Contents

Introduction to SURFCAM .................................................................................................................. 1
  System Requirements ......................................................................................................................... 2
  Software Interface Module ............................................................................................................... 2
  Installing SURFCAM ....................................................................................................................... 2
  Running SURFCAM .......................................................................................................................... 3
  Using the Get Started Project Files ................................................................................................. 3
  SURFCAM Help ................................................................................................................................ 4
  SURFCAM User Interface ................................................................................................................... 5

Using SURFCAM .................................................................................................................................. 11
  Creating part designs in SURFCAM ................................................................................................. 12
  Creating toolpaths ............................................................................................................................. 14
  Managing toolpaths ............................................................................................................................ 17
  Testing the accuracy of toolpaths ....................................................................................................... 18
  Generating NC code ......................................................................................................................... 19
  Communicating with the NC machine ................................................................................................ 19

Project 1 • Draw a 2D Part ................................................................................................................. 21
  Creating and modifying geometry in SURFCAM ............................................................................. 22
  Before you begin .............................................................................................................................. 25
  Step 1: Draw the part profile .......................................................................................................... 26
  Step 2: Draw the closed pockets ...................................................................................................... 37
  Step 3: Draw drilled holes ............................................................................................................... 49
  Step 4: Draw the stock profile ........................................................................................................ 55
  Step 5: Reposition the stock .......................................................................................................... 57

Project 2 • Mill a Production Part .................................................................................................... 61
  SURFCAM 2 Axis Milling .................................................................................................................. 62
  Before you begin ............................................................................................................................... 66
Introduction to SURFCAM

Surfware Inc. incorporates continuous revolutionary innovation into its SURFCAM product line to solve today’s biggest design and manufacturing concerns. SURFCAM utilizes cutting edge technologies for NC programming of 2-, 3-, 4- and 5-axis mills and lathes. This powerful full featured CAM system is easy to learn and a perfect fit for any shop's requirements to increase productivity and profitability.

This Get Started manual is for the Full, Educational, and Demo versions of SURFCAM. All three versions have the same 2D and 3D wireframe and surface design features. To obtain the most value, it is important to keep SURFCAM open as you read your Get Started manual.

Before you begin, you should have the following:

• Experience with the Microsoft® Windows® operating system
• Mechanical design experience
• Machine programming experience

Depending on which version of Windows you are running on your computer, you may notice a difference in the appearance of the menus and windows in this guide. These differences are merely visual and do not affect the projects you will complete in this guide.

• **Standard Version.** The standard version of SURFCAM saves design files with a .scprt file extension. These files can be opened with all three versions of SURFCAM.

• **Educational and Demo versions.** These versions of SURFCAM save files with a .dem file extension. These files can only be read by educational and demo versions of the software.

The Demo version lets you create a temporary toolpath and display it on the screen, but you cannot save the toolpath.

*Important:* Files with a .dem extension cannot be opened with the standard version of SURFCAM.
System Requirements

The following operating system components are required by SURFCAM:

- Microsoft® Windows® 2000 Service Pack 4; Windows® XP Professional Service Pack 1a or greater recommended
- Microsoft® Internet Explorer version 5.0 or greater
- DirectX® 9.0c or greater (see Installing SURFCAM for more information)

The following are minimum hardware requirements for SURFCAM:

- Intel® Pentium® III or Athlon® 800 MHz, 800 MHz or greater; Intel Pentium IV, 3.4 GHz recommended
- 32 MB video card compatible with DirectX 9.0c or higher capable of 16-bit color at a resolution of 1024 x 768; 256 MB capable of 32-bit color at 1280 x 1024 recommended
- RAM requirements of a minimum of 128 MB; 1 GB recommended
- CD-ROM or DVD drive
- Windows®-compatible mouse
- USB port

SURFCAM also requires full hardware acceleration. To enable hardware acceleration:
1. Right-click on your desktop and select Properties.
2. Click the Settings tab.
3. Click the Advanced button.
4. Click the Troubleshoot tab.
5. Move the hardware acceleration slider all the way to Full.
6. Click OK.

Software Interface Module

Surfware sends a Software Interface Module (SIM) with the first purchase of the standard version or the educational version of SURFCAM. Plug your SIM into a USB port on your computer.

Important: Surfware recommends that you insure the SIM for the cost of the SURFCAM system. If your SIM is lost, you will have to purchase a full SURFCAM system to replace the SIM.

Installing SURFCAM

The SURFCAM CD contains programs, neutral format files, sample files, and documentation. When you insert the SURFCAM CD into the drive, it will start automatically.

1. Insert the SURFCAM CD into the CD-ROM or DVD drive of your computer.
2. The CD includes DirectX. If not already installed, SURFCAM’s installation manager requires its installation before you are allowed to install SURFCAM.
3. Select the programs you want to install.
At the end of the installation, the What’s New document will display to let you know about the latest features and enhancements in SURFCAM.

**Running SURFCAM**

You can run several instances of SURFCAM at the same time. Only your computer system can limit the number of instances.

To run SURFCAM, double-click on the desktop icon or, from the Windows Start menu, select All Programs > SURFCAM VELOCITY > SURFCAM VELOCITY.

**Using the Get Started Project Files**

The SURFCAM files needed to complete the projects in this guide are located in the ..\SURFCAM\Samples folder. Two files are provided for each project. The first file is used to complete the project. The second file shows the completed project that you can examine before you begin the lesson.
SURFCAM Help

SURFCAM provides online help as well as this Get Started guide.

Online Help

To access the online help while you are in SURFCAM, do one of the following:

- Press the F1 key at any time
- On the Help menu, select Contents
- When a dialog is displayed on the screen, click the Help button or press F1 for specific help about the dialog options

Use the Contents, Index and Search tabs to navigate through the help. Use the Favorites tab to list topics you plan to visit again.

- **Contents.** Displays help topics in a chapter format. Select a book heading to view subheadings and topics available in that area of interest. Click a topic to display information in the right pane of the Help window.
- **Index.** Displays help topics in alphabetical order. Scroll through the list or type the first few letters of the keyword you are looking for. When an entry is highlighted, simply press Enter or click the Display button to display the help topic. You can also double-click on an entry to display the help topic.
- **Search.** Provides the ability to search for every instance of a specific word or phrase in the online help. Type a word or phrase (enclose a phrase in quotes to find the exact phrase), then press Enter or click the List Topics button.
- **Favorites.** Saves a list of help topics that you commonly refer to. To add a Favorites entry, use the Contents, Index or Search tabs to open the topic and then click the Add button.

Multimedia Tutorials

Also available on the Help menu, the multimedia tutorials give you a quick introduction to SURFCAM. These interactive tutorials let you see and then practice an operation.

About SURFCAM

This window displays product and license information about your current version of SURFCAM. A direct link to the Surfware web site is also provided.
Menu Region

The region at the top of the SURFCAM window contains menus and toolbars that display commands used to perform tasks in SURFCAM.

- **Main Menu Bar.** Contains menus used to select the system’s main functions.
- **SURFCAM Main Toolbar.** Contains shortcuts to frequently used commands such as New, Open, and Save.
- **SURFCAM Status Toolbar.** Contains shortcuts to commands used to change the system’s parameters for items frequently changed by the user such as Color, Layer, and Cview.
- **Submenu Toolbar.** When certain menu items are selected, a toolbar with commands related to the function displays.

To change the display of icons on the toolbars, right-click on any toolbar to display a context-sensitive menu. If you select Small Icons or Large Icons, your change will affect the display on all
toolbars. If you select No Text Labels to hide the display of text labels on icons, your change will only affect the toolbar where your pointer was positioned when you right-clicked.

Toolbars can also be repositioned. Point to the vertical bar at the left of the toolbar, then drag the toolbar to a new position. To restore toolbars to their original positions, right-click and select Restore Toolbar Defaults.

**The Main Menu**

The following table summarizes the eight items on the main menu:

<table>
<thead>
<tr>
<th>Menu</th>
<th>Keyboard Shortcut</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>File</td>
<td>F</td>
<td>Performs file operation functions such as opening, saving, and printing.</td>
</tr>
<tr>
<td>Create</td>
<td>C</td>
<td>Creates various types of geometric entities such as basic geometry, surfaces, text, dimensions, and vectors.</td>
</tr>
<tr>
<td>Edit</td>
<td>E</td>
<td>Modifies geometric entities such as trimming, extending, transforming, and deleting.</td>
</tr>
<tr>
<td>Display</td>
<td>D</td>
<td>Changes the display of geometric entities such as sizing, view orientations, and rendering.</td>
</tr>
<tr>
<td>NC</td>
<td>N</td>
<td>Performs various milling, turning, wire EDM, and mill/turn functions to generate toolpaths.</td>
</tr>
<tr>
<td>Analyze</td>
<td>Z</td>
<td>Displays measurement information about geometric entities such as coordinate location, distance, angle, tangency, and curvature.</td>
</tr>
<tr>
<td>Tools</td>
<td>O</td>
<td>Changes the default parameters in SURFCAM such as system units, axis, and the appearance of the SURFCAM window</td>
</tr>
<tr>
<td>Help</td>
<td>none</td>
<td>Gives access to online help, multimedia tutorials, and system information</td>
</tr>
</tbody>
</table>

**The SURFCAM Main Toolbar**

The buttons on the SURFCAM toolbar provide shortcuts to many commonly used commands.

Starting at the left, the File buttons are standard Windows® commands for opening, saving, and printing files. These commands are also available on the File menu.

Next, the display buttons let you change the display of objects in the graphics screen. These commands are also available on the Display menu.

The Delete and Transform buttons let you edit objects in the graphics screen.

At the right of the toolbar, the Operations Manager and Toolpath Verification buttons let you manage and verify NC toolpaths.
<table>
<thead>
<tr>
<th>Icon</th>
<th>Command</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>New</td>
<td>Opens a new file</td>
</tr>
<tr>
<td></td>
<td>Open</td>
<td>Opens an existing file</td>
</tr>
<tr>
<td></td>
<td>Save</td>
<td>Saves a file with the current filename. When a file has been modified but not saved, the icon displays a red figure.</td>
</tr>
<tr>
<td></td>
<td>Print</td>
<td>Prints using the default printer</td>
</tr>
<tr>
<td></td>
<td>Fit</td>
<td>Graphically shrinks or expands elements in the graphics space to fit the screen</td>
</tr>
<tr>
<td></td>
<td>Rotate</td>
<td>Graphically rotates elements in the graphics space</td>
</tr>
<tr>
<td></td>
<td>Zoom in</td>
<td>Graphically expands elements in the graphics space to a region defined by a selection box</td>
</tr>
<tr>
<td></td>
<td>Zoom out</td>
<td>Graphically shrinks elements in the graphics space by 10%</td>
</tr>
<tr>
<td></td>
<td>Pan</td>
<td>Graphically slides elements in the graphics space based on movement of the mouse</td>
</tr>
<tr>
<td></td>
<td>Last View</td>
<td>Displays up to 20 of the most recent screen displays</td>
</tr>
<tr>
<td></td>
<td>Next View</td>
<td>Re-displays up to 20 of the most recent screen displays in reverse of Last View</td>
</tr>
<tr>
<td></td>
<td>Repaint</td>
<td>Refreshes/redraws the display in the graphics space</td>
</tr>
<tr>
<td></td>
<td>Delete</td>
<td>Deletes elements, views and layers</td>
</tr>
<tr>
<td></td>
<td>Transform</td>
<td>Moves or copies elements to another coordinate value</td>
</tr>
<tr>
<td></td>
<td>Operations Manager</td>
<td>Manages generated toolpath</td>
</tr>
<tr>
<td></td>
<td>Toolpath Verification</td>
<td>Launches toolpath verification from outside the operations manager</td>
</tr>
</tbody>
</table>
The SURFCAM Status Toolbar

The buttons on the Status toolbar show the current system parameters. Click a button to change its value.

To hide the display of text labels, right-click on the toolbar and select No Text Labels.

<table>
<thead>
<tr>
<th>Icon</th>
<th>Command</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><img src="image" alt="Color: 40" /></td>
<td>Color</td>
<td>Sets the current system default color</td>
</tr>
<tr>
<td><img src="image" alt="View: 1" /></td>
<td>View</td>
<td>Changes the graphic view by its view number. You can press CTRL+1 through CTRL+8 to change to views 1 through 8.</td>
</tr>
<tr>
<td><img src="image" alt="View Association: WORLD" /></td>
<td>View Association</td>
<td>Sets Associativity to the active CVView</td>
</tr>
<tr>
<td><img src="image" alt="CView: 1" /></td>
<td>CVView</td>
<td>Set a construction/cut view by its view number</td>
</tr>
<tr>
<td><img src="image" alt="Layer: 1" /></td>
<td>Layer</td>
<td>Sets the current active layer by its number. You can place elements of a drawing on a maximum of 256 layers.</td>
</tr>
<tr>
<td><img src="image" alt="Wireframe: Shaded with wireframe" /></td>
<td>Wireframe Shaded</td>
<td>Toggles the display of elements in the graphics space between wireframe, shaded with wireframe, and shaded view</td>
</tr>
<tr>
<td><img src="image" alt="Coord: WORLD" /></td>
<td>Coord.</td>
<td>Sets the current coordinate system to World or View. Your choice controls which dialog is displayed when you need to enter the coordinates of a point.</td>
</tr>
<tr>
<td><img src="image" alt="Mask: OFF" /></td>
<td>Mask</td>
<td>Sets the system masking to control which elements cannot be selected in the graphics space. When Mask On is displayed, the unchecked items and colors in the Mask dialog cannot be selected. When Mask Off is displayed, any element or color can be selected in the graphics space.</td>
</tr>
<tr>
<td><img src="image" alt="D: 0.00000" /></td>
<td>Depth</td>
<td>Changes the current working depth (Z value) of entities when they are created in sketch mode</td>
</tr>
<tr>
<td><img src="image" alt="Scale: 1.62351" /></td>
<td>Scale</td>
<td>Sets the scale value for the graphics space</td>
</tr>
<tr>
<td><img src="image" alt="Line Type:" /></td>
<td>Line Type</td>
<td>Sets the line type for geometry as it is created</td>
</tr>
</tbody>
</table>
**Graphics Space**

The Graphics Space is the largest area of the screen. This is the working area where part geometry can be created, viewed, modified, or deleted.

**View Indicator**

The 3-dimensional design environment in SURFCAM uses two types of XYZ coordinate systems: world and local. World coordinates use the global origin, which never moves. Local coordinates use a local origin.

The View Indicator displays the origin of the world coordinate system. As you rotate the view in the Graphics Space, the orientation of the View Indicator changes.

**CView Indicator**

The Construction View (CView) Indicator displays the origin of the local coordinate system. Geometry is created on the CView. Before you create geometry, select the correct CView.

**Prompt Line**

At the lower left of the SURFCAM window, the prompt line displays instructions or additional information about menu commands. SURFCAM will prompt you for the actions you must take and display other important information while executing a command.

**Pointer Coordinates**

The coordinates of the current location of the pointer are displayed at the lower right of the SURFCAM window.
SURFCAM is a computer-aided design (CAD) program and a computer-aided manufacturing (CAM) program. You can design a part as well as create the toolpath to machine that part. This chapter describes the steps for design through manufacture using SURFCAM.

1. Create the part design
2. Create the toolpath
3. Manage toolpaths
4. Test the accuracy of toolpaths
5. Create NC code
6. Communicate with the NC machine
Creating part designs in SURFCAM

SURFCAM has full design capabilities that let you create basic 2D and 3D geometry such as lines, points and circles plus more complex 3D shapes such as NURB curves, splines, and a wide variety of surfaces. You can also create text and add dimensions to your drawing.

Use the commands on the Create menu to create geometry, splines, surfaces and dimensions.

![Create menu](image)

Use the commands on the Edit menu to modify a design after it is created.

![Edit menu](image)

Using designs from other programs

Designs from popular CAD programs can be opened in SURFCAM. The file can be opened as its own entity or appended to an existing SURFCAM part file.
Use the Open command on the File menu to open files of the following types:

- SolidWorks files (*.prt, *.sldprt)
- SolidWorks Assembly files (*.sldasm)
- Autodesk DXF and DWG files (*.dxf, *.dwg)
- IGES (*.igs, *.iges)
- ACIS (*.sat)
- Parasolid (*.x_t, *.x_b)
- Solid Edge (*.par)
- VDA-FS (*.vda)
- CADL (*.cdl)
- G-Code (*.ncc)
- Cam Profile (*.cm)
- Gear Profile (*.gr)
- Involute Curve (*.inv)
- ASCII Data (*.asc)

Special translators are also available for the following file types through separate purchase:

- CATIA 4 (*.model, *.exp)
- CATIA 5 (*.catpart)
- Mastercam (*.mc*)
- Pro/Engineer (*.prt, *.prt.*)
- STEP AP203/214 (*.step, *.stp)
- UG II (*.prt)
- Autodesk Inventor (*.ipt)
Creating toolpaths

When a design is complete you can begin applying toolpaths to it. Use the various machining modes on the NC menu to create the toolpaths.

**NC menu**

Machining modes

When you select a machining mode, its toolpath commands display on the submenu.

**3 Axis submenu**

To display NC toolbars

You can choose to display NC commands on toolbars instead of submenus.

1. On the Tools menu, click Options and then select NC Defaults.
2. Select Display Toolbar and click OK.
To activate the change, you need to restart SURFCAM. The next time you click a machining mode on the NC menu, the appropriate toolbar will display.

3 Axis toolbar

To customize an NC toolbar/menu
You can customize an NC menu to display only those commands you use most often.
1. On the Tools menu, click Customize and then NC Toolbar/Menu.

2. Click the machining mode you want to customize.
3. Select the commands you want to hide and then click the Remove button.
You can also insert separators or change the order of the commands.

4. When finished, click Close.

**Assigning toolpath parameters on operation pages**

Before you can create toolpaths, you must select the part geometry you want to cut. After you select the part geometry, SURFCAM displays a dialog where you enter values for the toolpath parameters. Each type of operation has a different dialog with different parameters. Typically, these dialogs have three tabs:

- **Tool Information tab**: Use this tab to select the tool and the material for the machining operation. The feed rates, plunge rates and speeds are calculated and displayed. You can change these calculated values if necessary. Changing a value may cause other values to change.

- **Cut Control tab**: Use this tab to set the parameters that control how the material is cut.

- **Defaults tab**: The parameters on this tab are the same for all the toolpath operations in that machining mode. Normally you do not need to change these values.

After you define your machining parameters, SURFCAM automatically creates the toolpath data and then draws the toolpath on the screen.

When the drawing of the toolpath is complete, SURFCAM displays the Keep Operation dialog. Click the Accept button to keep the toolpaths or click Reject to delete the toolpath.

The new toolpath is automatically assigned a name. A new icon with that name is added to the NC Operations Tree in the NC Operations Manager dialog.
Managing toolpaths

Use the NC Operations Manager to manage a single operation, a group of operations or several groups of operations.

Click **Operations Manager** on the **Main** toolbar to display the dialog.

![SURFCAM NC Operations Manager](image)

The names of the toolpaths are displayed in an Operations Tree in a group that is called a Setup Section. You can change the name of a toolpath on the screen.

The names display in the same sequence they were created. SURFCAM will cut the toolpaths in this sequence. To change the sequence, click on a toolpath and drag the icon to a different position.

You can create more setup sections if necessary. Then you can click on a toolpath and drag the icon to a different setup section. All toolpaths in one setup section use the same machine setup.
Testing the accuracy of toolpaths

Use SURFCAM Verify to test the accuracy of toolpaths. This program simulates the creation of a solid block or cylinder of material around the part. Then you can see a simulated tool remove the material from around the part.

The SURFCAM Verify command is available in the Operations Manager and on the Main toolbar.

SURFCAM Verify

When you select SURFCAM Verify on the Main toolbar, click the down arrow to select the Project, Setup Section or operation you want to verify.

Play | Verification Speed | Stop | Step Forward | Turbo
Generating NC code

A CNC machine tool needs the correct NC code to cut a part. The SURFCAM post processor systems have the machine configuration programs required to create this code. SURFCAM provides a large library of posts for a number of different machine types.

The post processor uses the toolpath information and the machine information in the post file to create the NC code. This code is saved in a file with an NCC extension.

After the NCC file is created, use the SURFCAM Editor to make any necessary adjustments to the file.

Communicating with the NC machine

Before a machine can cut a part, you must transfer the contents of the NCC file to the machine. One method of transfer is called Direct Numerical Control (DNC) communications. The computer with the NCC file is connected to the machine with an RS-232 connection. SURFCAM includes a program to transfer the NCC files to the machine controller.

Note: Errors can occur with RS-232 connections. Before SURFCAM dealers can help with the DNC program, the RS-232 connection must operate free from errors. If necessary, see your computer dealer or the machine tool dealer to correct any RS-232 errors.
Geometry provides the basis for all product design. SURFCAM offers a wide range of design tools for 2-dimensional geometry as well as 3-dimensional geometry and surfaces. This project shows you how to create and modify 2-dimensional geometry in a variety of ways to create a typical milled part.

1. Draw the part profile
2. Draw closed pockets
3. Draw drilled holes
4. Draw the stock profile
5. Reposition the stock to the origin
Creating and modifying geometry in SURFCAM

SURFCAM offers many design features that are often found in popular CAD systems. The intuitive interface makes it easy to quickly draw and modify geometry. This makes SURFCAM a complete system for both design and manufacture.

Creating geometry

SURFCAM offers many choices on the Create menu for easily creating 2-dimensional geometry.

Create Menu

Choices for creating 2D geometry include:

- Single points or multiple points, including point arrays
- Lines, including rectangles
- Arcs
- Circles
- Fillets
- Chamfers
- Splines

In this project, you will learn techniques for creating different types of points, lines, circles, fillets, and chamfers.

Editing geometry

SURFCAM also makes it easy to modify geometric elements after they are created. Use the commands on the Edit menu to modify 2-dimensional geometry.
Choices for editing 2-dimensional geometry include:
- Trimming and breaking elements
- Breaking an element into incremental sections
- Extending elements
- Moving and copying elements (transforming)
- Deleting elements
- Flipping arcs

In this project, you learn how to trim, move, and delete elements.

**Input methods**

As geometry is created, SURFCAM provides several input methods for locating elements exactly where you want them.
## Input Options

<table>
<thead>
<tr>
<th>Icon</th>
<th>Command</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><img src="image" alt="Sketch" /></td>
<td>Sketch</td>
<td>A grid and the sketch tool display in the graphics space. Any point on the grid can be selected for input. You can right-click in the graphics area to turn Snap Grid on or off. Grid properties are set in the Options dialog. On the Tools menu, click Options, then select Display &gt; Snap and Grid.</td>
</tr>
<tr>
<td><img src="image" alt="Point" /></td>
<td>Point</td>
<td>Locates the XYZ coordinate position of a point that already exists</td>
</tr>
<tr>
<td><img src="image" alt="End Point" /></td>
<td>End Point</td>
<td>Locates the XYZ coordinate position of an endpoint on the element you select. SURFCAM selects the endpoint nearest the position you clicked.</td>
</tr>
<tr>
<td><img src="image" alt="Center" /></td>
<td>Center</td>
<td>Locates the center of an arc, the center of a circle, or the midpoint of a line</td>
</tr>
<tr>
<td><img src="image" alt="Midpoint" /></td>
<td>Midpoint</td>
<td>Locates the midpoint of an arc or a line</td>
</tr>
<tr>
<td><img src="image" alt="Intersect" /></td>
<td>Intersect</td>
<td>Locates the XYZ coordinate position of a selected element and its intersection with another element. If two intersections are possible, SURFCAM selects the intersection nearest to the position you clicked. The two elements do not need to have a true intersect. Each element can be on a different plane. The intersection is projected based on the current CView. The intersection is located on the plane of the first element selected.</td>
</tr>
<tr>
<td><img src="image" alt="Relative" /></td>
<td>Relative</td>
<td>Locates a position that is a relative distance from a known location, which you select</td>
</tr>
<tr>
<td><img src="image" alt="Keyboard" /></td>
<td>Keyboard</td>
<td>Locates a position from X, Y, and Z coordinates entered in the World dialog. This location is based on World or View coordinates as set in the Status menu. When Coord is set to View, the coordinates use the current construction view (CView).</td>
</tr>
<tr>
<td><img src="image" alt="Quadrant" /></td>
<td>Quadrant</td>
<td>Selects one of four points on a circle. These points are 90° apart starting at the 3 o'clock position. To select one of these four points, click any point of the circle that is within 45° of the point you want. You can also select any of the same four points associated with an arc if they are within 45° of an end point of the arc.</td>
</tr>
</tbody>
</table>

On the same toolbar, two other options will either return you to the previous toolbar or exit the function and close the toolbar.

<table>
<thead>
<tr>
<th>Icon</th>
<th>Command</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><img src="image" alt="Go Back to Prev Toolbar" /></td>
<td>Go Back to Prev Toolbar</td>
<td>Exits the current function and returns to the previous toolbar</td>
</tr>
<tr>
<td><img src="image" alt="Exit Current Function" /></td>
<td>Exit Current Function</td>
<td>Completely exits the current function</td>
</tr>
</tbody>
</table>
Before you begin

The unit of measure for all projects in this Get Started guide is inch. Make sure your System Unit is set to Inch before you begin drawing geometry.

To set the system unit to Inch:
1. On the Tools menu, click Units.
2. Select Inch.

Units menu

In this project, the illustrations of toolbars do not show text labels. To control the display of text labels, right-click next to a toolbar and either check No Text Labels to hide the display of labels next to the toolbar icons or uncheck No Text Labels to display the labels.

If you are new to SURFCAM, it may be preferable to show text labels until you are familiar with the interface.
Step 1: Draw the part profile

The first step in this project is to draw the outer profile of the part. This profile is square-shaped with an open pocket along the lower edge.

You will begin by drawing all four outer lines at once. You will then offset three of the lines to form the profile of the open pocket at the bottom. You will finish by trimming the lines and rounding the corners with half-inch fillets.

**Draw the square**

Begin by drawing a rectangle that is 10 inches x 10 inches square with its lower left corner located at the origin. Although SURFCAM offers many ways to draw lines, you will create lines in this project using the following line commands:

- *Rectangle* will be used to create the four sides of the outer profile.
- *Offset* will be used to create the basic shape of the open pocket at the bottom of the profile.
On the Create menu, click Line

Click Rectangle

Set the dialog as shown

Click OK

---

Now that you’ve specified the size of the rectangle, you need to define the location of the corner point. In this case, you will enter coordinate values of 0,0,0 using the Keyboard command.

You will then fit the display in the graphics area to make it easier to see.

Since you will be creating more lines, you will use the Go Back to Prev. Toolbar command to take you back to the Create > Lines toolbar. This is faster and easier than returning to the Create menu.
5. Click Keyboard
6. Enter 0.0 for all the values
7. Click OK
8. Click Fit
9. Click Go Back to Prev. Toolbar
Draw the outline of the open pocket

Next you will offset some of the lines you just created to form the shape of the open pocket.

- The bottom line will be offset by a distance of **2 inches**.
- The left and right lines will be offset by a distance of **2.3 inches**.

The offset distance is easily changed. SURFCAM will retain the same offset distance until you decide to change it again.

After the offset distance is entered, SURFCAM will first prompt you to select the element to be offset. You will then be prompted to select the direction for the offset.

1. Click Offset
2. Click Change Offset
3. Enter an offset distance of 2.0 and click OK
4. Select the bottom line

5. Click inside the profile to indicate the offset direction

**Note:** Remember to follow the prompts when creating geometry

6. Click Change Offset again

7. Enter an offset of 2.3 and click OK
Select the left vertical line and click inside the profile.

Select the right vertical line and click inside the profile.

Now that you have the basic shape for the open pocket, the next step is to trim the middle of the lower line to create the opening and then round the corners.
**Trim and fillet the corners of the open pocket**

You will begin by trimming the middle section of the bottom line to the two offset lines. Then you will round all the corners with a **0.5** inch fillet.

1. **On the Edit menu, click Trim/Break**

2. **Click AutoTrim**

3. **Select the bottom line between the two offset lines**

Notice that the cursor has an image of a pair of scissors to indicate that you are in trimming mode.
Now that the middle segment of the line is trimmed, you can add fillets to the corners. The fillet command will also trim the lines at the same time.

4. On the Create menu, click Fillet

5. Click Change Radius

6. Enter a fillet radius of 0.5 and click OK
Click the Trim button until Trim 2 displays

Select the lines in the order shown below:
1 2 3 4 5 6 7 8
Chamfer the corners of the profile

The next step is to add a **0.125** inch chamfer to the corners of the outer profile. To accomplish this as quickly as possible, you will again chain the geometry and apply the chamfer to all corners at once.

1. On the Create menu, click Chamfer
2. Click Change Chamfer
3. Enter 0.125 for the First and Second elements picked and click OK
4. Click Chain
5. Select the beginning element as shown.

6. Select the ending element as shown.

7. Click Done.
Step 2: Draw the closed pockets

In this step, you will draw the two closed pockets inside the part profile.

The first pocket is quite simple since it’s a circle. For the upper pocket, you will learn another technique for offsetting geometry using the Transform command and then round the corners by adding arcs and fillets.

### Draw the round pocket

In this step, you will draw a 5.0 inch diameter circle in the center of the square profile.

1. On the Create menu, click Circle
1. Click Center/Diameter
2. Enter a diameter of 5.0 and click OK
3. Click Keyboard
4. Enter 5.0 for both the X value and the Y value
5. Click OK
Draw the outline of the upper closed pocket

Next, you will offset the top portion of the profile and the circle by 0.7 inch to draw the outline of the closed pocket.

1. On the Create menu, click Line
2. Click Offset

3. Click Change Offset

4. Enter an offset of 0.7 and click OK

Instead of selecting lines one by one, you will connect the lines in a single *chain* and offset all the lines at once.

A chain can be created from any number of elements that share end points. You simply select the starting element and the ending element and SURFCAM will connect all the elements in between automatically.

5. Click Chain
You want the chain to begin at the lower right corner of the profile geometry. The chain will end at the lower left corner.

**6** Select the right vertical line near the lower right corner

A small square highlights the start point for the chain.

**7** Select the left vertical line near the lower left corner

The geometry that will be included in the chain is highlighted, letting you verify your selection before moving to the next step.

**8** Click Done to create the chain
Click inside the profile to indicate the offset direction.

Click OK in the dialog to accept the defaults.

Next you will offset the circle by the same distance. SURFCAM retains the offset distance you entered earlier until you decide to change it.

On the Create menu, click Circle.
12. Click Offset

13. Select the circle

14. Click outside the circle to indicate the offset side

Trim and fillet the corners of the closed pocket

Next you will draw 0.75 inch arcs that are tangent to the offset lines and circle you just created. You will then trim the geometry to form the shape of the pocket.

1. On the Create menu, click Arc
1. Select the lines and circle in the order shown.
2. Click Tangent 2.
3. Click Change Radius.
4. Enter 0.75 for the Fillet Radius and click OK.
5. Select the lines and circle in the order shown.
On the Edit menu, click Trim/Break.

Click AutoTrim.

Select the lower half of each offset vertical line to trim them to the tangent arcs.
9 Select the lower half of the largest circle as shown

10 Select the other side of the circle as shown

A portion of the circle remains that needs to be deleted.
On the Main toolbar, click Delete.

Select the lower portion of the largest circle to delete it.

On the Create menu, click Fillet.

You will finish the pocket by adding 0.5 inch fillets in the upper corners.
14. Click Change Radius

15. Change the radius to 0.5 and click OK

16. Fillet the two top corners of the closed pocket
Step 3: Draw drilled holes

This part has two types of hole patterns: circular and rectangular. You will learn how to create a circular array of points and circles centered around the circular pocket. Then you will learn how to create a rectangular array of points and circles relative to the bottom left corner of the square part profile.

Draw points and circles in a circular array

You will create six points evenly spaced in a 360° circular array around the center of the round pocket. The radius of the pocket is 2.50 inches. To allow clearance for the 0.375 inch diameter drill holes, the radius of the array will be 2.85 inches.

1. On the Create menu, click Points
2. Click Circular
3. Set the dialog as shown and click OK
4 Click Center

5 Select the circle to create the points and circles
Draw points and circles in a rectangular array

You will now create points and 0.375 inch circles in each corner of the part. The overall size of the part is 10 inches x 10 inches. The holes will be situated at the corners of a 9 inch x 9 inch square, leaving 1/2-inch clearance from the outer edges of the part.

First, you will create a single point that is placed at a relative distance from the origin. This point will be used to identify the lower left corner for the new hole pattern.

1. On the Create menu, click Point.

2. Click Relative.

3. Enter 0.5 for the X-value and Y-value and click OK.
5. Select the origin as shown. Notice that the cursor changes when you hover the mouse over the origin.

6. On the Create menu, click Points.
Click Rectangular

Set the dialog as shown and click OK

Note: The exact value of the circle radius can be entered or you can use an equation to define the radius. In this case, you entered the diameter of the circle divided by two to calculate the radius.

Click Point

Select the point you just created
You have now completed drawing the part. Next, you will draw a profile of the stock around the part.
Step 4: Draw the stock profile

Since this part is ten inches square, you must create stock that is larger than the part. In this case, you will use standard **12 inch** square stock. You will draw the square stock profile the same way you drew the square profile of the part except that the lower left corner of the stock will be located to the left and below the origin by one inch.

1. On the Create menu, click Line
2. Click Rectangle
3. Set the dialog as shown and click OK
4. Click Keyboard

5. Set the dialog as shown and click OK
Step 5: Reposition the stock

In most circumstances, you will need to reposition geometry based on where it will be set up on the machine. For this project, you will move the part and the stock profile so that the lower left corner of the stock is located at the origin.

The Transform command is used to reposition, rotate, scale, mirror, or offset elements. You can find the Transform command on the Edit menu or on the Main toolbar.

1. On the Main toolbar, click Transform

2. Click Transform > Move:

3. Click Location

4. Click Visible to select all visible elements in the graphics area
Now you need to specify two locations that will define the distance and direction for the movement of the geometry. You will first use the **End Point** option to specify the point where the geometry will be moved from, based on the selection of an element. You will then use the **Sketch** option to select a point in the graphics area where you want the geometry moved to.

5  Click End Point

6  Select the lower left corner of the stock profile

This is the location you will move the geometry “from”.
Select the origin as shown

This is the location you will move the geometry “to”. Notice that the cursor image changes when you move the cursor close to the origin (as shown above).
This project is now complete.
To learn how to machine the part you just created, see *Project 2: Mill a Production Part*. 
Project 2 • Mill a Production Part

The goal of production milling is to produce high-quality parts with the shortest lead times. SURFCAM provides the tools that let you quickly create tool paths that are designed to reduce overall cycle times while minimizing wear on your machine and cutting tools.

This project shows you how to easily produce 2.5-axis machining operations for a typical production part, verify that tool path and create a setup sheet for the shop floor.

1. Face the material
2. Mill the pockets and profile
3. Drill the holes
4. Chamfer the part edges
5. Verify the tool path
6. Create a setup sheet
7. Generate the NC code
SURFCAM 2 Axis Milling

SURFCAM’s 2 Axis system is widely recognized as a high performance, cost effective product. Machining parameters are associative to the geometry so that the system can easily and efficiently regenerate the toolpath whenever changes are made to the tool or cut information. Optional 4- and 5-axis positioning and optimized drilling are also available for SURFCAM 2 Axis.

Typical 2 Axis Workflow

The creation of most 2 Axis operations follows the same basic workflow.

1. On the NC menu, click 2 Axis and then click the type of operation you want to create.

<table>
<thead>
<tr>
<th>File</th>
<th>Create</th>
<th>Edit</th>
<th>Display</th>
<th>NC (editable)</th>
<th>Analyze</th>
<th>Tools</th>
<th>Help</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td>2 Axis</td>
<td>TrueMill</td>
<td>TrueMill Reference</td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td>3 Axis</td>
<td>Pocket</td>
<td>Pocket Reference</td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td>4 Axis</td>
<td>Contour</td>
<td>Contour</td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td>5 Axis</td>
<td>Contour 3D</td>
<td>Contour 3D</td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td>Lathe</td>
<td>Drill</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td>Wire EDM</td>
<td>Pilot Hole</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td>Mill/Turn</td>
<td>Face Mill</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>Round Mill</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>Chamfer Mill</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>Groove Mill</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>Rest Material</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>Thread Mill</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>Options</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

2. The Select Chain toolbar is displayed and SURFCAM prompts you to select the beginning element. Select the elements you want to machine and, optionally, select the material boundary if you want to machine between the part boundary and the boundary of the stock material.

3. Click Done to open the operation dialog.
4. On the Tool Information tab, select a cutting tool and material, then set speeds and feed rates.
5. On the Cut Control tab, define the machining parameters for this operation.
6. Depending on the type of operation, additional tabs with parameters may be available.
7. When finished, click OK.
8. Accept or reject the new tool path.
Selection Methods

After a command is selected on the 2 Axis menu, you are prompted to select the elements to machine.

Most 2 Axis operations are based on chains that define the path the tool will follow for contouring operations or the boundary for pocketing operations. Chains are used to define the part boundaries you want to machine as well as the boundary of the stock material.

Drilling operations do not use chains. Instead, SURFCAM prompts you to select circles and/or points to define the drill locations.

<table>
<thead>
<tr>
<th>Icon</th>
<th>Command</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>![Done Icon]</td>
<td><strong>Done</strong></td>
<td>After chains are selected, click Done to continue with the machining operation.</td>
</tr>
<tr>
<td>![Single Icon]</td>
<td><strong>Single</strong></td>
<td>Select a single element or a series of single elements.</td>
</tr>
<tr>
<td>![Chain Icon]</td>
<td><strong>Chain</strong></td>
<td>Create a single chain from several connected elements. Select a point near the beginning of the first element, then select a point near the end of the last element. To chain all elements of a closed shape, click the beginning point twice.</td>
</tr>
<tr>
<td>![Vector Icon]</td>
<td><strong>Vector</strong></td>
<td>Create chain profiles that have vectors attached to them. When a Pocket or Contour operation is created, the operation automatically knows to cut on the sides defined by the vectors. To attach a vector to a profile, click Create &gt; Vector, then select from the Vector menu. Perpendicular Element is a popular choice.</td>
</tr>
<tr>
<td>![Auto Icon]</td>
<td><strong>Auto</strong></td>
<td>Select a single element to automatically create a chain from all connected elements. SURFCAM displays the Select toolbar to define the selection of elements. Click Done to return to the Select Chain toolbar.</td>
</tr>
<tr>
<td>![Plunge Icon]</td>
<td><strong>Plunge</strong></td>
<td>Select plunge locations for the cutting operation. SURFCAM displays the Select Point toolbar to define the plunge point coordinate. Click Done to return to the Select Chain toolbar. The cutting operation uses the closest user-defined plunge point and feeds to the cutting depth. If you do not specify plunge points, the operation automatically calculates plunge positions. SURFCAM ignores the Z axis coordinate of the plunge point, because the operation determines the depth to plunge.</td>
</tr>
</tbody>
</table>
**Select Chain Options**

<table>
<thead>
<tr>
<th>Icon</th>
<th>Command</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>🍃</td>
<td>Material</td>
<td>Use Material to chain the material boundary to define where material removal should occur between the Part and the Material boundary.</td>
</tr>
<tr>
<td>🏡</td>
<td>Part</td>
<td>Use Part to chain the part boundaries for contours and pockets. Part is the default chaining mode.</td>
</tr>
<tr>
<td>🔍</td>
<td>Chain Options</td>
<td>Displays the Chain Options dialog.</td>
</tr>
</tbody>
</table>

**SURFCAM Chain Options**

- **Chain Planar Elements**: When set to No, the system will chain in any direction. When set to Yes, the system maintains the chaining direction along the plane defined by the first entity chosen.
- **Limit Tangent Angle**: When set to No, the system will chain two elements that come together at any angle. When set to Yes, the system will chain only elements that meet at an angle less than the specified Angle Tolerance value.
- **Angle Tolerance**: This setting is available only when Limit Tangent Angle is set to Yes. Enter the limit angle for tangent elements.

When the Vector or Auto options are clicked on the Select Chain toolbar, the Select toolbar is displayed.

**Select toolbar**

**Select Options**

<table>
<thead>
<tr>
<th>Icon</th>
<th>Command</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>🌟</td>
<td>Done</td>
<td>After elements are selected, click Done to return to the Select Chain toolbar.</td>
</tr>
<tr>
<td>🎨</td>
<td>Single</td>
<td>Select a single element or a series of single elements.</td>
</tr>
</tbody>
</table>
## Select Options

<table>
<thead>
<tr>
<th>Icon</th>
<th>Command</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>![Within Icon]</td>
<td><strong>Within</strong></td>
<td>Select one or more elements enclosed completely within a rubber-band box.  A rubber-band box is a rectangle that you can stretch and position on the screen. Select a point on the screen, then click the left mouse button to anchor that position. Move the mouse to create a box. Click the left mouse button again when the box encompasses the region you want to select.</td>
</tr>
<tr>
<td>![Intersect Icon]</td>
<td><strong>Intersect</strong></td>
<td>Select one or more elements that are inside or <em>touching</em> a rubber-band box.</td>
</tr>
<tr>
<td>![Visible Icon]</td>
<td><strong>Visible</strong></td>
<td>Select everything that appears in the current work space.</td>
</tr>
<tr>
<td>![MultSelOn Icon]</td>
<td><strong>MultSelOn</strong></td>
<td>In most cases, clicking this command will toggle between MultSelOn and MultSelOff. If MultSelOn is displayed, the Select menu remains available as you select elements and Done also appears as a command on the menu. If MultSelOff is displayed, SURFCAM will move to the next menu after you select geometry.</td>
</tr>
<tr>
<td>![SelMode / DeselMode Icon]</td>
<td><strong>SelMode / DeselMode</strong></td>
<td>Click to toggle between SelMode and DeselMode. With SelMode you can select elements. With DeselMode you can deselect elements that you previously selected.</td>
</tr>
</tbody>
</table>
Before you begin

Make sure your unit of measure is set to Inch before you begin this project.

To set the system unit to Inch:
1. On the Tools menu, click Units.
2. Select Inch.

Open the project file
1. On the Main toolbar, click Open.
2. Open the file GetStarted 2D Milling-1.SCPRT

For this project you will learn how to use the following 2 Axis commands:
• Face Mill
• TrueMill
• Drill
• Chamfer Mill

For details about other commands on the NC > 2 Axis menu, please refer to SURFCAM Help.
Step 1: Face the material

Face milling typically uses face mills, shell mills, and fly-cutters to clear a relatively thin layer of material over the top of a part. This provides a uniformly flat starting surface for subsequent milling operations. SURFCAM designed its Face Mill operation to be as simple as possible, without losing any needed functionality.

SURFCAM’s Face Mill operation provides optimal performance as follows:

- Stepovers are adjusted to the part width to insure a constant width for all material cuts. A constant width of cut decreases the wear on the tool and machine.
- You can elect to reduce burrs with a climb cut along the part edge on the final cut (available only for the Zig and Zig Zag face methods).
- You can make use of the large side step (at shallow depths) capability of face mills, shell mills and fly-cutters. Side steps can be set to up to 90% of the tool diameter.
- The tool is never allowed to plunge into the material. Every face mill toolpath begins at the cut depth at a user-defined distance outside the selected geometry.

Facing Methods

- Zig: The tool is repositioned after each pass so that all cuts are created in the same direction.
- Zig Zag: The tool moves in a back-and-forth pattern.
- Spiral: The tool starts at the outside and moves inward, maintaining the cutting method (climb or conventional).

For this project, you will face the stock with a 3-inch face mill to remove 0.010 of material from the top of the part. You will use a Spiral facing method. The rapid plane is located 1 inch above the part.

On the NC menu, click 2 Axis and then click Face Mill.
By default, Chain is selected on the Select Chain toolbar. You will create a chain from the stock profile.

The Face Milling dialog displays. Next you will select a tool from the SURFCAM tool library. Since this is a facing operation, the system will automatically display a list of all face mills in the library.
Select the 3.0000 diameter, 5-flute carbide Face Mill

Click OK to return to the Tool Information tab

A tool number has already been assigned to this tool in the library. For this operation, you will specify that this is tool number 1. The next operation will use tool 2 and so on.
All other parameters should display as shown above. If your parameters do not match, please set them according to the dialogs shown in this project.
The tool path is generated according to the parameters in the dialog. SURFCAM will prompt you to either accept or reject the tool path. The tool path will be saved only if you accept it.

To make it easier to see each toolpath as it is created, you will use the Operations Manager to hide the display of a toolpath before you begin creating the next.
11. Click Accept

12. On the Main toolbar, click Operations Manager

13. Click Hide Toolpath and then click Done
Step 2: Mill the pockets and profile

TrueMill™ is Surfware’s high-speed machining solution for any machine tool. TrueMill™ enables the tool to maintain higher speeds because all abrupt direction changes are replaced with smooth, tangent tool motion. Smooth high-speed repositioning moves also replace the rapid moves, allowing the machine to stay in the high-speed mode continuously during the entire cycle. Because of these enhancements, toolpath feed rates can be increased and cycle times reduced.

Tool Engagement Angle

The tool engagement angle (TEA) is a key setting in TrueMill. SURFCAM creates the toolpath so that the actual engagement angle of the tool never exceeds the value you specify. The Tool Engagement and Maximum Stepover settings work together. Set one to dynamically calculate the other. For example, if you are using a 1-inch end mill and set Maximum Stepover to 0.5, Tool Engagement calculates to 90 degrees. If you set Tool Engagement to 90, Maximum Stepover calculates to 0.50.

For this project, you will use a \( 120^\circ \) tool engagement angle. The part has a total depth of 1 inch. You will use a \( 1/2\text{-inch} \) maximum depth of cut using a \( 3/4\text{-inch end mill} \). Other settings you will use include:

Minimize Plunges

In a pocket, the tool may isolate itself in one area. The tool may retract and move to the next area and plunge to continue the pocket. Check Minimize Plunges to reduce the number of retract and plunge moves.

Final Finish Cut Overlap

When lead-in and lead-out moves share the same location, the tool may leave an undesirable mark. When Final Finish Cut Overlap is used, the final finish pass continues past the start location by the value you enter before the lead-out move takes place.
On the NC menu, click 2 Axis and then click TrueMill

Click Material

Create a chain on the stock profile

Since you will be machining the area between the part profile and the stock profile, you must define the Material boundary for this operation. Then you will select the part boundaries you want to machine. To create the part boundaries as quickly as possible, you will use the Auto option.
Select one element on each pocket and the outer part profile.

Click Done on the Select toolbar.

Click Done on the Select Chain toolbar.

Click Select Tool.
Get Started with SURFCAM

Select the .75 inch, 4-flute end mill and click OK

Enter 2 for the Tool Number, Length Offset, and Diameter Offset
12. Click the Common to > tab

13. Enter a Depth of 1.0

14. Select Minimize Plunges
The rough passes will leave 0.01 stock on the walls and no stock on the floors. You will then finish only the walls.
Open the Operations Manager

19

Accept the toolpath

20

Hide the toolpath and click Done
Step 3: Drill the holes

A single drilling operation in SURFCAM can include multiple tools, such as spot drill, drill and tap, with separate drilling properties for each tool. These tools and their unique machining properties are used to create a hole process. A hole process can be saved and used for future drilling operations.

Masking

Masking provides control over the selection of elements by eliminating specific attributes and/or elements from selection. This lets you work more easily with the elements of the drawing that are not masked.

To make it easier to select the holes for drilling, you will mask from selection any circles that do not have a diameter of $0.375$ inch. Instead of typing the diameter values manually, you can select a hole in the graphics area to extract the diameter information.

On Main Toolbar 2, click the arrow to the left of the Mask icon.
2. Click More to expand the masking options.

3. Click the selection arrow for Min diameter under Circle.
Select one of the drilled holes in the graphics area to enter the diameter information in the dialog.

Click OK to enable the Mask Selection Settings.
Create the Hole Process

For this project, you will first spot drill, then drill, then countersink all of the 3/8-inch holes using a single hole process. You will use SURFCAM’s hole sorting capabilities to calculate the shortest toolpath length before you create the final toolpath.

1. On the NC menu, click 2 Axis and then click Drill

2. Click Visible to select all visible holes with a 0.375 diameter

3. Click Done
Get Started with SURFCAM

4. Click Select Tool

5. Select the .5000 diameter, 120 degree Spot Drill and click OK
Click Add After

Click Select Tool

Click the Drill tool type

Click 1/8 to narrow the tool search

Select the .3750 diameter Drill and click OK
11. Click Add After

12. Click Select Tool

13. Click the Countersink tool type

14. Select the .7500 diameter, 90 degree Countersink and click OK
Select each tool in the list and enter the appropriate Tool Number and Length Offset as shown.

<table>
<thead>
<tr>
<th>Tool</th>
<th>Cycle</th>
<th>Total Depth</th>
</tr>
</thead>
<tbody>
<tr>
<td>3</td>
<td>Spotdrill</td>
<td>0.10825</td>
</tr>
<tr>
<td>4</td>
<td>Drill</td>
<td>0.10825</td>
</tr>
<tr>
<td>5</td>
<td>Countersink</td>
<td>0.28125</td>
</tr>
</tbody>
</table>

- **Tool Number**: 5
- **Length Offset**: 5
16. Click the Cycle Control tab

17. Select the Spot Drill to display its cycle properties

18. Enter 0.125 for Spot Drill Diameter

19. Select the Drill

20. Set Depth to: Shoulder

21. Enter 1.0 for Hole Depth
Next you will sort the holes in a variety of ways to determine which hole pattern produces the shortest possible toolpath.

- **Manual Sorting** arranges the holes in the same sequence they were created or, if holes were selected using the Single option on the Select menu, in the same sequence the holes were selected.
- **Shortest Distance** arranges holes by the shortest total distance.
- **Orthogonal Sorting** arranges holes vertically and horizontally according to the Bidirectional and Use Band Widths options.
- **Reverse Direction** reverses the order of the hole pattern after a tool change. Otherwise, the hole pattern is drilled in the same order each time. This setting only applies when multiple tools are used.
With Orthogonal Sorting selected, click Calculate Toolpath Length.

Select Shortest Distance and then click Calculate Toolpath Length.

Select Reverse Direction and then click Calculate Toolpath Length.

This is the shortest toolpath length, so click OK to create the toolpath.
Accept the toolpath

Open the Operations Manager and hide the toolpath

Click the Mask icon to turn masking off
Step 4: Chamfer the edges

Chamfering is useful for removing burrs, trimming sharp edges, or adding a relief angle. For this project, you will add a 0.075-inch 45° chamfer around the top edge of the outer part profile using a 0.25-inch chamfer tool.

1. On the NC menu, click 2 Axis and then click Chamfer Mill

2. Create a chain on the part profile and then click Done
Select the 0.2500 diameter, 45 degree Chamfer tool and click OK

Enter 6 for the Tool Number, Length Offset, and Diameter Offset
5. Click the Cut Control tab

6. Enter 0.075 as the Chamfer size

7. Click the Chamfer icon to change the measurement of the chamfer to be vertical

8. Click OK

Next you need to indicate which side of the profile will be chamfered. The tool should travel along the outside of the profile.
Click outside the profile to indicate the chamfer side

Accept the toolpath
Step 5: Verify the tool path

Verification lets you dynamically simulate the toolpath before you output the NC code to the machine. Before verifying the toolpath you just created, you need to create the simulation stock. Since you already have geometry that defines the stock profile, SURFCAM can automatically extract the size and location data from the geometry. You only need to enter the stock depth. For this project, the actual part is 1 inch thick and the stock is 2 inches thick.

1. Open the Operations Manager
2. Right-click on Setup One and select Edit Setup Information from the menu
Click the Stock tab

Set Stock Type to Box

Click the Geometry Bounding Box button

Enter -2.0 for the Corner 1 Z location

Click Add and then click OK
The Verification toolbar is now displayed. When different tools are used in one setup, it can be an advantage to assign a different color to each tool so you can see exactly where material removal is occurring. You can easily change a tool's color before you verify the toolpath.
Click Edit Tool List

9

Double-click on a tool color and then choose a new color

10

Click the Drill Tools tab to change the colors of the drill tools

11

When finished, click OK

12
Click Play to start the verification

To exit the verification, click Toolpath Verification on the Main toolbar
Step 6: Create a setup sheet

You can quickly create a setup sheet for the shop floor that can be viewed with any Web browser. Tooling information (with or without graphics), operation information, or both can be output with the click of a button. The pages are created in the same folder as your part file.
Set the dialog as shown and click OK.

An operations list and a tooling list are automatically created and displayed.

**OPERATIONS LIST**

<table>
<thead>
<tr>
<th>Date: Thu May 11 2006</th>
</tr>
</thead>
<tbody>
<tr>
<td>Time: 10:43:45</td>
</tr>
<tr>
<td>Output Filename: INCSetup_One.INC</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Tool Number</th>
<th>Operation</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Zaxis Face Mill</td>
</tr>
<tr>
<td>2</td>
<td>Zaxis TrueMill Pocket</td>
</tr>
<tr>
<td>3</td>
<td>Zaxis Hole Process</td>
</tr>
<tr>
<td>4</td>
<td>Zaxis Hole Process</td>
</tr>
<tr>
<td>5</td>
<td>Zaxis Hole Process</td>
</tr>
<tr>
<td>6</td>
<td>Zaxis Chamfer Mill</td>
</tr>
</tbody>
</table>

**TOOLING LIST**

<table>
<thead>
<tr>
<th>Description</th>
<th>3.0000 dia - 5 flt - Carbide Face Mill</th>
</tr>
</thead>
<tbody>
<tr>
<td>Tool Number</td>
<td>1</td>
</tr>
<tr>
<td>Length Register</td>
<td>1</td>
</tr>
<tr>
<td>Diameter Register</td>
<td>1008</td>
</tr>
<tr>
<td>Diameter</td>
<td>3.0000</td>
</tr>
<tr>
<td>Corner Radius</td>
<td>0.0313</td>
</tr>
<tr>
<td>Flute Length</td>
<td>0.3200</td>
</tr>
<tr>
<td>Total Length</td>
<td>1.7500</td>
</tr>
<tr>
<td>Number of Flutes</td>
<td>5</td>
</tr>
<tr>
<td>Program Point</td>
<td>Tip</td>
</tr>
</tbody>
</table>
Step 7: Generate the NC code

As a last step in this project you will generate the NC code for the created operations by running the post processor. You can either post out a single operation or multiple operations that are under a setup. You can use any post from the list of currently installed post processors.

You will post process all the operations that have been created in this project using the standard FANUC 15M-B post processor:

1. Open the Operations Manager
2. Select Setup One
3. Select FANUC
4. Click Post
Now the generated code is shown in the SURFCAM NC Editor (EditNC).

```
%0
(Surfware, Inc.)
(5703 Corsa Ave)
(Westlake, Ca. 91311)
G17 G40 G80 G90
T1 M6
M3 S299
G00 G54 X-1.5569 Y12.3687
G43 Z1. H1
M8
G00 Z-0.01
G01 X13.5559 F6.0
G00 X13.6
Y-0.3688
X13.5569
G01 X-1.5559
G00 X-1.6
Y11.2688
G01 X13.6
G00 Y0.7312
G01 X-1.6
G00 Y10.1687
G01 X13.6
G00 Y1.8312
```

You may now save the NC file and transfer it to the CNC machine.
Project 3 • Design a 3D Part

SURFCAM offers surfacing tools for all your 3D design needs—from simple fillets to turbine blades to complex blends. Use these tools to create your own 3D parts or modify existing part models. This project shows you how to create and modify 3-dimensional surfaces in a variety of ways to design an MP3 player.

1. Create the dial surfaces
2. Create the display area
3. Add the cover
4. Trim the surfaces
5. Add the buttons
Creating and modifying surfaces in SURFCAM

Surfaces represent the 3-dimensional shape of a part. In SURFCAM, surfaces are displayed using U and V flow lines that show the outer boundary and inner shape of the surface. Because surfaces are used to describe complex 3D shapes, each surface has a direction arrow that serves two purposes. First, it determines which side of the surface will be used for filleting, offsetting, single surface machining, and cutter intersections. Second, it determines the direction of the cutter path for single surface machining operations.

Creating surfaces

Surfaces are created using commands on the Create > Surface toolbar.

Create > Surface toolbar

SURFCAM supports the following types of surfaces:

<table>
<thead>
<tr>
<th>Icon</th>
<th>Command</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Points</td>
<td>Creates a surface through a set of points</td>
</tr>
<tr>
<td></td>
<td>Cross Section</td>
<td>Creates a surface from a set of contours</td>
</tr>
<tr>
<td></td>
<td>Drive Curve</td>
<td>Creates a surface by sweeping one or more contours along a curve</td>
</tr>
<tr>
<td></td>
<td>Offset</td>
<td>Creates a surface by offsetting an existing surface</td>
</tr>
<tr>
<td></td>
<td>Fillet</td>
<td>Creates a constant or variable fillet surface tangent to two surfaces</td>
</tr>
<tr>
<td></td>
<td>Primitives</td>
<td>Creates a cylinder, cone, sphere, torus, wedge, cuboid or planar surface</td>
</tr>
<tr>
<td></td>
<td>Revolution</td>
<td>Creates a surface by rotating a contour around an axis</td>
</tr>
<tr>
<td></td>
<td>Extrude</td>
<td>Creates a surface by extruding, or pushing, a profile along a vector</td>
</tr>
<tr>
<td></td>
<td>Blend</td>
<td>Creates a surface between two to four curves or surfaces</td>
</tr>
<tr>
<td></td>
<td>Trim Plane</td>
<td>Creates a planar surface at the edge of an existing surface or within a closed profile</td>
</tr>
</tbody>
</table>

The Options command lets you control how many U and V curves are used to display the surface, the surface tolerance, and options for controlling drive curve surfaces.

Modifying surfaces

Surfaces can be modified using standard commands on the Edit menu such as Trim/Break or Transform or by using commands on the Edit > Surfaces toolbar.
Edit > Surfaces toolbar

The following commands give you more control over surface arrows and surfaces:

<table>
<thead>
<tr>
<th>Icon</th>
<th>Command</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Side</td>
<td>Side</td>
<td>Changes the surface arrow to the other side of the surface</td>
</tr>
<tr>
<td>Direction</td>
<td>Direction</td>
<td>Changes the surface arrow to point in the opposite direction</td>
</tr>
<tr>
<td>Corner</td>
<td>Corner</td>
<td>Changes the location of the surface arrow so the machining will start at a different corner of the surface</td>
</tr>
<tr>
<td>Arrow</td>
<td>Arrow</td>
<td>Hides or displays the surface arrow</td>
</tr>
<tr>
<td>Twist Vectors</td>
<td>Twist Vectors</td>
<td>Changes the curvature within a surface without affecting the surface boundary</td>
</tr>
<tr>
<td>Polygon</td>
<td>Polygon</td>
<td>Creates a polygon on an existing NURBS or parametric surface</td>
</tr>
<tr>
<td>Display</td>
<td>Display</td>
<td>Changes the number of U and V flow lines used to display the surface</td>
</tr>
<tr>
<td>Align</td>
<td>Align</td>
<td>Aligns the surface arrows of one or more surfaces automatically</td>
</tr>
<tr>
<td>Untrim</td>
<td>Untrim</td>
<td>Untrims a previously trimmed surface or surface edge</td>
</tr>
<tr>
<td>Color</td>
<td>Color</td>
<td>Changes the color of surfaces to help separate models that consist of inner and outer surfaces. SURFCAM determines which surfaces are above, or outside, others and colors them accordingly</td>
</tr>
<tr>
<td>Check</td>
<td>Check</td>
<td>Detects defective or invalid surfaces and optionally changes their color or moves them to a different layer</td>
</tr>
</tbody>
</table>

Before you begin

Make sure your unit of measure is set to Inch before you begin this project.

To set the system unit to Inch:
1. On the Tools menu, click Units.
2. Select Inch.

Open the project file
1. On the Main toolbar, click Open.
2. Open the file GetStarted 3D Part-1.SCPRT
3. On the Main toolbar, click Shaded with Wireframe.
For this project you will learn how to create the following surface types:
- Cross Section
- Drive Curve
- Fillet
- Extrude
- Blend

For details about other commands on the Create > Surface menu, please refer to SURFCAM Help.

**Step 1: Create the dial surfaces**

This project file already contains the geometry you need to begin creating surfaces. You will start by extruding the circle for the dial along the **Z-axis by 0.50 inch**.

To extrude a surface, you need a profile to extrude and a vector to extrude along. SURFCAM lets you extrude along the X, Y, or Z axis or you can define your own axis.

When a profile is extruded, you also have the option to cap either end of the extrusion with a planar surface. You can cap the start of the extrusion, the end, or both. In this step, you will cap the start (or bottom) of the extrusion.

After the profile is extruded and capped, you will create a fillet surface with a radius of **0.05 inch** between the two surfaces.
3. Click Keyboard

4. Select Z-Axis and click OK

5. Change Extrude Length to 0.50, set Cap Surface to Start and click OK

6. Click the circle twice to chain the geometry

7. Click Done to create the surfaces
Next you will create a fillet surface at the bottom of the dial opening. First, you need to change the side of the surface arrows so they point towards the inside so that the fillet will be created on the inside of the surfaces. In the illustration below, you can see that the arrows are currently pointing towards the outside.

**Note:** The direction of the surface arrows may differ depending on how the chain was selected. Therefore the surfaces that you created may have a different direction than that shown below. Both arrows should point towards the inside before the fillet is created.

8. On the Edit menu, click Surfaces

9. Click Side
Select each surface to change the arrow side to point towards the inside as shown.

Now you are ready to fillet the surfaces.

On the Create menu, click Surface.
12 Click Fillet

13 Change Fillet Radius to 0.05, set Trim Surface to Both and click OK

14 Select the two surfaces to create the fillet
Step 2: Create the display area

In this step you will use a Drive Curve surface to create the walls of the display area and a Trim Plane surface for the floor.

For a Drive Curve surface, you need two profiles. The first profile provides the path, or drive curve, for the second profile, which describes the shape of the final surface. You also need to identify an attachment point on the drive curve that controls where the surface will start.

To create a Trim Plane surface, you need a single planar profile.

1. Click Drive Curve

It will be easier to select the correct elements if you zoom the display. You can either click the Zoom In icon on the Main toolbar or you can use the scroll wheel on your mouse to dynamically zoom in and zoom out. Simply position your cursor in the center of the area you want to zoom and roll the scroll wheel backward to zoom in or forward to zoom out.

2. Click the bottom profile twice to create a chain for the drive curve

3. Click the ends of the elements shown to create a chain for the cross section
4. Click End Point

5. Select the end point of the cross section

6. Click Done
Next you will create the planar surface at the bottom of the display opening.

1. Click **Trim Plane**
2. Click **Chain**
3. Click the bottom profile twice to chain the geometry
4. Click **Done** to create the surface
5. Click **Go Back to Prev. Toolbar**
Step 3: Add the cover

Next you will create the cover with a Cross Section surface. For this type of surface you need at least two profiles in the major (primary) direction and at least two profiles in the minor (secondary) direction to form a grid. The more profiles you use, the more accurate the surface.

1. Click Cross Section

2. First click Manual and then click Grid

You are prompted to select cross-section elements in the major direction. You will select the three elements that form the longest shape of the cover. You will use the Single command to select the elements one by one.

3. Click Single and then select the three elements shown

4. Click Done to confirm the selection of major elements
You are now prompted to select elements in the minor direction. You will select the two elements at the front and the back of the cover.

5. Click Single and then select the two end profiles.

6. Click Done to create the surface.

Select beginning element of cross section - 3 in Minor direction or press 'done'.
You will finish the cover by creating planar surfaces over the ends. You can use an open or a closed contour to create a planar surface.

7. Click Go Back to Prev. Toolbar

8. Click Trim Plane

9. Make sure Cap is selected and then select the two end profiles
Step 4: Trim the surfaces

Now you will trim the surfaces to each other to create the openings in the cover for the dial and the display area. You can trim surfaces just like any other geometry using the Trim/Break command on the Edit menu.

1. On the Edit menu, click Trim/Break

2. Click Trim 2. This button toggles between Trim 1 and Trim 2

3. On the Main toolbar click Wireframe and then press Ctrl + 5 to change to the right side view

4. Select the cover surface and the dial surface in the areas you want to keep
Select the cover surface and the display surface in the areas you want to keep.

Change back to a shaded view and press Ctrl + 7 to change to the isometric view.
Step 5: Add the buttons

The last step in this project is to add the buttons to the MP3 player. For this step, you will use geometry and surfaces that are currently invisible on separate layers.

First you will trim the cover to create openings for the two buttons. Then you will use a Blend surface to create the bottom of the button.

Finally, you will use the button surfaces on the left side of the player to create a mirror copy on the right.

1. Click Set Layer
2. Click the “I” next to Trim Curves to change it to “V” (Visible), then click OK
3. Press Ctrl + 1 to change to the top view
On the Edit menu, click Trim/Break and then click Trim 1

Select the cover surface in the area you want to keep

Click Multiple
7. Click Within and then draw a rubber band box around the button geometry.

8. Click Done to trim the cover.

9. Click Set Layer and make the Buttons layer visible, then click OK.
The Blend command creates a surface blended between two to four curves or surfaces. In this example, you will use the splines on the lower edges to create the blend surface. To prevent the selection of surfaces by mistake, you will mask all surfaces.
12. Click the arrow to the right of the Mask icon, uncheck Surface and click OK.

13. Select the four splines around the bottom edge of the button.

14. When prompted by the Method dialog, select Bicubic and click OK.
The last step is to copy and mirror the button surfaces to the right side of the MP3 player.
Next you need to define the mirror line. You will use the end points of the vertical element in the center of the player.
This project is now complete. The finished MP3 player is shown below.
An excellent surface finish is of critical importance in surface milling applications. SURFCAM provides the tools that let you rough your parts in such a way that finish passes produce optimal results. A wide variety of milling operations are provided that let you machine your parts your way. This project shows you how to generate optimized roughing passes using multiple tools in one easy step. A consistent step height is produced across all surfaces before you proceed with the pre-finishing and finishing operations.

1. Rough and pre-finish in one step
2. Finish the camera body
3. Finish the lens housing
4. Trace the blend fillets
5. Verify the toolpath
SURFCAM 3 Axis Milling

SURFCAM generates optimized 3-axis toolpaths on any surface.

Typical 3 Axis Workflow

The creation of most 3 Axis operations follows the same basic workflow.

1. On the **NC** menu, click **3 Axis** and then click the type of operation you want to create.

   ![3 Axis Menu](image)

2. The **Select** toolbar is displayed and SURFCAM prompts you to select surfaces to cut across.

3. If an operation was selected that includes roughing passes (SRM, RapidRough, Z Rough, Plunge Rough), you must define the size and shape of the initial stock using the Material Information dialog.

   ![Material Information](image)

4. On the **Tool Information** tab, select a cutting tool and material, then set speeds and feed rates.

5. On the **Cut Control** tab, define the machining parameters for this operation.
6. Depending on the type of operation, additional tabs with parameters may be available.
7. When finished, click **OK**.
8. Accept or reject the new toolpath.

**Before you begin**

Make sure your unit of measure is set to Inch before you begin this project.

**To set the system unit to Inch:**
1. On the **Tools** menu, click **Units**.
2. Select **Inch**.

**Open the project file**
1. On the **Main** toolbar, click **Open**.
2. Open the file **Get Started 3D Milling-1.SCPRT**
3. On the **Main** toolbar, click **Shaded with Wireframe**.

For this project you will learn how to use the following 3 Axis commands:
• RapidRough
• Planar
• 3D Offset
• Pencil Cut

For details about other commands on the NC > 3 Axis menu, please refer to SURFCAM Help.
Step 1: Rough and pre-finish in one step

RapidRough optimizes 3D material removal by automatically synchronizing multiple tools and incorporating TrueMill™ technology. TrueMill is the only toolpath generator that intelligently controls the tool’s engagement with the material to dynamically manage tool load and enable the use of extremely aggressive machining parameters.

RapidRough technology is a single step roughing through pre-finishing operation for complex multi-surface parts. By using a synchronized multiple tool strategy, as explained below, optimal results are achieved. Up to eight tools can be used in a single RapidRough operation.

First tool

Rough from top to bottom at the largest tool’s maximum capable depth of cut.

The first tool then machines from bottom to top reducing its own steps in preparation for the next tool.

Second tool

Rough from top to bottom wherever the previous tool couldn’t fit.

The second tool then machines from bottom to top reducing its own steps in preparation for the next tool (or finishing operation).
For this project, you will use two tools: a 0.750 inch bullmill for the initial roughing phase and a 0.375 inch bullmill for the second roughing phase.

After all cutting passes are complete, a 0.020 inch stock allowance will remain on the surfaces with a uniform step height that will not exceed 0.050 inch.

Key RapidRough settings include:

• **Maximum Final Step Height**: This option on the Common\Setup tab controls the final step height after the RapidRough operation is complete. The final step height may be less than this height, but never more.

• **Minimum Stock Removal**: This option on the Cut Control tab prevents the tool from cutting minimal amounts of material on steps. These conditions typically occur on steep surfaces. The tool will skip any steps that have less than the Minimum Stock Removal amount.

• **Minimum Radius**: This option will prevent the cutter from making excessive small movements, especially in corners and narrow passages that are only slightly larger than the tool diameter. This option saves time and reduces tool wear by eliminating inefficient tool motion during a radial cut.
1. On the NC menu, click 3 Axis and then click RapidRough.

2. Click Visible.

3. Set Material Type to Extents Box and click OK.
4. Change XY Offset and Z Offset to 0.10

5. Click the Calculate Bounding Box button and then click OK

6. Set the Setup\Common tab as shown
Click the Tool 1 tab, then click Select Tool and choose the 3/4 inch Bullmill as shown.

Set the Tool Information tab as shown.
Set the Cut Control tab as shown.

Click the Add Tool button to add a second tool.
1 | Get Started with SURFCAM

11. Click Select Tool and choose the 3/8 inch Bullmill as shown.

12. Set the Tool 2 Tool Information tab as shown.

13. Set the Tool 2 Cut Control tab as shown.
Note: In this example, the display of rapid moves has been turned off. To control the display of rapid moves, click Options on the Tools menu. Select Display > Toolpath and then check or uncheck the Draw Rapid Moves option.
Open the Operations Manager and hide the new toolpath.
Step 2: Finish the camera body

Next, you will use a Planar operation to pre-finish all the surfaces of this part. The Planar operation creates parallel cuts across the surfaces based on a selected start point, a specified direction of cut and the side to machine.

For this project, you will use a **0.250 inch ball mill** and create cutting passes at a **45 degree** angle with a **0.030 inch** step increment between each pass.

1. On the NC menu, click 3 Axis and then click Planar
2. Click Visible
Click Select Tool and choose the 0.2500 diameter 4 flute Ball mill.

Set the Tool Information tab as shown.
Set the Cut Control tab as shown.

You are prompted to select a point where the cutting passes will begin. You will use snap points on the grid to indicate the start point for the toolpath, a second point that indicates the 45 degree angle for the cutting passes and a third point that indicates the side where machining passes will be created.
Click Sketch

Right-click on the graphics area and select Snap Grid

Notice that the cursor image has changed to indicate that you are in Snap Grid mode. You will now select two grid points that will define the 45 degree angle for the cutter path.

Select the toolpath start point

Select the point in the direction of cut
The first and second points form an imaginary line that the cutting passes will follow. Now you need to indicate the side where machining will take place.

11. Select the offset side as shown

12. Accept the toolpath

13. Open the Operations Manager and hide the new toolpath
Step 3: Finish the lens housing

You will use a 3D Offset operation to create finish passes on the lens housing. 3D Offset creates concentric offset cutting passes based on the shape of the outer contour of the part. The toolpath maintains a constant scallop height, for a consistent finish. This technique can be used to finish other areas of the camera as well.

For this project, you will use a **0.250 diameter ball mill** with a **0.0200** inch step increment to cut the lens housing from the **outside in**. You will do this by specifying a bounding curve that defines the shape of the cutting passes and is also used to contain them.

1. On the NC menu, click 3 Axis and then click 3D Offset

2. Click Visible
Note: You will use the same tool from the Planar operation. There is no need to select a new tool.

Set the Tool Information tab as shown
Set the Cut Control tab as shown

Click OK

You are prompted to select the beginning element for the bounding curves. You will use the outer circle of the lens housing.
6. Chain the outer circle by clicking twice on the circle.

7. Click Done to create the chain and process the toolpath.

8. Accept and then hide the toolpath.
Step 4: Trace the blend fillets

You will complete the machining of this part by using a small tool to finish all the blend fillets. The Pencil Cut operation removes material left by previous operations along the concave intersection of two surfaces, or material left in “crease like” regions in a single surface. Normally, in order for the Pencil Cut operation to remove material, a tool must be selected that has a diameter less than the one used in the previously performed operation.

You will use a **0.1875 diameter ball mill** for this operation. SURFCAM will use the diameter of the ball tool to calculate curves where the ball is in contact with two surfaces at once.

1. On the NC menu, click 3 Axis and then click Pencil Cut

2. Click Visible
Click Select Tool and choose the 0.1875 diameter 4 flute Ball mill.

Set the Tool Information tab as shown:

You will use the default settings on the Cut Control tab.
5 Click OK

6 Accept the toolpath
Step 5: Verify the toolpath

For 2 Axis and 3 Axis milling operations, you have the option to enable perfectly accurate verification, even when you zoom in on the model. When this option is ON, you must wait a moment when you Zoom or Pan during the verification. The speed of your computer processor affects performance.

With this option checked, the following verification buttons are disabled:

- Remove Fragments
- Tool Solid
- Tool Translucent
- Tool Wireframe
- Stock Solid
- Stock Translucent
- Stock Wireframe

1. On the Tools menu, click Options and then select Verification
2. Select True Solids Verification - 2/3 Axis Milling Operations and click OK
Next you will create stock for verification. You can automatically create a block of material based on the size of the part. For this project, you will create a bounding box around the part that is offset by 0.10 inch.
1. Change Offset to 0.10 and click Geometry Bounding Box.
2. Change Corner1 Z to 0.0 and then click Add.
3. Click OK.
Note: To more easily see the difference between each operation, click Edit Tool List and change the colors of the different tools. For details on how to edit tool colors, refer to Project 2: Mill a Production Part.
Appendix A: TrueMill™ Parameter Setup

TrueMill is designed to cut faster with greater depths of cut than traditional toolpaths. So how do you set the spindle speed, feed rate, tool engagement and other machining parameters to get the greatest benefit for your machining needs? Here is a simple technique to get started.

**Increase the Chip Load**

Start with your usual cutting parameters, but *double* the tooling manufacturer’s recommended chip load. Make straight line test cuts along the outside edge of the material.

**Increase the Spindle Speed**

As you make the test cuts, monitor the spindle load meter. If the load is less than the desired amount, incrementally increase the Spindle Speed while maintaining the increased chip load from step 1. The feedrate will therefore increase.

**Test Parameter Combinations**

Test combinations of increased Spindle Speed, deeper Depth of Cut, and larger Tool Engagement Angle until the desired spindle load is achieved. TrueMill typically allows you to safely utilize a spindle load of 100%.

**Update the TrueMill Operation Page**

Use the final optimal parameters on the TrueMill operation page every time you use that specific combination of machine, cutter and material—regardless of the shape of the part.
Appendix B: TrueMill™ Sample Testing Results

Use the information on the following pages as a guideline for improving your productivity with TrueMill™. Information is organized by common types of material, machine tools and cutting tools. These test results should be used for information purposes only. Your actual results will vary depending on the capabilities of your machine tool and the types of cutters you are using.
<table>
<thead>
<tr>
<th>Material</th>
<th>Machine Brand</th>
<th>Rated HP</th>
<th>Tool Type/Make</th>
<th>Tool Dia.</th>
<th># of Flutes</th>
<th>Spindle Speed</th>
<th>Feedrate (effective)</th>
<th>Axial DOC</th>
<th>TEA</th>
</tr>
</thead>
<tbody>
<tr>
<td>Aluminum 6061 T6</td>
<td>Chiron FZ15W Fanuc 18M</td>
<td>29</td>
<td>Iscar Carbide Endmill</td>
<td>.500&quot;</td>
<td>3</td>
<td>12000</td>
<td>855 IPM</td>
<td>.500&quot;</td>
<td>120°</td>
</tr>
<tr>
<td>DMG DMU 50 Heidenhain TNC530</td>
<td></td>
<td>17 (13 kW)</td>
<td>Iscar Carbide Endmill</td>
<td>12mm</td>
<td>3</td>
<td>10000*</td>
<td>15000 mm/min</td>
<td>9mm</td>
<td>122°</td>
</tr>
<tr>
<td>FADAL CAT 40 (year 1983)</td>
<td></td>
<td>20</td>
<td>Kennametal Carbide, uncoated, 37° helix</td>
<td>.500 in</td>
<td>3</td>
<td>10000*</td>
<td>400 IPM*</td>
<td>.500&quot;</td>
<td>101°</td>
</tr>
<tr>
<td>FADAL CAT 40 (year 1993)</td>
<td></td>
<td>20</td>
<td>DataFlute Carbide, uncoated, 37° helix</td>
<td>.500 in</td>
<td>3</td>
<td>10000*</td>
<td>525 IPM*</td>
<td>.500&quot;</td>
<td>100°</td>
</tr>
<tr>
<td>Haas VF2 SS</td>
<td></td>
<td>29</td>
<td>OSG Blizzard #204149</td>
<td>.500 in</td>
<td>3</td>
<td>12000*</td>
<td>756 IPM</td>
<td>.500&quot;</td>
<td>120°</td>
</tr>
<tr>
<td>Haas VF2 SS</td>
<td></td>
<td>29</td>
<td>Iscar Carbide Endmill</td>
<td>.500 in</td>
<td>3</td>
<td>12000*</td>
<td>840 IPM</td>
<td>.500&quot;</td>
<td>120°</td>
</tr>
<tr>
<td>Haas VF3 SS</td>
<td></td>
<td>29</td>
<td>OSG Blizzard #204149</td>
<td>.500 in</td>
<td>3</td>
<td>12000*</td>
<td>630 IPM</td>
<td>.500&quot;</td>
<td>120°</td>
</tr>
<tr>
<td>Haas VF3 SS</td>
<td></td>
<td>29</td>
<td>Iscar Carbide Endmill</td>
<td>.500 in</td>
<td>3</td>
<td>12000*</td>
<td>825 IPM</td>
<td>.500&quot;</td>
<td>120°</td>
</tr>
<tr>
<td>Haas VF4 (year 1997)</td>
<td></td>
<td>20</td>
<td>OSG Blizzard #204149</td>
<td>.500 in</td>
<td>3</td>
<td>10000*</td>
<td>300 IPM*</td>
<td>.500&quot;</td>
<td>120°</td>
</tr>
<tr>
<td>Haas VF5</td>
<td></td>
<td>30</td>
<td>Kennametal Carbide, uncoated, 37° helix</td>
<td>.500 in</td>
<td>3</td>
<td>7500*</td>
<td>400 IPM</td>
<td>.500 in</td>
<td>100°</td>
</tr>
<tr>
<td>Haas VM2</td>
<td></td>
<td>20</td>
<td>OSG Blizzard #204149</td>
<td>.500 in</td>
<td>3</td>
<td>12000*</td>
<td>1000 IPM*</td>
<td>.500 in</td>
<td>120°</td>
</tr>
<tr>
<td>Haas VM3</td>
<td></td>
<td>20</td>
<td>Iscar Finishred</td>
<td>12 mm</td>
<td>4</td>
<td>12000*</td>
<td>1000 IPM*</td>
<td>.500 in</td>
<td>120°</td>
</tr>
<tr>
<td>Hurco VMX 42</td>
<td></td>
<td>24</td>
<td>DataFlute Carbide Endmill</td>
<td>.500 in</td>
<td>3</td>
<td>10000*</td>
<td>500 IPM*</td>
<td>.500 in</td>
<td>95°</td>
</tr>
<tr>
<td>Material</td>
<td>Machine Brand</td>
<td>Rated HP</td>
<td>Tool Type/Make</td>
<td>Tool Dia.</td>
<td># of Flutes</td>
<td>Spindle Speed</td>
<td>Feedrate (effective)</td>
<td>Axial DOC</td>
<td>TEA</td>
</tr>
<tr>
<td>-------------------</td>
<td>---------------</td>
<td>----------</td>
<td>----------------</td>
<td>----------</td>
<td>-------------</td>
<td>---------------</td>
<td>----------------------</td>
<td>------------</td>
<td>-----</td>
</tr>
<tr>
<td>Aluminum 6061 T6</td>
<td>Mazak NEXUS VCN 510C</td>
<td>20</td>
<td>OSG Blizzard #204149</td>
<td>.500 in</td>
<td>3</td>
<td>12000*</td>
<td>420 IPM*</td>
<td>.500 in</td>
<td>120°</td>
</tr>
<tr>
<td></td>
<td>Mori-Seiki NV4000DCG (VMC)</td>
<td>25</td>
<td>Generic bargain endmill</td>
<td>.500 in</td>
<td>3</td>
<td>12000*</td>
<td>630 IPM</td>
<td>.500 in</td>
<td>120°</td>
</tr>
<tr>
<td></td>
<td>Mori-Seiki NH4000DCG (HMC)</td>
<td>25</td>
<td>Generic bargain endmill</td>
<td>.500 in</td>
<td>3</td>
<td>14000*</td>
<td>784 IPM</td>
<td>.500 in</td>
<td>120°</td>
</tr>
<tr>
<td>Steel A2 Rockwell 27-30</td>
<td>Cincinnati FTV 640</td>
<td>20</td>
<td>DataFlute SSST40250C1</td>
<td>.250 in</td>
<td>4</td>
<td>7154</td>
<td>65 IPM</td>
<td>.350 in</td>
<td>100°</td>
</tr>
<tr>
<td></td>
<td>Cincinnati FTV 640</td>
<td>20</td>
<td>DataFlute SSST40375C1</td>
<td>.375 in</td>
<td>4</td>
<td>5100</td>
<td>55 IPM</td>
<td>.400 in</td>
<td>100°</td>
</tr>
<tr>
<td></td>
<td>Cincinnati FTV 640</td>
<td>20</td>
<td>Helical 14317</td>
<td>.500 in</td>
<td>4</td>
<td>4200</td>
<td>55 IPM</td>
<td>.400 in</td>
<td>100°</td>
</tr>
<tr>
<td>Steel 1018</td>
<td>Kia VX500</td>
<td>20</td>
<td>DataFlute Carbide Endmill</td>
<td>.500 in</td>
<td>4</td>
<td>3100</td>
<td>50 IPM</td>
<td>.700 in</td>
<td>120°</td>
</tr>
<tr>
<td></td>
<td>Haas VM3</td>
<td>20</td>
<td>Iscar Finishred 12 mm</td>
<td>.500 in</td>
<td>4</td>
<td>12000</td>
<td>1000 IPM</td>
<td>.500 in</td>
<td>64°</td>
</tr>
<tr>
<td>Steel 1020</td>
<td>Haas VF2</td>
<td>15</td>
<td>Iscar Carbide Endmill</td>
<td>.500 in</td>
<td>4</td>
<td>7500</td>
<td>500 IPM*</td>
<td>.500 in</td>
<td>36°</td>
</tr>
<tr>
<td>Steel 1040</td>
<td>Haas VF2 SS</td>
<td>29</td>
<td>Iscar Carbide Endmill</td>
<td>.500 in</td>
<td>3</td>
<td>7500*</td>
<td>560 IPM</td>
<td>.500 in</td>
<td>27°</td>
</tr>
<tr>
<td>Steel 4140</td>
<td>Haas VM3</td>
<td>20</td>
<td>Iscar Finishred 12 mm</td>
<td>.500 in</td>
<td>4</td>
<td>9700</td>
<td>850 IPM</td>
<td>.500 in</td>
<td>64°</td>
</tr>
<tr>
<td>Steel 4142</td>
<td>Haas VF3</td>
<td>20</td>
<td>Iscar ECPI (Chip Splitter) E3L 500-1.0/1.5C500 IC900</td>
<td>12mm</td>
<td>3</td>
<td>5500</td>
<td>396 IPM</td>
<td>.375 in</td>
<td>45°</td>
</tr>
<tr>
<td>Material</td>
<td>Machine Brand</td>
<td>Rated HP</td>
<td>Tool Type/Make</td>
<td>Tool Dia.</td>
<td># of Flutes</td>
<td>Spindle Speed</td>
<td>Feedrate (effective)</td>
<td>Axial DOC</td>
<td>TEA</td>
</tr>
<tr>
<td>-----------------------</td>
<td>---------------</td>
<td>-------------</td>
<td>----------------------</td>
<td>-----------</td>
<td>-------------</td>
<td>------------------</td>
<td>----------------------</td>
<td>------------</td>
<td>-----</td>
</tr>
<tr>
<td>Steel X40CrMoV51</td>
<td>Tostec OPTIMA</td>
<td>37.5 (28 kW)</td>
<td>Walter FROH#16</td>
<td>32mm</td>
<td>4</td>
<td>2120</td>
<td>1272 mm/min</td>
<td>10mm</td>
<td>89°</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>Carbide inserts</td>
<td>32mm</td>
<td>4</td>
<td>4977</td>
<td>3285 mm/min</td>
<td>5mm</td>
<td>89°</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td>16mm</td>
<td>2</td>
<td>3006</td>
<td>956 mm/min</td>
<td>5mm</td>
<td>89°</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td>16mm</td>
<td>2</td>
<td>3006</td>
<td>1203 mm/min</td>
<td>5mm</td>
<td>89°</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td>16mm</td>
<td>2</td>
<td>3006</td>
<td>1203 mm/min</td>
<td>5mm</td>
<td>89°</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>Walter FROH#12</td>
<td>12mm</td>
<td>4</td>
<td>3185</td>
<td>1020 mm/min</td>
<td>4mm</td>
<td>89°</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>Carbide inserts</td>
<td>12mm</td>
<td>4</td>
<td>3185</td>
<td>1020 mm/min</td>
<td>4mm</td>
<td>89°</td>
</tr>
</tbody>
</table>
### Symbols
- 2 Axis workflow 62
- 3 Axis workflow 130
- 3D Offset 146

### A
- AutoTrim 32

### B
- Blend surface 121

### C
- Chain 40
- Chamfer 35
- Chamfer Mill 92
- Circle 37
- Create menu 22
- Cross Section surface 116
- CView Indicator 9

### D
- Direct Numerical Control (DNC) 19
- DirectX 2
- Drill 80
- Drive Curve surface 113

### E
- Edit menu 22
- Extrude surface 108

### F
- Face Mill 67
- Fillet 33
- Fillet surface 112

### G
- geometry 22
- Go Back to Prev. Toolbar 28

### H
- hardware acceleration 2
- Hole Process 83

### K
- Keyboard 28

### L
- Line 27

### M
- machining modes 14
- main menu 6
- Mask 80
- Mirror 127

### N
- NC menu 14
- NC Operations Manager 17

### O
- Offset 26
- Operations Manager 17

### P
- Pencil Cut 150
- Planar 141
- Point 51
- Points 49
- post processor 19
- prompt line 9

### R
- RapidRough 132
- Rectangle 26
- RS-232 19

### S
- Select Chain toolbar 63
- Select toolbar 64
- Set Layer 121
- Setup Section 17
- Setup Sheet 101
- SIM 2
- Sketch 59
- Software Interface Module 2
- Status toolbar 8
- SURFCAM toolbar 6
- SURFCAM Verify 18
- system requirements 2

### T
- toolbars
  - Status 8
  - SURFCAM 6
- tool engagement angle (TEA) 73
- Transform 57
- Trim/Break 32
- Trim Plane surface 115
- TrueMill 73
- True Solids Verification 153

### U
- user interface 5

### V
- Verify 18
- View Indicator 9