future sessions.

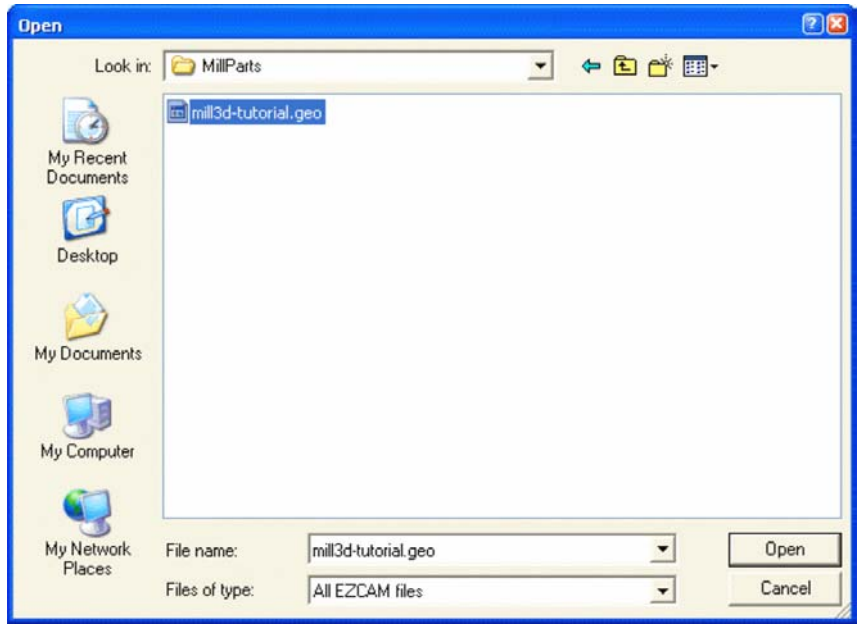
10. After the preferences have been correctly set, click **OK**.



Picture 1-3

LOADING THE GEOMETRY

This section shows how to load the tutorials existing geometry file. The file named "MILL3D-TUTORIAL.GEO" was copied to your computer by the setup program and is located in the "EZCAMW \ MILLPARTS" folder.

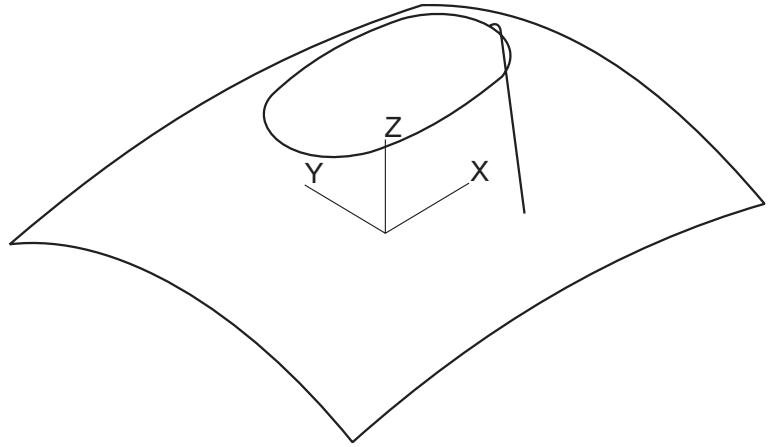


Picture 1-4

1. Select "**Open**" command from the "**File**" menu to open the file dialog. In **Picture 1-4** you can see the standard Windows XP dialog. 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

4. Select the file "**MILL3D-TUTORIAL.GEO**" and click the "**Open**" button. The imported geometry should appear as in **Picture 1-5**.



Picture 1-5

Imported Geometry

CREATE THE FIRST SURFACE (COONS TYPE)

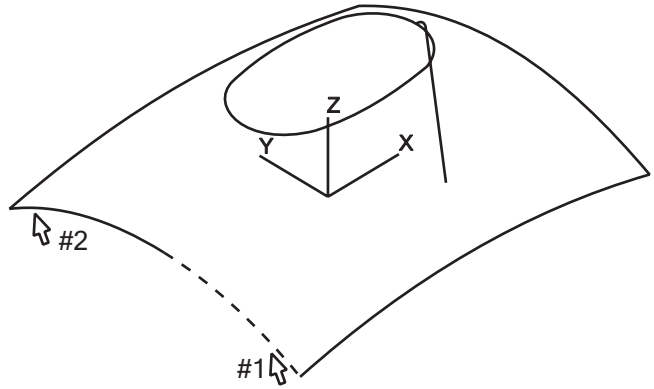
This step creates the curved main surface that represents the bottom of the part. As we will use the "Coons" surface type for this, we have to define four curves (two longitudinal and two transverse) to make up the surface. These curves each define one edge of the surface with coincidental endpoints/start points.

1. Select the "**New**" command from the "**Curves**" menu or click the corresponding button to start creation of the first longitudinal curve. In the dialog that opens press **OK** to confirm "**Crv1**" as the new ID.

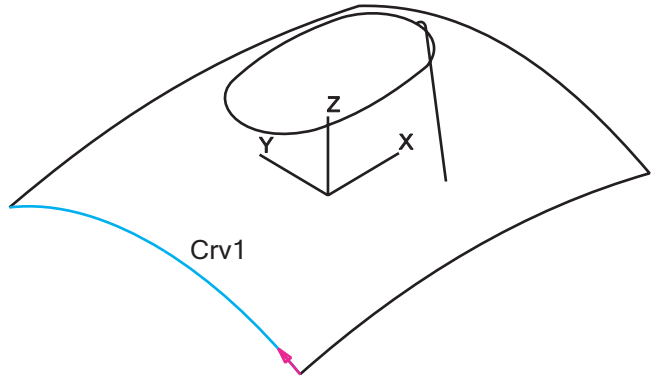

New

2. Select "**Chain**" from the "**Curve**" menu or click the corresponding button. Move the cursor to **Position #1** as shown in **Picture 1-6** and click once. The arc is then highlighted and selected as the first element of the chain operation. Then move the cursor to Position #2 and again do a single click. This selects the second arc as the end of the chain and the software automatically completes the curve as shown in **Picture 1-7**.

Chain



Picture 1-6



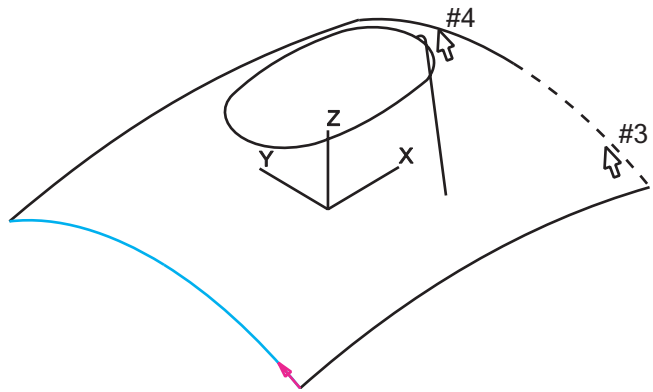
Picture 1-7

3. Again, select the "**New**" command to create the second longitudinal curve and confirm "**Crv2**" as the new ID.

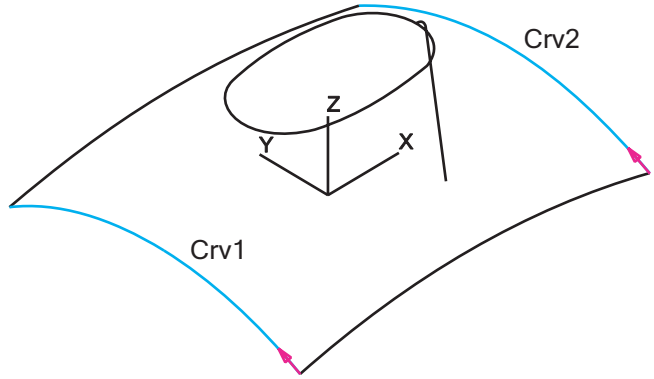

New

4. Select "**Chain**" command and move the cursor to **Position #3** and **#4** as shown in **Picture 1-8.** Click once at each location and the system automatically completes the curve as shown in **Picture 1-9**.


Chain



Picture 1-8



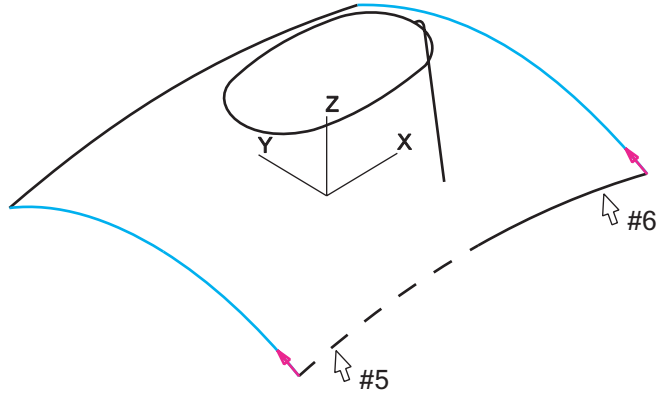
Picture 1-9

5. Select the "**New**" command for the first transversal curve and confirm "**Crv3**" as the curve ID.

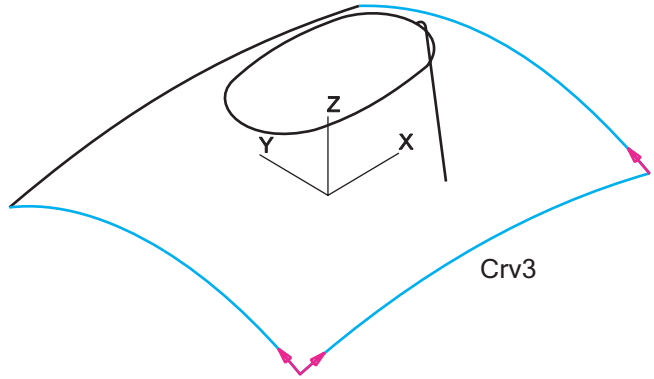

New

6. Select "**Chain**" and move the cursor to **Position #5** and **#6** as shown in **Picture 1-10.** Click once at each location and the system automatically completes the curve as shown in **Picture 1-11**.


Chain



Picture 1-10



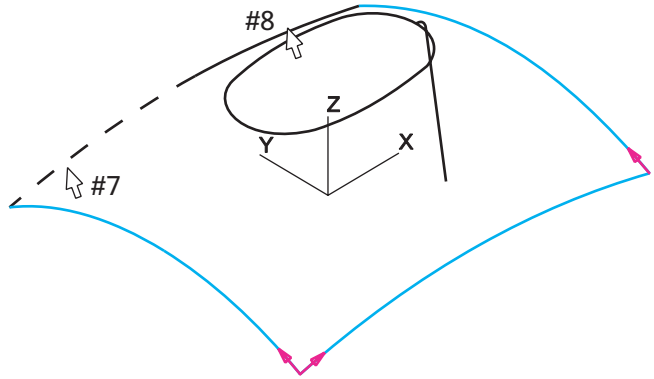
Picture 1-11

7. Select the "**New**" command for the second transversal curve and confirm "**Crv4**" as the curve ID.

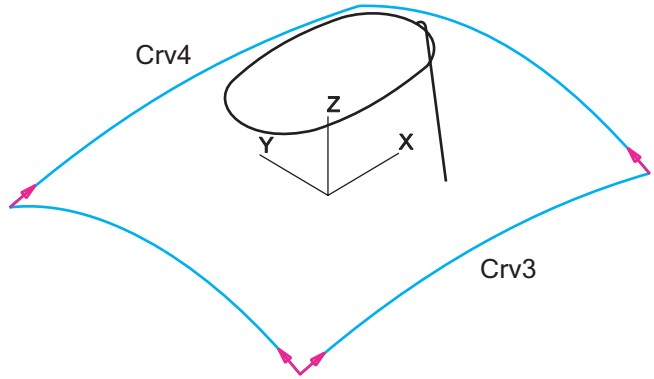

New

8. Select "**Chain**" and move the cursor to **Position #7** and **#8** as shown in **Picture 1-12.** Click once at each location and the system automatically completes the curve as shown in **Picture 1-13**.


Chain

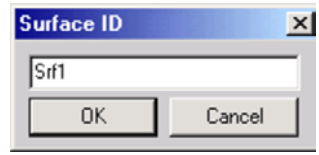


Picture 1-12

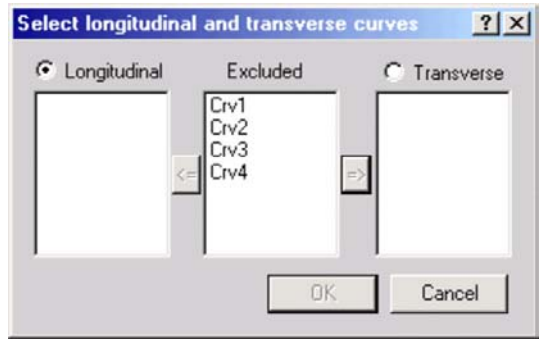


Picture 1-13

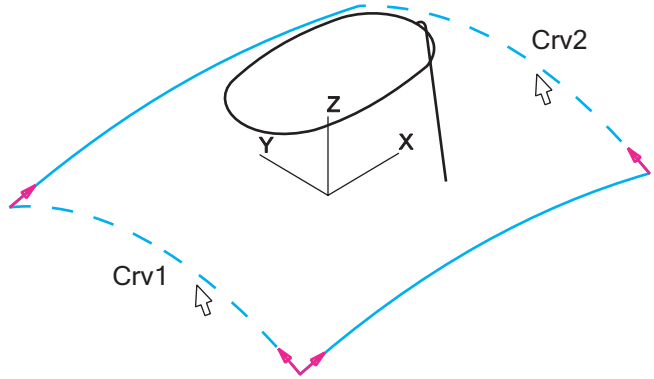
9. Now we use the previously defined curves to create our first surface using the "Coons" surface type. Select "**Coons**" from the "**Surfaces**" menu and press OK to confirm "**Srf1**" as the surface ID.



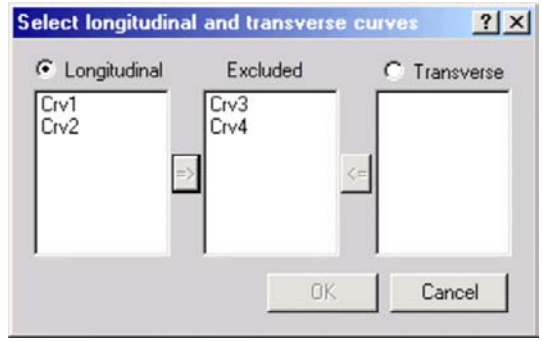
The "**Select longitudinal and transverse curves**" dialog opens with all available curves listed in the "**Excluded**" section.



As the "**Longitudinal**" option button is already activated by default, use the cursor to select the longitudinal surface curves "**Crv1**" and "**Crv2**" from the screen. The dialogs curve list is updated automatically as shown below. The selected curves are highlighted as shown in **Picture 1-14**.

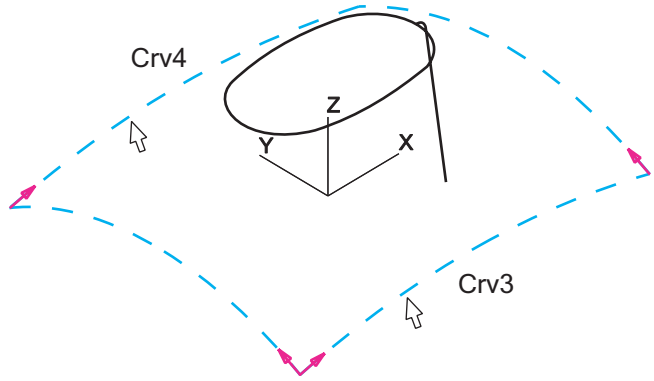


Picture 1-14

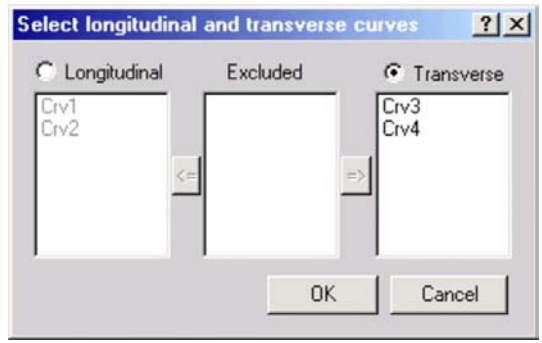


You can also select the curves directly in the dialog by marking them in the "Excluded" list section and then pressing the <= or => buttons on either side to move them to the desired section.

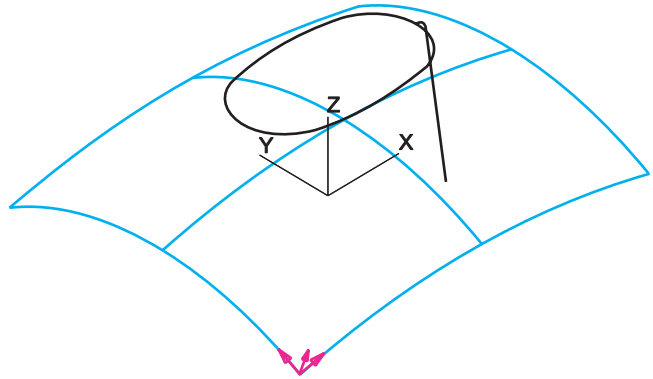
Now activate the "**Transverse**" option button on the surface dialog and select the transversal surface curves "**Crv3**" and "**Crv4**" from the screen as shown in **Picture 1-15**.



Picture 1-15



Press **OK** on the dialog to confirm curve selection and to start the internal surface creation process. The newly created surface is displayed as shown in **Picture 1-16**.



Picture 1-16

It is very important to know that the sequence of curve selection is very important for the surface creation process. If, for example, you've got more than 2 longitudinal curves, these curves should be selected in the same sequence as they are to be connected for the surface.

See "**Surfaces Menu/ Coons Surface**" topic in the online help for more information about "Coons" surfaces.

CREATE THE SECOND SURFACE (TRANSLATION TYPE)

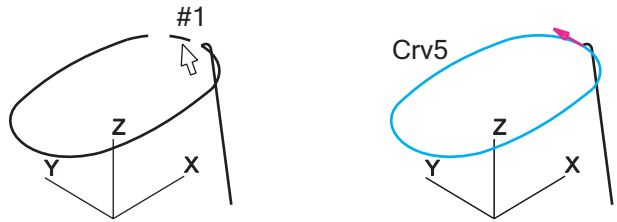
The second surface to be created represents the sidewalls of the protrusion from the main surface. For the "Translation / Normal" surface type we have to create two surface curves, the so-called "Base" curve and a single "Drive" curve.

1. Select the "**New**" command from the "**Curves**" menu to start "Base" curve creation. Press OK to confirm "**Crv5**" as the new ID.


New

2. Select "**Chain**" command and double-click the cursor at **Position #1** as shown in **Picture 1-17**. The arc is selected as the first element of the chain operation and the software automatically completes the curve.


Chain



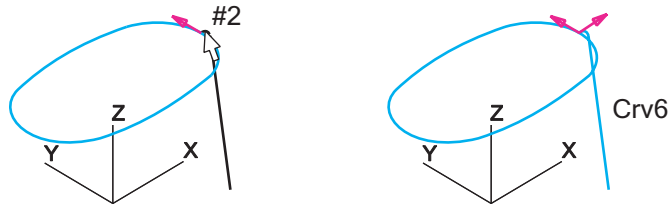
Picture 1-17

3. For the "Drive" curve select the "**New**" command again and confirm "**Crv6**" as the new ID.


New

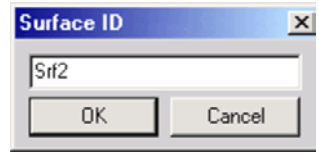
4. Select "**Chain**" command and double-click at begin of the small arc as indicated by **Position #2** shown in **Picture 1-18**. The software automatically completes the curve.

Chain

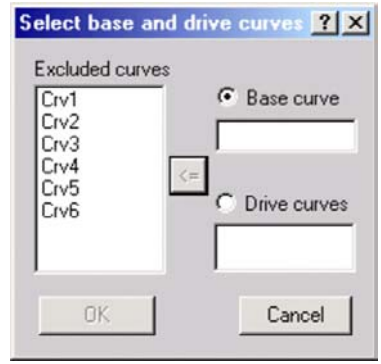


Picture 1-18

5. To create the surface, select "**Transformation / Normal**" from the "**Surfaces**" menu and confirm "**Srf2**" as the new surface ID.

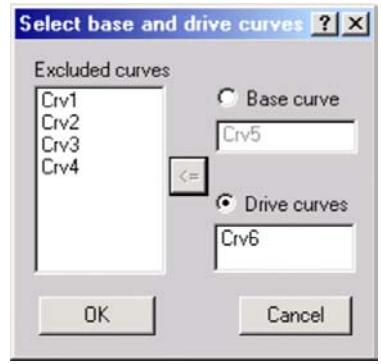


Now the "**Select base and drive curves**" dialog opens on the screen. All available curves are listed in the "**Excluded curves**" section. The "**Base curve**" option button is already activated by default.

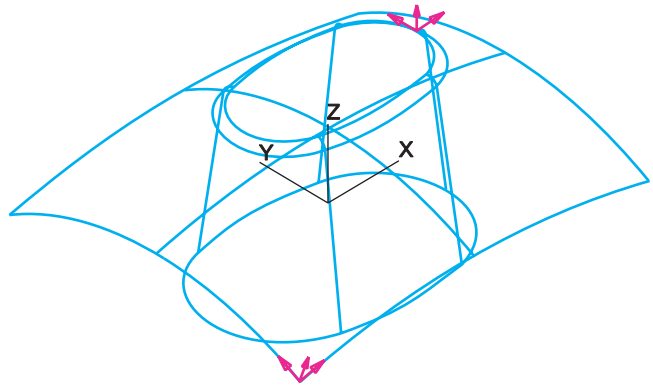


Use the cursor to select "**Crv5**" as base curve. The curve is highlighted as shown in **Picture 1-19** and the "**Drive Curves**" option button is automatically activated on the dialog. Continue with selecting "**Crv6**" as the drive curve.

Picture 1-19



Press **OK** on the dialog to confirm curve selection and to start the surface creation process. The newly created surface is displayed as shown in **Picture 1-20**.



Picture 1-20

CREATE THE THIRD SURFACE (RULED TYPE)

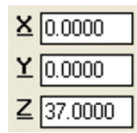
This step creates the flat surface that fills the top of the previously created surface. For this we will use the already existing curve "Crv5" that represents the outside profile and a second, single point curve, which has to be created yet. This new curve is located at the center of the new surface and defined by a single point only.

1. Select the "**New**" command from the "**Curves**" menu and confirm "**Crv7**" as the curve ID.

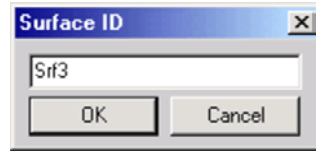

New

2. Select the "**Linear**" command from the "**Curves**" menu. Type "**0**" in the "**X**" field, "**0**" in the "**Y**" field and "**37**" in the "**Z**" field of the **Value Entry Box**. Then press the ENTER key to confirm these coordinates as the first and only point of the new curve. A small triangle representing the defined point location is then displayed on the screen.

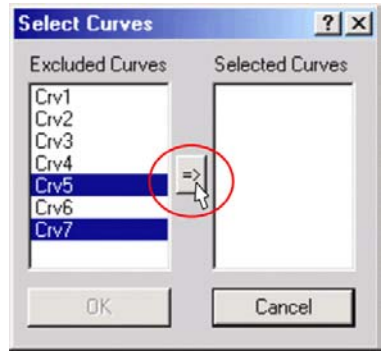

Linear

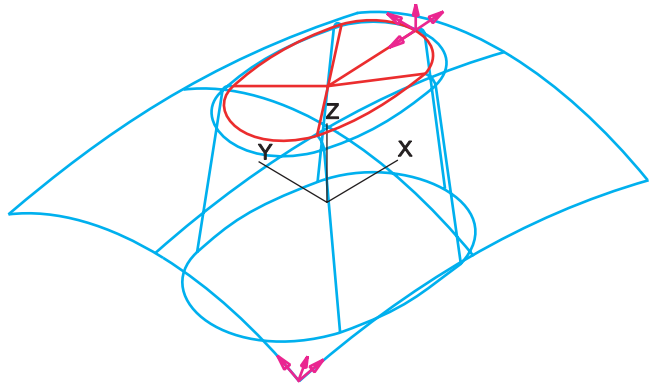


3. To create the surface, select "**Lofted / Ruled**" from the "**Surfaces**" menu and confirm "**Srf3**" as the surface ID.



When the "**Select Curves**" dialog opens, hold the "**Ctrl**" key while using the cursor to select "**Crv5**" and "**Crv7**" as shown below. Then click the small arrow button in the center of the dialog to move these two curves to the "Selected Curves" list. Press OK to confirm selection and start surface creation process. The resulting surface is displayed in **Picture 1-21** (in red for better visibility).





Picture 1-21

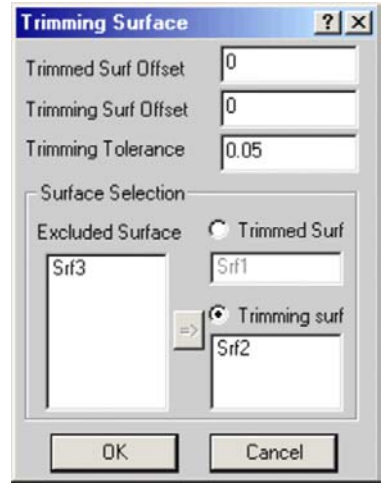
SURFACE TRIMMING

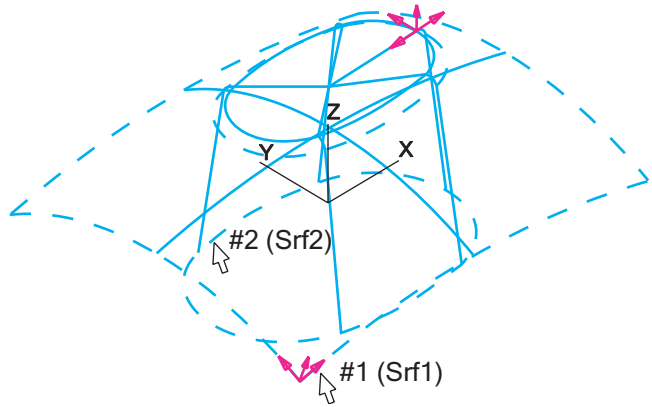
Before we continue with surface creation, the two surfaces "**Srf1**" and "**Srf2**" have to be trimmed at their intersection. This is accomplished in two steps; first "Srf1" is trimmed onto "Surf2" and second "Srf2" onto "Srf1".

1. **Step1**: Trim "**Srf1**" onto "**Srf2**".
   Select the "**Trim Surface**" command from the "**Surfaces / Trimmed Surface**" menu. The "Trimming Surface" dialog opens with the "Trimmed Surf" option button activated by default.

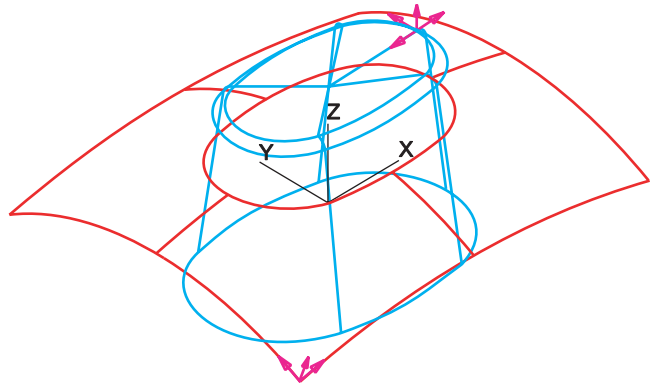
Trim Surface

Ensure that all "Offset" settings are "**0**" and the "Trimming Tolerance" is set to "**0.05**" mm.
Use the cursor to select "**Srf1**" as the "Trimmed Surf" (see position #1 in **Picture 1-22**). Then select "Srf2" as the "Trimming Surface" (see position #2). The selected surfaces are highlighted and the dialog is updated automatically. Press **OK** to start the trimming process. See **Picture 1-23** for the result of the first trim operation.





Picture 1-22

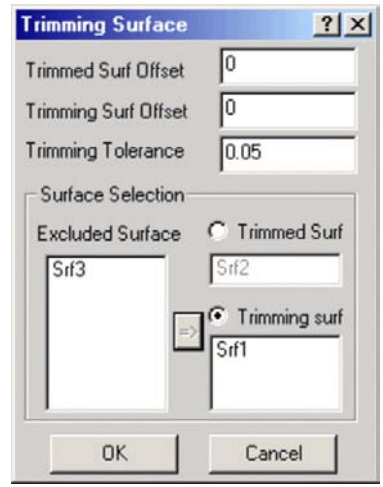


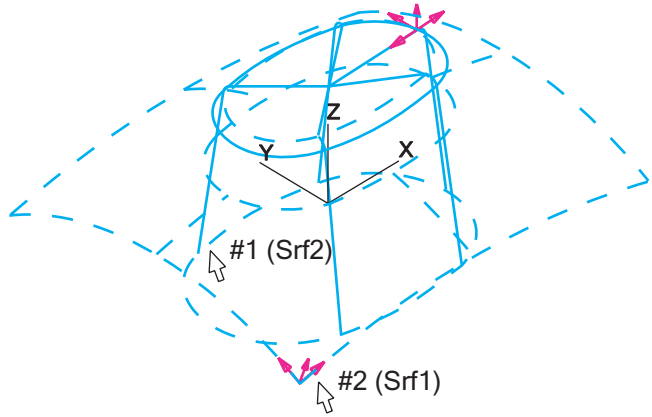
Picture 1-23

2. **Step2**: Trim "**Srf2**" onto "**Srf1**".
   Select the "**Trim Surface**" command again.

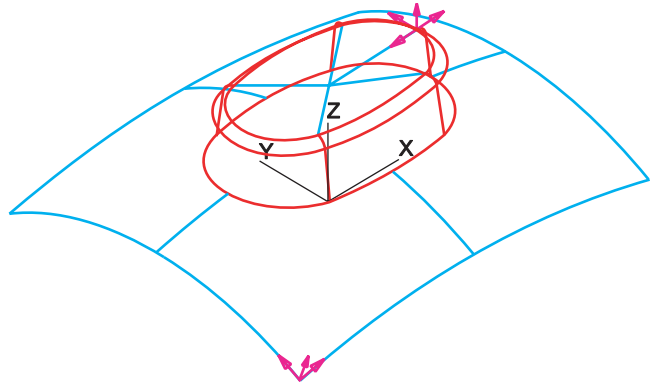

Trim Surface

Use the cursor to select "**Srf2**" as the "Trimmed Surf" (see position #1 in **Picture 1-24**). Then select "Srf1" as the "Trimming Surface" (see position #2). Press **OK** to start the trimming process. See **Picture 1-25** for the result of the second trim operation.





Picture 1-24



Picture 1-25

CREATE THE FOURTH SURFACE (RULED TYPE)

The machining cycle used for roughing requires the parts body being completely enclosed. Therefore we now create two more curves to create one more ruled surface. At this point we first hide (blank) all existing curves and surfaces for better visualization of the following steps.

1. Select the "**Blank**" command from the "**Edit**" menu. Check that the "**Verify**" mode button is pressed (set to "On").


Blank


Verify Mode

2. As we want ho hide all existing curve and surface entities, the easiest way is to activate ("check") the "**Curves**" and "**Surfaces**" discrimination options in the "**Edit / Discrimination**" menu. Alternatively you can select the corresponding icons on the left side of the screen (default location).


Discrim. Curves


Discrim. Surfaces

3. Now select the "**Select All**" command from the "**Edit**" menu. This selects all entities according to the currently active discrimination modes (curves & surfaces). Press **ENTER** to confirm selection and "**Redraw**" to clear the screen. Now you should only see the plain geometry that we imported at the very begin of the tutorial.


Select All


Enter

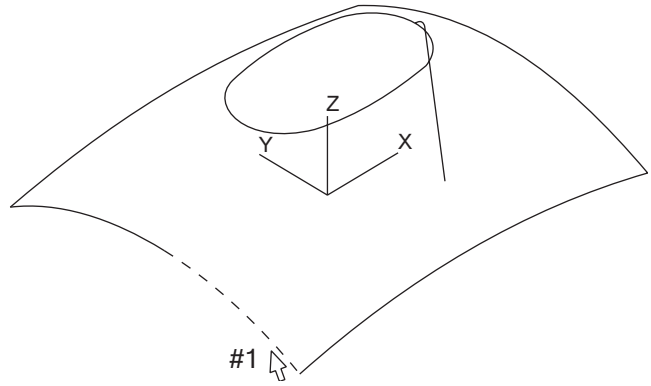

Redraw

4. To create the new surface we need two more curves. Select the "**New**" command from the "**Curves**" menu and confirm "**Crv8**" as the new ID.

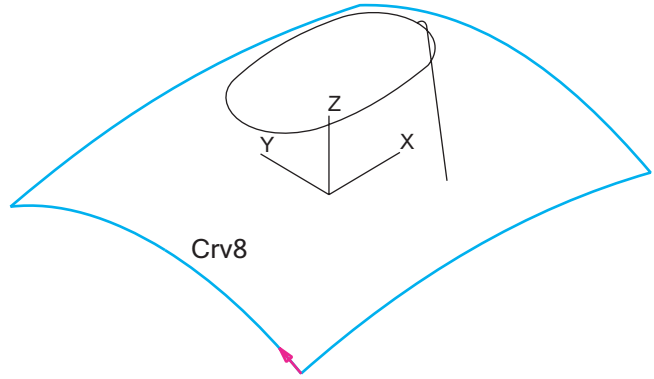

New

5. Select "**Chain**" command and double-click the cursor at **Position #1** shown in **Picture 1-26**. The system automatically completes the curve as shown **Picture 1-27**.

Chain



Picture 1-26



Picture 1-27

6. The second curve is simply a rectangle located at **Z0** level, representing the outside boundary of the new surface. Select the "**New**" command from the "**Curves**" menu and confirm "**Crv9**" as the new ID.


New

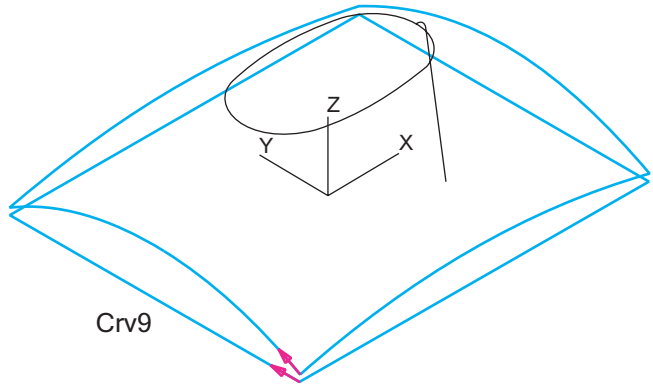
7. We will use the "**Linear**" link type command from the "**Curves**" menu to connect the corner locations of the rectangle.


Linear

For the first corner type "**-60**" in the "**X**" field of the **Value Entry Box**. Use the **TAB** button to switch to the "**Y**" field and type "**-50**". Go to the "**Z**" field and type "**0**". Then press **ENTER** to verify the first corner location. Continue to define the other corner locations using the coordinates for **P2-P5** as provided in the table below.

	X	Y	Z
P2	-60	50	0
P3	60	50	0
P4	60	-50	0
P5	-60	-50	0

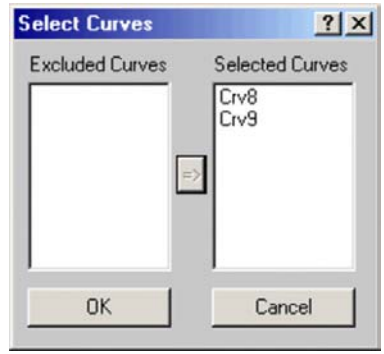
The resulting curve is displayed in **Picture 1-28**.

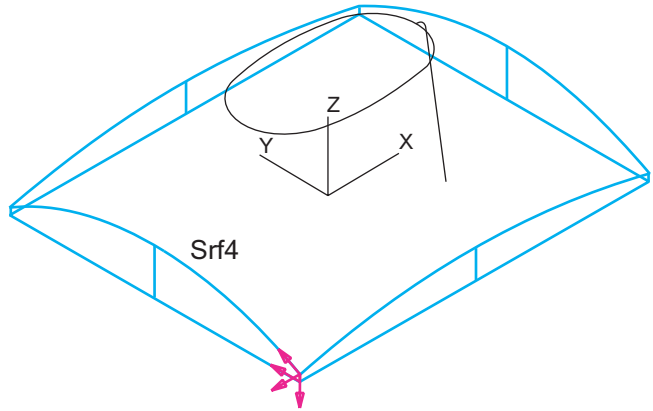


Picture 1-28

8. To create the surface, select "**Lofted / Ruled**" from the "**Surfaces**" menu and confirm "**Srf4**" as the surface ID.

When the "**Select Curves**" dialog opens, hold the "**Ctrl**" key while using the cursor to select "**Crv8**" and "**Crv9**". Then click the small arrow button in the center of the dialog to move these two curves to the "Selected Curves" list. Press OK to confirm selection and start surface creation process. The resulting surface is displayed in **Picture 1-29**.





Picture 1-29

CREATE THE BOUNDARY CURVE

As the roughing cycle needs to have a two-dimensional border, this step will create a curve that represents the rectangular machining boundary. As we will use a 14mm DIA bull nose end mill for roughing we define the boundary with enough space around the part (15mm) in order to allow the tool to rough the vertical surfaces up to the final Z depth.

1. Select the "**New**" command from the "**Curves**" menu and type "**Boundary**" as the new ID.


New

2. We will use the "**Linear**" link type command from the "**Curves**" menu to connect the boundary corners.

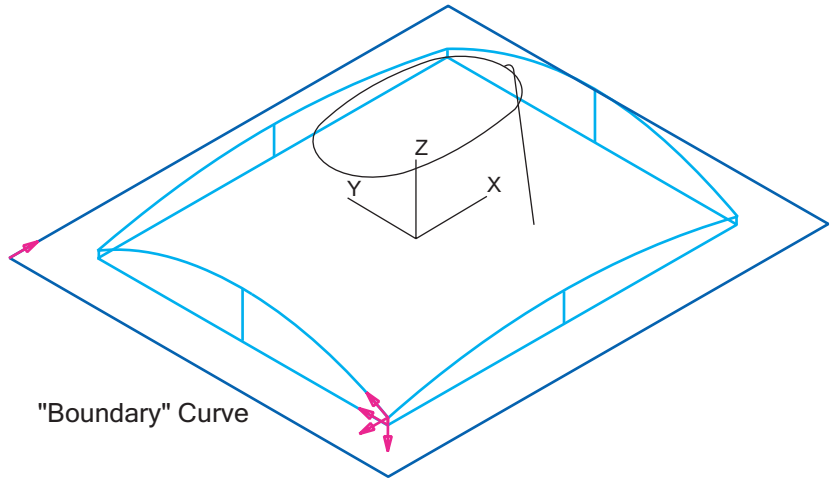

Linear

For the first corner type "**-75**" in the "**X**" field, "**65**" in the "**Y**" field and "**0**" in the "**Z**" field of the **Value Entry Box**. Then press **ENTER** to verify the first boundary corner.

Continue to define the other corner locations using the coordinates for **P2-P5** as provided in the table below.

	X	Y	Z
P2	75	65	0
P3	75	-65	0
P4	-75	-65	0
P5	-75	65	0

The resulting curve is displayed in **Picture 1-30**.



Picture 1-30

ARRANGE SCREEN ENTITIES FOR MACHINING

Before continuing we will hide ("blank") all entities except the boundary curve and the surfaces that will be machined. The easiest way to accomplish this is first to hide the complete screen content and then show ("unblank") the desired entities only.

1. Select the "**Blank**" command from the "**Edit**" menu and check that the "**Verify**" mode button is pressed (set to "On"). Now select the "**Select All**" command and press **ENTER** to confirm selection. Use "**Redraw**" to clear the screen that should be completely empty afterwards.


Blank


Verify Mode

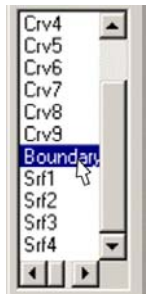

Select All


Enter

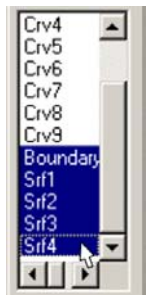

Redraw

2. Now we want to make the boundary curve and surfaces visible again. Upon selecting the "**Unblank**" command, all previously "hidden" curves and surfaces are displayed in the "Selection List Box" on the right side of the screen.


Unblank

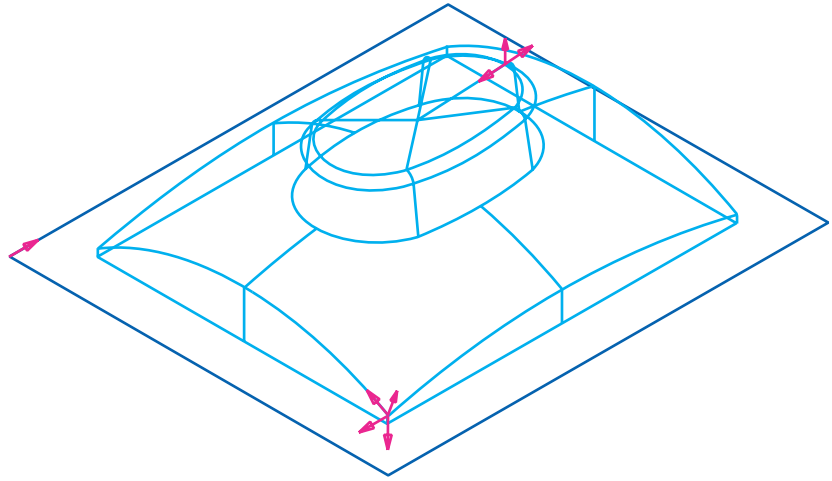


Use the cursor to select the "**Boundary**" curve entry.



Now hold the "**SHIFT**"-button pressed while selecting the last entry "**Srf4**". All entries between the two selected items will be highlighted. Press **ENTER** to confirm selection.

Now your screen display should look as shown in **Picture 1-31**.

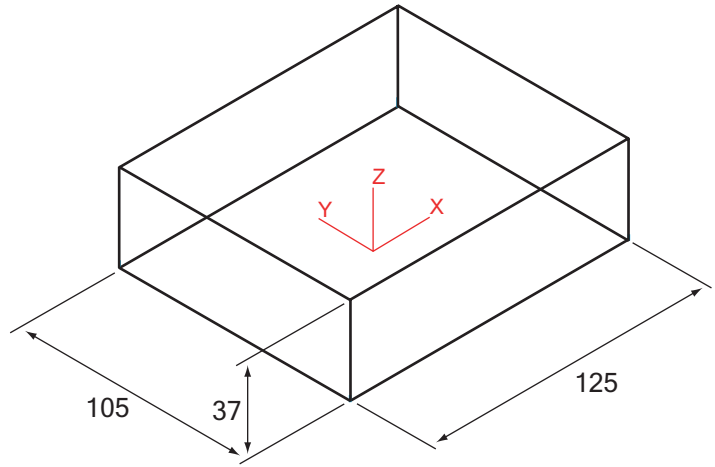


Picture 1-31

CREATING THE PART PROGRAM

Now as the curves and surfaces for the sample part are created we continue with the definition of the Work Steps that are necessary to machine the part. Each Work Step uses a specific machining cycle (Contour, Constant-Z, etc.) with its associated tool settings and machining parameters to machine the assigned curves and/or surfaces. Once all necessary Work Steps have been defined the complete part program can be visually checked using the 3D solid simulation. If everything is ok, the CNC-Code is created after selecting the postprocessor for the desired machine.

We assume that the parts stock size is 125x105x38. The work zero point is located at the bottom of the stock center. See **Picture 1-32**.



Picture 1-32

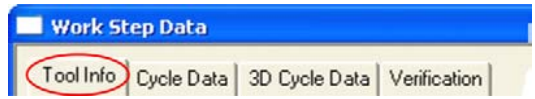
The part program will consist of these 4 Work Steps:

1. Surface roughing using the “Constant-Z” cycle.
2. Roughing the vertical surfaces on the outside of the part using the standard 2D contouring cycle.
3. Finishing the vertical surfaces (2D contouring cycle).
4. Surface finishing using the “Zig-Zag” cycle.

CREATING WORK STEP #1 (SURFACE ROUGHING)

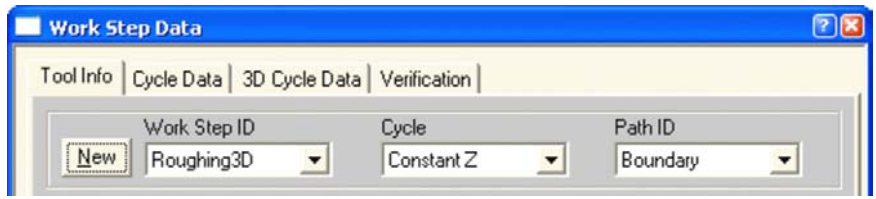
Now we create the first work step that will do the roughing of the part's top surfaces. Because of the constant cutting depth, the "Constant-Z" cycle is the first choice for roughing. This cycle can create a pocketing type of tool path, removing the material from outside to inside. We will use a 16mm DIA bull nose mill with a 4mm corner radius for machining, stepping down in increments of 2mm.

1. Select the "**Data**" command in the "**Machining**" menu to open the "Work Step Data" dialog and go to the "**Tool Info**" tab.

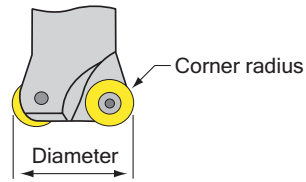


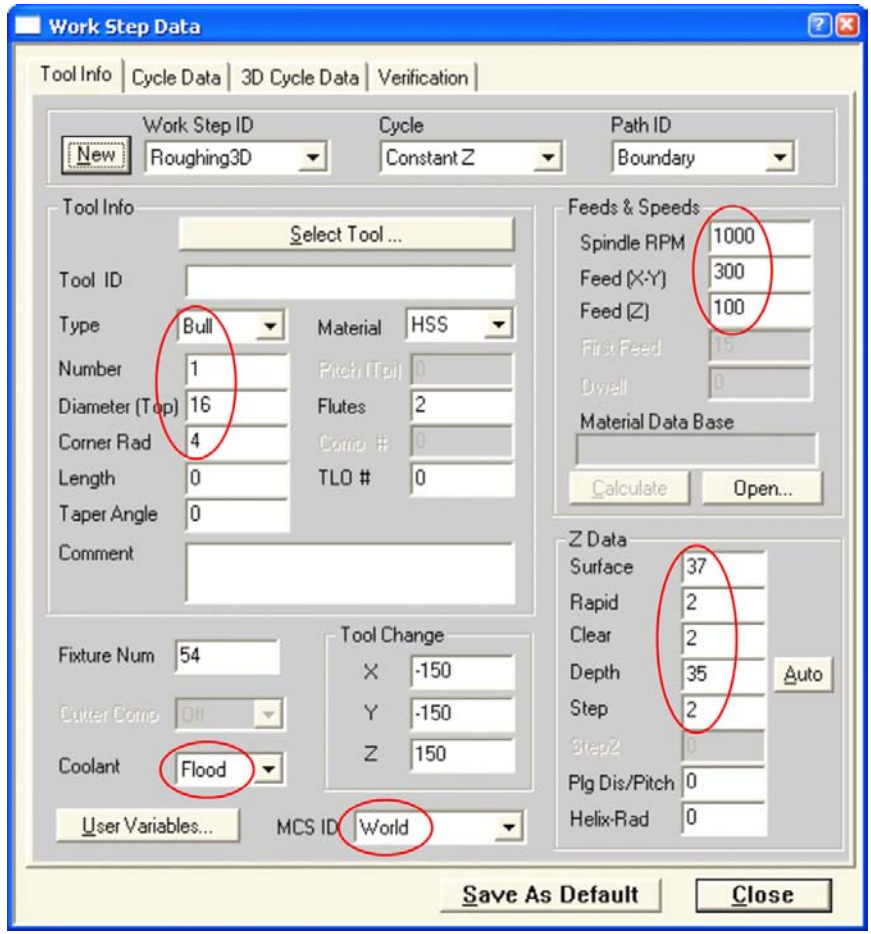
2. Press the "**New**" button and input "**Roughing3D**" as the new Work Step ID and confirm with **OK**.

3. Select "**ConstantZ**" from the cycle list and the "**Boundary**" curve from the Path ID list.



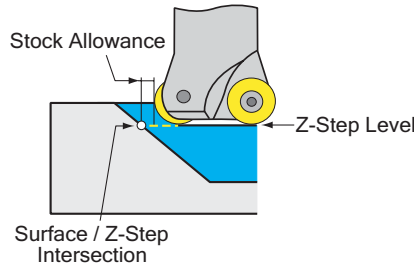
4. Next step is to input tool and technology settings as provided by the following table. See also **Picture 1-33**.

Dialog Field	Value	Comment
Type	BULL	
Number	1	Number of tool in tool magazine
Diameter (Top)	16	Full outside diameter of tool
Corner Rad.	4	Insert corner radius
Coolant	Flood	Activate coolant
Spindle RPM	1000	Sets spindle RPM to 1000 / minute
Feed (XY)	300	Cutting feed rate in XY plane (mm/minute)
Feed (Z)	100	Cutting feed rate for Z depth moves
Z-Surface	37	Defines absolute Z-axis position where calculation of the Z steps will start. It is important to know that if the system does not find any surface intersections at the first Z-Step level, tool path calculation is stopped.
Z-Rapid	2	Incremental rapid positioning distance.
Z-Clear	2	Incremental distance above Z-Surface. The system changes from rapid to feed rate moves
Z-Depth	35	Total incremental depth. Although part height is 37 mm, we want to stop 2mm above the lowest part to avoid the vertical sidewalls to be machined.
Z-Step	2	Incremental step for “ConstantZ” depth moves.


Work Step Data
Tool Info
Cycle Data
3D Cycle Data
Verification
New
Work Step ID
Roughing3D
Cycle
Constant Z
Path ID
Boundary
Tool Info
Select Tool ...
Tool ID
Type
Bull
Material
HSS
Number
1
Pitch (Tpi)
0
Diameter (Top)
16
Flutes
2
Corner Rad
4
Comp #
0
Length
0
TLO #
0
Taper Angle
0
Comment
Feeds & Speeds
Spindle RPM
1000
Feed (X-Y)
300
Feed (Z)
100
First Feed
15
Dwell
0
Material Data Base
Calculate
Open...
Z Data
Surface
37
Rapid
2
Clear
2
Depth
35
Auto
Step
2
Step2
0
Plg Dis/Pitch
0
Helix-Rad
0
Fixture Num
54
Tool Change
X
-150
Y
-150
Z
150
Cutter Comp
Off
Coolant
Flood
User Variables...
MCS ID
World
Save As Default
Close

Picture 1-33

5. Switch to the "**Cycle Data**" tab and check that the values and settings as listed in the table below are set correctly.

Dialog Field	Value	Comment
Stock Allowance	1	2D finish allowance that is applied to the boundary profile that is created at every Z-Step level. 
Step Over	4	XY plane step over.
Outside-In Milling	☑	The tool path is moving from the outside boundary to the pocket center. Only used when "Operation" option on "3D Cycle Data" tab set to "Pocket".

6. Continue to the "**3D Cycle Data**" tab to input the values as shown in the table below. Finally close the Work Step Data dialog using the "**Close**" button.

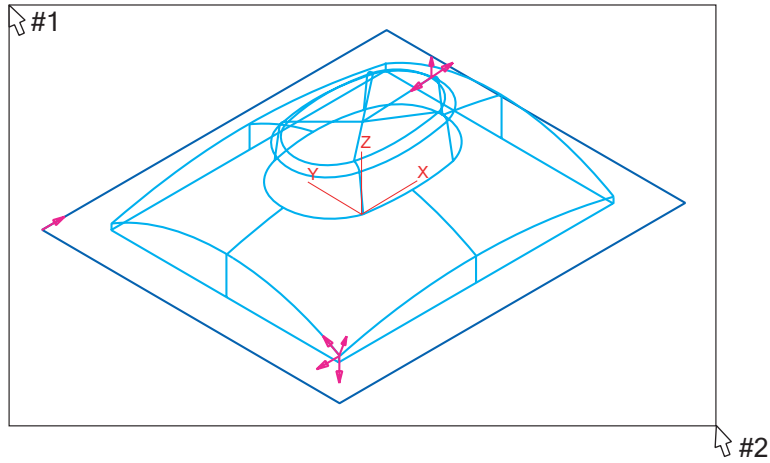
Dialog Field	Value	Comment
Fin. Allowance (Work Step Surfaces)	0	No surface finish allowance since the "ConstantZ" cycle only supports 2D boundary offsetting. (See "CycleData / Stock Allowance")
Operation	Pocketing	Create boundary-parallel tool moves.
Chordal Tolerance	0.2	This parameter controls the maximum deviation of the actual tool path from the surface profile.

7. Now we define the surfaces that will be machined by the current Work Step. Select the "**Select Surfaces**" command from the "**Machining**" menu and check that the "Verify" mode button is pressed (set to "On").


Select Surfaces


Verify Mode

Use the cursor to drag a frame around the desired surfaces on the screen as shown in **Picture 1-34**. All surfaces that are completely encircled by this frame are selected for machining. This process can be repeated as multiple times.



Picture 1-34

At the same time the selected surfaces are highlighted in the list on the right side of the screen. Press ENTER to confirm the surface selection.

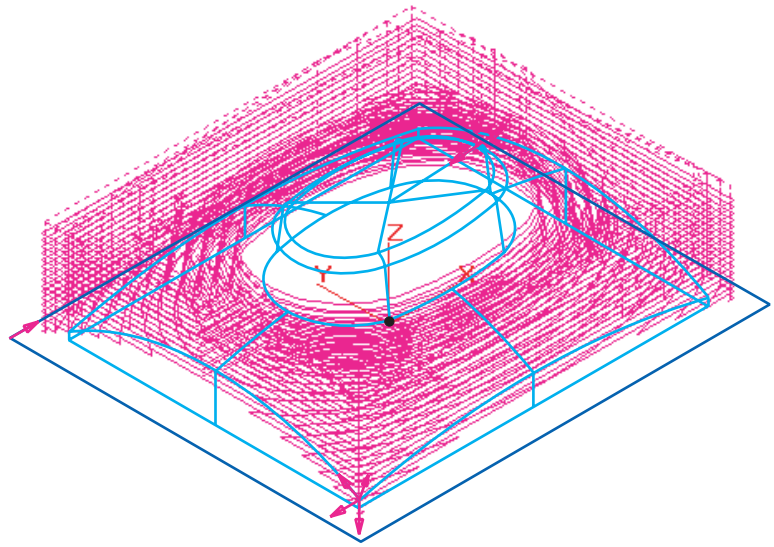
Surface list


Enter

For more information about selecting the machining surfaces see the corresponding "Select Surfaces" topic in the online help.

8. To ensure that the first Work Step was created correctly click the "**Verif**y" button. The system calculates the cutter path as shown in **Picture 1-35**. Depending on the currently used hardware it may take some seconds until the tool path will be drawn on the screen.


Verify



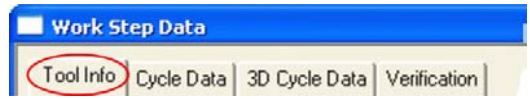
Picture 1-35

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

CREATING WORK STEP #2 (2D CONTOUR ROUGHING)

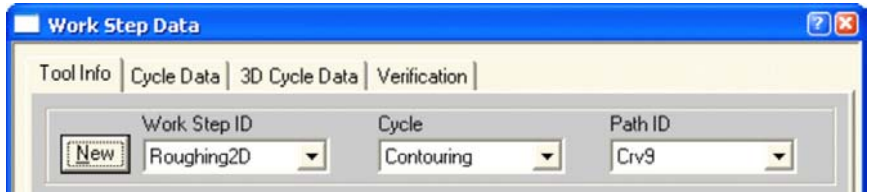
The second Work Step is defined for roughing the rectangular outside of the part using the standard 2D contouring cycle. We will use a 20mm DIA end mill to create a single pass leaving 0.3mm finish allowance along the profile. For this example we will not use any cutter compensation but simply let the software compute the centerline tool path.

1. Select the "**Data**" command in the "**Machining**" menu to open the "Work Step Data" dialog and go to the "**Tool Info**" tab.



2. Press the "**New**" button and input "**Roughing2D**" as the new Work Step ID and confirm with **OK**.

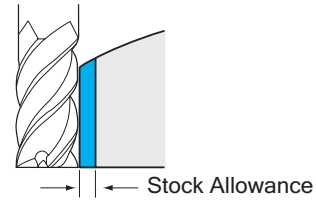
3. Select "**Contouring**" from the cycle list and the "**Crv9**" curve from the Path ID list.



4. Next step is to input tool and technology settings to the appropriate fields as shown in the table below.

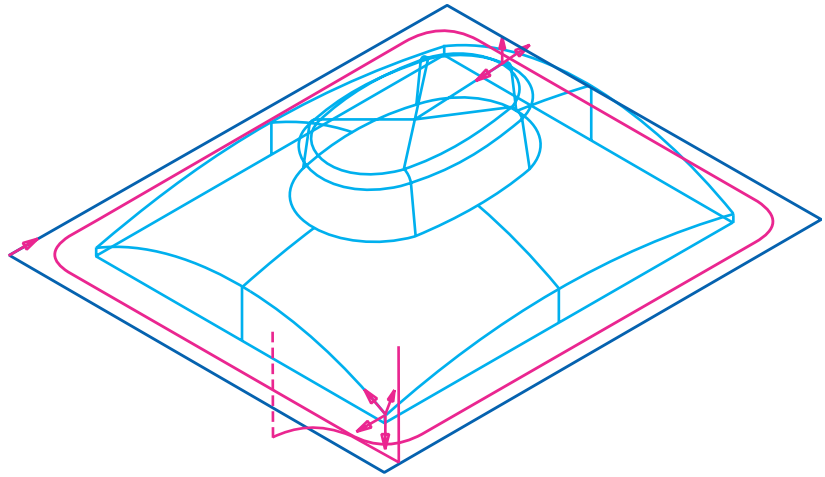
Dialog Field	Value	Comment
Type	FLAT	
Number	2	Number of tool in tool magazine.
Diameter (Top)	20	Full outside diameter of tool.
Cutter Comp	OFF	No cutter compensation (G41 or G42).
Coolant	Flood	Activate coolant.
Spindle RPM	800	Sets spindle RPM to 800 / minute.
Feed (XY)	250	Cutting feed rate in XY plane (mm/minute).
Feed (Z)	200	Not important for cutting because only used for move from Z-Clear to final depth.
Z-Surface	0	Defines absolute Z-axis reference plane at the bottom of the part.
Z-Rapid	39	Incremental rapid positioning distance above Z-Surface (part height 37mm + 2mm rapid clearance).
Z-Clear	2	Incremental distance above Z-Surface. The system changes from rapid to feed rate moves.
Z-Depth	0	No depth is assigned here since Z-Surface represents the final depth and therefore no depth move required.
Z-Step	0	No Z-axis steps

5. Switch to the "**Cycle Data**" tab and check that the values and settings as listed in the table below are set correctly. Then close the Work Step Data dialog using the "**Close**" button.

Dialog Field	Value	Comment
Offset Direction	LEFT	Sets offset direction to left in respect to curve direction.
Stock Allowance	0.3	2D finish allowance that is used to offset the tool path from machined curve. 
Ramp Status	OUT	Activates "Ramp-Out" move only.
Out Radius	15	Sets ramp-out radius.
Out Angle	90	Sets sweep angle of ramp-out move.
Lead Status	In	The tool path is moving from the outside boundary to the pocket center. Only used when "Operation" option on "3D Cycle Data" tab set to "Pocket".
In Length	15	Sets length of linear retract move.
In Angle	0	Sets angle of linear retract move.

6. Click the "**Verify**" button to calculate and display the cutter path as shown in **Picture 1-36**.


Verify



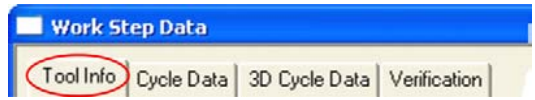
Picture 1-36

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

CREATING WORK STEP #3 (2D CONTOUR FINISHING)

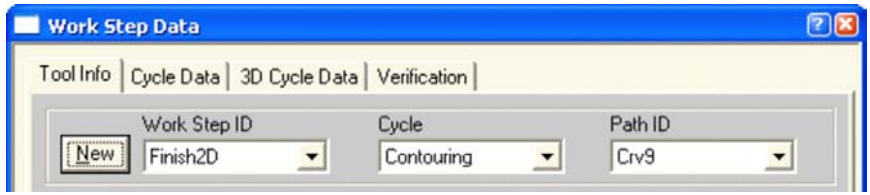
The third Work Step is defined for 2D finishing the outside of the part. This time we will use a 14mm DIA end mill to remove the 0.3mm finish allowance that remained from the previous Work Step.

1. Select the "**Data**" command in the "**Machining**" menu to open the "Work Step Data" dialog and go to the "**Tool Info**" tab.



2. Press the "**New**" button and input "**Finish2D**" as the new Work Step ID and confirm with **OK**.

3. Select "**Contouring**" from the cycle list and the "**Crv9**" curve from the Path ID list.



4. Next step is to input tool and technology settings to the appropriate fields as shown in the table below (only settings that are different from the previous Work Step are displayed).

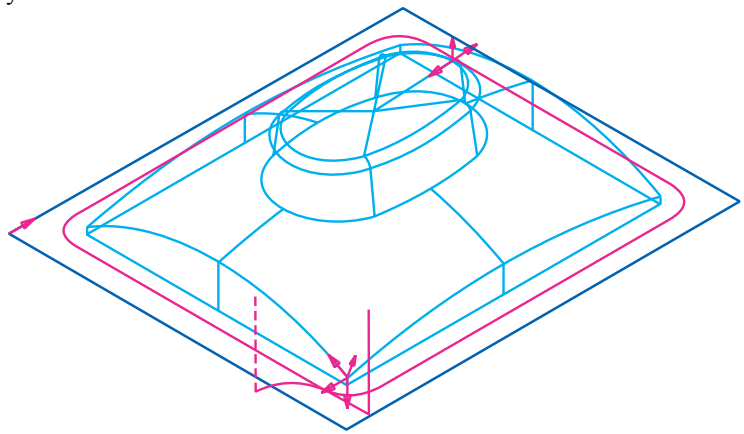
Dialog Field	Value	Comment
Number	3	Number of tool in tool magazine
Diameter (Top)	14	Full outside diameter of tool
Spindle RPM	1100	Sets spindle RPM to 1100 / minute
Feed (XY)	350	Cutting feed rate in XY plane (mm/minute)

5. Switch to the "**Cycle Data**" tab and check that the values and settings as listed in the table below are set correctly. Close the Work Step Data dialog using the "**Close**" button.

Dialog Field	Value	Comment
Stock Allowance	0.0	No stock allowance for finishing

6. Click the "**Verify**" button to calculate and display the cutter path as shown in **Picture 1-37**.

Verify



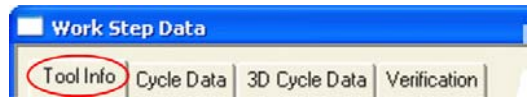
Picture 1-37

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

CREATING WORK STEP #4 (SURFACE FINISHING)

Last Work Step is defined to finish the top-level surfaces of the part. We use the "Zig-Zag" cycle in combination with the "Variable Step Over" option to ensure good surface quality along the steep sidewalls of the protruding area of the parts center section.

1. Select the "**Data**" command in the "**Machining**" menu to open the "Work Step Data" dialog and go to the "**Tool Info**" tab.



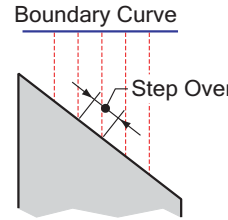
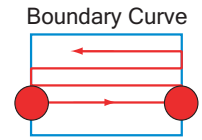
2. Press the "**New**" button and input "**Finish3D**" as the new Work Step ID and confirm with **OK**.

3. Select "**Zig Zag**" from the cycle list and the "**Crv9**" curve from the Path ID list as the boundary profile for this Work Step.

4. Next step is to input tool and technology settings to the appropriate fields as shown in the table below.

Dialog Field	Value	Comment
Type	BALL	 Diameter
Number	4	Number of tool in tool magazine
Diameter (Top)	12	Full outside diameter of tool
Spindle RPM	1400	Sets spindle RPM to 1400 / minute
Feed (XY)	500	Cutting feed rate in XY plane (mm/minute)
Feed (Z)	300	Cutting feed rate for Z depth moves
Z-Surface	37.5	Define the absolute Z-axis position high enough to ensure that all machined surface areas are lying well below the Z level specified here.
Z-Rapid	2	Incremental rapid positioning distance.
Z-Clear	2	Second incremental rapid positioning distance. Switch from rapid to feed rate.
Z-Depth	0	No depth for this cycle
Z-Step	0	No step for this Work Step

5. Switch to the "**Cycle Data**" tab and check that the values and settings as listed in the table below are set correctly

Dialog Field	Value	Comment
Stock Allowance	0	No 2D finish allowance.
Step Over	0.5	XY plane step over.
Cut Angle	0	Define angle that cutting direction is parallel to X-axis.
Variable Step Over	☑	Forces the software to calculate the step over directly along the machined surfaces instead of projecting it from the XY plane. 
Finish Path	☐	No final finish pass along the boundary curve.
Face Milling	☑	The "Zig-Zag" cycle's tool path extends outside the boundary curve by the diameter of the tool. 

6. Continue to the "**3D Cycle Data**" tab to input the values as listed in the table below. Finally close the Work Step Data dialog using the "**Close**" button.

Dialog Field	Value	Comment
Fin. Allowance (Work Step Surfaces)	0	No surface finish allowance.
Chordal Tolerance	0.2	This parameter controls the maximum deviation of the actual tool path from the surface profile.
Curve Step	0.5	When projecting tool paths onto a surface, this parameter specifies the distance between these points. The tool makes linear moves through each of these projected points along the surface. A smaller Curve Step gives more tool path points and a better finish; however, more points also gives very large files and longer processing times

7. Now we define the surfaces that will be machined by the current Work Step. Select the "**Select Surfaces**" command from the "**Machining**" menu and check that the "**Verify**" mode button is pressed (set to "On").


Select Surfaces

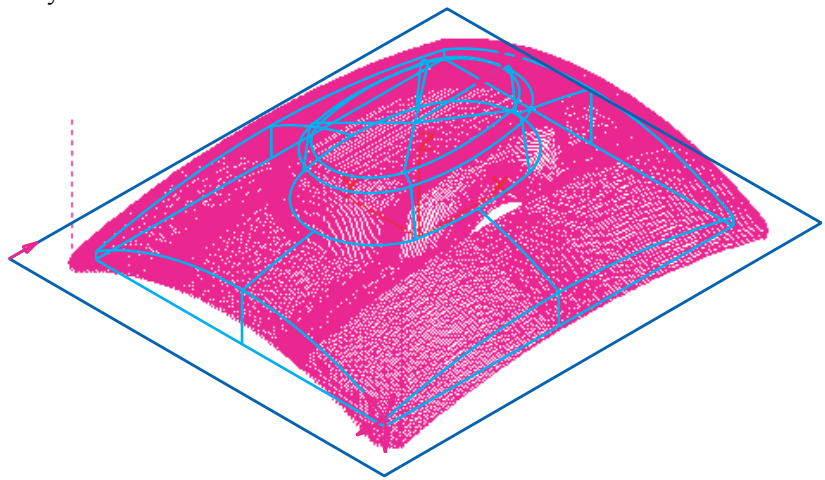

Verify Mode

In Work Step #1 we explained how to select surfaces by dragging a frame on the screen. Now we will use the selection list box on the right side of the screen. Select the first surface in the list using the cursor. Then select the last surface in the list while holding the SHIFT-button. Now all surfaces should be highlighted. Press ENTER to confirm the surface selection.


Enter

7. Click the "**Verify**" button to calculate and display the cutter path as shown in **Picture 1-38**.

Verify



Picture 1-38

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

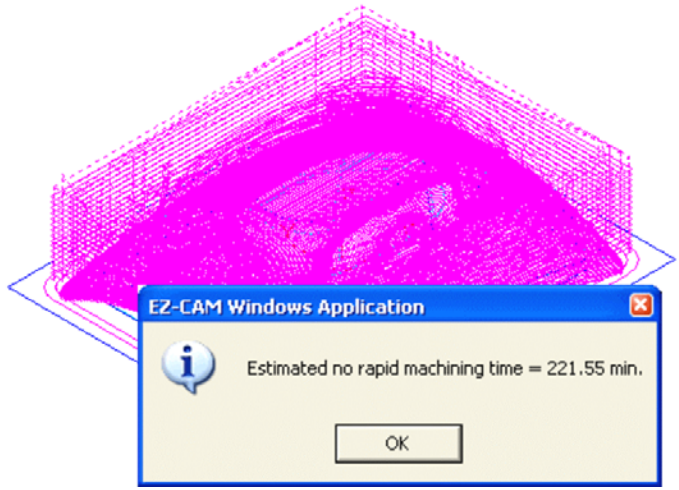
ESTIMATING TOTAL MACHINING TIME

The "**Verify All**" command in the "**Post**" menu is used to estimate the total machining time. 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.

To get the same view as shown in **Picture 1-39**, switch to isometric using the "**View Isometric**" command, then select "**Verify All**" to start tool path calculation. To close the dialog click OK.


View Isometric


Verify

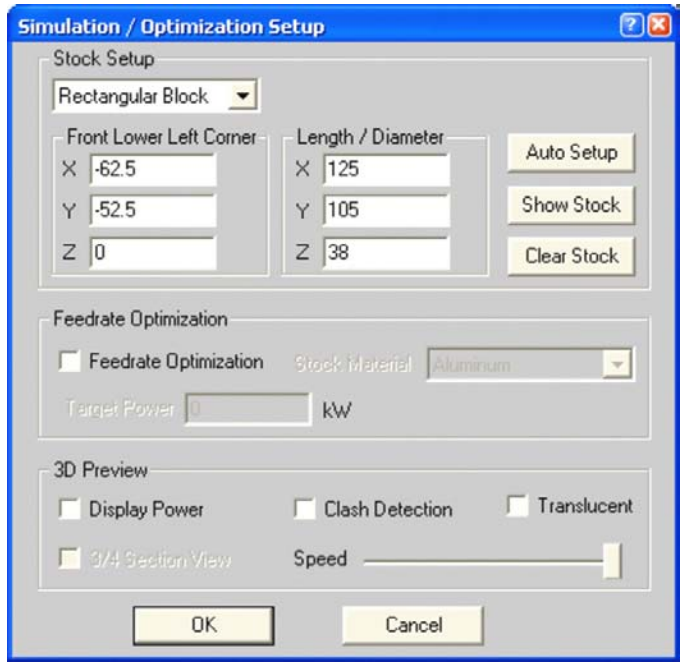


Picture 1-39

3D SOLID 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. 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 Setup" has been defined when the "Preview 3D" command is called, 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 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 1-40**. Close the dialog with **OK**.

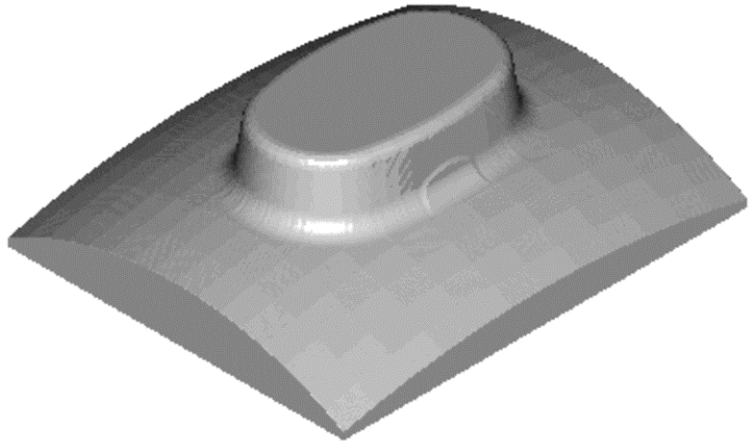


Picture 1-40

2. Select the "**Isometric View**" command and then start the simulation using "**Preview 3D**" command from the "**Machining**" menu or the corresponding button. See **Picture 1-41**.


View Isometric


Preview 3D



Picture 1-41

3. Once the simulation is finished you can change the on-screen view by using the dynamic view commands (Rotate, Pan, Zoom) from the "View / Dynamic Viewing" menu. Use the "**Redraw**" command to return to the standard screen display.


Dynamic Rotate


Dynamic Zoom


Dynamic Pan


Redraw

Use the "**Rapid Cut**" command from the "**Machining**" menu if you want to see the preview result instead of watching the tool moves. The solid representation of the part is then calculated in memory and displayed on the screen.

SAVING THE PART

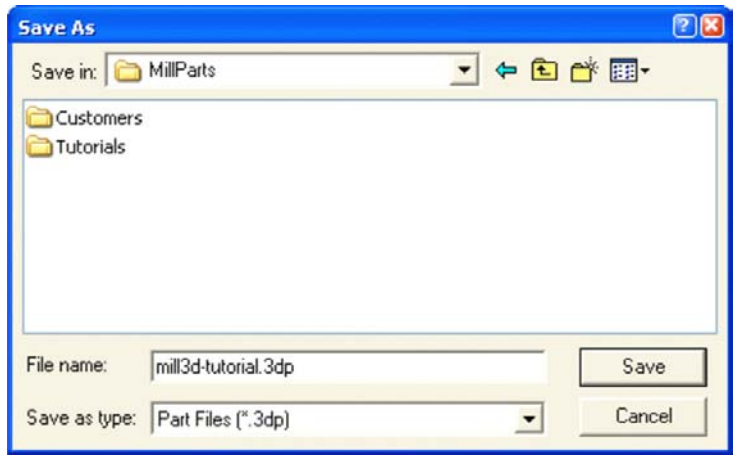
It is very important to save the newly created or edited part from memory to disk periodically during a session as well as at the end to ensure that no information is lost. The EZ-CAM “Save” and “Save as” commands under the File menu transfer files from system memory to a hard disk or other media. In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

File Type : **GEOMETRY**
Extension : **GEO**
Data : Geometry Elements (lines, arcs, etc.), Curves, 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.



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 "**mill3d-Tutorial**" in the "**File Name**" box and click the "**Save**" button. The file extension is added automatically.

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 first saving the part file, the software automatically inserts corresponding entries for geometry and surface files using the same name into the "Save" menu. The "Save All" option from the "File" menu or the corresponding toolbar button can then be used to save all files at the same time.

Save All

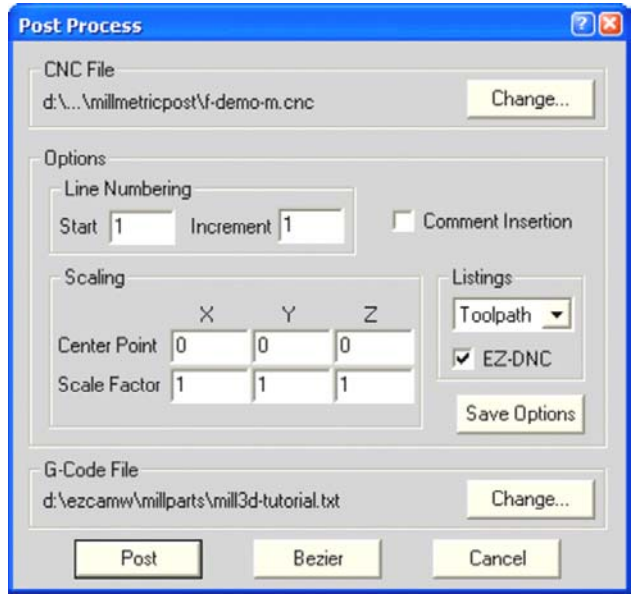
<u>This command overwrites any existing files without any screen prompt !!</u>

You can use this command anytime for fast saving of your work.

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. This will open the "**Post Process**" dialog.



Picture 1-42

2. First a postprocessor has to be loaded. 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 postprocessor.

   Standard postprocessor directories created by the EZ-CAM setup:

INCH	<DRIVE>: \ EZCAMW \ EZCAM13 \ MILLINCHPOST
METRIC	<DRIVE>: \ EZCAMW \ EZCAM13 \ MILLMETRICPOST

3. Select the "**Toolpath**" option from the "**Listings**" list box. This will display the tool path parallel to the part program creation.

4. Activate (check) the "**EZ-DNC**" option. This will automatically start the "EZ-DNC Express" application when part program creation is finished and load the newly created file for sending it to 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. As default it displays the part files name and directory for the computed program file.

> Ensure that part file and postprocessor share the same dimension unit ("Metric" 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.
>
> See online help for more information about the "Setup" dialog located in the "View" menu.

6. Click the "**Post**" to start posting. The Processing window will be displayed showing messages followed by listings of ASCII code created. When all Work Steps have been processed, a final message dialog box is shown.

6. Click **OK** to close the message dialog box.

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