SmartCAMcnc

Exploring SmartCAM® Milling V11.5

Production Milling™

Advanced Milling™

FreeForm Machining™

Doc SC005-MRS

Originally Published: November 1997
Update Published: December 2004

SmartCAMcnc

Contents

2 Working with SmartCAM

3 Using Job Operations

4 Working with Elements

Exploring SmartCAM Advanced Milling

7 Using Profile Curves to Create Meshes

8 Creating Meshes

9 Editing Meshes

10 Creating and Editing Elements from Solids

11 Working with Surfaces

12 Using Planar Cuts and Cavity Roughing

Exploring SmartCAM FreeForm Machining

13 Using Profile Curves to Create Surfaces

14 Editing Surfaces

15 Machining Surfaces

Appendix A: Self-Test Answers

Glossary

Index

Welcome

Welcome to the exploration of SmartCAM Milling. Before you begin this exploration, take a few minutes to read through this introduction.

Overview

There are three sections to this course: Production Milling, Advanced Milling, and FreeForm Machining. You may need to use only one or all sections of the manual, depending on the course you are attending. However, you need to know Production Milling before learning the Advanced Milling and FreeForm Machining material.

Using Your Exploring Manual

This manual is designed to be used, not simply read. It is yours to keep. It is organized into units that cover different parts of the system. The individual lessons in each unit contain the following material:

- Lesson Objectives
- Feature Overviews
- Points to Remember
- Self-Tests
- Challenge Projects

Exploring Prerequisites

While SmartCAM is easy to understand and operate, you will benefit from having some experience with computers and machine tools. You will find it helpful to be familiar with Microsoft Windows and its corresponding commands:

You should also be familiar with these topics:

- Hard disk organization
- Computer Numerical Control (CNC) programming practices
- Basic machine tool terminology

Following the SmartCAM Milling Exploring Program

SmartCAM Milling exploring sessions are intended to give you a broad introduction to the SmartCAM Milling software features while helping you build a solid foundation of skills.

This will enable you to easily learn more advanced skills as you use SmartCAM, on your own, to accomplish specific work requirements.

Exploring Production Milling Objectives

- Become acquainted with SmartCAM.
- Work with SmartCAM.
- Use Job Operations.
- Work with elements.
- Generate and verify roughing toolpath.
- Use utilities.
- Generate code.

Exploring Advanced Production Objectives

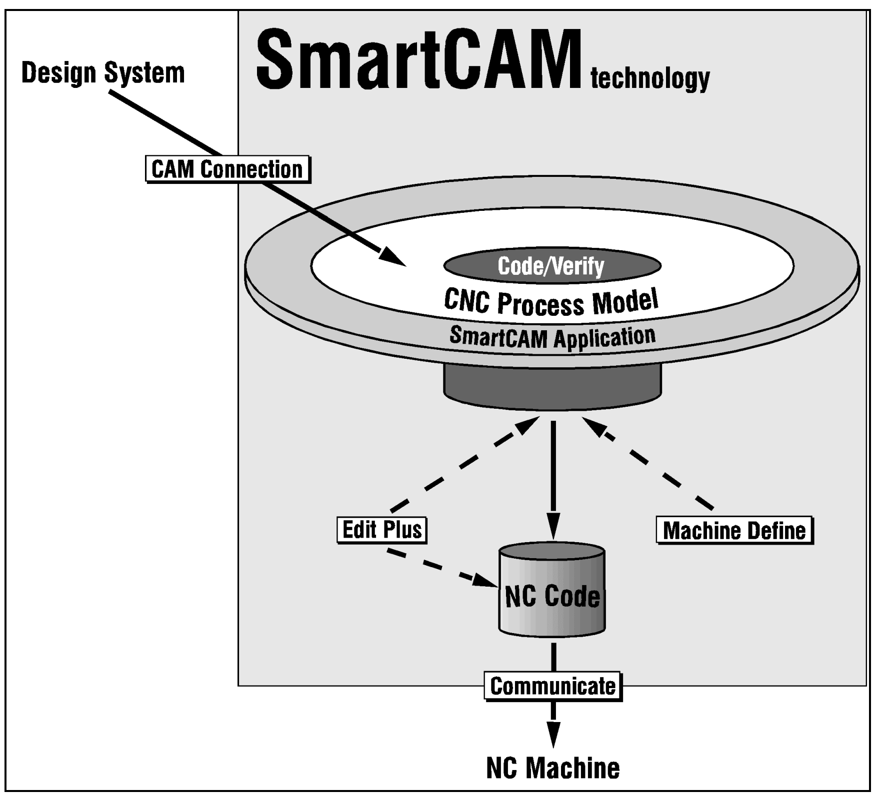
- Use profile curves to create meshes.
- Create different kinds of meshes.
- Edit meshes.
- Create and edit elements from solids.
- Work with surfaces.
- Make planar cuts and perform cavity roughing.

Exploring FreeForm Machining Objectives

- Use profile curves to create surfaces.
- Work with surfaces.
- Edit surfaces.
- Create and edit elements from solids.
- Machine surfaces.

Learning SmartCAM Technology

SmartCAM consists of several modules that work together to provide a single method for changing a design into machine code. These modules are shown in the following figure and are described in the next section.



SmartCAM CNC Process Modeling

The SmartCAM approach to CNC machining captures the mental model you have when you create a part and display it on your computer screen as a graphic model. This enables you to create, change, and interact with the model of the manufacturing process. This model is called the *CNC Process Model*. The part geometry and toolpath are incorporated as you build the model, and you can view the toolpath at any time. When the machining process is correct, you generate code directly from the model.

Considerations such as feeds and speeds, tool availability, fixtures, and machine idiosyncrasies are part of the model. When changes are required, you can easily revise the model and generate code again.

When you add a new machine, you simply select the new machine and template files and regenerate the code.

SmartCAM's Components

Each SmartCAM application is composed of two pieces that work together: a CNC Process Model and a job operations setup.

CNC Process Model

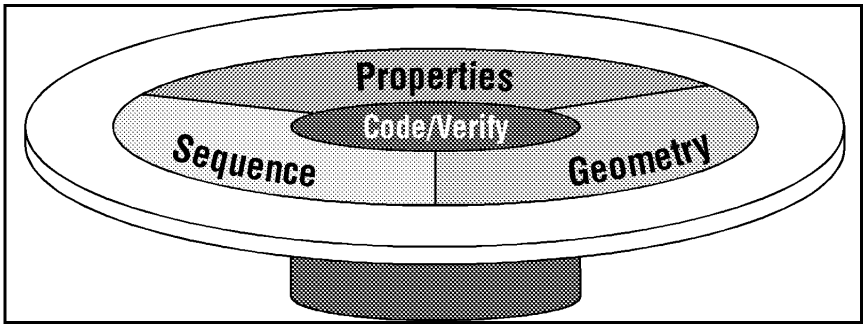
A SmartCAM CNC Process Model is a dynamic, sequential toolpath database. Part geometry is immediately converted into toolpath, sequenced in the way the machine will cut the part. Any changes you make to the model immediately update the database.

Because there is no separation of part geometry and toolpath, you do not need to wait until you finish creating the geometry to specify its sequence and properties. You can make changes at any time and instantly view the resulting tool motion.

As you build a CNC Process Model, you incorporate the following:

- **Sequence** (when the operation should occur)—the order that machining operations and toolpath become part of the model.
- **Properties** (how the operation should occur)—machining parameters, such as tool selection, depths, tool offset direction, and machine-control behavior, are assigned to the toolpath.
- **Geometry** (where the operation should occur)—elements defining the toolpath, such as linear or circular cutting, rapid traverses, and lead-in moves, are added to the model.

As you build and save a model, it is stored in a process model file. All process model files have a `.pm4` extension.



Job Operations Setup

An important part of the CNC Process Model is the information about the tools and operations you use to machine the part. This information, known as the *job operations setup*, is stored in the job operations file. All job operations files have a `.jof` extension. Each process model is linked to a `.jof` file.

The job operations setup links tools and operations together as *process steps*. As you develop a process model, you assign steps to the toolpath elements. Each step contains all the parameters for a specific tool and a specific operation. When you generate NC code, SmartCAM accesses the `.jof` file and uses the tool and operation parameters. This information can be printed and used by the machine operator to set up the machine.

Applications

Your SmartCAM application provides the graphic environment for creating a CNC Process Model and NC code. Use the various tools available in the application to define the process for machining the part. Change the sequence, properties, or geometry as needed, and view the results before you create code. When the model is complete, select the Code option from the Process menu to generate machine-ready NC code from the process model.

Edit Plus

Edit Plus is an ASCII text editor that you can use for a variety of editing tasks. Edit Plus's features include search and replace, repeat with axis increments, and absolute-to-incremental conversion. These features speed up the editing process.

Communicate

For Windows users, Communicate provides various ways to communicate with your CNC machine using RS-232 or parallel communication formats. You can punch a tape, read a tape, or send code directly to or from a machine's controller.

CAM Connection

CAM Connection translates files from CAD systems into information SmartCAM can use to create geometry for a CNC Process Model.

Machine Define

Machine Define specifies and configures a machine file that sets the parameters relating to your machine's code requirements. Machine (`.smf`) files for several popular CNC machines come with your SmartCAM system. If these do not fit your needs, you can easily modify them to do the job.

SmartCAM Visual CTK

SmartCAM Visual CTK is a drag-and-drop WYSIWYG (what you see is what you get) development environment that enables you to create dialog boxes and control panels for your macros. This helps you to integrate your customized productivity solutions into the SmartCAM interface.

Using the Documentation

Exploring SmartCAM Milling provides an overview of SmartCAM, describes the SmartCAM workplace environment, and includes exercises to help you get acquainted with your SmartCAM software.

Your SmartCAM documentation package also includes these documents:

- *Installation Guide for All SmartCAM Products* contains instructions on how to install the SmartCAM software and customize your SmartCAM display.

These documents can now be found online:

- *User Guide* provides information about the tools you use to create a CNC Process Model.
- *SmartCAM Edit Plus User Guide* describes how to use the SmartCAM text editor.
- S*martCAM Communicate User Guide* provides an overview of RS-232 communications and an explanation of how to use the Communicate utility.
- *SmartCAM CAM Connections* shows how to convert both CAD-prepared drawing files into SmartCAM CNC Process Model files and SmartCAM CNC Process Model files into CAD files.
- *SmartCAM CAM Connection Reference Manual* provides information about how the CAM Connection application translates CAD input files into CAM output files. It also contains information about customizing the CAM Connection environment and running the application non-graphically.

- *SmartCAM Code Generation Guide* provides in-depth information about generating NC code from your CNC Process Model, reference sections for machine files and template files, and an explanation of how to use the Machine Define utility.
- *SmartCAM Customization Manual* contains information on how to customize SmartCAM windows, hot keys, icons, and macro commands.

Using Online SmartCAM Manuals

SmartCAM manuals are available for use online. You can search for a specific term, see graphics that explain advanced applications, look for an explanation of a specific error message, and more. Online documentation is quick and easy to use, and it answers your questions as they come up while you are working at the computer. The online reader provides an opportunity to search all or any of the manuals at one time for words, phrases, even wild card expressions.

Using Online Help

You can find more information about a topic or an input field using these methods:

- Use context-sensitive Help by performing these tasks:
  a. Press **Shift +F1** to display a  cursor. Place the cursor anywhere, and click the left mouse button. A Help topic is displayed with an overview.
  b. To learn more about the topic you chose, select the underlined text for this topic in Help. A Help topic is displayed that has an overview and buttons for **Fields** and **How To**. Each input field is described in Fields. The How To provides operational information about how to perform a process.
- Use Search to find information about specific input fields. You enable Search when you press **F1**, select **Search**, and then enter the word to learn about.
- Use Glossary to find a word and how it is used. You can also press **F1** and select the title *Glossary*. The Glossary contains conceptual information.

Documentation Conventions

Information in SmartCAM manuals is presented in a consistent way, using the following conventions:

- Inch and metric formats are both given where appropriate. The metric measurement is enclosed in square brackets ([ ]) following the inch measurement, for example, 4.700 [120].

- Sample files are provided in both inch and metric formats. File names for metric models are similar to the corresponding inch file names except that they begin with the letter *m*. Metric files are stored in directories named with an `_m`. The metric parts are not exact conversions; they are similar parts with appropriate metric tooling.
- Points to Remember are at the end of every lesson. This provides an opportunity for you to review the important parts of each lesson.
- Self-Tests are at the end of Production MillingTurning units and at the end of Advanced Milling and Freeform Machining explorations. These tests enable you to check your understanding of the *Points to Remember.* Some self-tests contain *Challenge Projects,* which enable you to apply your new skills in a challenging and practical project.
- Commands are presented exactly as you should enter them. Be sure to include all spaces.
- If your keyboard has a Return key, press that key whenever instructed to press Enter.
- Combinations of keys appear with a plus (+) between the keys; for example, **Alt+4** or **META+4** means you should hold down the ALT or META key and press the **4** key at the same time.
- On some keyboards, the META key is labeled EXTENDED CHAR.
- Selections you should make to move through a procedure appear in **bold typeface**.
- A series of selections used in directions will appear as in the following example:

  Select **Edit—Geo Edit—Blend**.

  This tells you to select **Edit** from the top menu bar, **Geo Edit** from the Edit menu, and then **Blend** from the list of Geo Edit modeling tools.

Exploring SmartCAM Production Milling

Exploring SmartCAM Production Milling

Welcome

Production Milling is a highly productive solution for applications that require 2 1/2-axis machining with fourth axis positioning. As the first tier in the SmartCAM family of milling applications, it offers the functionality you need today, plus an easy, long-term growth path for your future needs. Like all SmartCAM applications, it does more than reduce your programming time. It helps you improve your machining processes and move your products to market faster.

Units in This Exploration

- Becoming Acquainted with SmartCAM
- Working with SmartCAM
- Using Job Operations
- Working with Elements
- Generating and Verifying Roughing Toolpath
- Generating Code

1

Becoming Acquainted with SmartCAM

Overview

It is important for you to become acquainted with the SmartCAM environment before you perform Milling operations. This unit introduces you to the basic SmartCAM workplace, terms, files, and view manipulation techniques.

Lessons for This Unit

- Learning the SmartCAM Workplace
- Working with Files
- Manipulating the View

Learning the SmartCAM Workplace

Objectives

This lesson shows you how to perform these tasks:

- Start SmartCAM.
- Identify and define SmartCAM workplace areas.
- Identify and define SmartCAM controls.

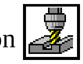
Overview

You can use the menu bar to access most of the tools in SmartCAM. You can select items in the menu bar to open pull-down menus. When you select an item, a list of items is displayed that enables you to open toolboxes, dialog boxes, and submenus. When you choose a toolbox, its title is displayed on the workbench. Depending upon your screen resolution, the last three to five toolboxes you used are displayed on the workbench. A list of tools is displayed in the toolbox, which is below the workbench.

Starting SmartCAM

If SmartCAM is not installed on your computer, see the *Installation Guide for All SmartCAM Products* for directions on installing the software.

Perform these tasks to start SmartCAM:

1. Locate the **SmartCAM Production Milling** icon  in the SmartCAM program group.

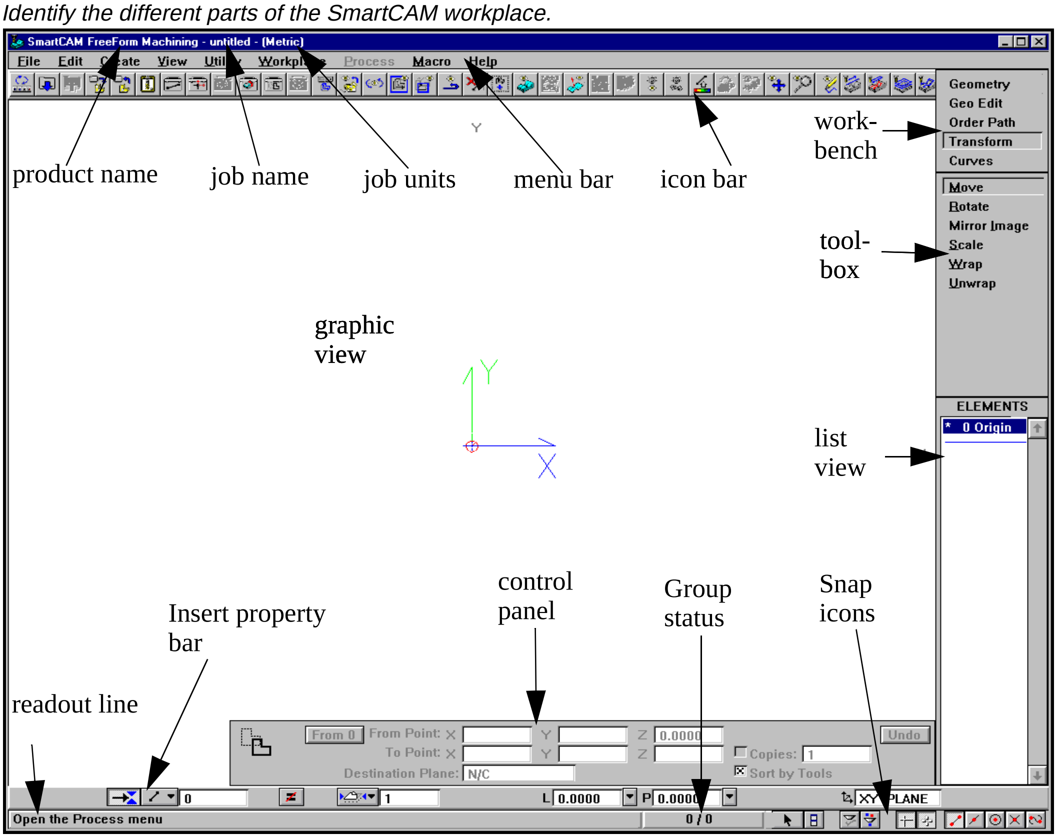
2. Use the left mouse button to select the icon (double-click in Windows NT 3.5.1 or single-click in Windows NT 4 or Windows 95).

The SmartCAM application window is displayed on the screen.

Using Workplace Areas

Many of the terms used to describe the SmartCAM workplace are based on common terms and procedures used in the CNC machining world.

Figure 1-1
Identify the different parts of the SmartCAM workplace.

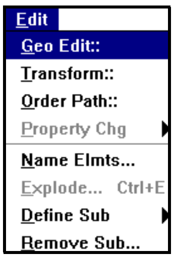


Menu Bar

The menu bar is displayed across the top of the SmartCAM workplace. Select items from the menu bar to open pull-down menus, from which you can open toolboxes, dialog boxes, or submenus. Take a few minutes to get acquainted with the layout of each pull-down menu.

Some menu choices are dimmed (a light shade of gray) and cannot be selected for one or more of these reasons:

- The job operations file is not active.
- The function requires an active group.
- The process model is empty (no geometry).
- The function requires a layer or a step to be active.



Pull-Down Menus

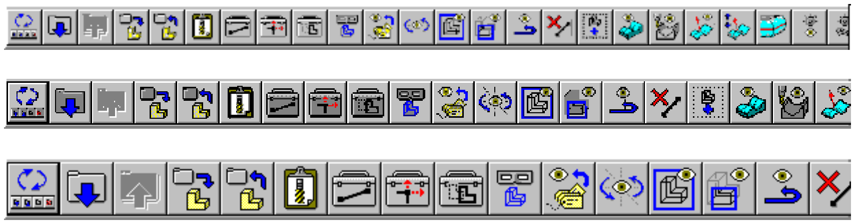
Pull-down menus are accessed through the menu bar. Perform these tasks to select from the Edit pull-down menu:

1. Use one of these methods to open the Edit pull-down menu:
   - Use the mouse to position the cursor over the Edit topic in the menu bar and press the left mouse button.
   - Use the keyboard by pressing the Alt key and the underlined letter (E) in the name of the menu item.
2. Slide the cursor down the list of items, and select the one you want. For example, select **Geo Edit** from the **Edit** pull-down menu. Notice that **Geo Edit** is displayed on the workbench, the tools for editing geometry are displayed in the toolbox, and the control panel for the highlighted tool is displayed at the bottom of the screen.

You can access these elements from pull-down menus.

- **Toolboxes** are indicated by a square made of four dots after the item name. When this item is selected, a list of tools is displayed below the workbench. If you select Geo Edit, a new list of tools is displayed on the side of the screen.
- **Dialog boxes** are indicated by an ellipsis (...) after the item name. When this type of item is selected, a dialog box is displayed. You can move the dialog box anywhere on the screen by placing the cursor on its title bar, pressing and holding the left mouse button, and moving the mouse. If you select a menu option, such as Edit—Name Elmts, a dialog box is displayed at the bottom of the screen.
- **Submenus** are indicated by a triangle after the item name. When a submenu item is selected, another menu is displayed. You can then select an item from this second menu. For example, if you select Edit—Property Chg, a submenu is displayed on the side.

Icon Bar



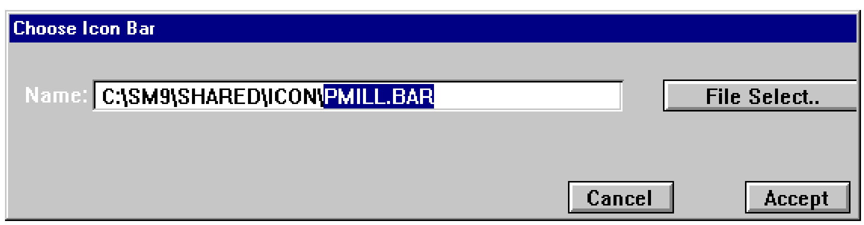
The icon bar provides shortcuts for accessing SmartCAM menus, macros, toolboxes, modeling tools, and functions. You can position it at different locations in the SmartCAM workplace, add icons to it, and size it to meet your needs. You have three size choices: small, medium, and large (represented above, respectively).

Selecting an Icon Bar

You can choose to display a different icon bar on the screen. Perform these tasks to select an icon bar to display:

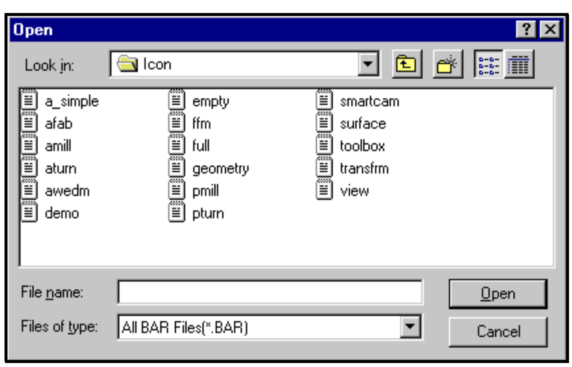
1. Select **Utility—Icon Bar—Choose**. The **Choose Icon Bar** dialog box is displayed.

Figure 1-2
Set the values on the Choose Icon Bar dialog box.



2. Select the **File Select** button to choose the icon bar to work with. The **Open** dialog box is displayed.

Figure 1-3
Set the values on the Open dialog box.



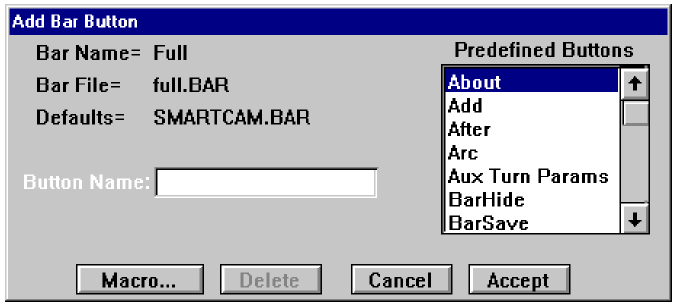
3. Select an icon bar file.
4. Select the **Open** button. The **Choose Icon Bar** dialog box becomes active.
5. Select the **Accept** button.

Adding an Icon to the Icon Bar

Perform these tasks to add an icon to the icon bar:

1. Select **Utility—Icon Bar—Button Add**. The **Add Bar Button** dialog box is displayed.

Figure 1-4
Set the values on the Add Bar Button dialog box.



2. Select the icon to add from the **Predefined Buttons** list.
3. Select the **Accept** button.

Note Shift-clicking an existing or blank icon also enables you to add or replace an icon.

Configuring an Icon Bar

Perform these tasks to set the position and size of the icon bar:

1. Select **Utility—Icon Bar—Configure**. The **Configure Icon Bar** dialog box is displayed.

Figure 1-5
Set the values on the Configure Icon Bar dialog box.

2. Turn on a **Position** option switch to specify where on the screen to display the icon bar.

3. Turn on an **Icon Size** option switch to specify the size of the icons. The smaller the icon size, the more graphic view space is available.
4. Turn on the **Visible** on/off switch to make the icon visible.
5. Select the **Accept** button.

Viewing Large Icon Bars

The `.bar` files supplied with SmartCAM may contain more icons than you can display at certain combinations of screen resolution, bar position, and icon size. Perform these tasks to view the entire icon bar:

1. Select **Utility—Icon Bar—Configure**. The **Configure Icon Bar** dialog box is displayed.
2. Set the **Position** option switch to **Float**.
3. Select the **Accept** button.

   Note If you have not used the floating icon bar before, it will appear with only six icons visible.

4. Move your cursor over the edge of the icon bar.
5. When your cursor changes to a double arrow shape, click with your left mouse button and drag the icon bar box out until all the icons are displayed.

Editing an Icon Bar

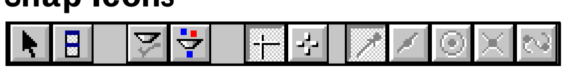
You can change any of the icons that are displayed on the icon bar. One way is to open your `.bar` file in a text editor and edit it in that way. However, you can perform these tasks to edit your icon bar from within SmartCAM:

1. Shift-click on an icon. The **Edit Icon Bar** dialog box appears.

   Note This procedure replaces the selected icon with the newly selected icon. The original icon will not be shifted over, it will be replaced.

2. Select the name of the icon you want to display in its place.
3. Select the **Accept** button.

Snap Icons



Snap icons enable you to use selection modes and point settings to enter coordinate values in input fields and snap to existing geometry.

Snap Selection Modes

Snap uses these selection modes:

- **Snap mode** requires that you activate the Snap Mode icon . When Snap mode is on, the pointer is displayed as a short cross hairs. Use the Snap Point Settings icons, which are on the readout line, to set the active point settings and then snap to an element.

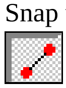
  Note If no snap points are found within the element picking distance, Snap returns the coordinates of the last point that you selected.

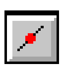
- **Free Coordinate mode** requires that you activate the Free Coordinates Mode icon . When Free Coordinate mode is on, the pointer is displayed as a large cross hairs. As you move the cross hairs across the graphic view, the coordinates are displayed on the readout line. The Linear Increment setting in the Increment dialog box determines the interval at which coordinates are updated.

- **Automatic mode** requires that you activate the Snap Mode icon and the Free Coordinates Mode icon. The system switches between the modes depending on the distance of the pointer from the nearest available point. When the pointer is within the pick limit from an existing point, the Snap mode is in effect. You are able to snap only to those element types that are indicated by the Snap Point Settings icons.

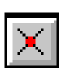
Snap Point Settings

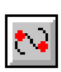
Snap uses these point settings, any or all of which can be active at one time:


Endpoint uses the snap points at the start and end points of all elements in the process model.


Midpoint uses the snap points at the midpoint of all elements in the process model, except for polylines and splines.


Center Point uses the snap points at the center point of all arc elements in the process model.


Intersection uses the snap points at each intersection of lines or arcs in the process model. Intersection does not recognize the intersection of polylines and splines.

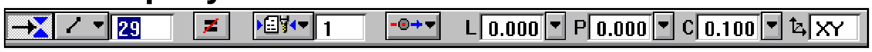

Control Point uses the snap points at each control point on a spline or polyline element in the process model.

Readout Line

The readout line is always visible below the graphic view. The information displayed to the left depends on the task in progress. As you move the cursor over any menu, icon, toolbox, or tool, the readout line displays a short statement identifying its use. The readout line displays information about the selected element when you use Snap mode.

The information displayed to the right is the group status. The number to the left of the slash reflects the quantity of entities in the active group. The number to the right of the slash reflects the total number of entities in the model.

Insert Property Bar



The Insert property bar is positioned either below or above the graphics view. It displays information about the insert position, assignment, association, and properties.

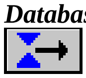
Its appearance changes somewhat between inserting with a step or on a layer because not all fields apply to both conditions.

Chapter 2 presents detailed information about the Insert property bar.

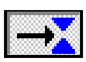
Insert Positions

These icons control where the new geometry resides:

Database Location

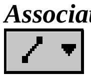


The **Before** icon inserts geometry into the database before the current position.



The **After** icon inserts geometry into the database after the current position.

Association



The **Element** icon specifies that the insert position is relative to an element.

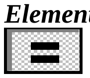
The **Profile** icon specifies that the insert position is relative to a profile.

The **Layer** icon specifies that the insert position is relative to a layer.

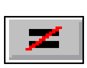
The **Step** icon specifies that the insert position is relative to a step.

The **Tool** icon specifies that the insert position is relative to a tool.

Element Property

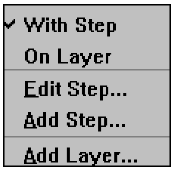


The **Match Element** icon updates the properties of an inserted element to match those of an element that you select.



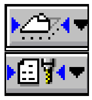
The **Don’t Match Element** icon does not match the properties of an inserted element to another element.

Insert Assignments



Assignment

These icons control what the new geometry will be assigned to:



The **On Layer** icon assigns the new geometry to a layer.

The **With Step** icon assigns the new geometry to the chosen step, but it does not match the properties of an inserted element to another element.

Edit/Add

These menu items enable you to manipulate step and layer information from the Insert property bar:

The **Edit Step...** item opens the Edit Process Step dialog box, which enables you to view or change step properties.

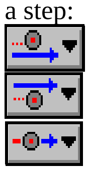
The **Add Step...** item opens the Add Process Step dialog box, which enables you to create new process steps.

The **Add Layer...** item opens the Add Layer dialog box, which enables you to create a new layer with full control over layer properties.

Insert Properties

Offset

Offset is relative to the direction of the geometry. These icons control the properties of the new geometry, and they are available only when you insert with a step:



The **Offset Left** icon sets the tool to the left of the geometry.

The **Offset Right** icon sets the tool to the right of the geometry.

The **No Offset** icon centers the tool on the geometry.

Level

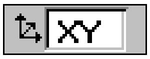
The **L** selector switch accepts a value for the level of the geometry on a specific workplane. You can enter a value in the input field, snap to a level in the graphic view, use the default level, or select one of the five different most recently used levels.

Profile Top

The **P** selector switch accepts a value for the top of the material. You can turn it on or off. If you turn it on, you can select one of the five different most recently used profile top settings, or enter a value in the input field.

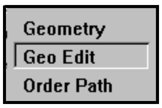
Clear

The **C** selector switch accepts a value for the height the tool retracts between profiles. If you turn it on, you can select one of the five different most recently used profile top settings, or enter a value in the input field. This is available only if you are inserting with a step.



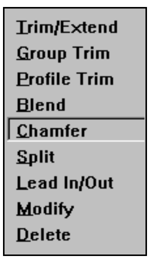
Work Plane

This input field accepts a value for the workplane to which the insert will correspond. The list view displays defined workplanes when you select this input field. This field accepts both keyboard input or an input made by selecting from the defined workplanes displayed in the list view.



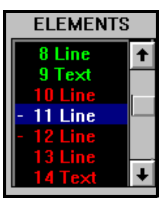
Workbench

The workbench provides quick access to frequently used sets of procedures, called *toolboxes*. Depending upon your screen resolution, the three to five most recently used toolbox titles are resident on the workbench. To add a new toolbox title to the workbench, select it from a menu. You can select any of the five last used toolboxes by pressing the **Alt** key and numbers **1**, **2**, **3**, **4**, or **5**.



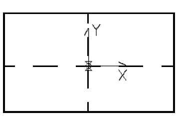
Toolbox

The toolbox that is displayed corresponds to the active workbench selection (toolbox title). The toolbox contains sets of modeling tools that you can use to create and change the model that you are building. When you select a tool from the toolbox, the list view and the control panel that correspond to it are displayed.



List View

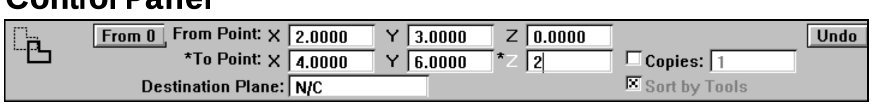
The list view displays information for the tool that you select from the open toolbox. For example, Element Seq is the selected tool, so element information is displayed sequentially in the list view.



Graphic View

The graphic view provides a visual representation of the process model. Additions or changes you make to the graphic view are reflected immediately. The displayed model may be viewed from any angle, and rotated, moved, enlarged, or panned. You can also view the toolpath and make changes quickly and easily.

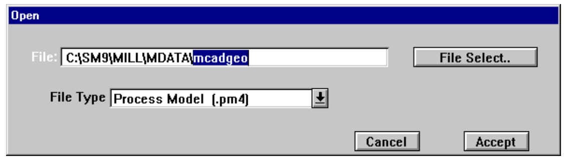
Control Panel



Control panels are displayed at the bottom of the screen. Each tool has its own control panel. In the upper left corner of each control panel is a copy of the tool's icon image.

An asterisk (*) in the control panel indicates that after you set that switch or field, SmartCAM has enough information to complete the action.

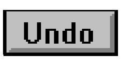
Dialog Box



Dialog boxes open temporarily to enable you to select or control specific events. A dialog box can be displayed anywhere on the screen. It is closed when its action is complete or is cancelled. If a dialog box is open, you must complete it or cancel it to do anything else with the model. For example, this dialog box is opened when you select Open from the File menu. It remains open until you press the Accept button or the Cancel button.

Using Controls

Controls interact with SmartCAM control panels and dialog boxes.



Action Buttons

Action buttons initiate a specific action or function.

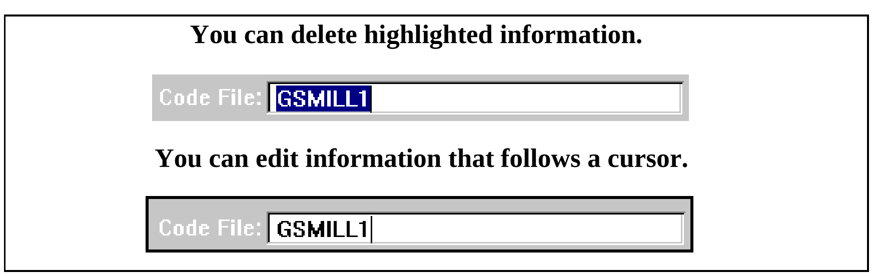
Select a button by placing the cursor over it and pressing the left mouse button or by pressing the Tab key until the button is highlighted and then pressing the Enter key.

Input Fields

Use input fields to enter information, such as names and numbers. Select the field by placing the cursor over it and pressing the left mouse button. Then enter the desired name or number. You can switch between input fields by pressing the Tab key.

Selecting the name of the input field highlights its contents. If the current information in the input field is highlighted, it is deleted when you enter a new value.

Figure 1-6
Delete or edit information in input fields.

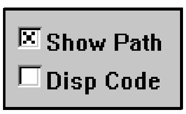


Selecting in the input field places the cursor to the right of the last character in the input field. If only a flashing bar (cursor) is visible, the current value can be appended to or edited by using the arrow and Backspace keys.

Most of the input fields accept expressions in place of direct input. SmartCAM solves the expression and uses the solution. For example, to specify a Z coordinate location that is one inch from the end point of an existing element, perform these tasks:

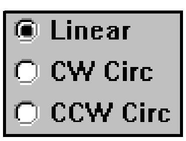
1. Select the Z input field label to make the input field active.
2. Select the end of the existing element (with Snap on) to enter the Z coordinate of the existing element end point.
3. Enter **+1** in the input field after the element input coordinate.

When you accept the coordinates, SmartCAM solves the expression and uses the resulting Z coordinate.



On/Off Switches

On/Off switches turn a function or an input field on or off. The switch has an X in the box when it is turned on, and the box is empty when the switch is turned off. Use the mouse to place the cursor over the switch, and press the left mouse button to turn the switch on or off. In this example, Show Path is on and Disp Code is off. You can have both on/off switches on at the same time.



Option Switches

Option switches turn on one option from a group. Place the cursor over the option switch, and press the left mouse button to toggle the switch on or off. Unlike on/off switches, only one option switch can be on at a time. In this example, With Step is on and On Layer is off.

Selector Switches

Selector switches contain a short list of options or choices for a field. Selector switches are displayed in many control panels and dialog boxes. Choosing the selector switch causes the option list to be displayed.

Use these methods to set selector switches:

- Move the cursor over the name of the selector switch, and then press the mouse button to cycle through the different options.
- Move the cursor over the selector switch option, and then press the mouse button to display the list of options. Select the option you want. If there are many options, use the scroll bar to display all the options.
- Press the Tab key until the selector switch is highlighted, and then press the Up or Down arrow key to set the correct option.

Using Hot Keys

Many SmartCAM tasks are assigned to hot keys that provide a quick way of performing a task or setting a mode of operation. **[Alt]+[Letter]** selects a pull-down menu from the menu bar. **[Letter]** should be the first letter of the pull-down menu you want to display. Press **[Alt]** and the appropriate letter key at the same time.

For information about customizing hot key assignments, refer to the *shortcut key* topic in the online Help.

Points to Remember

- You cannot select dimmed menu items.
- There are several types of controls that you can use to interact with control panels and dialog boxes: action buttons, input fields, on/off switches, option switches, and selector switches.
- The icon bar can be thoroughly customized to meet your needs. It can be resized, moved, reorganized, and changed.
- An asterisk (*) in the control panel indicates that after you set that switch or field, SmartCAM has enough information to complete the action.
- Each tool has its own control panel.
- To enter information in an input field, you must first select it.

Working with Files

Objectives

This lesson shows you how to perform these tasks:

- Identify SmartCAM file types.
- Open a file.
- Save a file.
- Create a new file.

Overview

Now that you are familiar with the SmartCAM workplace, you are ready to experiment with SmartCAM files using the model files provided with SmartCAM Production Milling.

Learning SmartCAM File Types

SmartCAM creates and uses process model and job operations files.

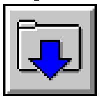
Process Model

The process model file graphically represents the part, material block, clamps, and fixtures. It also contains the sequence of machine operations.

Job Operations File

The job operations file contains all of the step information used in a job: tool information, feeds, and speeds.

Opening a File



Model File: PMVIEWS.PM4

When you first open your application, most of the options on the menu bar and the icon bar are dimmed, and you cannot select them. This is because you have not yet opened a model file.

The files referenced in this manual are in the `\train` directory of your SmartCAM CD. If your CD drive is labeled drive `d`, the files with English unit data are in `d:\train\mill_e`, and metric data files are in `d:\train\mill_m`.

Note The model file references in this book are to English unit files. Although the metric unit files are not directly referenced, the metric file naming convention is such that a metric file name can easily be derived from the name of the referenced English unit file. An "m" has been added as the last character before the period in the metric file name. For example, where the English unit file name is `PMVIEWS.PM4`, the metric file name would be `PMVIEWSM.PM4`. If the English file name already uses the limit of 8 characters and needs to be truncated to add the "m," the metric name has been altered some to maintain its mnemonic significance.

If your CD drive is labeled differently, insert that letter in place of "d." If you have copied these files to a local or network drive, it is necessary to know that path to use the files referenced in this book.

Open a file by performing these tasks:

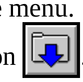
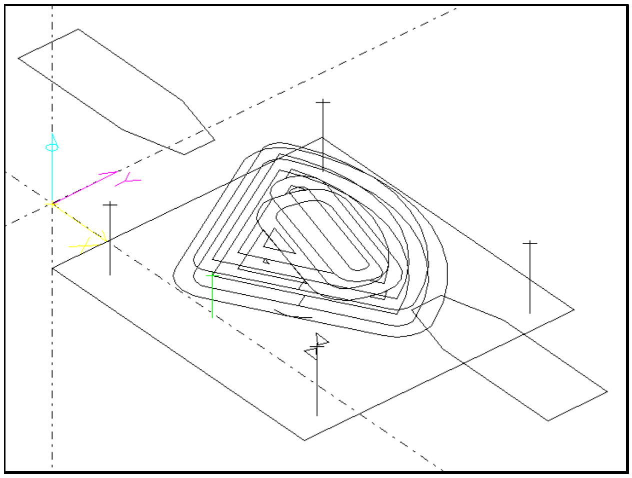
1. Access the **Open** dialog box by performing one of these tasks:
   - Select **Open** from the **File** menu.
   - Select the **File—Open** icon .
2. Select the **Process Model** (`.pm4`) selector switch. The 2-D (`.sh2`) and 3-D (`.sh3`) options are used for converting files created in previous versions of SmartCAM.
3. Select **File Select** to display a list of available files in the **Open** dialog box.
4. Select model file **`PMVIEWS.PM4`**.
5. Select the **Accept** button.

Figure 1-7
Open PMVIEWS.



Saving a File

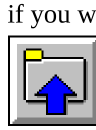
To avoid losing your work, save your model frequently while you are working on it. Avoid saving model files that are shipped with the software because they are often used for multiple lessons. However, if you do save changes to one of these files on your local drive, you can return to the `\train` directory of your SmartCAM CD for a clean copy of the file.

There are two ways to save files: File—Save and File—Save As.

Using File—Save 

When you select Save from the File menu, the open process model file and job operations setup file are saved immediately with the current filenames, which overwrites the existing file.

Note Use File—Save As if you want to save the file with a different name.

Using File—Save As 

When you select Save As from the File menu, a dialog box opens, enabling you to save the process model file, the job operations file, or both with a new file name. This enables you to create a new file without changing the file already on disk. The new file and path you assigned in the Save As dialog box become the active file and path for SmartCAM.

Creating a New File

Use New to create a new CNC Process Model. When you select New, the existing process model closes, and SmartCAM opens a new, untitled process model. Perform these tasks to create and save a new file:

1. Select **File—New**.
2. Select **File—Save As**.

Points to Remember

- ❑ Process model files graphically represent the part.
- ❑ Job operation files contain step information.
- ❑ Use File—Save to overwrite the existing file.
- ❑ Use File—Save As to save a file with a new name.
- ❑ A clean copy of each training file is in the `\train` directory of your CD.

Manipulating the View

Objectives

This lesson shows you how to perform these tasks:

- Window in.
- Name views.
- Get views.
- Change the orientation of the view.
- Change the display of the view.

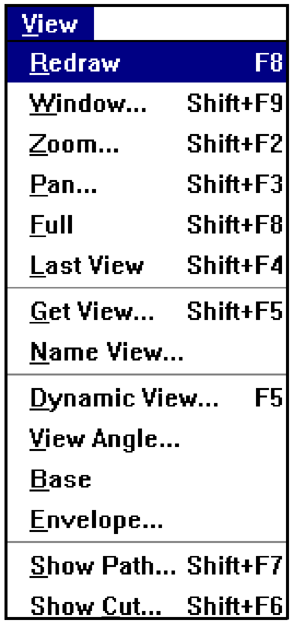
Overview

Use the View menu to change the view of the model.

Using the View Menu

Use the View menu to change the graphic view of your model.

Figure 1-8
Specify options from the View menu.



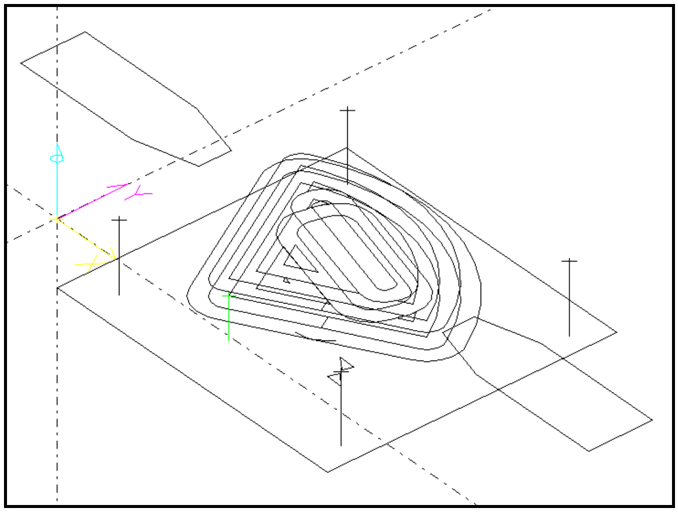
Using Window

Model File: PMVIEWS.PM4

Use Window to magnify part of the graphic view to show more detail. Perform these tasks to practice using Window:

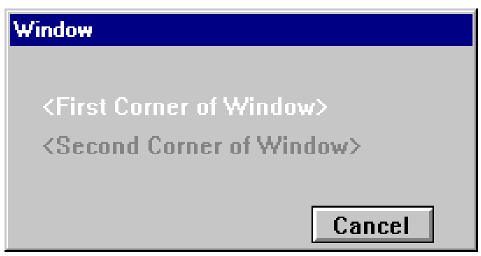
1. Open the model file **`PMVIEWS.PM4`**.

Figure 1-9
Open PMVIEWS.



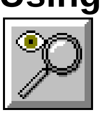
2. Select **View—Window**. The **Window** dialog box is displayed.

Figure 1-10
Specify the first and second corners of a window.



3. Use the mouse to select the first corner of the window, and then drag the mouse, stretching a box around the area to magnify.

Using Zoom

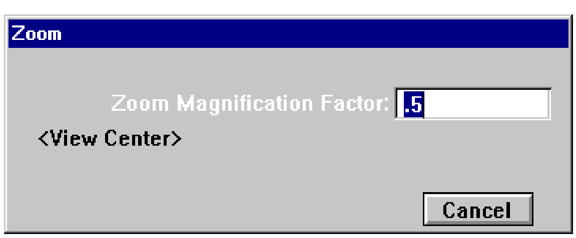


Model File: PMVIEWS.PM4

Use Zoom to magnify a specific area. Perform these tasks to use Zoom:

1. Select **View—Zoom**. The **Zoom** dialog box is displayed.

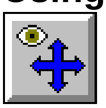
Figure 1-11
Specify a zoom magnification factor in the Zoom dialog box.



2. Set the **Zoom Magnification Factor** input field. The **<View Center>** prompt is highlighted.

3. Select a point in the graphic view. The graphic view is zoomed around this point.

Using Pan



Model File: PMVIEWS.PM4

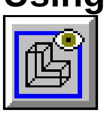
Use Pan to move the view along the distance between two points you specify. Perform these tasks to use Pan:

1. Select **View—Pan**. The **Pan** dialog box is displayed.

Figure 1-12
Specify a point to pan from and a point to pan to.

2. Select a point in the graphic view to pan from.
3. Select a point in the graphic view to pan to. The graphic view pans from the first point to the second point.

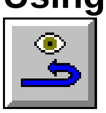
Using Full



Model File: PMVIEWS.PM4

Select **View—Full**. SmartCAM fits all of the non-hidden geometry into the graphic view.

Using Last View



Model File: PMVIEWS.PM4

Select **View—Last View**. The previous view is displayed. You can use this option to alternate between two convenient views.

Using Get View

Model File: PMVIEWS1.PM4

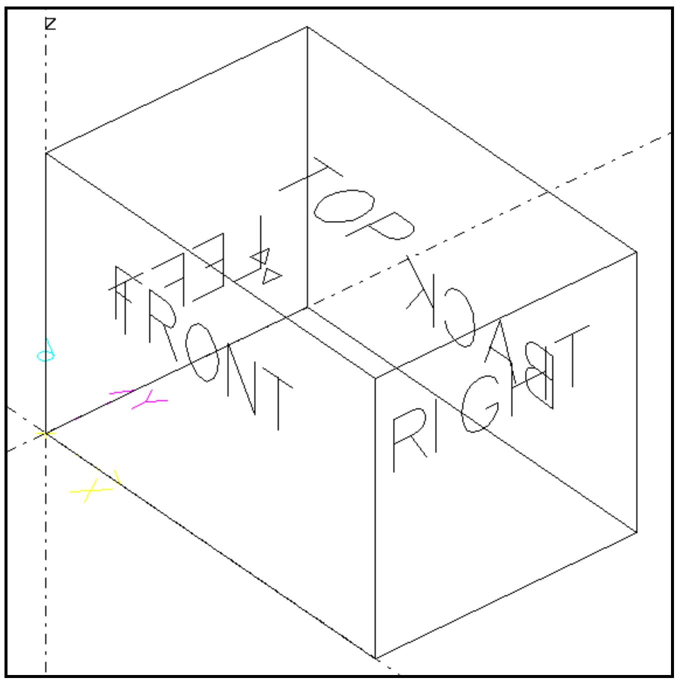
Use the Get View feature to view different orientations that you saved. By default, four different views are available (Top, Right, Front, Isometric). You can use the Get View dialog box or these hot keys to view your model from different orientaions:

- F9—Change the graphic view to the full Top view of the part.
- F10—Change the graphic view to the full Front view of the part.
- F11—Change the graphic view to the full Right view of the part.
- F12—Change the graphic view to the full Iso view of the part.

Perform these tasks to use the Get View dialog box:

1. Open the model file **PMVIEWS1.PM4**.

Figure 1-13
Open PMVIEWS1.



2. Select **View—Get View**. The **Get View** dialog box is displayed.
3. Select the view you want.

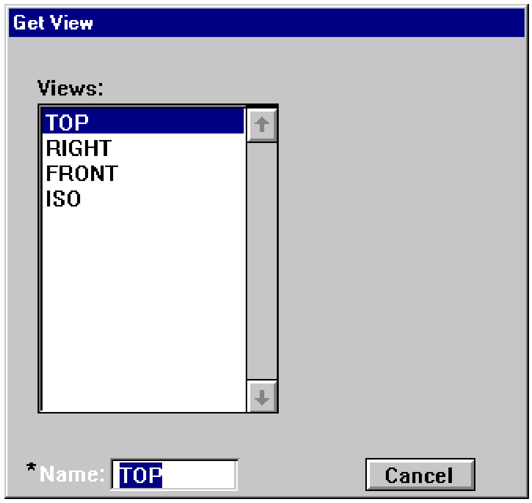
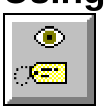


Figure 1-14
Specify the view that you want.

Using Name View



Model File: PMVIEWS1.PM4

Use the Name View feature to save different orientations by assigning them names. You can access these views later using Get View. Perform these tasks to use Name View:

1. Create the view for your current display.
2. Select **View—Name View**. The **Name View** dialog box is displayed.

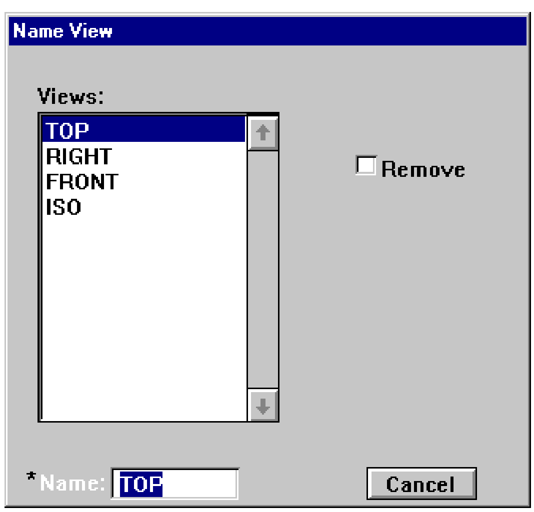
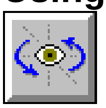


Figure 1-15
Set the Name input field to save the new view.

3. Set the **Name** input field to save the new view.

Using Dynamic View



Model File: PMVIEWS1.PM4

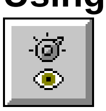
Use the Dynamic View feature to change the orientation of the view in the active window. You can rotate, spin, pan, or zoom using the mouse. Perform these tasks to use Dynamic View:

1. Select **View—Dynamic View**. The **Dynamic View** dialog box is displayed.

Figure 1-16
Specify a way to move the view in the Dynamic View dialog box.

2. Select **Rotate**, **Flat Spin**, **Pan**, or **Zoom**:
   - **Rotate** moves the view as a three-dimensional object around a center point.
   - **Flat Spin** moves the view as a two-dimensional object around its visual center point.
   - **Pan** shifts the view on the screen.
   - **Zoom** magnifies or shrinks the view.
3. Select and hold the left mouse button while moving the mouse. The dynamic view changes.
4. Select the **Accept** button when you are satisfied with the view.

Using Display Modes



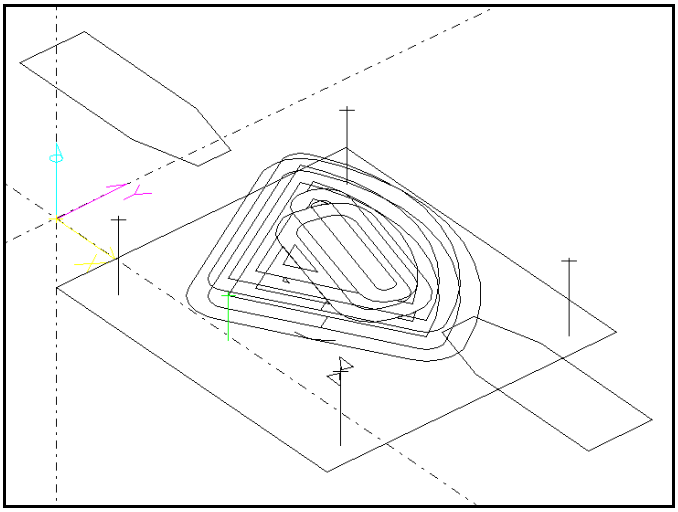
Model File: PMVIEWS.PM4

Use the Display Modes dialog box (Utility—Display Modes) to control how geometry is displayed in the graphic view.

Perform these tasks to use Display Modes:

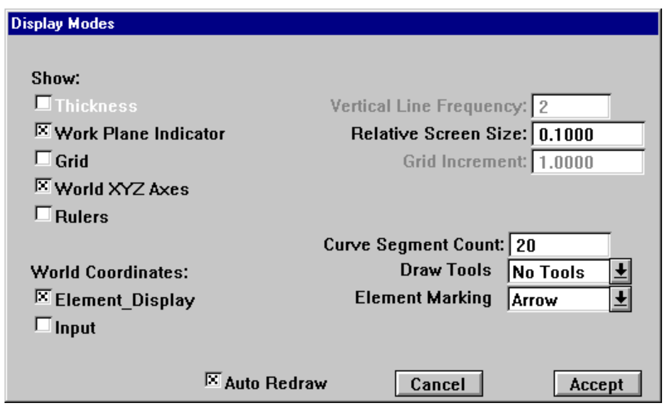
1. Open the model file **`PMVIEWS.PM4`**.

Figure 1-17 Open PMVIEWS.



2. Select **Utility—Display Modes**. The **Display Modes** dialog box is displayed.

Figure 1-18 Set the values on the Display Modes dialog box.



3. Set the values on the dialog box. Although you typically accept most default settings, the following list contains commonly changed settings:

   - **Thickness** displays the distance between the Z level and the Prof Top values when turned on.
   - **Work Plane Indicator** displays the work plane indicator when turned on.
   - **World XYZ Axes** displays the World Coordinate System when turned on.
   - **Element Marking** displays elements in the active group with one of these options: **Arrow**, **Dotted**, or **Both**.

Points to Remember

- ❑ Use the View menu to access options for manipulating the view.
- ❑ Window and Zoom magnify specific areas of the view.
- ❑ Pan and Dynamic View manipulate the orientation of the view.
- ❑ Save a view with Name View.
- ❑ Use the Display Modes feature to control how the geometry is displayed in the graphic view.
- ❑ You can show thickness, work plane, grid, world area, and rulers.

Self-Test

Directions

Test your understanding of the concepts and procedures in this section by answering the following questions. The answers for each self-test are in *Appendix A* of this manual.

_______________ **1. It is possible to select dimmed menu items.**

a) true
b) false

_______________ **2. What does an asterisk (*) in the control panel indicate?**

a) SmartCAM enters a system default for that switch or field.
b) SmartCAM requires one more digit for that switch or field.
c) SmartCAM completes the action after you set that switch or field.
d) SmartCAM completes the action after you enter one more value and set that switch or field.

_______________ **3. Which of these workplace items does each control panel correspond to?**

a) workbench
b) toolbox
c) tool
d) dialog box

_______________ **4. What do process model files graphically represent?**

a) part
b) feed
c) machine code
d) b and c

_______________ **5. What do job operation files contain?**

a) layer information
b) step information
c) both a and b
d) neither a nor b

_______________ **6. Which View features magnify specific areas of the view?**

a) Window
b) Zoom
c) both a and b
d) neither a nor b

_______________ **7. Which View features manipulate the orientation of the view?**

a) Window
b) Zoom
c) both a and b
d) neither a nor b

_______________ **8. What feature do you use to control how the geometry is displayed in the graphic view?**

a) Show/Mask
b) Display Modes
c) Increment
d) Status

2

Working with SmartCAM

Overview

After you become acquainted with SmartCAM and can perform basic tasks such as navigating in the workplace, opening files, and manipulating the view, you are ready to work with SmartCAM. This entails manipulating elements, which represent toolpath or boundary definitions and element properties.

Lessons for This Unit

- Inserting Elements and Setting the Active Step/Layer
- Using Levels, Clearance, and Profile Top
- Working with Work Planes and Tool Planes
- Using Snap
- Grouping Elements
- Using Edit Filters

Inserting Elements and Setting the Active Step/ Layer

Objectives

This lesson shows you how to perform these tasks:

- Using the Insert property bar to set the insert position.
- Using the Insert property bar to set the insert assignment.

Overview

Elements represent toolpath or boundary definitions within the model database. If a process step is assigned to an element, it is toolpath.

A process step is a combination of tool information and an operation. Any geometry created with a process step represents toolpath. Tool information includes these qualities:

- Tool (mill, drill, bore, ream, or tap)
- Type (ball nosed, flat, or radiused cutter)Shape (round)
- Diameter
- Length
- Cutter side angle or taper angle
- Number of flutes
- Optional custom tool graphics
- Operation information, which includes the following:

- Feed rate in IPM, IPR, or IPT
- Spindle speed in SFM or RPM

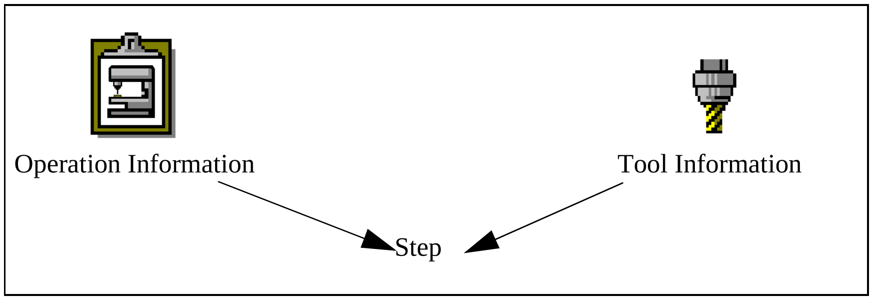


Figure 2-1
A step is made of operation and tool information.

When you need to model part boundaries, fixtures, or other details that are not toolpath, assign elements to different layers. Layer information is CAD geometry. Each layer in a model has one of 16 colors, and the maximum number of layers is 99. Items such as clamps, fixtures, and material boundaries are examples of geometry to draw on layers because they do not represent toolpath. In addition, no tool or operation information is associated with layer geometry.

Elements are displayed in the following figure.

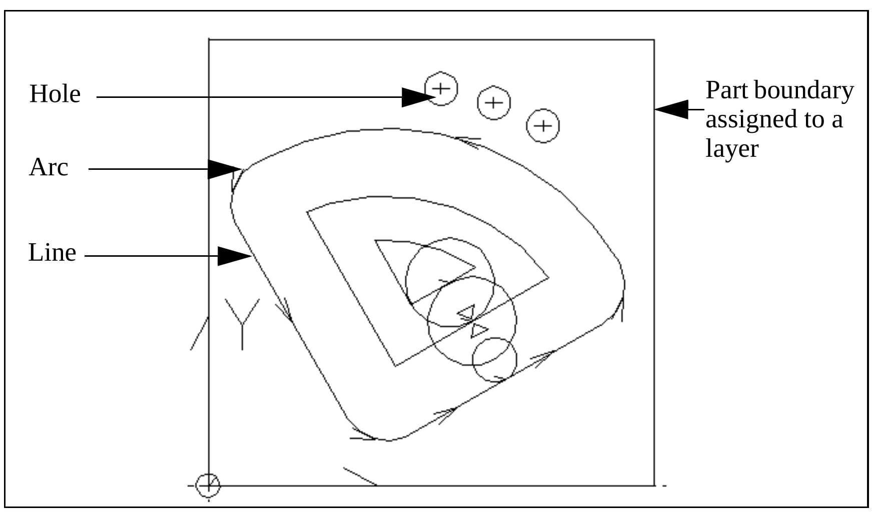


Figure 2-2
These are element samples.

Use the Insert property bar to add new elements to the database at a specified insert location with the specified properties.

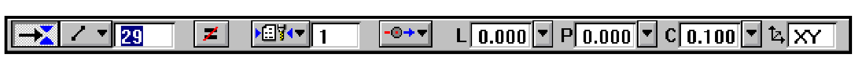


Figure 2-3
The Insert property bar is very important for creating elements.

You can set insert locations with either a process step or a layer. Just remember that geometry associated with a process step represents toolpath and geometry associated with a layer does not. Use one of the methods in the Insert property bar to set the insert location. A horizontal line in the list view indicates the insert location. You can use the Insert property bar to set the active process step or layer, and the offset, work plane, level, prof_top, and clear values.

The active settings on the Insert property bar affect only element creation. To change the insert sequence for existing elements, use Edit—Order Path. To change other insert properties for existing elements, use Edit—Property Change.

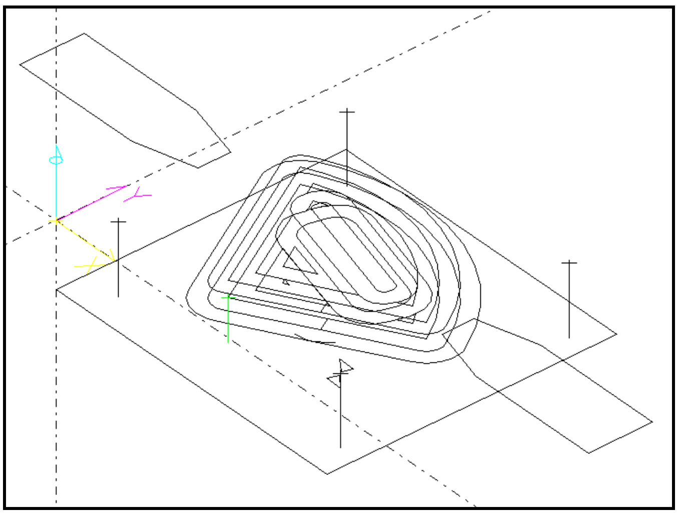
Inserting Geometry

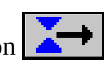
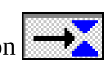
Model File: PMVIEWS.PM4

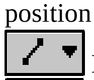
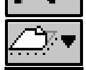
Perform these tasks to insert geometry:

1. Open the model file **`PMVIEWS.PM4`**.

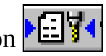
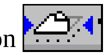
Figure 2-4 *Open PMVIEWS.*



2. Set the **Before** icon  or **After** icon  on the **Insert** property bar to indicate if the insert point is to be before or after the selected element, process step, tool, layer, or profile.
3. Specify with an Association icon on the **Insert** property bar, the insert position that is relative to a tool. These icons are valid options:

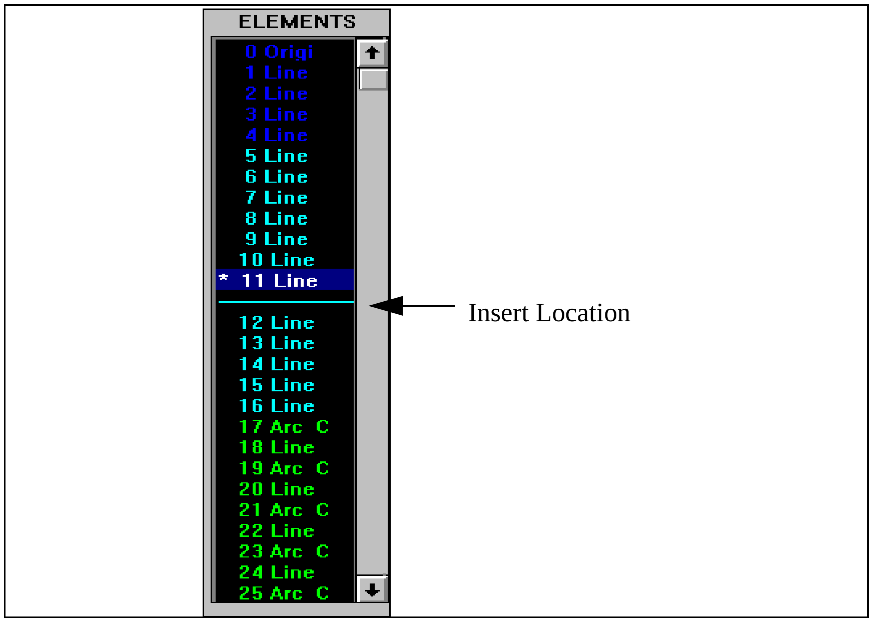
    Element
    Profile
    Layer
    Step
    Tool

   Notice that the list view is updated to correspond to the icon you select.
4. Select from the list view or graphic view the element, profile, layer, process step, or tool to insert the element before or after.

5. Set the **With Step** icon  or the **On Layer** icon  on the **Insert** property bar.

6. Select the step or layer to use from the list view or graphic view.

Figure 2-5
Identify the insert location in the list view.

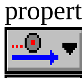


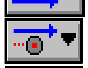
7. Set the **L (Level)** input field on the **Insert** property bar. This is the Z-axis position for the geometry.

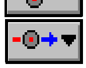
8. Set the **C (Clear)** input field on the **Insert** property bar. This is the primary retract plane for the tool. You can set this value only when you insert with a step. Setting a clearance value for a step ensures that subsequent uses of inserting on a layer use this data for layer geometry.

9. Set the **P (Profile Top)** input field on the Insert property bar. This identifies the material top level for geometry.

10. Set one of the **Offset** icons on the Insert property bar to indicate the offset property for the new geometry along its direction:

     The **Offset Left** icon sets the tool to the left of the geometry.

     The **Offset Right** icon sets the tool to the right of the geometry.

     The **No Offset** icon centers the tool on the geometry.

    Note Offset icons are available only if you are inserting with a step.

11. Use the **Create** menu to build the geometry, see *Creating Elements*, on page 4-3.

Points to Remember

- Layer geometry is not associated with a step and cannot generate code.
- Toolpath geometry is created with a step and can generate code.
- The Offset and Clear selector switches are available only when you insert with a step.
- To input the number of a profile, step, layer, or an element, use the Before or After icons with step and layer input fields.

Using Levels, Clearance, and Profile Top

Objectives

This lesson shows you how to perform these tasks:

- Use levels.
- Define the clearance height.
- Define the profile top.
- Set the Insert property bar.

Overview

Use levels to set the Z-height of the geometric elements you draw. When you draw an element with a step, SmartCAM sets the Z-height for the tip of the tool to the level you set in the Level input field.

When you draw an element with a step, you use the C (Clear) selector switch to set the Z-height for how far the tool retracts after making a cut. You use the P (Profile Top) selector switch to identify the material top for geometry.

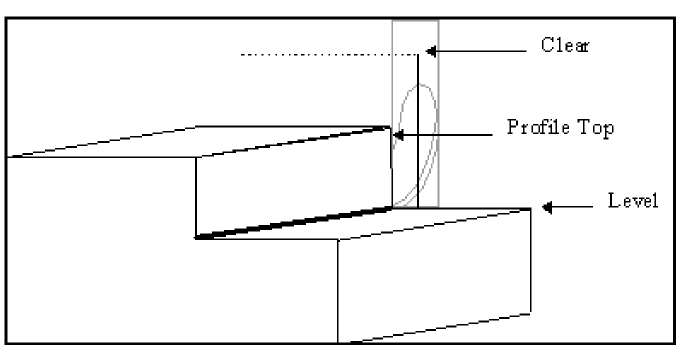
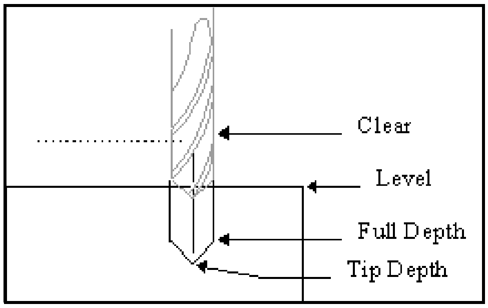


Figure 2-6
Notice the Clear, Profile Top, and Level of a simple operation.

Figure 2-7
Notice the Clear, Level, Full Depth, and Tip Depth of a simple drilling operation.

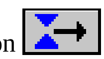
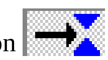
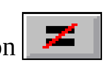
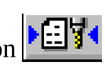


Setting the Insert Property Bar

When you set level, clearance, and profile top on the Insert property bar, you must turn on the on/off option for the Clear and Prof_Top fields to be active.

Setting Levels, Clearance, and Profile Top

Perform these tasks to set levels,clearance, and profile top:

1. Activate the **Before** icon  or **After** icon  on the **Insert** property bar to indicate if the insert point is to be before or after the selected element, process step, tool, profile, or layer.
2. Select the **Element** icon  on the **Insert** property bar.
3. From the list view or graphic view, select the element that you want to place the new element before or after.
4. Make sure the **Don't Match Element** icon  is active on the **Insert** property bar. This prevents the new element from adopting the properties of the preceding element.
5. Activate the **With Step** icon  on the **Insert** property bar, and select a milling tool from the list view.
6. Set the **L (Level)** selector switch on the **Insert** property bar. It specifies the Z-level at which the tip of the tool cuts.
7. Turn on **C (Clear)**, and set the selector switch on the **Insert** property bar. It specifies how far the tool retracts after making a cut.
8. Turn on **P (Profile Top)**, and set the selector switch on the **Insert** property bar. It specifies the top edge of the element.

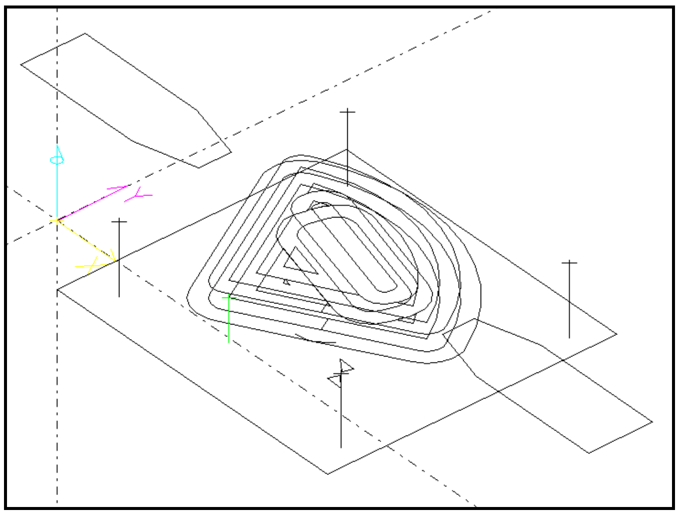
Checking Levels, Clearance, and Profile Top

Model File: PMVIEWS.PM4

When you insert on a layer, SmartCAM uses the levels , clearance, and profile top values that you set in a step to create geometry. You can verify this by performing these tasks:

1. Open the model file **`PMVIEWS.PM4`**.

Figure 2-8
Open PMVIEWS.



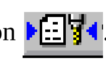
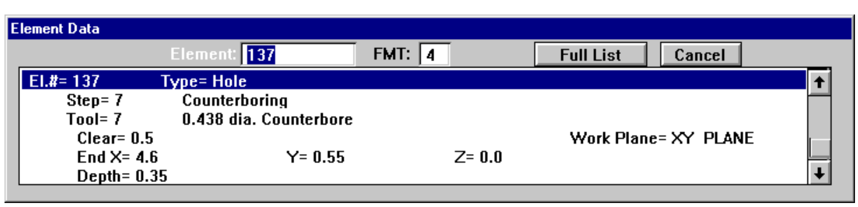
2. Set the **With Step** icon .
3. Select the step for which to set the levels, clearance, and profile top.
4. Select **Utility—Element Data**. The **Element Data** dialog box is displayed.
5. Notice that the level, clearance, and profile values are the same as the values that you entered when you inserted with a step.

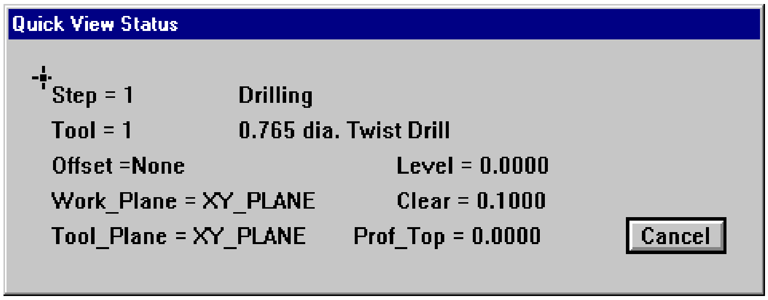
Figure 2-9
Verify your data with the Element Data dialog box.



6. Select **Utility—Status**. The **Quick View Status** dialog box is displayed.

   This dialog box lets you view the operation's status during SmartCAM operation without interrupting the current task or losing control panel inputs.

Figure 2-10
View the status of the current operation with the Quick View Status dialog box.



7. Select the **Cancel** button when you finish viewing the status.

Points to Remember

- ❑ Clear defines the height to which the tool retracts after making a cut.
- ❑ Level defines the Z-level at which the tip of the tool cuts.
- ❑ Profile Top defines the Z-height of the top of your part.
- ❑ Use Status to view the operation's status.
- ❑ Using Status does not interrupt the current task or lose control panel inputs.

Working with Work Planes and Tool Planes

Objectives

This lesson shows you how to perform these tasks:

- Understand work plane and tool plane terminology.
- Create work planes.
- Delete work planes.
- Create tool planes.

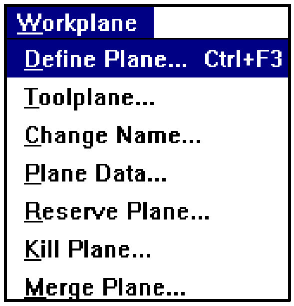
Overview

Work planes determine the plane orientation of planar elements such as lines, arcs, and ellipses. SmartCAM has three pre-defined work planes that are reserved: XY, YZ, and XZ. You cannot modify these pre-defined work planes. You can assign any work plane the reserved status. You can also define work planes that are not reserved by defining the position and orientation of the Local Coordinate System (LCS). An LCS has a red Z axis, a blue X axis, and a green Y axis. SmartCAM also has a World Coordinate System (WCS), which is the system origin point.

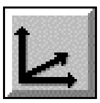
Each work plane has a corresponding tool plane. A tool plane defines the plane that the cutter axis is normal (perpendicular) to. While the tool plane and the work plane are usually parallel, they can have different orientations.

Use the Workplane menu to create or manipulate work planes or tool planes.

Figure 2-11
Use the Workplane menu.



Creating Work Planes



Model File: PMWKPLANE.PM4

You can create work planes when you transform elements and have the Suppress Planes option turned off. You can also create work planes in one of these ways: three points, line and angle, and rotation.

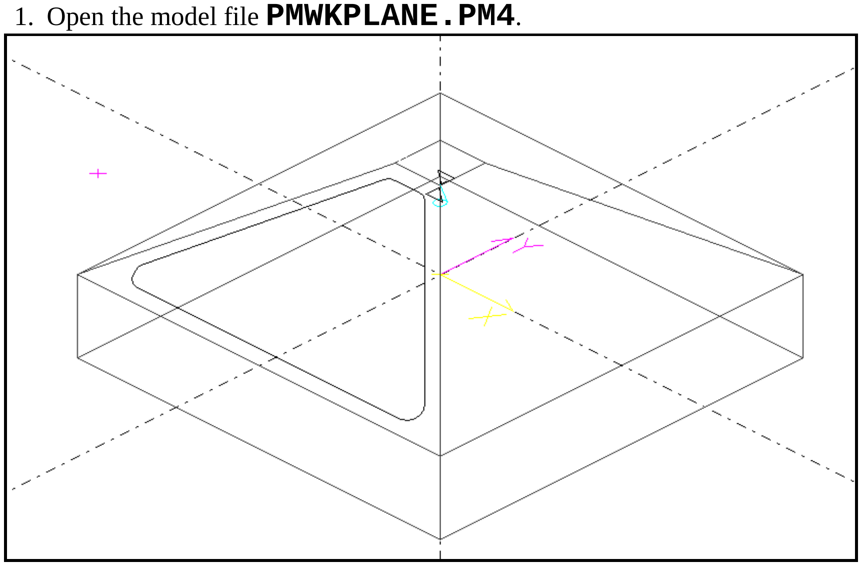
Use Define Plane to create a new work plane for the model. When you define and accept a work plane, it becomes the active work plane. To make an existing work plane active, use the Insert property bar.

Defining a Work Plane from Three Points

Perform these tasks to define a work plane from three points:

1. Open the model file **`PMWKPLANE.PM4`**.

Figure 2-12
Open PMWKPLANE.



2. Select **Workplane—Define Plane**. The **Define Plane** dialog box is displayed.

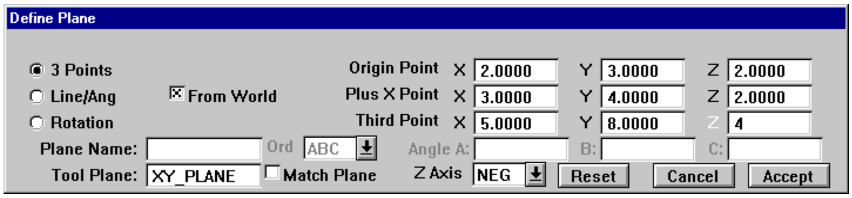


Figure 2-13
Set the values on the Define Plane dialog box.

3. Turn on the **3 Points** option switch.
4. Turn on the **From World** on/off switch to use the WCS as the reference for setting the new work plane toolpaths. When this on/off switch is turned off, the points are positioned using the active work plane as a reference.
5. **Set the Origin Point**, **Plus X Point**, and **Third Point** input fields.
6. Toggle the **Z Axis** selector switch to change the orientation of the Z-axis work plane (POS or NEG).
7. Turn on the **Match Plane** on/off switch. This ensures that the work plane has the same plane as the tool plane.
8. Enter a name for the plane in the **Plane Name** input field. Notice that the **Tool Plane** input field has the same plane name when you press the Enter key.
9. Select the **Accept** button.

Defining a Work Plane from a Line/Angle

Perform these tasks to define a work plane from a line and an angle:

1. Select **Workplane—Define Plane**. The **Define Plane** dialog box is displayed.

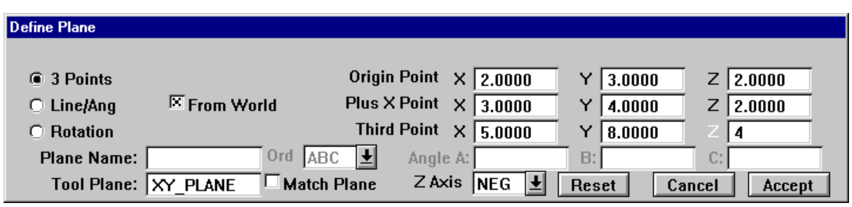


Figure 2-14
Set the values on the Define Plane dialog box.

2. Select **Line/Ang**.
3. Turn on the **From World** on/off switch to use the WCS as the reference for setting the new work plane's **Origin Point** input fields. When this switch is toggled off, the points are positioned using the active work plane as a reference.
4. Set the **Origin Point** by using the **X**, **Y**, and **Z** input fields.
5. Set the **Plus X Point** by using the **X**, **Y**, and **Z** input fields.

6. Set the **Angle A** input field to indicate the angle of rotation for the work plane. The angle A is rotated around the X axis.
7. Toggle the **Z Axis** selector switch to change the orientation of the work plane.
8. Turn on the **Match Plane** on/off switch to enable the work plane to have the same plane as the tool plane.
9. Enter a name in the **Plane Name** input field. Notice that the same name is displayed in the **Tool Plane** input field when you press the Enter key.
10. Select the **Accept** button.

Defining a Work Plane for a Rotation

Perform these tasks to define a work plane for a rotation:

1. Select **Workplane—Define Plane**. The **Define Plane** dialog box is displayed.

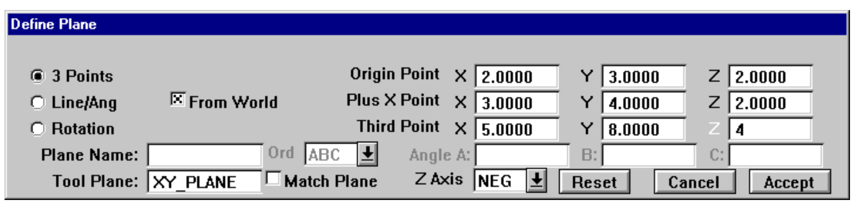
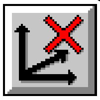


Figure 2-15
Set the values on the Define Plane dialog box.

2. Turn on the **Rotation** option switch.
3. Turn on the **From World** on/off switch to use the WCS as the reference for setting the new work plane toolpath's **Origin Point**, **Plus X Point**, and **Third Point** input fields. When this switch is turned off, the points are positioned using the active work plane as a reference.
4. Use the **Origin Point** input fields to indicate the Origin Point.
5. Use the **Angle A**, **B**, and **C** input fields to indicate the angles of rotation for the work plane. The angles A, B, and C are rotated around the X, Y, and Z axes, respectively.
6. Toggle the **Ord** selector switch if you want to change the order in which the angles are rotated.
7. Toggle the **Z Axis** selector switch to change the orientation of the work.
8. Turn on the **Match Plane** on/off switch to ensure that the work plane has the same plane as the tool plane.
9. Enter a name in the **Plane Name** input field. Notice that the same name is dispayed in the **Tool Plane** input field when you press the Enter key.
10. Select the **Accept** button.

Killing a Plane



Use Kill Plane to delete work planes that have no elements associated with them. This frees computer memory so you can add more elements to the model. Only unused work planes are shown in the dialog box list. You cannot kill reserved work planes. Perform these tasks to kill a plane:

1. Select **Workplane—Kill Plane**. The **Kill Plane** dialog box is displayed.

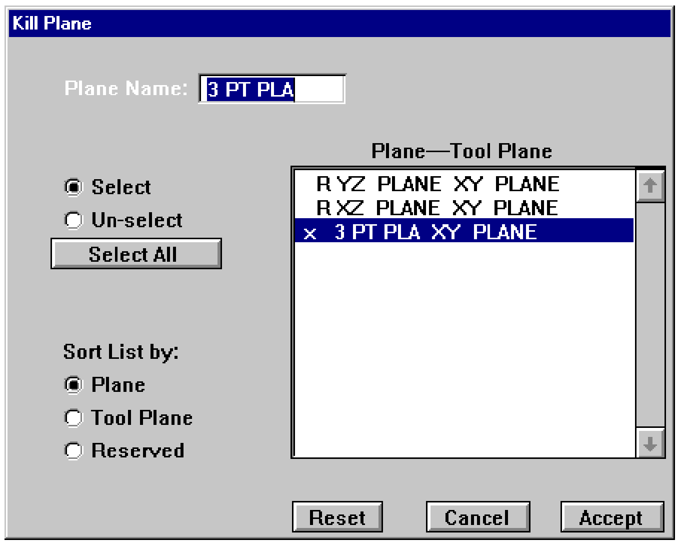
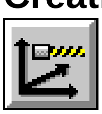


Figure 2-16 *Select the plane to delete on the Kill Plane dialog box.*

2. Select the name of the tool plane to delete from the **Plane—Tool Plane** list.
3. Press the Enter key.
4. Select the **Accept** button.

Creating a Tool Plane

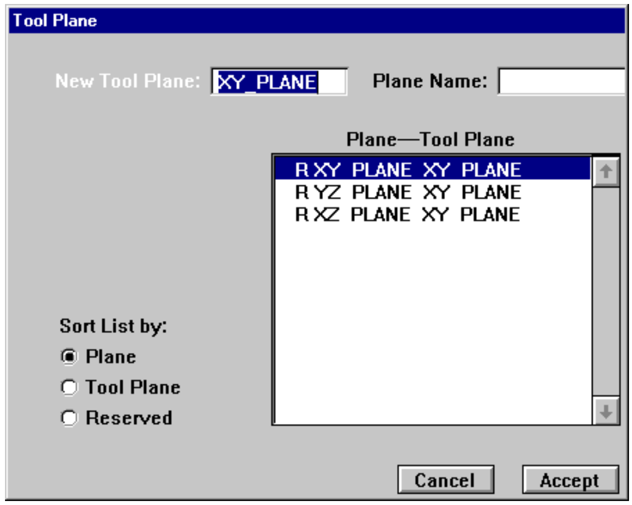


Create a tool plane to define the plane that the cutter axis is normal (perpendicular) to. Use the Tool Plane dialog box to assign a tool plane to an existing work plane. You can position the tool plane at any angle to the work plane, but the tool plane and the work plane are usually parallel.

Perform these tasks to create a tool plane:

1. Select **Workplane—Tool Plane**. The **Tool Plane** dialog box is displayed.

Figure 2-17
Set the name of the tool plane in the Tool Plane dialog box.



2. Select the name of the tool plane from the **Plane—Tool Plane** list. (Press the Enter key.) This name is entered in the **New Tool Plane** input field.
3. Select the work plane to assign to the tool plane from the list.
4. Select the **Accept** button.

Points to Remember

- Work planes determine the orientation of geometry.
- Tool planes determine the plane the cutter axis is perpendicular to.
- Delete unused work planes using Kill Plane.

Using Snap

Objectives

This lesson shows you how to perform these tasks:

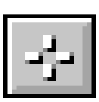
- Learn Snap selection modes.
- Use Snap.

Overview

Snap enables you to use points and other element properties from existing geometry to enter coordinate values in input fields. When you use it, the cursor snaps to the place identified by the active point setting. Use the Snap Mode icons, which are on the readout line, to set the mode and the active point settings.

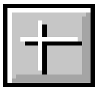
Snap Selection Modes

Snap controls whether the system uses points on elements or specific coordinates when you select coordinate values from the graphic view. Snap uses these selection modes:

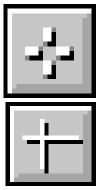


- **Snap mode** requires that you activate the Snap Mode icon. When Snap mode is on, the pointer is displayed as a short cross hairs. Use any of the snap point settings icons (End Point, Midpoint, Center Point, and so forth), which are on the readout line, to set the active point settings and then snap to an element.

  Note If no snap points are found within the element selection distance, Snap returns the coordinates of the last point you selected.



- **Free Coordinates mode** requires that you activate the Free Coordinates Mode icon in a feature that permits it, such as Create Geometry. When Free Coordinates mode is on, the pointer is displayed as a large cross hairs. As you move the cross hairs across the graphic view, the coordinates are displayed in the readout line. The Linear Increment setting in the Increment dialog box determines the interval at which coordinates are updated.



- **Automatic mode** requires that you activate the Snap Mode icon and the Free Coordinates Mode icon. The system switches between modes depending on the distance of the pointer from the nearest available point. When the pointer is within the selection limit of an existing point, Snap mode is in effect. You are able to snap only to those element types that are indicated by the Snap Point Settings icons.

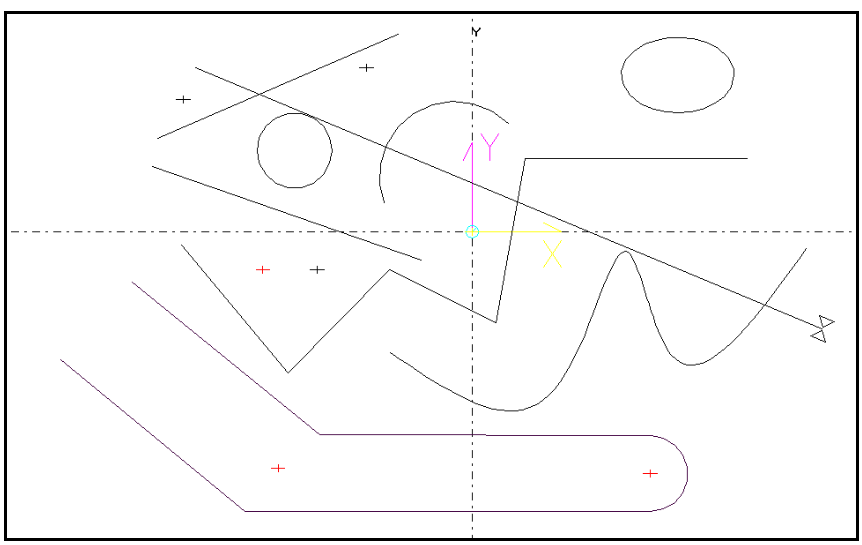
Operating Snap

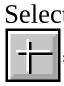
Model File: PMSNPGRP.PM4

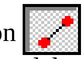
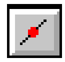
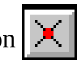
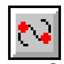
Snap point settings control which element points are used for snap points. You can use any combination of the snap point settings, but you must always have at least one setting turned on. Practice using Snap mode and the Snap point settings by performing these tasks:

1. Open the model file **`PMSNPGRP.PM4`**.

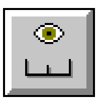
Figure 2-18 *Open PMSNPGRP.*



2. Select the **Snap Mode** icon  and the **Free Coordinates Mode** icon , so that you are in Automatic Mode.
3. Select **Create—Geometry—Line**, and select an input field. Notice that large cross hairs are displayed in the graphic view.

4. Select the **End Point** icon  and snap to the start and end points of all elements in the process model.
5. Select the **Midpoint** icon  to snap to the midpoint of all elements in the process model, except for polylines and splines.
6. Select the **Center Point** icon  to snap to the center point of all arc elements in the process model.
7. Select the **Intersection** icon  and snap to the intersection of lines or arcs in the process model. Intersection does not recognize the intersection of polylines and splines.
8. Select the **Control Point** icon  and snap to points at each control point on a spline or polyline element in the process model.
9. Keep the model file open.

Incrementing with Snap



Model File: PMSNPGRP.PM4

Use Increment, which is on the Utility menu to set the precision of coordinates and pointer movement in the graphic view, the pick distance for Snap, and the default behavior for Z-level settings on the Create control panels.

You increment the pointer coordinates and angles in the graphic view when you adjust the Linear Increment and Angular Increment values on the Increment dialog box. To increment coordinates and angles in the graphic view with snap, perform these tasks:

1. Open the model file **PMSNPGRP.PM4**.
2. Select **Utility—Increment**. The **Increment** dialog box is displayed.

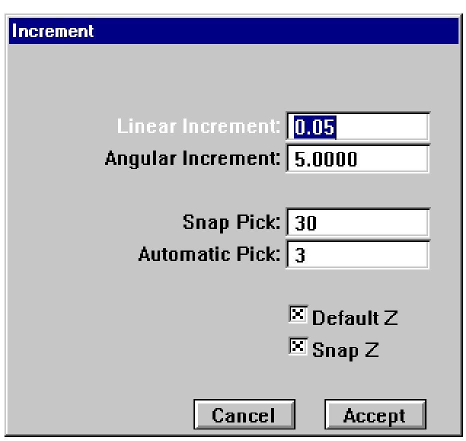


Figure 2-19
Set the values on the Increment dialog box.

3. Set **Linear Increment** to the minimum distance for pointer movement between coordinate display updates when you are using Snap Free Coordinate mode. Set this distance to the smallest value necessary for the precision required by the part geometry.
4. Set **Angular Increment** to the minimum degrees for pointer movement between angle display updates. Set this distance to the smallest value necessary for the precision required by the part geometry.
5. Set **Snap Pick** to the maximum distance in pixels that the pointer can be from an element to select it when you use Snap in Snap mode. This value is also used by the Group arrow and it is normally about 30 pixels.
6. Set **Automatic Pick** to the distance in pixels that the pointer must be from existing elements to switch from Snap mode to Free Coordinate mode when you use Snap in automatic mode. This value is normally between three and five pixels.

   Note The acceptability of the **Snap Pick** and **Automatic Pick** values is affected by your monitor resolution.
7. Turn on the **Default Z** on/off switch to change Z and Level default values on control panels to the current value.
8. Turn on the **Snap Z** on/off switch to update the Z input field to reflect the specified Z level if you use Snap to enter coordinates.

Points to Remember

- ❑ Snap enables you to use points and other element properties from existing geometry to enter coordinate values in input fields when you create or edit geometry.
- ❑ If no snap points are found within the element picking distance, Snap returns the coordinates of the last point that you selected.
- ❑ You can use any combination of the snap point settings, but you must always have at least one setting turned on.
- ❑ You increment the pointer coordinates and angles in the graphic view when you adjust the Linear Increment and Angle Increment values on the Increment dialog box.
- ❑ Select Snap Pick or Automatic Pick to set the Snap mode distance.
- ❑ Turn on Default Z to change the Z and Level default values on control panels to the current value.
- ❑ The acceptability of the Snap Pick and Automatic Pick values is affected by your monitor resolution.

Grouping Elements

Objectives

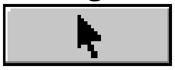
This lesson shows you how to practice these tasks:

- Group elements using the Group Arrow icon.
- Group elements using the Group tool palette.
- Include and restrict the types of elements that you group using the Group tool palette.

Overview

Use the Group icon or the Group tool palette to create an active group of elements.

Using the Group Arrow Icon



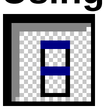
Use the Group Arrow icon to add, edit, or delete elements from the active group or to create a new active group if you operate without the Group tool palette showing.

Select the Group Arrow icon in the readout line below the graphic view. The arrow stays active until you initiate a different function by selecting a control panel or clicking an input field in the active control panel.

Elements in the active group are marked with arrowheads or as dotted lines. To change the active group indicator, change the Element Marking setting in the Utility—Display Modes dialog box; see *Using Display Modes,* on page 1-27.

Note You cannot group hidden or filtered elements. To add hidden elements to a group, first use Utility—Show/Mask to make them visible; see *Using Show/Mask,* on page 4-48. To change the filtering criteria, select the Edit Filter icon  from the readout line or the Group tool palette to open the Edit Filter dialog box.

Using the Group Tool Palette



Model File: PMGROUP.PM4

Use the Group tool palette anytime to group elements, profiles, boxes, steps, layers, or tools. You can activate the Group tool palette by selecting the Group Tool Palette icon that is on the readout line. You can place the Group tool palette anywhere on the screen, and it remains open until you close it.

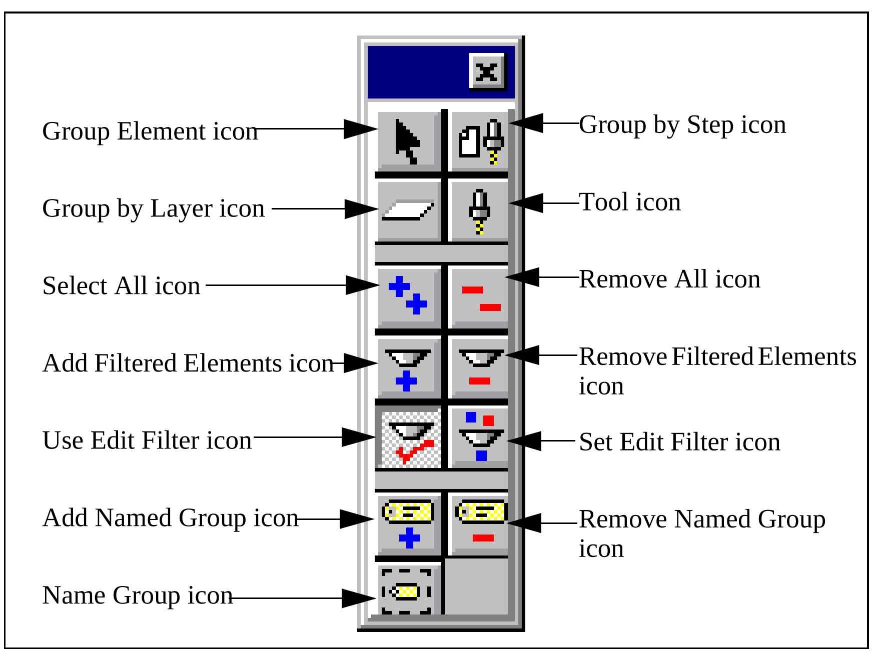


Figure 2-20 *Learn to identify the icons in the Group tool palette.*

This is the basic procedure for using the Group tool palette:

1. Open the model file **PMGROUP.PM4**.

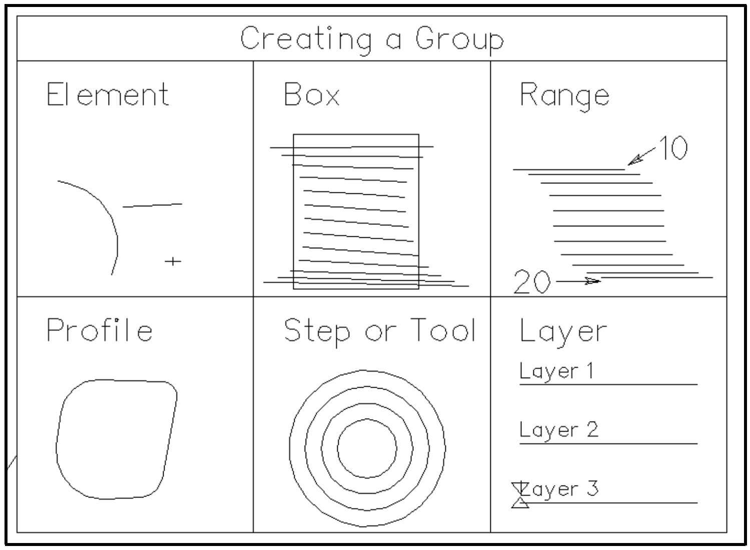
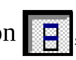
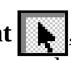


Figure 2-21
Open PMGROUP.

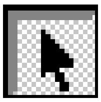
2. Select the **Group Tool Palette** icon , which is on the readout line.
3. Select the **Group Element** , **Group by Step** , **Tool** , or **Group by Layer**  icon to group elements. Use the **Group by Step**, **Tool**, or **Group by Layer** icon to indicate the step, tool, or layer to work with from either the graphic view or the list view.

Note Press the Control (Ctrl) key while performing any of the following selection methods to remove elements from the active group. Each selection method behaves the same for ungrouping as as it does for grouping.

- **Element—**Single-click an element to add it to the group.
- **Profile—**Double-click an element in a profile to add it to the group.
- **Layer** or **Step—**Triple-click any element associated with the desired layer or step to add it to the group.
- **Range—**Single-click the first element in the range, and Shift-click the last element in the range to add it to the group.
- **All Visible Geometry—**Double-click in an area that does not contain geometry.
- **Elements Partially in a Box—**Click and drag from one corner of the box to the opposite corner. All elements at least partially in the box are added to the active group.

- **Elements Completely in a Box—**Shift-click and drag from one corner of the box to the opposite corner. All elements completely within the box are added to the active group.

4. Use the **Remove All** icon  to remove all elements from the active group.



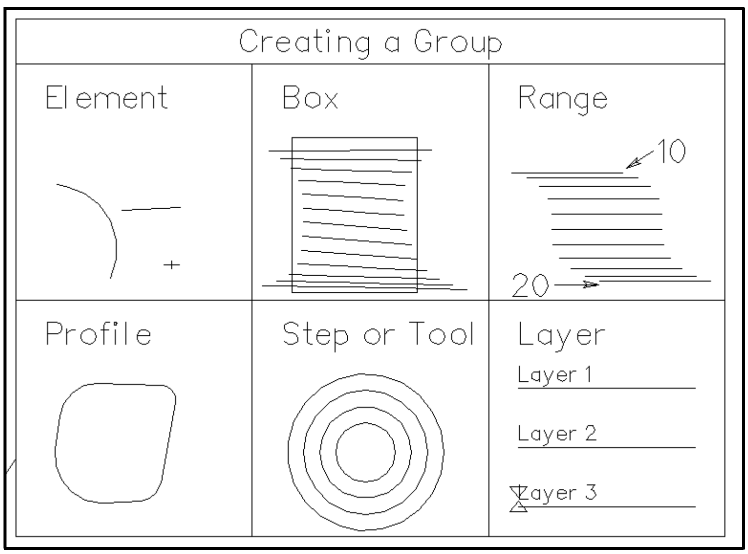
Grouping Geometry by Element

Model File: PMGROUP.PM4

Grouping by element enables you to group or ungroup single elements from the screen as you use the mouse. Perform these tasks to group geometry by element:

1. Open the model file **`PMGROUP.PM4`**.

Figure 2-22
Open PMGROUP.

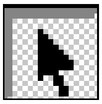


2. Select the **Group Tool Palette** icon . The **Group** tool palette is displayed.
3. Select the **Group Element** icon .
4. Move your cursor over an element and select it by clicking your left mouse button, which adds it to the group. Otherwise, Control-click on a grouped element to remove it from the group.

   You can select an element in one of these ways:

   - Select the element from the graphic view.
   - Select the element from the list view.

Grouping Geometry by Box

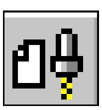


Grouping by box groups or ungroups multiple elements by defining a region or box on the screen with two corner locations. Elements inside or crossing the boundary of the box are grouped or ungrouped. Perform these tasks to group by box:

1. Select the **Group Tool Palette** icon . The **Group** tool palette is displayed.
2. Select the **Group Element** icon .
3. Click and drag the cursor in the graphics view to create a selection box to select geometry. There are two variations on box selection:
   - **Box Complete** is invoked by pressing the Shift key while making the box selection. This adds only elements that are entirely within the selection box.
   - **Box Partial** is the default method of box selection. This adds all elements with any part inside the box.

Note Hold the Control (ctrl) key while performing either a Box Complete or a Box Partial selection to ungroup items.

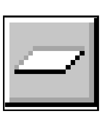
Grouping Geometry by Step



Grouping by step enables you to group or ungroup all elements that are defined with a selected step number. Perform these tasks to group geometry by step:

1. Select the **Group Tool Palette** icon . The **Group** tool palette is displayed.
2. Select the **Group by Step** icon from the **Group** tool palette.
3. Select the step to add or remove. You can select a step in one of these ways:
   - Select an element associated with the step from the graphic view.
   - Select the step from the list view.

Grouping Geometry by Layer



Grouping by layer enables you to group or ungroup all elements that are on the selected layer. Perform these tasks to group geometry by layer:

1. Select the **Group Tool Palette** icon . The **Group** tool palette is displayed.
2. Select the **Group by Layer** icon from the **Group** tool palette.
3. Select the geometry.

- Add geometry to the active group by selecting, from the list view, the layer that contains geometry you want added, or select geometry in the graphics view that is on the layer you want added to the active group.
- Remove geometry on a layer from the active group by pressing the Control key while selecting geometry from that layer in the graphics view or selecting that layer in the list view.

Saving Grouped Geometry

Save grouped geometry by performing these tasks:

1. Select the **Group Tool Palette** icon. The **Group** tool palette is displayed.
2. Select a grouping method and group your elements.
3. Select the **Name Group** icon on the **Group** tool palette. The **Name Group** dialog box is displayed.

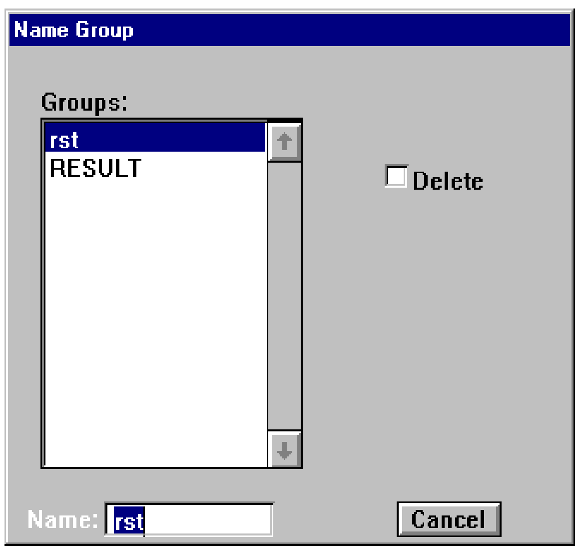


Figure 2-23 *Set the values on the Name Group dialog box.*

4. Set the **Name** input field, for example, rst.
5. Press the Enter key.

Adding and Removing Grouped Geometry

Retrieve grouped geometry by performing these tasks:

1. Select the **Group Tool Palette** icon. The **Group** tool palette is displayed.
2. Select the **Add Named Group** icon on the **Group** tool palette. The **Add Named Group** dialog box is displayed.

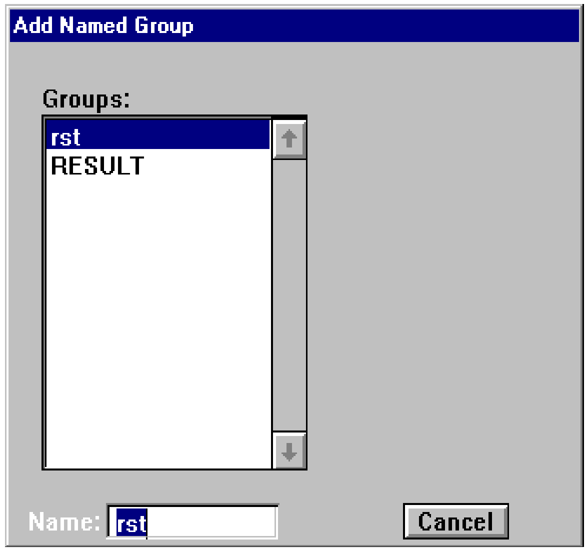
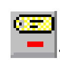


Figure 2-24 *Set the values on the Add Named Group dialog box.*

3. Set the **Name** input field to **rst**, or select **rst** from the **Groups** list.

   Note This is only an option if you named a group *rst* in the saving grouped geometry lesson above.

4. Press the Enter key.

5. Select the **Remove Named Group** icon . The Remove Named Group dialog box is displayed.

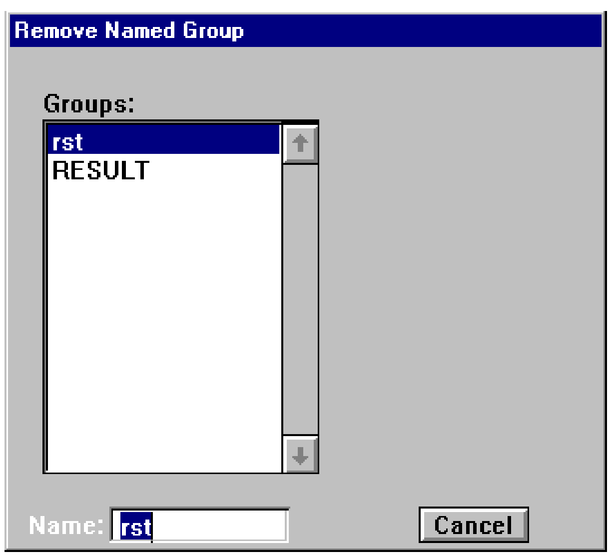


Figure 2-25 *Set the values on the Remove Named Group dialog box.*

6. Set the **Name** input field to **rst**, or select **rst** from the **Groups** list.

7. Press the Enter key.

Points to Remember

- ❑ You must create a group before you can perform procedures that involve more than one element.
- ❑ Tools that require groups are dimmed if no group is active.
- ❑ Group by element, box, range, profile, step, tool, or layer.
- ❑ You can place the Group tool palette anywhere on the screen.
- ❑ Single-click an element to add it to a group.
- ❑ Double-click a profile to add it to a group.
- ❑ Triple-click any element associated with the desired layer or step to add it to the group.
- ❑ Control-click items to remove them from a group.
- ❑ Hold the Shift key when using the group box feature to toggle from partial select to complete.

Using Edit Filters

Objectives

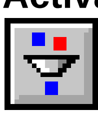
This lesson shows you how to perform these tasks:

- Activate edit filters.
- Turn edit filters on and off.
- Add filtered elements.
- Remove filtered elements.

Overview

Use Edit Filter to set the selection filtering criteria when you select an active group, view element data, or use Snap mode to enter coordinate values. When you use the filter, group, snap, and element data recognize only the element types that you specify in the Edit Filter dialog box.

Activating Edit Filters for an Active Group

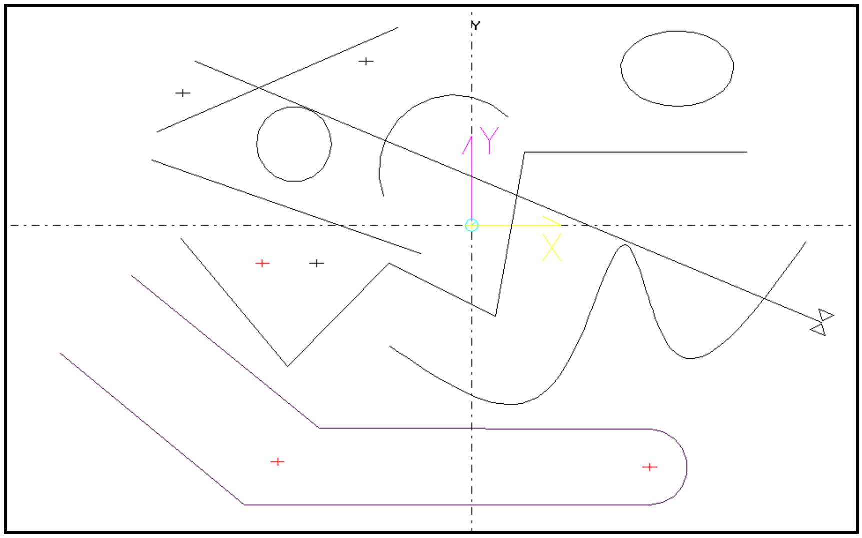


Model File: PMSNPGRP.PM4

Perform these tasks to activate edit filters:

1. Open the model file **PMSNPGRP.PM4**.

Figure 2-26
Open PMSNPGRP.



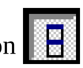
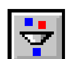
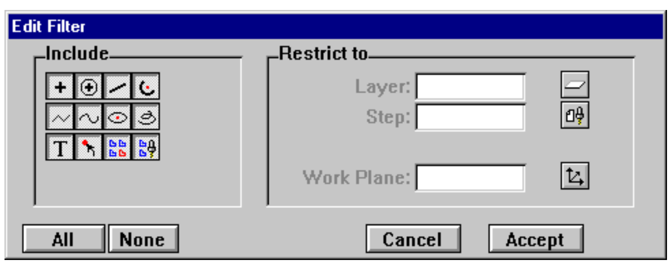
2. Select the **Group Tool Palette** icon  to open the **Group** tool palette.
3. Select the **Edit Filter** icon  from the **Group** tool palette. The **Edit Filter** dialog box is displayed.

Figure 2-27
Enter Include and Restrict values on the Edit Filter dialog box.

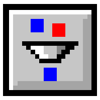


4. Under the **Include** heading, specify the element types for the system to recognize for the group functions. You can choose all element types by selecting the **All** button or exclude all element types by selecting the **None** button.

   Note All element types are activated by default. To select one element type, select the **None** button. Then specify the element type to include.

5. Under the **Restrict to** heading, you can choose to limit element selections by a specific layer, step, or work plane.
6. Select the **Accept** button.

Activating Edit Filters for Snapping

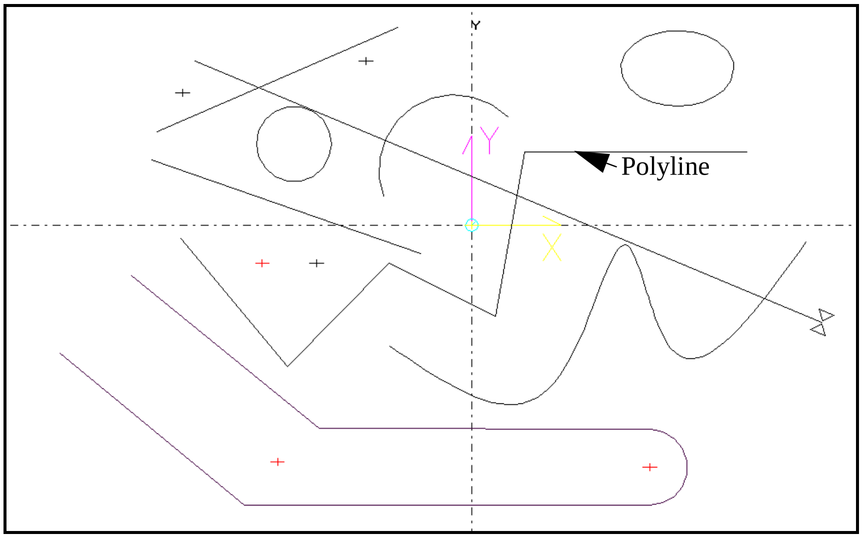


Model File: PMSNPGRP.PM4

Perform these tasks to activate edit filters:

1. Open the model file **`PMSNPGRP.PM4`**.

Figure 2-28 *Open PMSNPGRP.*



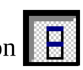
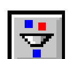
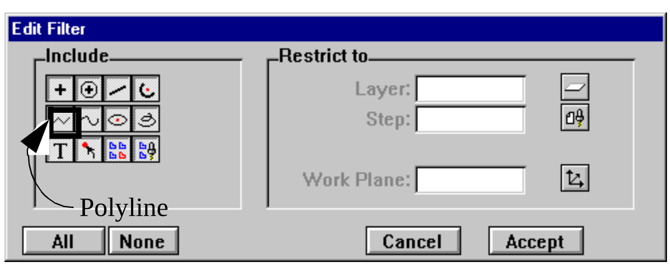
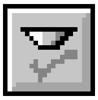
2. Select the **Group Tool Palette** icon  to open the **Group** tool palette.
3. Select the **Edit Filter** icon  from the **Group** tool palette. The **Edit Filter** dialog box is displayed.

Figure 2-29 *Enter Include and Restrict to values on the Edit Filter dialog box.*



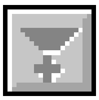
4. Include only polyline elements.
5. Select the **Accept** button.
6. Set the Snap mode to **Automatic**, and turn on all of the snap point settings that are on the readout line.
7. Select **Create—Geometry—Line**.
8. Move the cursor in the graphic view. Notice that the automatic snapping recognizes only the polyline element because the edit filter is on.

Turning Edit Filters On and Off



The on/off switch for using the edit filter is turned on when you complete the Edit Filter dialog box and select the Accept button. However, you can turn the filter on and off manually by selecting the Use Edit Filter icon.

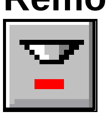
Adding Filtered Elements



Select the Add Filtered Elements icon, which is on the Group tool palette. You can select these elements:

- Points
- Holes
- Lines
- Arcs
- Text
- Splines
- Polylines
- Ellipses
- Helixes
- User Commands
- Sub Calls
- Drill Calls
- Layers
- Work Planes
- Steps

Removing Filtered Elements



If necessary, use the Remove Filtered Elements icon to remove elements that are defined by the Edit Filter dialog box.

Points to Remember

- Select the Set Edit Filter icon to quickly open the Edit Filter dialog box.
- Select the All button on the Edit Filter dialog box to include all element types in a group.
- Restrict element selection by a specific layer, step, or work plane.
- You can use the Snap modes alone or with the Edit Filter dialog box to limit the element types that you snap to.

Self-Test

Directions

Test your understanding of the concepts and procedures in this section by answering the following questions. The answers for each self-test are in *Appendix A* of this manual.

_______________ **1. Layer geometry generates code.**

a) true
b) false

_______________ **2. Step geometry generates code.**

a) true
b) false

_______________ **3. Which input field defines the height that the tool retracts to after making a cut?**

a) Level
b) Clear
c) Prof_Top
d) Insert

_______________ **4. Which input field defines the height of the top of your part?**

a) Level
b) Clear
c) Prof_Top
d) Insert

_______________ **5. What type of plane determines the orientation of geometry?**

a) kill
b) tool
c) work
d) home

______________ **6. What type of plane determines the plane the cutter axis is perpendicular to?**

a) kill
b) tool
c) work
d) home

______________ **7. Which SmartCAM feature enables you to use points and other element properties from existing geometry to enter coordinate values in input fields when you create or edit geometry?**

a) Group
b) Snap
c) Edit Filters
d) Kill Plane

______________ **8. Using Status interrupts the current task.**

a) true
b) false

______________ **9. How do you know if tools require groups if there are no active groups?**

a) Tools are dimmed.
b) Geometry is dimmed.
c) both a and b
d) neither a nor b

3

Using Job Operations

Overview

This unit shows you how to use the Job Operation Planner to build a new job operations setup, add process steps and tools, and print the Job Operations Setup Reports.

Lessons for This Unit

- Using the Job Operation Planner
- Using the Material Librarian
- Printing Job Operations Setup Reports

Using the Job Operation Planner

Objectives

This lesson shows you how to perform these tasks:

- Create a `.jof` file.
- Add and edit process steps.
- Remove and move steps and tools.
- Renumber steps.
- Sort steps and tools.

Overview

Use the Job Operation Planner to enter the units and machine files to use for code generation. You can also use it to create and edit process steps, which are combinations of tool and operation information. All of the step information you enter is then contained in a Job Operation File (`.jof`). SmartCAM generates a `.jof` when you save your model file (`.pm4`).

Every .jof includes some general information about the job to be completed. The Job Information section of the Job Operation Planner contains this data. Much of the data in this section is optional. You can leave it out to save time creating the process model and generating code. However, you must set the units in the Job Information section. It is also helpful to set the machine files at this point. You must also set the speed and feed information on the Edit Process Step dialog box to view toolpath.

The Job Information section includes the following basic information about the job:

- Units
- Part description
- Machine Define file (.smf)
- Machine Template file (.tmp)
- Material description

The steps used in the process model include the following information:

- Feedrate and units
- Spindle speed and units
- Turret and tool number
- Lead angle
- Tool shape and size

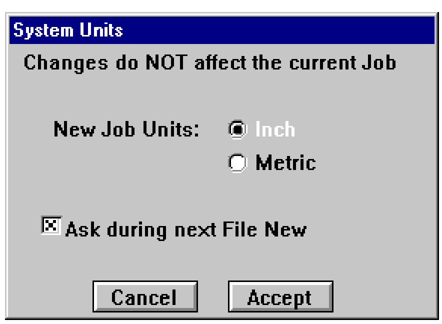
Creating a .jof File

Perform these tasks to create a .jof file:

1. If you use the same units for every job, perform these tasks:

   a. Select **Utility—System Units**. The **System Units** dialog box is displayed.

Figure 3-1
Select Utility—System Units to display the System Units dialog box.



   b. Set the **New Job Units** option switch to **Inch** or **Metric**.

   c. Turn off the **Ask during next File New** on/off switch.

   d. Select the **Accept** button.

2. If you use different units for different jobs, perform these tasks:

   a. Select **File—New**. The **New Job Units** dialog box is displayed.

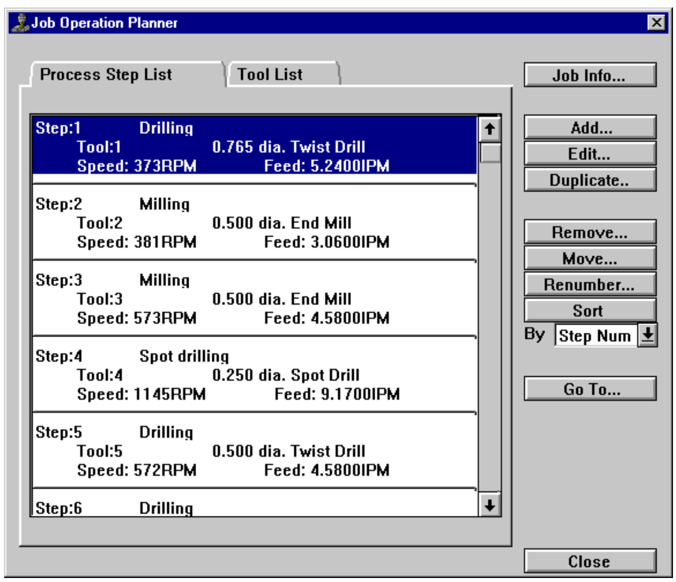
Figure 3-2
File—New invokes the New Job Units dialog box.

b. Set the **New Job Units** option switch to **Inch** or **Metric**.

c. Select the **Accept** button.

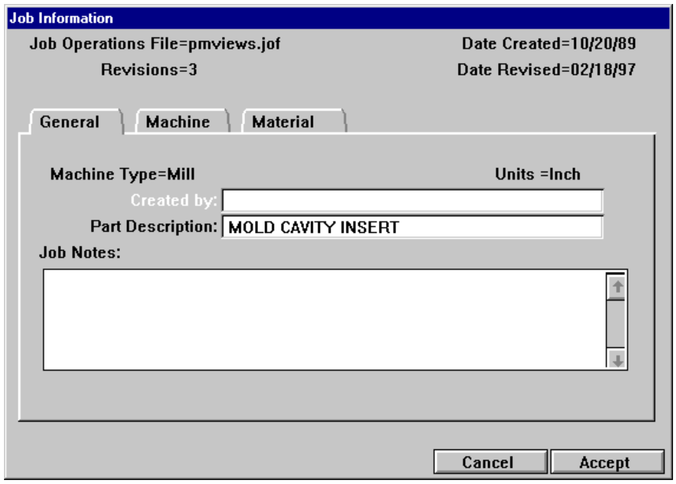
3. Select **File—Planner**. The **Job Operation Planner** dialog box is displayed.

Figure 3-3
File—Planner invokes the Job Operation Planner dialog box.



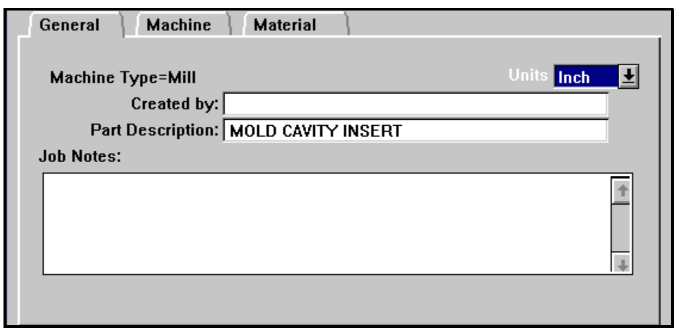
4. Select the **Job Info** button in the upper right corner of the **Job Operation Planner** dialog box. The **Job Information** dialog box is displayed.

Figure 3-4
The Job Info button invokes the Job Information dialog box.



5. Select the **General** tab. The **General** page is displayed.

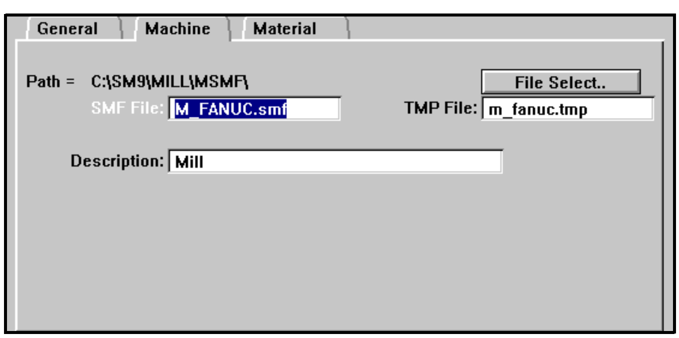
Figure 3-5
The General tab invokes the General page.



6. Set the **Created by**, **Part Description,** and **Job Notes** input fields. These fields are optional.

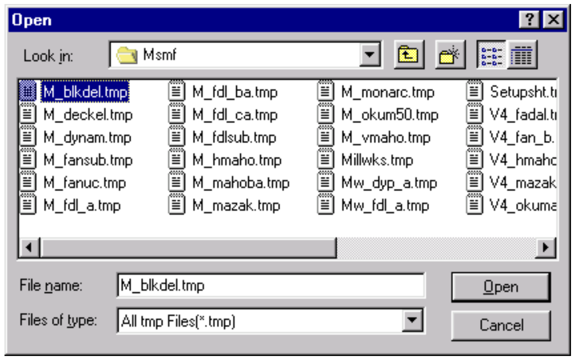
7. Select the **Machine** tab on the **Job Information** dialog box. Input fields for machine defined files and templates files are displayed.

Figure 3-6
The Machine tab invokes the Machine page.



8. Place the cursor in the **SMF File** input field, and select the **File Select** button. The **Open** dialog box is displayed.

Figure 3-7
Select an SMF file to use from the MSMF directory.



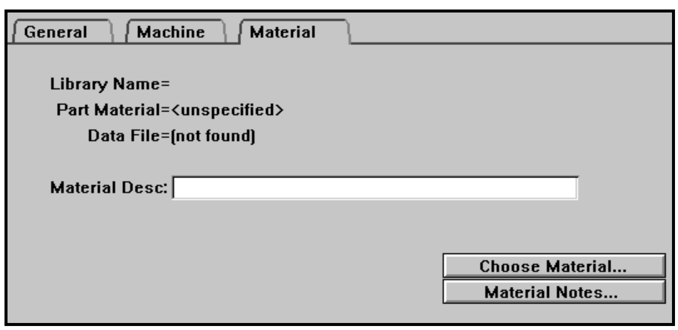
9. Select an SMF file to use from the MSMF directory (for example: `V4_FADAL.SMF`).

10. Select the **Open** button.

11. With the cursor still in the **SMF File** input field, press the Enter key, or click the right mouse button. The **TMP File** input field is then set. The name of the template file used is taken from question #1 of the .smf file.

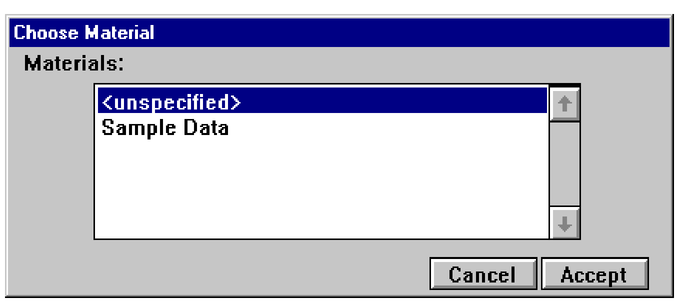
12. Select the **Material** tab. The **Material** page is displayed.

Figure 3-8
Use the Material page to set the material description.



13. Select the **Choose Material** button. The **Choose Material** dialog box is displayed.

Figure 3-9
Use the Choose Material dialog box to specify the kind of material to use.

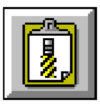


14. Select the name of the material to use.

Note Material options are set in the Material Librarian. For more information about setting these options, see *Setting Up a File with the Material Librarian,* on page 3-29.

15. Select the **Accept** button.
16. Set the **Material Desc** input field with the description of the material that you will machine the part from. This step is optional.
17. Select the **Accept** button on the bottom of the **Job Information** dialog box, see Figure 3-4.

Adding a Process Step



Remember, a step is a combination of tool and operation information. You must complete the tool description and feed/speed values to create a complete step.

Creating a Step Using Add

Perform these tasks, using a new file and either the values that are displayed in the first two steps in Figure 3-10 or values that you choose:

1. Select **File—Planner**. The **Job Operation Planner** dialog box is displayed.

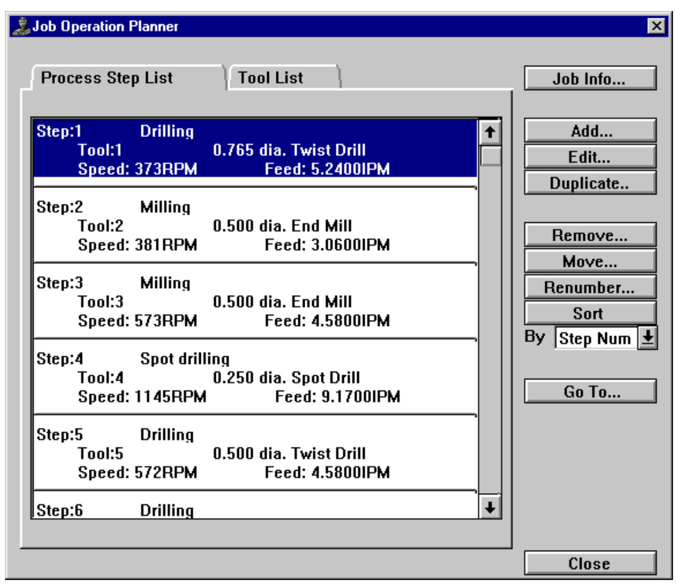
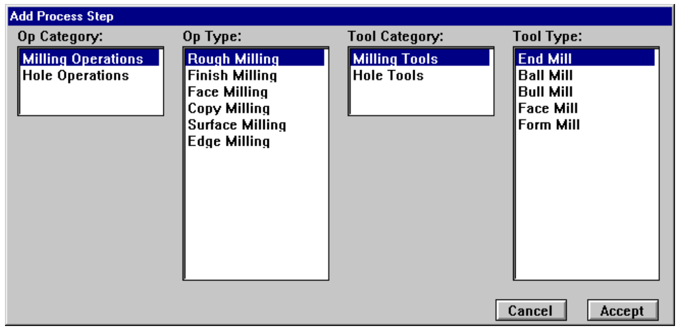


Figure 3-10 *File—Planner invokes the Job Operation Planner dialog box.*

2. Select the **Add** button in the **Job Operation Planner** dialog box. The **Add Process Step** dialog box is displayed.

Figure 3-11 Select tool information with the Add Process Step dialog box.

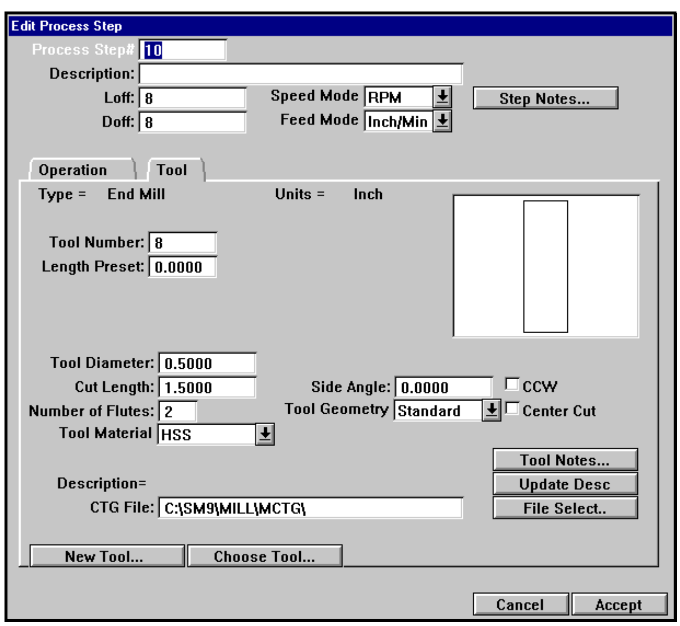


3. Complete the **Add Process Step** dialog box by selecting one operation or tool from each of the four fields:

   - **Op Category** sets the operation category to use (for example, **Milling Operations**).
   - **Op Type** sets the operation type to use. The listed operation type depends on the operation category (for example, **Rough Milling**).
   - **Tool Category** sets the tool to use. The list of tool categories depends on the operation category selected (for example, **Milling Tools**).
   - **Tool Type** sets the tool type to use. The list of tools depends on the selected tool category (for example, **End Mill**).

4. Select the **Accept** button on the bottom of the **Add Process Step** dialog box. The **Edit Process Step** dialog box is displayed.

Figure 3-12 Select the Accept button to open the Edit Process Step dialog box.



5. Set the **Process Step #** input field if needed; otherwise, a default value is placed in the input field.

   Note During each session, SmartCAM assigns colors to any steps that have numbers greater than 255. However, color assignments are not saved with these steps. Therefore, the colors for these steps may be different for each SmartCAM session.

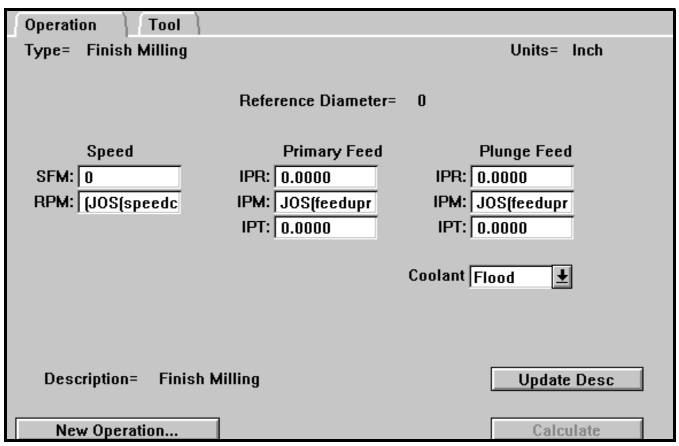
6. Set the **Speed Mode** and **Feed Mode** input fields. These determine the units to output.

7. Select the **Tool** tab, and complete the **Tool** page.

8. Select the **Operation** tab, and complete the **Operation** page.

   Note The **Feed** and **Speed** input fieldsare for Showpath (displaying the toolpath) purposes only. Code output depends on your machine, .smf file, and .tmp file.

Figure 3-13
Use the Operation page to enter operation information for the process step.



9. Select the **Accept** button on the bottom of the **Edit Process Step** dialog box, see Figure 3-12. A new step is listed in the **Process Step List** page.

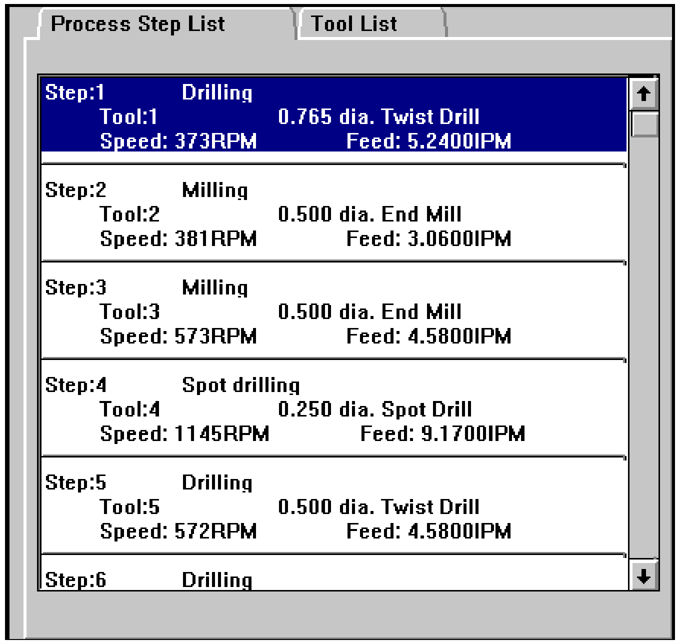


Figure 3-14
The Process Step List page is updated.

Creating a Step Using Duplicate

You can create a step using the Duplicate button on the Job Operation Planner dialog box only if other steps exist. When you select the Duplicate button, the selected step is copied to a new number. You can then change either the operation values or the tool parameters.

Perform these tasks to duplicate a step:

1. Select **File—Planner**. The **Job Operation Planner** dialog box is displayed.

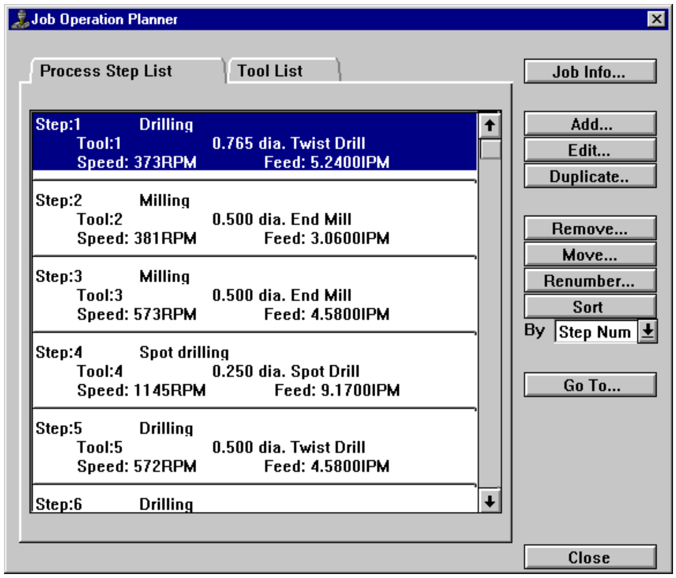


Figure 3-15
File—Planner invokes the Job Operation Planner dialog box.

2. Select the step to duplicate from the **Process Step List** tab. The step is highlighted.

3. Select the **Duplicate** button on the right side of the **Job Operation Planner** dialog box. The **Edit Process Step** dialog box is displayed.

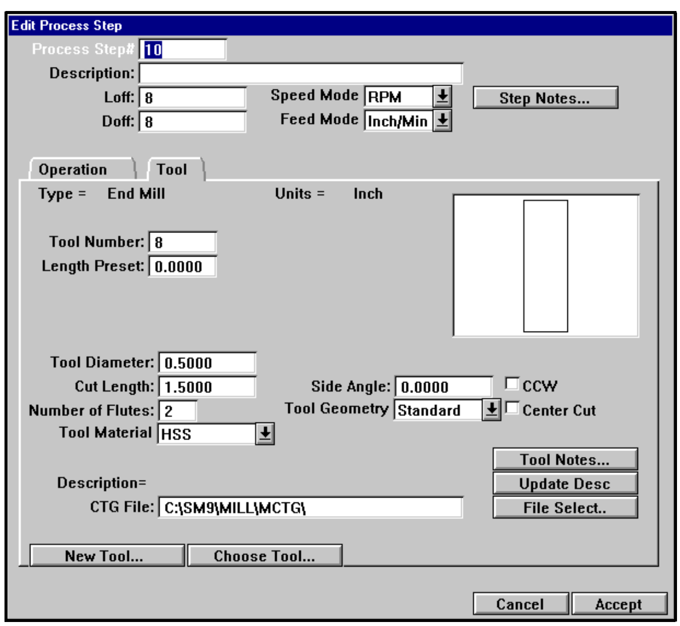


Figure 3-16
Select the Duplicate button to open the Edit Process Step dialog box.

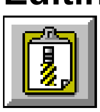
4. Change either the operation parameters or add new tool parameters.

Note Changes you make to the operation parameters are reflected only in the new step you create. However, changes you make to tool parameters are reflected in every step that uses that tool. Therefore, it is not advisable to change tool parameters when you duplicate a step. If you need a different tool for your step, select a different number tool for your new step or create a new tool. For more information about new tools, see *Using the New Tool Button,* on page 3-15.

- If you want to change operation parameters, perform these tasks:
  a. Select the **Operation** page.
  b. Make changes to the input fields as needed. The **Process Step #** input field increments to the next available number. You can change it to any number not used by a step.
- If you want to add new tool parameters, perform these tasks:
  a. Select the **Tool** page.
  b. Select the **New Tool** button. The **Choose Tool Type** dialog box is displayed.
  c. Select a new tool.
  d. Select the **Accept** button.
  e. Complete the input fields on the **Tool** page to describe the new tool.

5. Select the **Accept** button.

Editing a Process Step

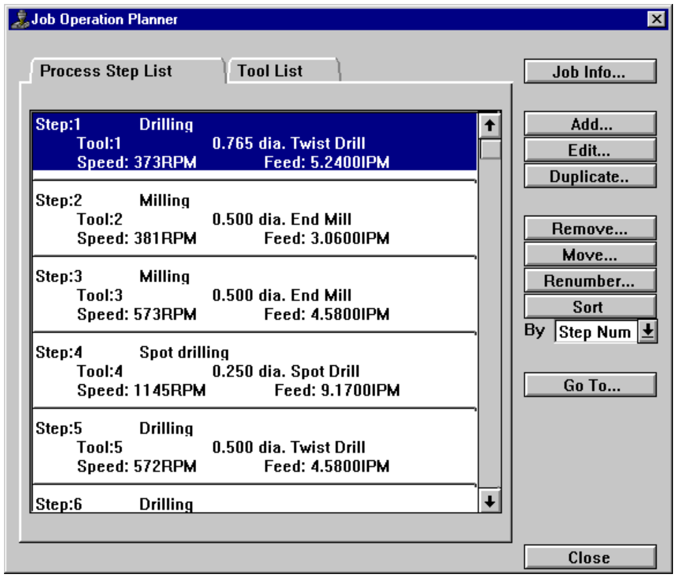


You can edit any step using the Job Operation Planner. You can use the Edit button or you can double-click a step to access the Edit Process Step dialog box.

By using the Edit button to edit a step, you can change the operation and tool descriptions to new values. Perform these tasks to edit a process step:

1. Select **File—Planner**. The **Job Operation Planner** dialog box is displayed.

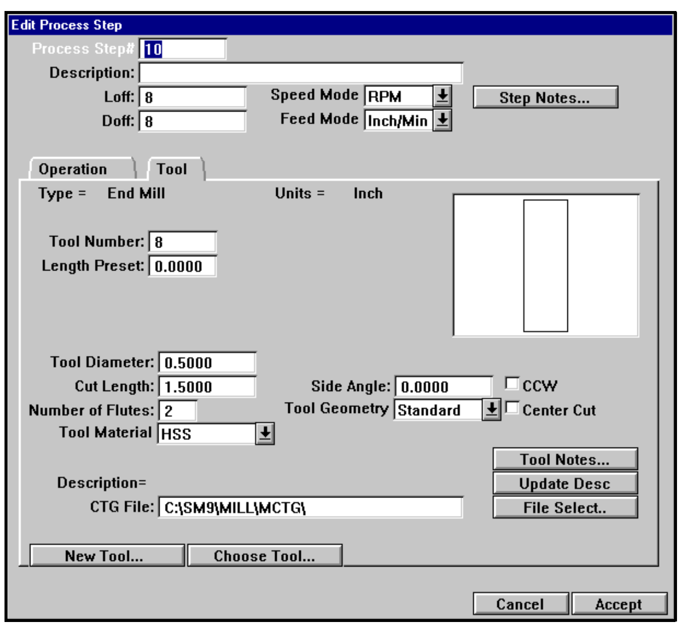
Figure 3-17
File—Planner invokes the Job Operation Planner dialog box.



2. Select the step to edit. The step is highlighted.

3. Select the **Edit** button on the right side of the **Job Operation Planner** dialog box. The **Edit Process Step** dialog box is displayed.

Figure 3-18
Select the Edit button to open the Edit Process Step dialog box.



4. Make changes to the input fields on either the **Tool** or **Operation** page.

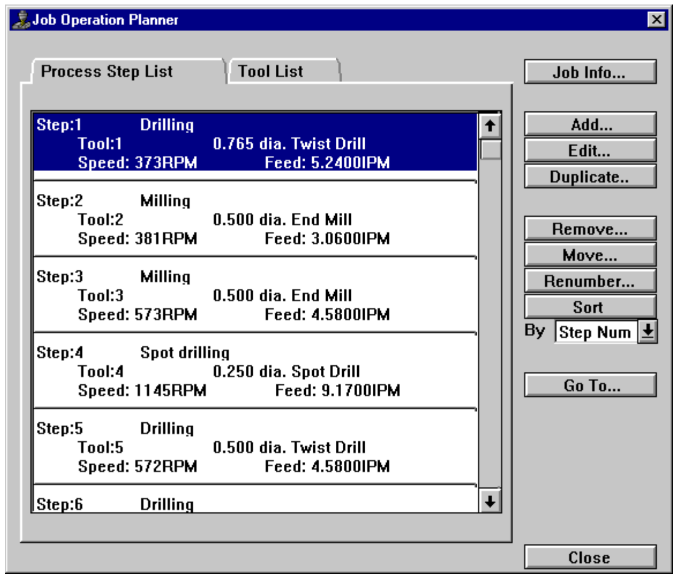
5. Select the **Accept** button when you are done.

Using the New Tool Button

Use the New Tool button on the Tool page of the Edit Process Step dialog box to use a new tool when you edit an existing step. For example, you can switch from a milling tool to a hole tool. Perform these tasks to select a new tool to use:

1. Select **File—Planner**. The **Job Operation Planner** dialog box is displayed.

Figure 3-19 *File—Planner invokes the Job Operation Planner dialog box.*



2. Select a step to edit. The step is highlighted.
3. Select the **Edit** button on the right side of the **Job Operation Planner** dialog box. The **Edit Process Step** dialog box is displayed.

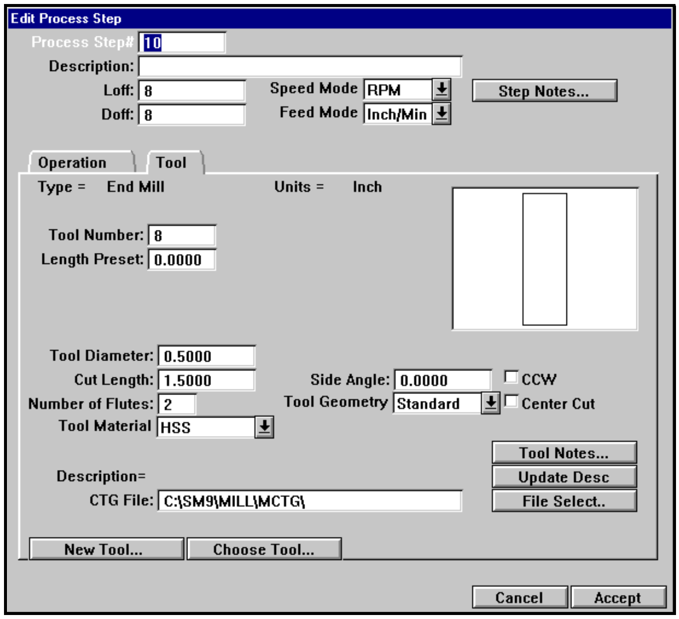


Figure 3-20
Select the Edit button to open the Edit Process Step dialog box.

4. Select the **Tool** tab. The **Tool** page is displayed.

5. Select the **New Tool** button on the bottom of the **Tool** page to define a new tool to use with this step. The **Choose Tool Type** dialog box is displayed.

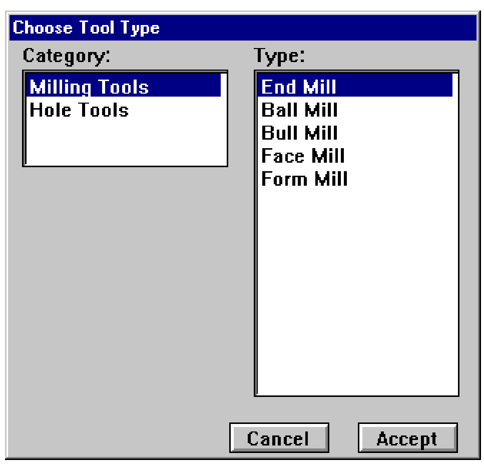


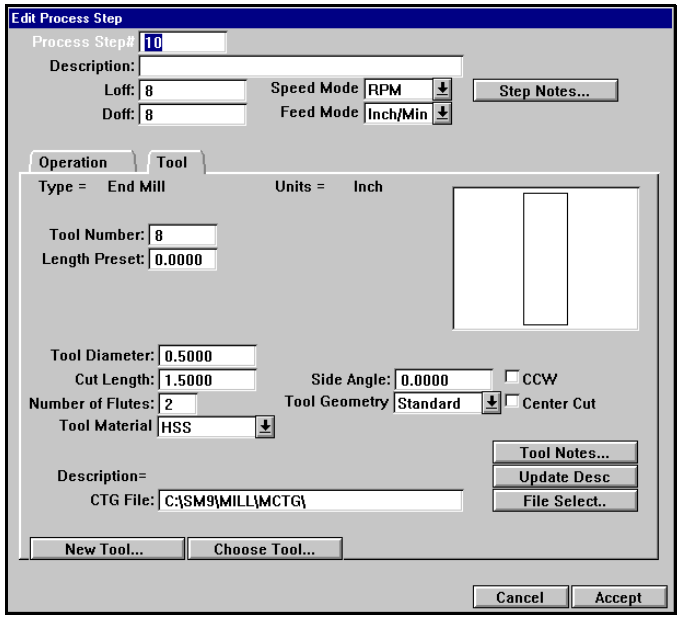
Figure 3-21
Use the Choose Tool Type dialog box to select a new tool category and type.

6. Select a new tool category. Remember, the available tool categories are based on the operation that you chose when you created the step.

7. Select a new tool type.

8. Select the **Accept** button on the bottom of the **Choose Tool Type** dialog box. The **Edit Process Step** dialog box is displayed.

Figure 3-22
Select the Accept button to open the Edit Process Step dialog box.



9. Complete the input fields on the **Tool** page to describe the new tool.
10. Select the **Accept** button.

Using the Choose Tool Button

Use the Choose Tool button to use an existing tool when you edit a step. You can either copy an existing tool to a new tool number or use it directly. Perform these tasks to choose a tool:

1. Select **File—Planner**. The **Job Operation Planner** dialog box is displayed.

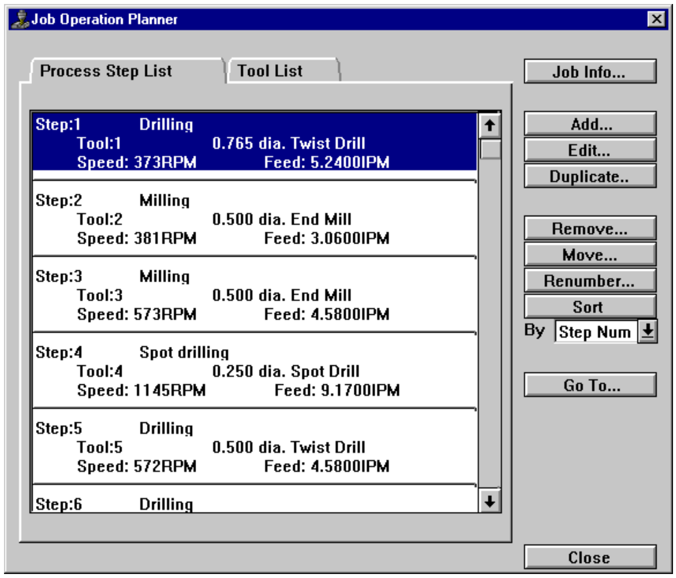


Figure 3-23
File—Planner invokes the Job Operation Planner dialog box.

2. Select a step to edit. The step is highlighted.
3. Select the **Edit** button on the right side of the **Job Operation Planner** dialog box. The **Edit Process Step** dialog box is displayed.

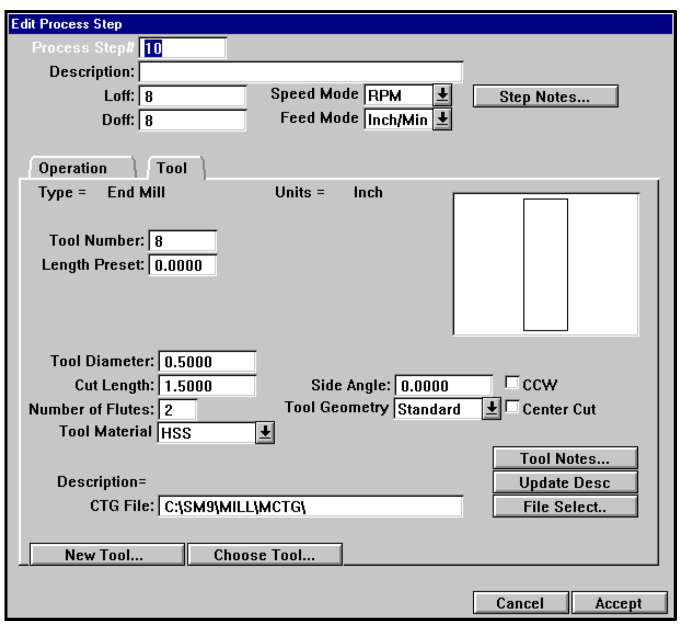
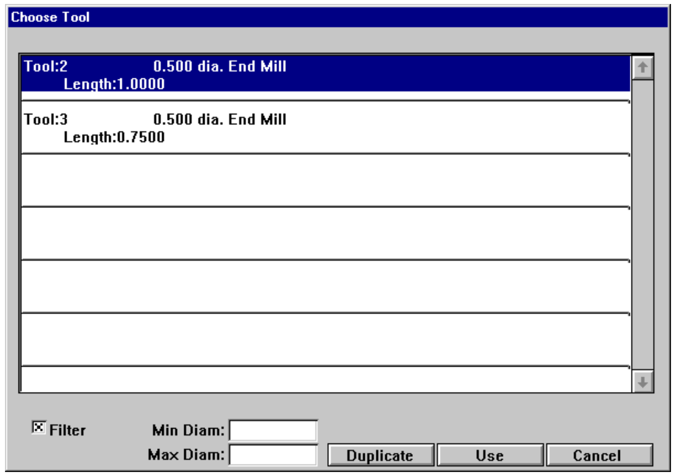


Figure 3-24
Select the Edit button to open the Edit Process Step dialog box.

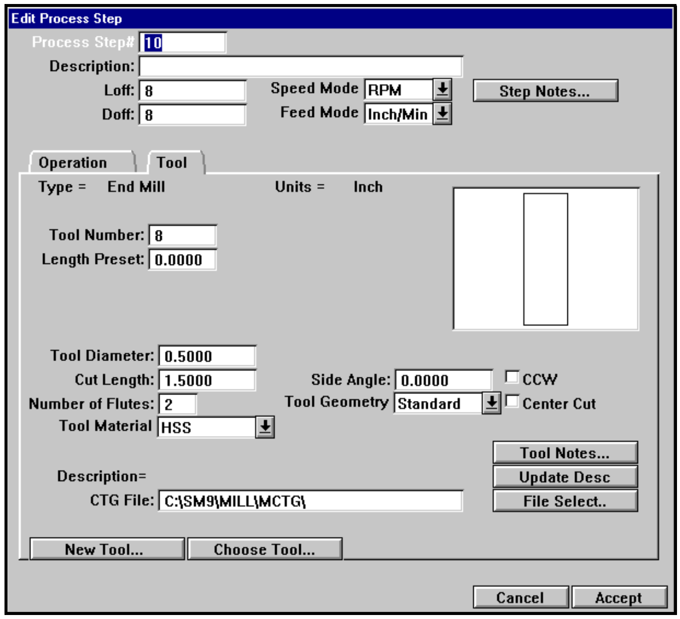
4. Select the **Tool** tab. The **Tool** page is displayed.
5. Select the **Choose Tool** button on the bottom of the **Tool** page to select an existing tool to use with this step. The **Choose Tool** dialog box is displayed.

Figure 3-25
Use the Choose Tool dialog box to select an existing tool.



6. Set **Filter** to list the tool types you need to view.
   - Filter **on** causes this dialog to display only tools that match the original tool. For example, if the tool being edited is a ball end mill, only existing ball end mills will be listed.
   - Filter **off** causes all tools in the current job (those listed in the **Job Operation Planner**) to be listed.
7. Select a tool from the list of existing tools.
8. Select either the **Duplicate** or **Use** button. The **Edit Process Step** dialog box is displayed. Values for the selected tool are set in the input fields, as follows:
   - The **Duplicate** button uses the attributes of the selected tool to create a new tool and assigns a new tool number to this new tool.
   - The **Use** button applies the existing tool to the step.

Figure 3-26
Select the Duplicate or Use button to open the Edit Process Step dialog box.



9. Select the **Accept** button to complete the edit process.

Finding Help About Tool and Operation Tabs

If you are using the Edit Process Step dialog box, you can find more information about a tool or one of the input fields using these methods:

- Use context-sensitive Help by performing these tasks:

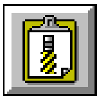
    a. Press **SHIFT +F1** to display a  cursor. Place the cursor anywhere on the Tool or Operation tab and click. A Help topic about Edit Process Step is displayed with an overview of information. If you select the **Fields** button for this topic, a new Help topic is displayed that describes the input fields for the general parameters.

    b. To learn more about either the Tool or Operation tab, select the underlined text for a tab in the Help topic. A Help topic is displayed that has an overview and buttons for **Fields** and **How To**. Each input field is described in Fields. The How To provides operational information about how to change the tab.

- Use **Search** to find information about specific input fields. You enable Search when you press **F1**, select **Search**, and then enter the word to learn about.

- Use **Glossary** to find a word and how it is used. You can also press **F1** and select the title **Glossary**. The Glossary contains conceptual information that applies in a broad context. Use **Search** or the **Fields** button when you need information about specific input fields.

Removing Steps and Tools



You can remove unused steps and tools from the Job Operation Planner dialog box. Only the steps or tools that are not used or selected as active are listed for possible removal.

Perform these tasks to remove unused steps and tools from the Job Operation Planner dialog box:

1. Select **File—Planner**. The **Job Operation Planner** dialog box is displayed.

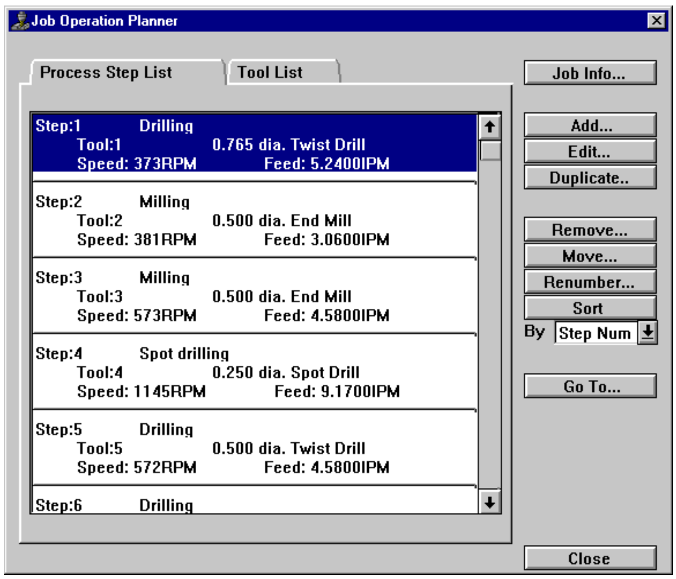


Figure 3-27 *File—Planner invokes the Job Operation Planner dialog box.*

2. Select the **Remove** button on the right side of the **Job Operation Planner** dialog box. The **Remove** dialog box is displayed.

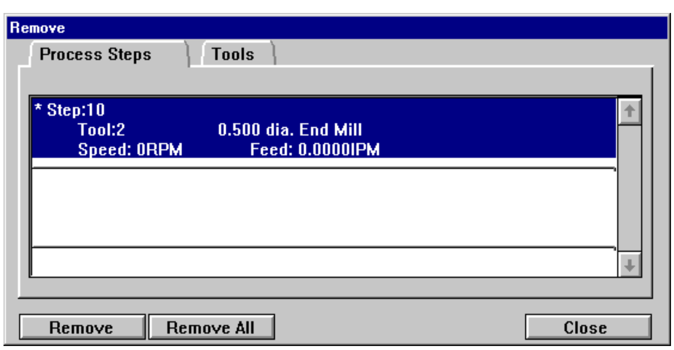


Figure 3-28 *Use the Remove dialog box to specify steps or tools to remove from the Job Operation Planner.*

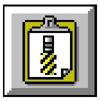
3. Select the **Process Steps** tab to remove a step, or select the **Tools** tab to remove individual tools.

4. Select the step or tool to remove.

5. Select the **Remove** or **Remove All** button:

   - The **Remove** button removes the highlighted step or tool.
   - The **Remove All** button removes all unused steps or tools, depending on the tab that is selected.

Moving Steps and Tools



You can sequence one step or tool at a time using Move.

Moving Steps

Perform these tasks to move a step to a different place in the sequence:

1. Select **File—Planner**. The **Job Operation Planner** dialog box is displayed.

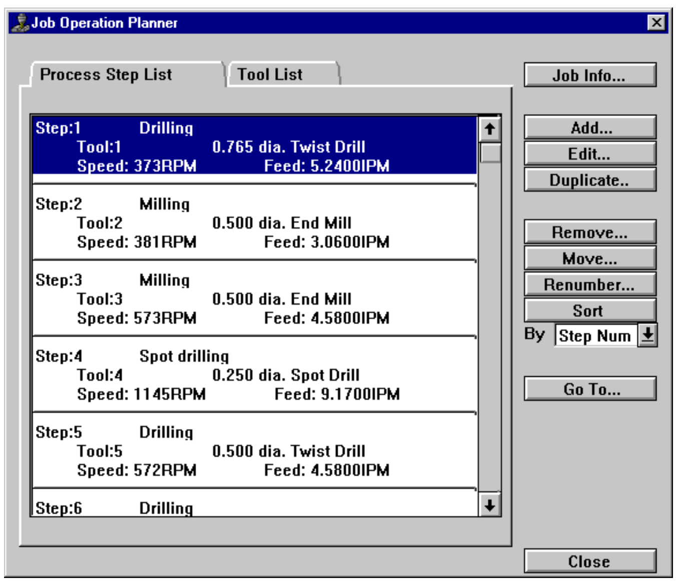


Figure 3-29
File—Planner invokes the Job Operation Planner dialog box.

2. Select the **Process Step List** tab to move a step.
3. Select the step to move.
4. Select the **Move** button on the right side of the dialog box. The **Move Process Step** dialog box is displayed.

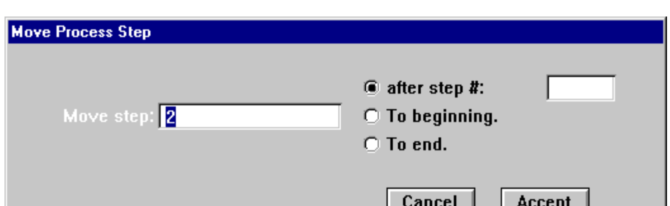


Figure 3-30
Use the Move Process Step dialog box to specify where to move the step.

5. Specify where to move the step. You can move the step to these locations:
   - After a specified step number
   - To the beginning of the list of steps
   - To the end of the list of steps
6. Select the **Accept** button. The step is moved to its new location.

Moving Tools

Perform these tasks to move tools after a specified tool number, to the beginning of the list of tools, or to the end of the list of tools:

1. Select **File—Planner**. The **Job Operation Planner** dialog box is displayed.

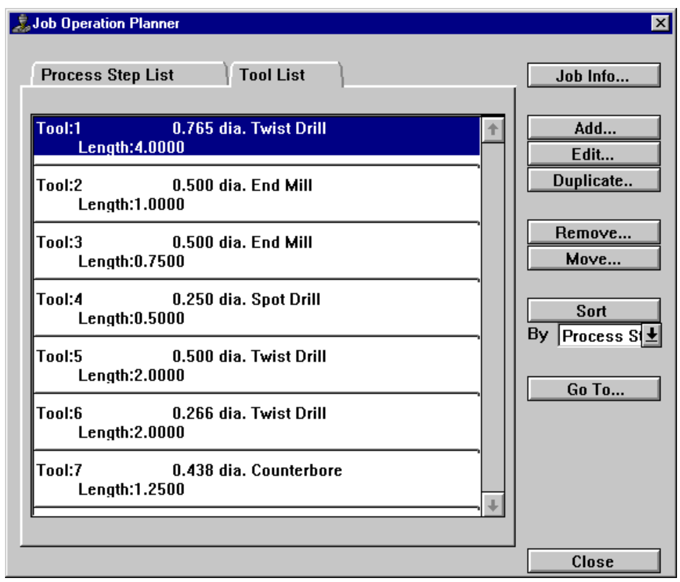


Figure 3-31 *File—Planner invokes the Job Operation Planner dialog box.*

2. Select the **Tool List** tab to move a tool. The **Tool** page is displayed.
3. Select the tool to move.
4. Select the **Move** button on the right side of the dialog box. The **Move Tool** dialog box is displayed.

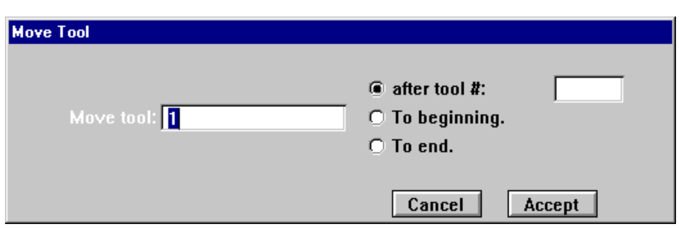
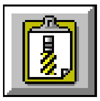


Figure 3-32 *Use the Move Tool dialog box to specify where to move the tool.*

5. Specify where to move the tool. You can move the tool to these locations:
   - After a specified tool number
   - To the beginning of the list of tools
   - To the end of the list of tools
6. Select the **Accept** button. The tool is moved to its new location.

Renumbering Steps



Perform these tasks to update or adjust step numbers:

1. Select **File—Planner**. The **Job Operation Planner** dialog box is displayed.

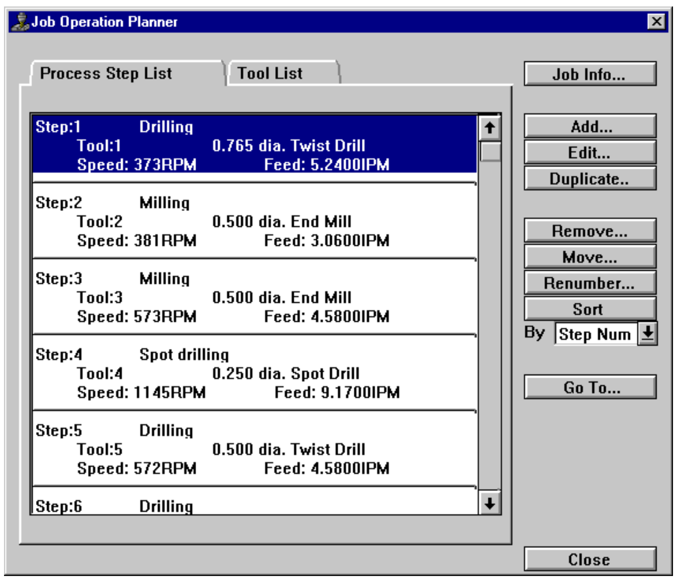


Figure 3-33
File—Planner invokes the Job Operation Planner dialog box.

2. Select the **Process Step List** tab. The **Process Step List** page is displayed.
3. Select the **Renumber** button on the right side of the dialog box. The **Renumber Process Step** dialog box is displayed.

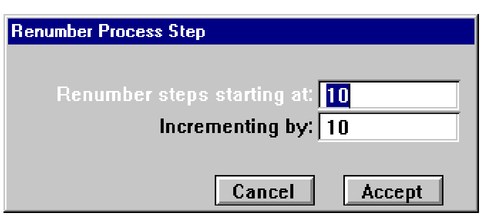
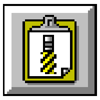


Figure 3-34
Set the start and increment numbers by which to renumber steps.

4. Set the starting value for renumbering. This is the number of the first step.

5. Set the value to increment by between each step.
6. Select the **Accept** button.

Sorting Steps and Tools



You can sort steps and tools in different ways. Sort steps by the step number or the order used in the process model. Sort tools by the steps they are used in, the order they are used in the process model, or by the tool number.

Sorting Steps

Perform these tasks to sort steps:

1. Select **File—Planner**. The **Job Operation Planner** dialog box is displayed.

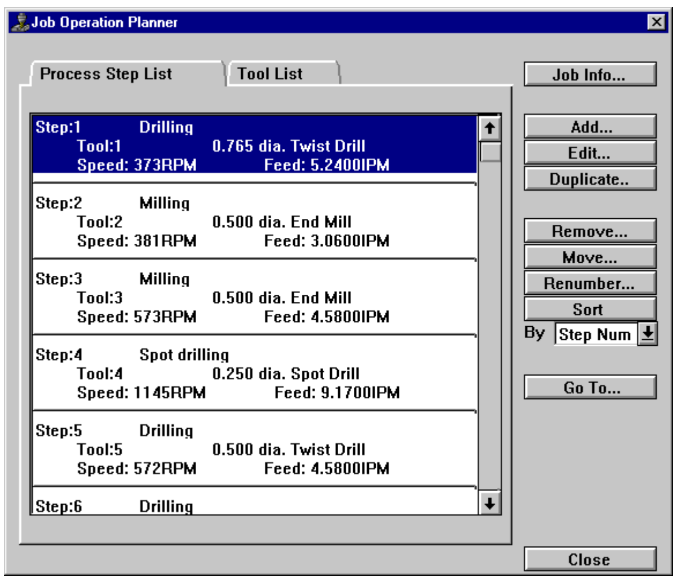


Figure 3-35
File—Planner invokes the Job Operation Planner dialog box.

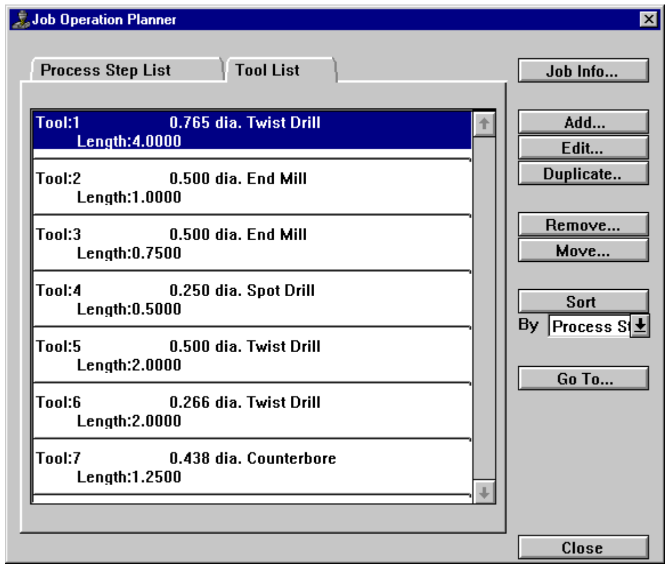
2. Select the **Process Step List** tab to sort the steps.
3. Set the **By** selector switch to one of the following methods for sorting the steps:
   - **Step Num** to sort by the step numbers
   - **Proc Mdl** to sort by the order used in the process model
4. Select the **Sort** button to sort the steps.

Sorting Tools

Perform these tasks to sort tools:

1. Select **File—Planner**. The **Job Operation Planner** dialog box is displayed.

Figure 3-36
File—Planner invokes the Job Operation Planner dialog box.



2. Select the **Tool List** tab to sort the tools.
3. Set the **By** selector switch to one of the following methods for sorting the tools:
   - **Step Num** to sort by the step numbers they were used in
   - **Proc Mdl** to sort by the order used in the process model
   - **ToolNumber** to sort by the tool number
4. Select the **Sort** button to sort the tools.

Points to Remember

- ❑ Machine files must be set on the Job Information dialog box.
- ❑ A tool description and feed/speed values must be set to complete a process step.
- ❑ A step must exist if you want to use the Duplicate feature.
- ❑ If you make changes to a tool parameter when you use the Duplicate feature, the changes are reflected in every step that uses that tool.
- ❑ Only the steps or tools that are not used or selected as active are listed for possible removal.
- ❑ You can move only one step or tool at a time using the Move feature.
- ❑ Sort steps by step number or order used in the process model.
- ❑ Sort tools by the steps they are used in, the order they are used in the process model, or the tool number.

Using the Material Librarian

Objectives

This lesson shows you how to perform these tasks:

- Set up a material librarian file, which contains process options based on a specific part material.
- Open the material librarian file in the Job Operation Planner.

Overview

Use the Material Librarian to build and edit material information that the Job Operation Planner can use to calculate process step information like feeds and speeds. This keeps you from entering and reentering the same tool and process information for a specific material type. In some cases, if the Material Librarian does not contain an exact match, the resulting process options may be interpolated from available information.

Setting Up a File with the Material Librarian

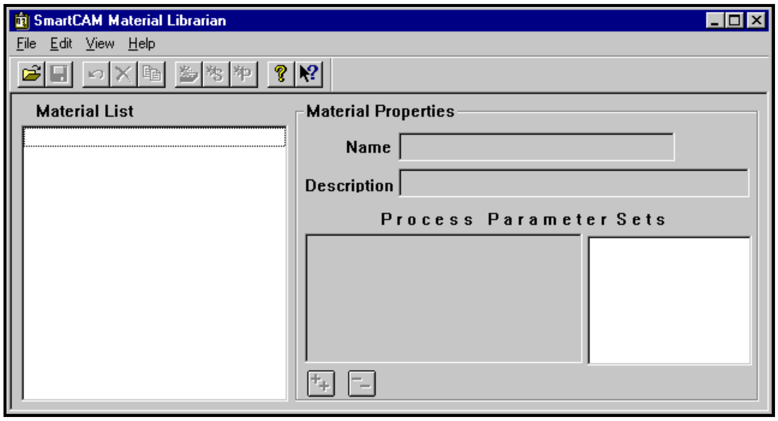
Perform these tasks to set up a file with the Material Librarian:

- Add material.
- Insert a parameter set.
- Insert a nested parameter set.
- Add a tool diameter parameter set.
- Define the process parameters.
- Duplicate a parameter set, if necessary.

Adding Material

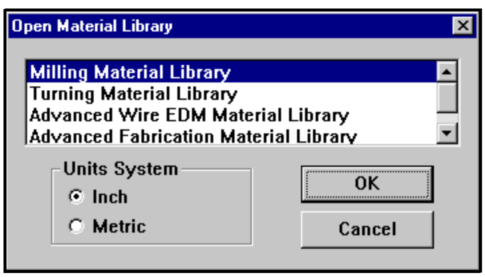
1. Select the **Material Librarian** icon from the SmartCAM program group. The **SmartCAM Material Librarian** is displayed.

Figure 3-37 *Open the SmartCAM Material Librarian.*



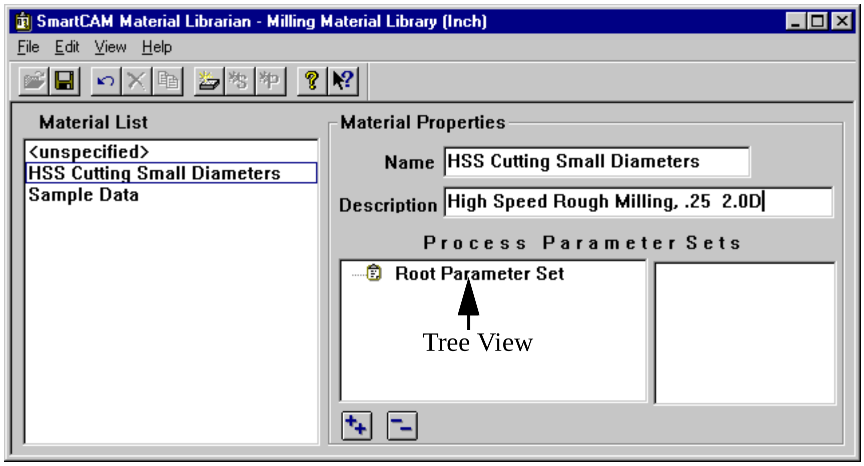
2. Select **File—Open Library**. The **Open Material Library** dialog box is displayed.

Figure 3-38 *Open the Open Material Library dialog box.*



3. Select **Milling Material Library**.
4. Specify the **Units System** to use as the default.
5. Select the **OK** button. The updated **SmartCAM Material Librarian** dialog box is displayed.
6. Select **Edit—Add Material**.
7. Change the name of the material. The new material is added in alphabetic order.
8. Enter the material description.

Figure 3-39
Set the name and description.

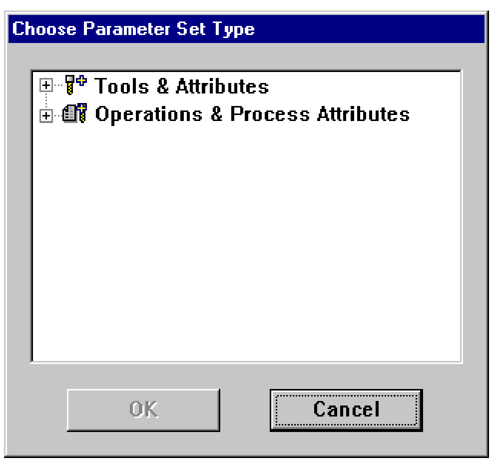


9. Keep the Material Librarian open and continue with the next procedure.

Inserting a Parameter Set

1. Highlight **Root Parameter Set**, which is displayed in the tree view, see Figure 3-39.
2. Select **Edit—Insert Parameter Set**. The **Choose Parameter Set Type** dialog box is displayed.

Figure 3-40
Open the Choose Parameter Set Type dialog box.



3. Select the plus sign that is next to **Operations & Process Attributes**. The section is expanded.
4. Select **Operation** from the list.

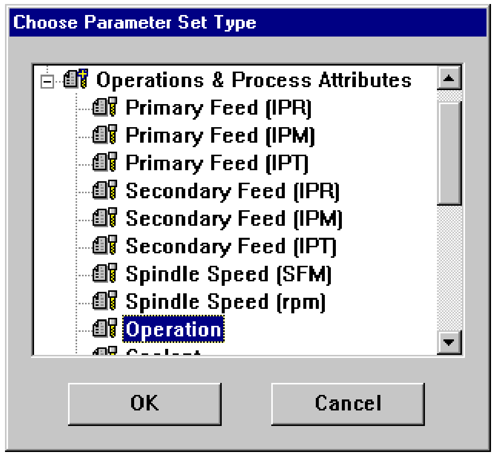


Figure 3-41 *Select Operation from the list.*

5. Select the **OK** button. The parameter set is added, and the **Choose Parameter Set Type** dialog box is closed.

6. Set the **Operation** selector switch to **Rough Milling**. This specifies the operation type for the parameter set.

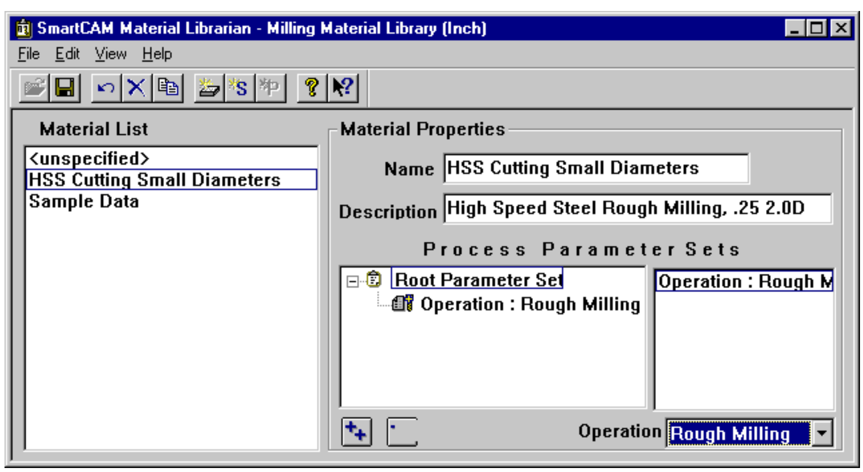


Figure 3-42 *Set the Operation selector switch.*

7. Keep the Material Librarian open and continue with the next procedure.

Inserting a Nested Parameter Set

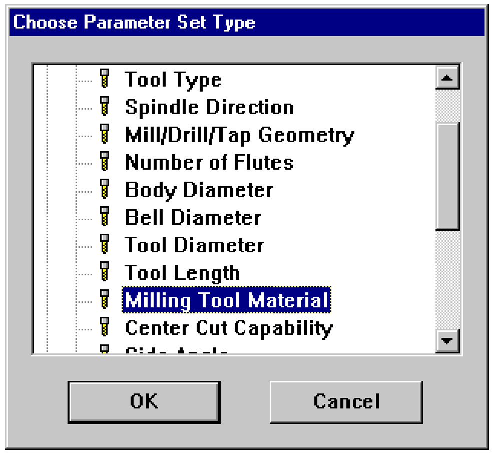
1. Select **Operation: Rough Milling** from the root parameter set in the tree view.

2. Select **Edit—Insert Parameter Set**. The **Choose Parameter Set Type** dialog box is displayed.

3. Select the plus sign that is next to **Tools & Attributes**. The section is expanded.

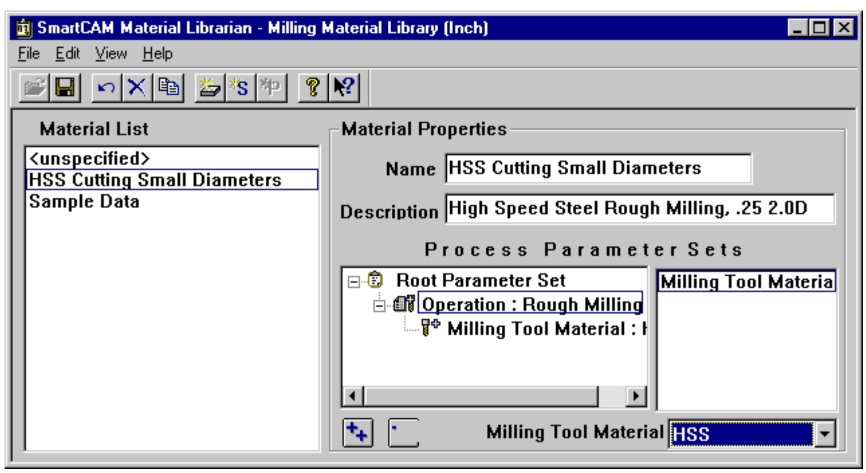
4. Select **Milling Tool Material** from the list.

Figure 3-43
The Tools and Attributes section is expanded.



5. Select the **OK** button. The parameter set is added, and the **Choose Parameter Set Type** dialog box is closed.
6. Set the **Milling Tool Material** selector switch to **HSS**. This specifies the operation type for the parameter set.
7. Press Enter.

Figure 3-44
Set the Tool Material value.



8. Keep the Material Librarian open, and continue with the next procedure.

Adding a Tool Diameter Parameter Set

1. Select **Milling Tool Material: HSS** from the tree view.
2. Select **Edit—Insert Parameter Set**. The **Choose Parameter Set Type** dialog box is displayed.

3. Select the plus sign that is next to **Tools & Attributes**. The section is expanded.

4. Select **Tool Diameter** from the list.

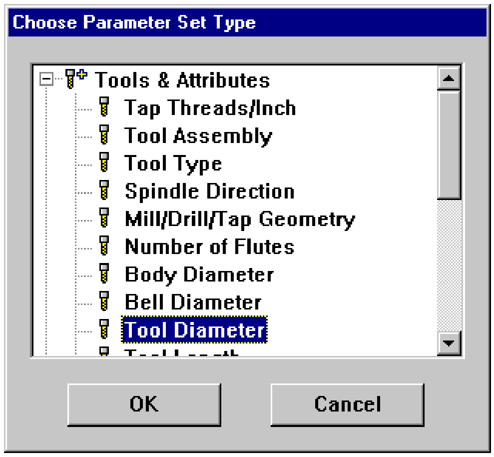


Figure 3-45
Select Tool Diameter.

5. Select the **OK** button. The parameter set is added, and the **Choose Parameter Set Type** dialog box is closed.

6. Set the **Tool Diameter** input field to **.25**.

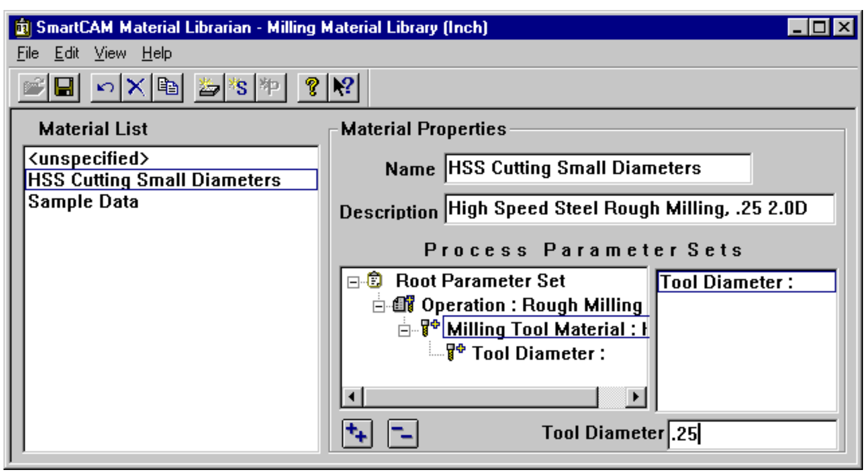


Figure 3-46
Set the Tool DIameter value.

7. Press Enter.

8. Keep the Material Librarian open and continue with the next procedure.

Defining Process Parameters

Process parameters are the last level of information that you add to a material library. These are usually manufacturing parameters such as feeds and speeds.

1. Select **Tool Diameter: .25** from the tree view.

2. Select **Edit—Insert Process Parameter**. The **Choose Process Parameter Type** dialog box is displayed.

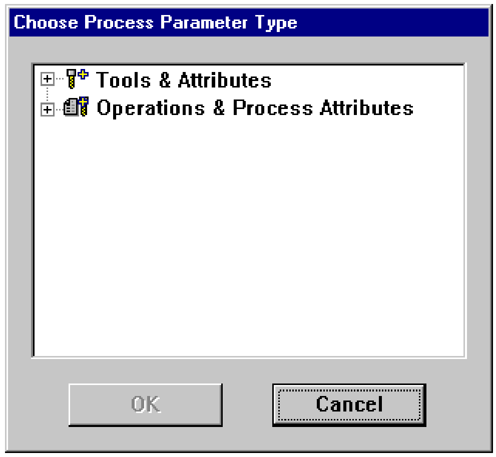


Figure 3-47
Open the Choose Process Parameter Type dialog box.

3. Select the plus sign that is next to **Operations & Process Attributes**. The section is expanded.

4. Select **Primary Feed.**

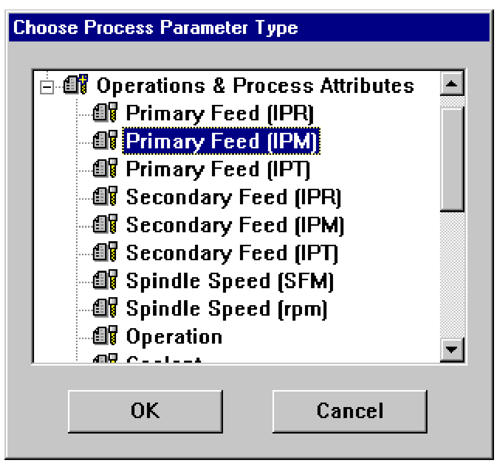
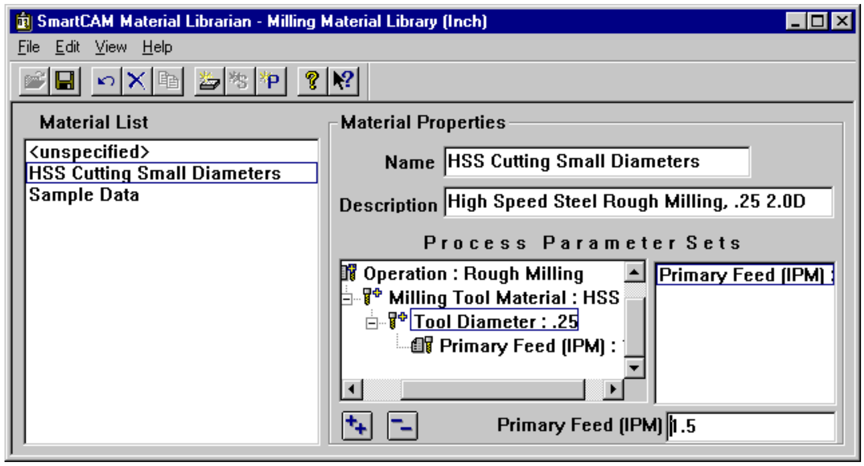


Figure 3-48
Select Primary Feed.

5. Select the **OK** button. The process parameter is added, and a box is drawn around it, indicating that this process parameter is the active parameter set.

6. Set the **Primary Feed** input field to the feed rate that you want to use.

Figure 3-49
Set the Primary Feed input field to the rate that you want to use.



Note SmartCAM identifies both primary and secondary feeds for milling operations. Primary feeds are the horizontal feed rates, and secondary feeds are the vertical rates, also called plunge feeds.

7. Repeat steps 2 through 6, substituting Spindle Speed to set the speed.

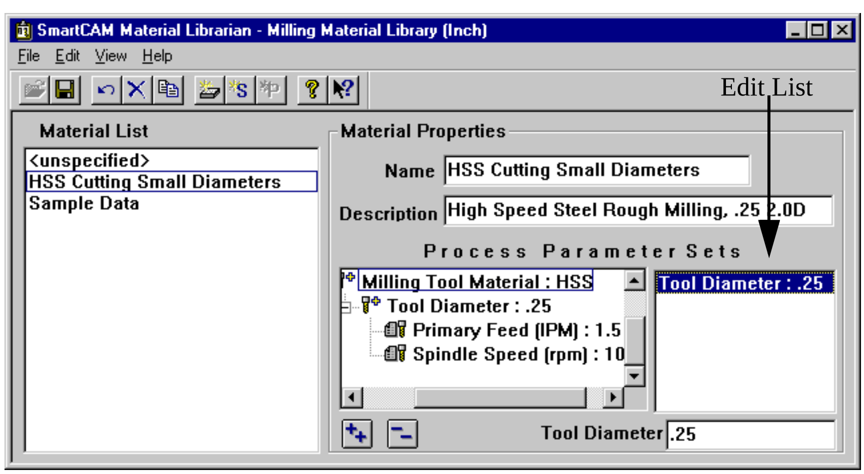
Duplicating a Parameter Set

A material library can be simple or complex. To reduce the time it takes to build a complex library, build an initial parameter set. Then duplicate the parameters, so you can change their attributes.

1. Select **Milling Tool Material: HSS** from the tree view.
2. Select **Tool Diameter: .25** from the edit list.

Note You can edit only values that are displayed in this list.

Figure 3-50
Select Tool Diameter: .25 from the edit list.



3. Select **Edit—Duplicate** to copy the **Tool Diameter: .25** parameter set and all of its parameters.
4. Select the new **Tool Diameter: .25**. (It is displayed below the original.)
5. Set the **Tool Diameter** input field to **2** to change the tool diameter.
6. Press Enter.
7. Select **Tool Diameter 2** from the tree view. The **Primary Feed** and **Spindle Speed** for **Tool Diameter 2** are displayed in the edit list.
8. Select **Primary Feed** from the edit list.
9. Set the **Primary Feed** input field to the new feed rate.
10. Press Enter.
11. Select **Spindle Speed** from the edit list.
12. Set the **Spindle Speed** input field to the new spindle speed.
13. Press Enter. The feeds and speeds for the operation are set. You can repeat the duplicating process to add as many tool size parameter sets as you need.

Note Although you have not defined the feeds and speeds for tool diameters between .25 and 2.0, SmartCAM interpolates feeds and speeds for those tools.

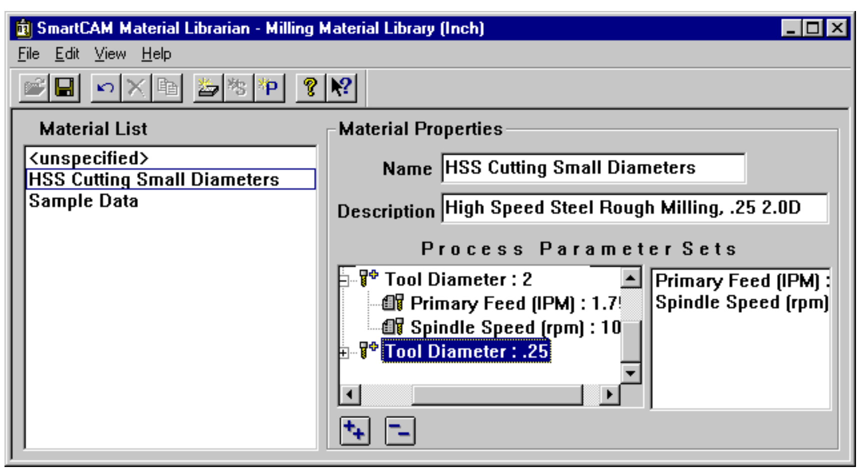


Figure 3-51 *The feeds and speeds for the operation are set.*

Viewing Results

To view the material that you added to the material library, select the plus sign that is next to **Tool Diameter: .25** from the tree view.

Saving and Exiting

To save the material library and exit the Material Librarian, perform these tasks:

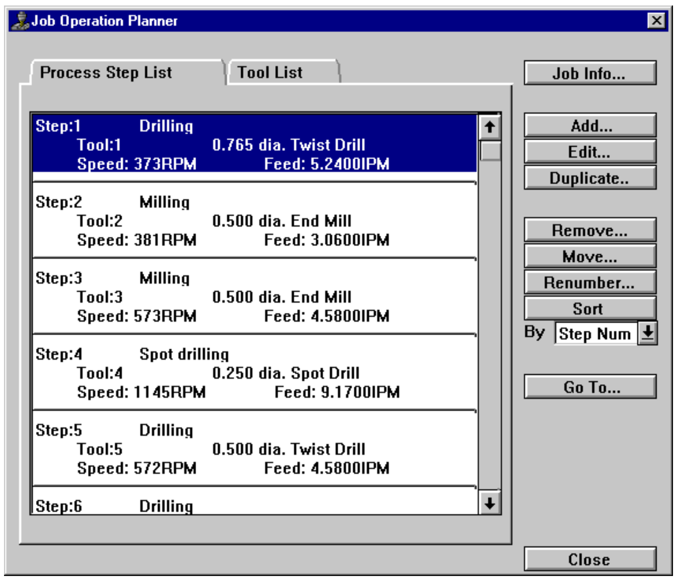
1. Select **File—Save Changes** to save the file.
2. Select **File—Exit** to close the Material Librarian.

Opening a Material Librarian File

Once you set a material librarian file, you can open it in the Job Operation Planner. To open a material librarian file, perform these tasks:

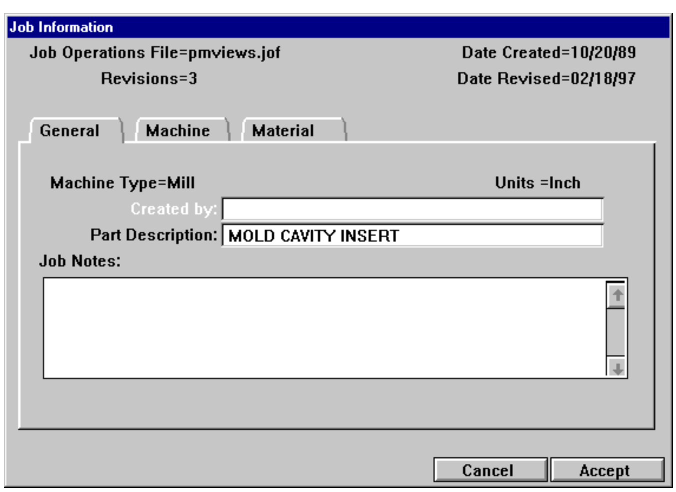
1. Select **File—Planner** from the Production Milling application. The **Job Operation Planner** dialog box is displayed.

Figure 3-52
File—Planner invokes the Job Operation Planner dialog box.



2. Select the **Job Info** button. The **Job Information** dialog box is displayed.

Figure 3-53
The Job Info button invokes the Job Information dialog box.



3. Select the **Material** tab. The **Material** page is displayed.

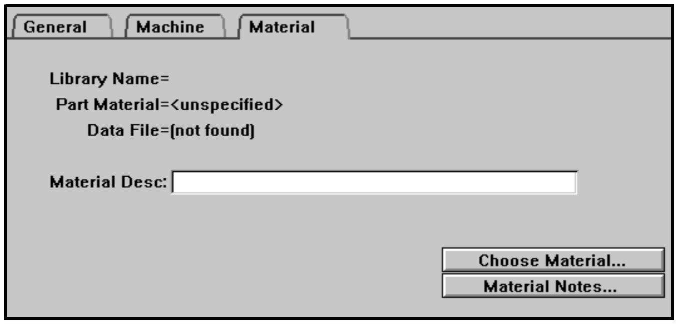


Figure 3-54
Use the Material page to set the material description.

4. Select the **Choose Material** button. The **Choose Material** dialog box is displayed.

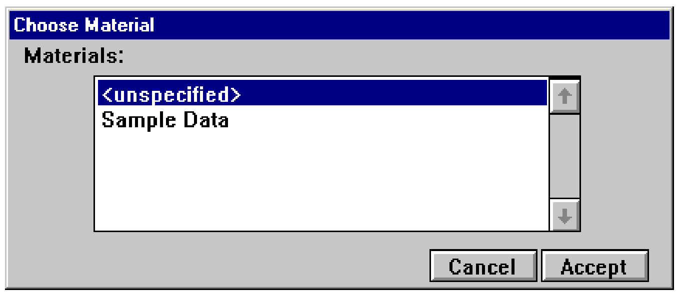


Figure 3-55
Use the Choose Material dialog box to specify the file that you set up in the Material Librarian.

5. Select the name of the file that was set up in the Material Librarian.
6. Select the **Accept** button.
7. Select the **Accept** button on the **Job Information** dialog box.
8. Select the **Close** button on the **Job Operation Planner** dialog box.

Points to Remember

- ❑ The Material Librarian icon is found in the SmartCAM program group and not in the Production Milling application.
- ❑ The Material Librarian enables you to choose tools, operations, and processes for specific material.
- ❑ Access a Material Librarian file from the Material tab of the Job Information dialog box.

Printing Job Operations Setup Reports

Objectives

This lesson shows you how to perform these tasks:

- Print a job information report.
- Print a tooling report.
- Print a step report.

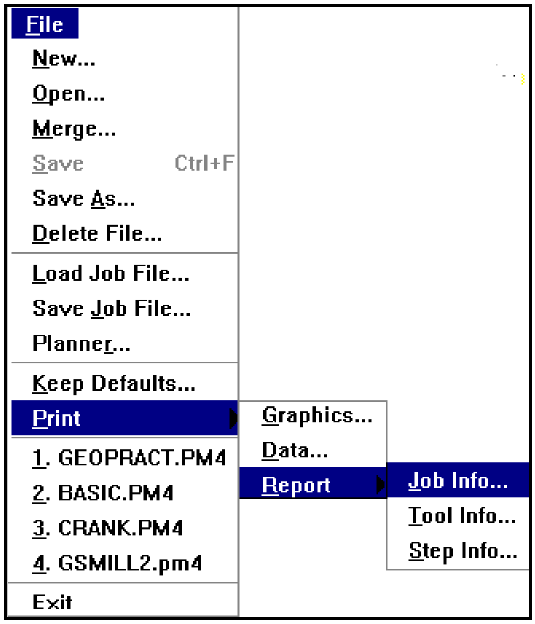
Overview

Use reports to provide operator instructions and machine setup information.

Select File—Print—Report to open the job operations setup reports submenu. There are three types of reports you can create in the job operations setup:

- **Job Info** opens the Print Job Info Report dialog box so you can print general information, machine information, and part material information.
- **Tool Info** opens the Print Tooling Report dialog box so you can print information about the tools in the job operations setup.
- **Step Info** opens the Print Step Report dialog box so you can print information about the steps in the job operations setup.

Figure 3-56
Select File—Print—Report, and specify the type of report to work with.



Printing a Job Information Report

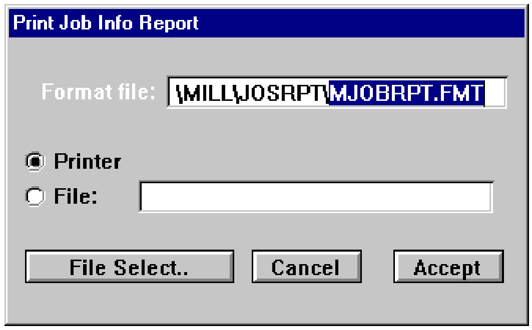
Use Job Info to print job information or create a report file that contains job information for the open process model. The report can contain job information, machine information, and part material information.

Printing Job Information

Perform these tasks to print job information:

1. Select **File—Print—Report—Job Info** to open the **Print Job Info Report** dialog box.
2. Specify the path and filename of the report format file, or use **File Select** to choose a filename from the list of `.fmt` files on disk.
3. Specify the destination for the report, **Printer** or **File**. If you select **File**, enter the path and filename in the corresponding input field, or use **File Select** to choose a filename from the list of `.rpt` files on disk.
4. Select the **Accept** button to produce the report.

Figure 3-57
Use the Print Job Info Report dialog box to create operator instructions.



Printing Machine Setup, Operation, and Tool Information

To print reports with full information for machine setup and operation, use the Step Info Report. To print reports about the tools in the job operations setup, use the Tool Info Report.

A standard set of format files is located in the report subdirectory of the SmartCAM installation directory. To search for a format (`.fmt`) file, place the cursor in the Format File input field and then select the File Select button.

Printing a Tooling Report

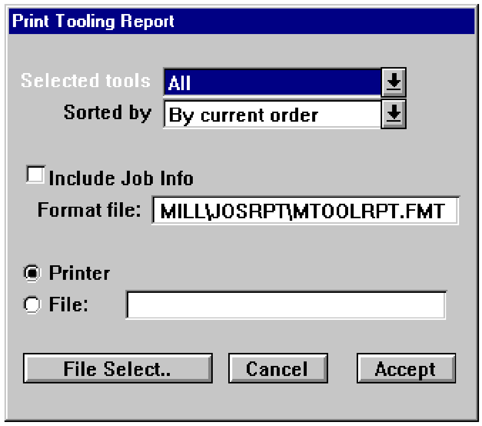
Use Tooling Report to see information about the tools in the active job operations setup or to create a report file that contains tooling information. You can view information about all tools in the job operations setup, or you can limit the report to only those tools used in steps or in the active process model. You can sort the tooling report using one of these options:

- current order
- tool number
- step list order
- process model order

To print a tooling report, perform these tasks:

1. Select **File—Print—Report—Tool Info** to open the **Print Tooling Report** dialog box.

Figure 3-58
Use the Print Tooling Report dialog box to limit the report.



2. Enter the name of the format file to use for the report. To search for a file, select the **File Select** button.
3. Select **Printer** to print a paper copy of the report or **File** to print to a file. If you print to a file, enter the path and name of the file.
4. Turn on the **Include Job Info** on/off switch to include general job information with the tooling report. When the switch is off, the report prints only the tooling information.
5. Select the **Accept** button to print the tooling report or create the print file.

Printing a Step Report

Use Step Report to see information about the steps used in the active job operations setup or to create a report file that contains step information. You can print information about all steps in the job operations setup or limit the report to only those steps that are used in the active process model. You can sort the step report using one of several options.

1. Select **File—Print—Report—Step Info** to open the **Print Step Report** dialog box.

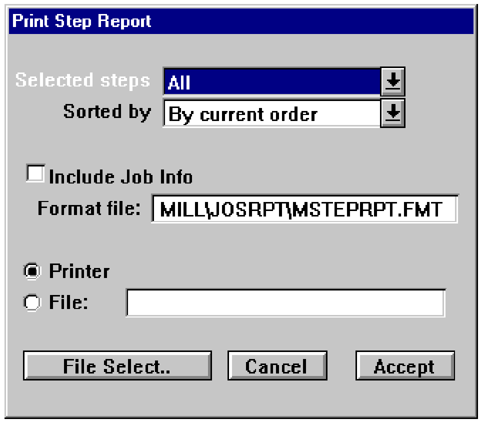


Figure 3-59
Use the Print Step Report dialog box to limit the report to specific steps.

2. Select the scope and sorting criteria for the report from the selector switches.
3. Enter the name of the format file to use for the report. To search for a file, use the **File Select** button to choose from the `.fmt` files on disk.
4. Turn on the **Include Job Info** on/off switch to include general job information with the step report. When the switch is off, the report prints only the step information.
5. Select the **Printer** option switch to print a paper copy of the report. Otherwise, select the **File** option switch to print to a file. If you print to a file, enter the path and name of the file.
6. Select **Accept** to print the step report or create the print file.

Points to Remember

- ❑ Use the Job Information Report to print job information or create a report file that contains job information for the open process model.
- ❑ Use the Tooling Report to print information about the tools in the active job operations setup or to create a report file that contains tooling information.
- ❑ Use the Step Report to print information about the steps used in the active job.

Self-Test

Directions

Test your understanding of the concepts and procedures in this section by answering the following questions. The answers for each self-test are in *Appendix A* of this manual.

______________ **1. What pieces of job information are required?**

a) unit
b) machine file
c) both a and b
d) neither a nor b

______________ **2. What values must be set to complete a process step?**

a) unit
b) machine file
c) both a and b
d) neither a nor b

______________ **3. How many steps or tools can you move at a time when you use the Move feature?**

a) 0
b) 1
c) 2
d) unlimited

______________ **4. You cannot remove active steps from a job.**

a) true
b) false

_______________ **5. Where do you define material information that you want to use on a regular basis?**

a) Material tab of the Job Information dialog box
b) Material Librarian
c) Job Information Report
d) anywhere in the planner

_______________ **6. Once material information is set up, how do you access it from Production Milling?**

a) Material tab of the Job Information dialog box
b) Material Librarian
c) Job Information Report
d) anywhere in the planner

_______________ **7. Which report should you print if you want to know about the job information for the open process model?**

a) Job Information Report
b) Tooling Report
c) Step Report
d) Open Process Model Report

Challenge Project

Directions

Use the Add, Edit, and Duplicate functions that are available in the Job Operation Planner. Practice using the key input fields.

Perform these tasks:

1. Use the following table to create the specified steps.

Step #	Tool Shape/Type	Diameter	Speed	Feed
5	Flat	0.75	1000 RPM	8 IPM
10	Ball	0.5	800 RPM	5 IPM
15	Bull Nose	0.375 w/ 0.06 corner radius	1000 RPM	3 IPM
20	Twist Drill	0.25	900 RPM	.03 IPM
25	Face Mill	2	1500 RPM	18 IPM

2. Select the **Renumber** button to change the step numbers to 1, 2, 3, 4, and 5.
3. Select the **Edit** button to change step number 2 to a .625 diameter and 1-inch cut length.
4. Select the **Duplicate** button to create a new step that has the same operation properties as step 1, but uses a new tool with the following description: 1-inch diameter flat end mill with a 2-inch cut length.

4

Working with Elements

Overview

Elements are the entities that SmartCAM places in the database, displays in the list view, and displays in the element data list. Element types include: lines, arcs, polylines, splines, user events, sub calls, holes, points, ellipses, and helixes.

Lessons for This Unit

- Creating Elements
- Viewing Element Data
- Editing Geometry
- Changing Properties and Attributes
- Using Show Path
- Using Order Path
- Transforming Geometry
- Importing a CAD File
- Exporting a SmartCAM File

Creating Elements

Objectives

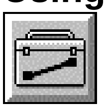
This lesson shows you how to perform these tasks:

- Create lines.
- Create arcs.
- Create wall offsets.
- Create holes.
- Create rapid points.
- Create start profiles.
- Create line profiles.
- Create arc profiles.
- Create polylines.
- Create splines.
- Create ellipses.
- Create user events.
- Create rectangles.
- Create polygons.
- Create lead ins/lead outs.

Overview

With SmartCAM, you can create a variety of geometric elements from various points. Before you create geometry, use the Insert property bar to identify the insert location and the properties for the new geometry. If a step is active, the geometry results in CNC code when you use the Generate Code feature. Create geometry when a layer is active to describe non-coded areas of a part, such as fixtures, clamps, material boundaries, and construction geometry.

Using the Geometry Toolbox



Use the Geometry toolbox modeling tools to create point rapid, hole, line, and arc elements.

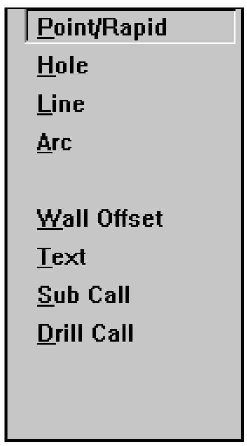
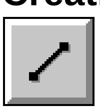


Figure 4-1 *Use the Geometry toolbox to create geometry.*

Creating Lines



Model File: PMLINES.PM4

Use Line to insert a single line element. To create a line, you do not always need to know all of the values in the control panel before SmartCAM can solve for it. Perform these tasks to create lines:

1. Open the model file **`PMLINES.PM4`**.

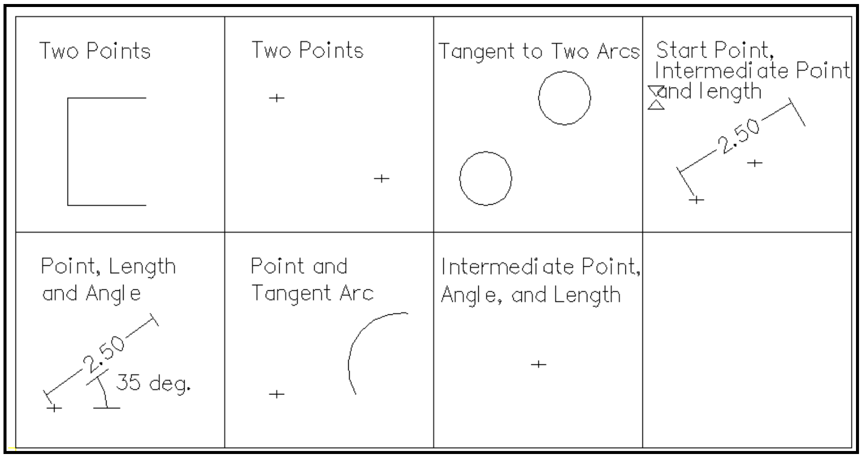
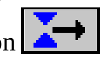
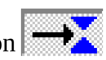
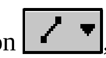
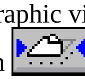
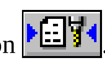


Figure 4-2
Open PMLINES.

2. Set the insert location.

   ■ Set the **Before** icon  or **After** icon  to indicate if the insert point is to be before or after the selected element, process step, tool, or layer.

   ■ Select the **Element** icon , and select the element to be before or after in the list view or graphic view.

   ■ Select the **On Layer** icon  or the **With Step** icon .

   ■ If you inserted on a layer, set the **Layer Number** input field on the Insert property bar. If you inserted on a step, set the step number in the list view.

   ■ If you are inserting with a step, set the **Level**, **Prof Top**, **Clear**, and **Offset** input fields on the Insert property bar.

   ■ If you are inserting on a layer, set the **Level** and **Prof Top** input fields on the Insert property bar.

Figure 4-3
Set the Insert property bar.

3. Select **Create—Geometry**.

4. Select **Line** from the toolbox. The **Line** control panel is displayed.

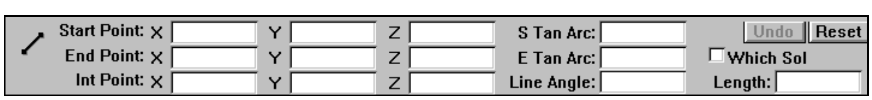


Figure 4-4
Set the values on the Line control panel.

5. Create the lines in the model file by setting point values in the input fields or by using a point, a distance, and an angle. Lines can be made tangent to arcs by using the **S Tan Arc** and **E Tan Arc** input fields.

Figure 4-5
This line has two known points.

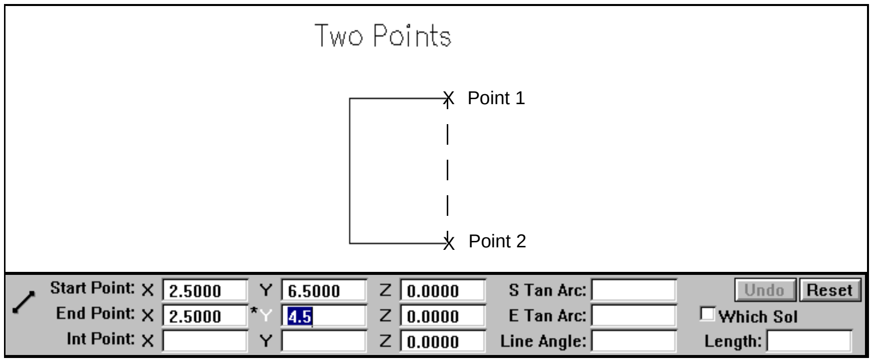


Figure 4-6
This line has two known points.

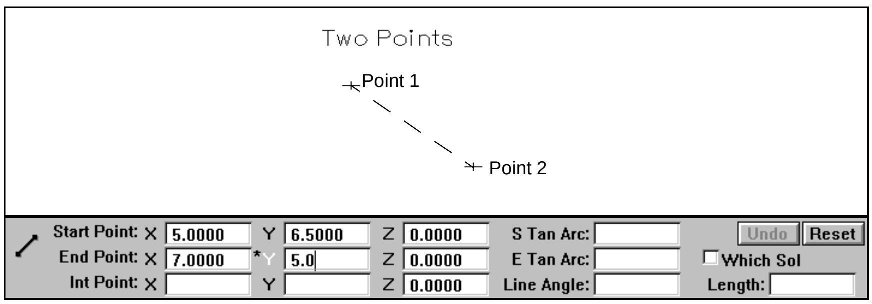


Figure 4-7
This line is tangent to two arcs.

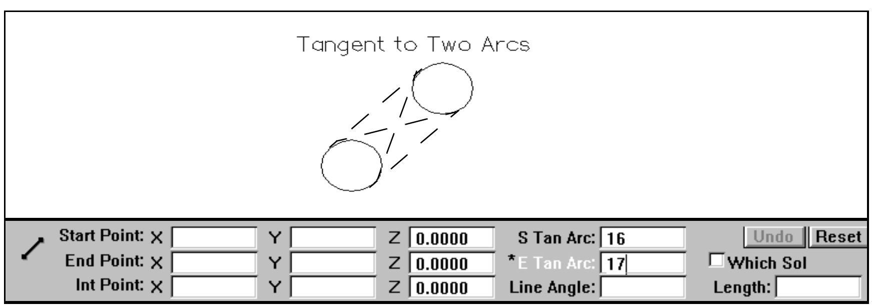
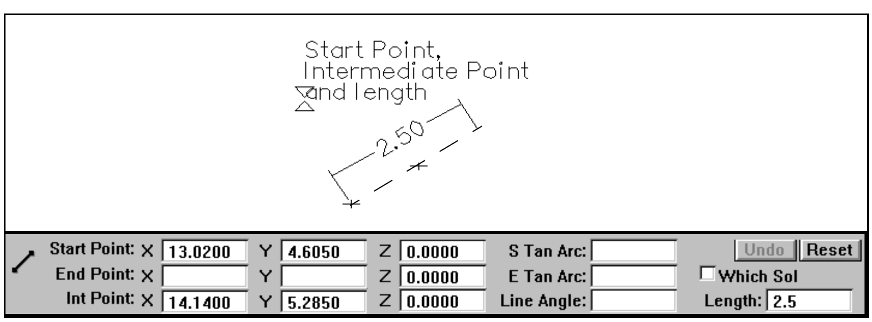


Figure 4-8
This line has a known start point, intermediate point, and length.



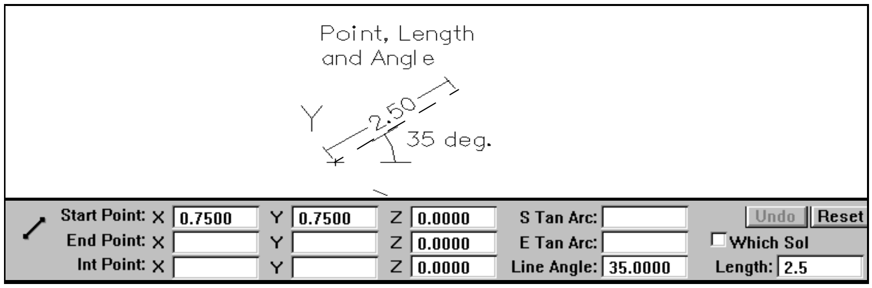


Figure 4-9
This line has a known start point, length, and angle.

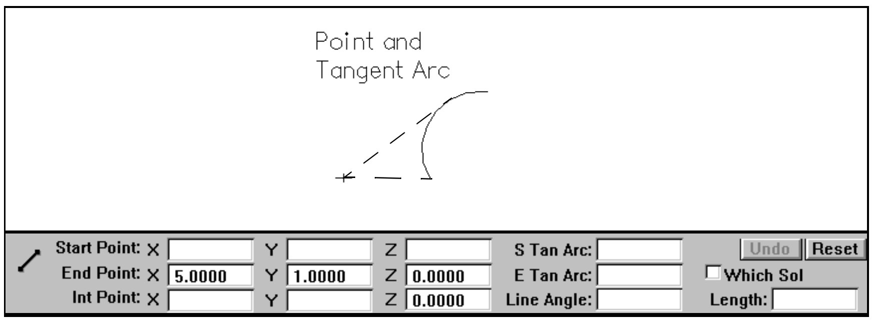


Figure 4-10
This line has a known end or intermediate point, and a tangent arc.

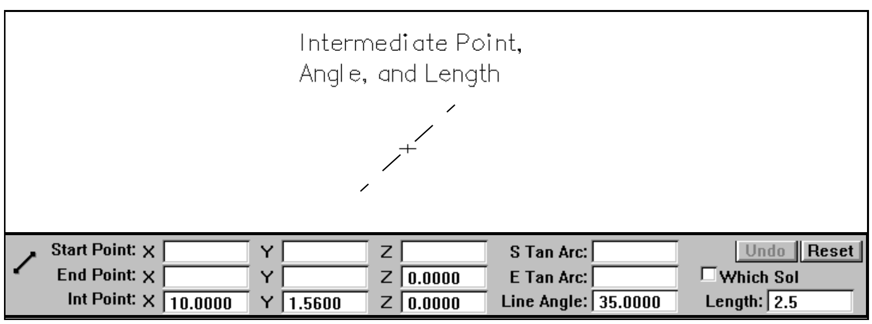
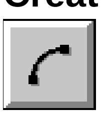


Figure 4-11
This line has a known intermediate point. Specify any angle and length.

Creating Arcs



Model File: PMARCS.PM4

If you associate an arc with a step, a circular cutting move results. Perform these tasks to create full or partial arcs:

1. Open the model file **`PMARCS.PM4`**.

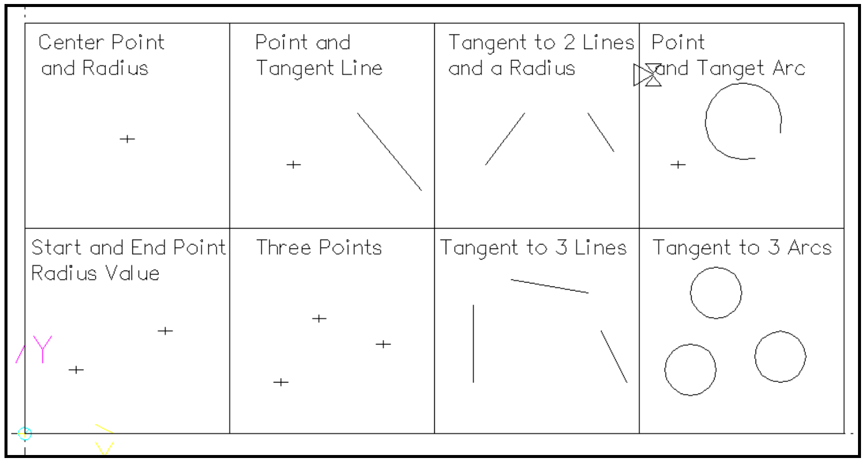


Figure 4-12
Open PMARCS.

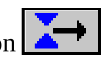
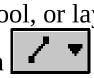
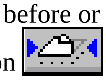
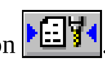
2. Set the insert location.
   - Set the **Before** icon  or **After** icon  on the Insert property bar to indicate if the insert point is to be before or after the selected element, process step, tool, or layer.
   - Select the **Element** icon  on the Insert property bar.
   - Select the element to be before or after in the list view.
   - Select the **On Layer** icon  or the **With Step** icon .
   - If you are inserting with a step, select a tool from the list view. If you are inserting with a layer set the **Layer Number** input field on the Insert property bar.
   - If you are inserting with a step, set the **Level**, **Prof Top**, **Clear**, and **Offset** input fields on the Insert property bar. If you are inserting with a layer, set the **Work Plane**, **Layer**, **Level** and **Prof Top** input fields on the Insert property bar.

Figure 4-13
Set the Insert property bar.

3. Select **Create—Geometry**.
4. Select **Arc** from the toolbox. The **Arc** control panel is displayed.

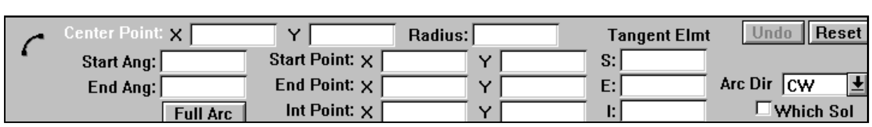


Figure 4-14
Set the values on the Arc control panel.

5. Locate the position of the line by inserting X and Y values in the input fields for the arc's center point. Use the **Tangent Element** input fields to define tangent conditions for the arc.
6. Set the **Arc Direction** selector switch to **CW** or **CCW**:

- **CW** generates a clockwise arc.
- **CCW** generates a counterclockwise arc.

7. Set the **Radius** input field.
8. Set the **Start Ang** and **End Ang** input fields or select **Full Arc**.

Figure 4-15 *Create an arc that has a known center point and any radius that you specify.*

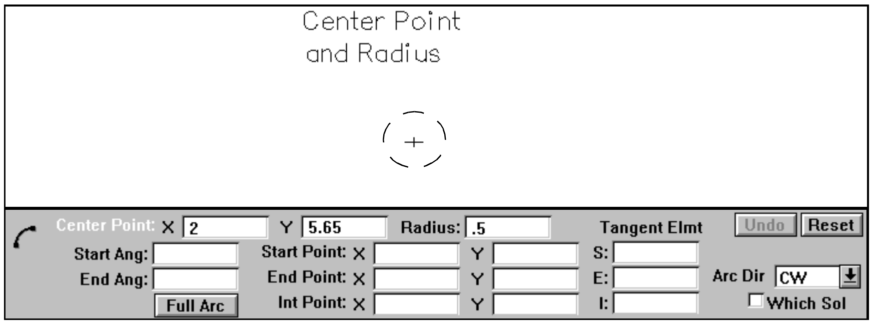


Figure 4-16 *Create an arc that has a center point and a tangent line.*

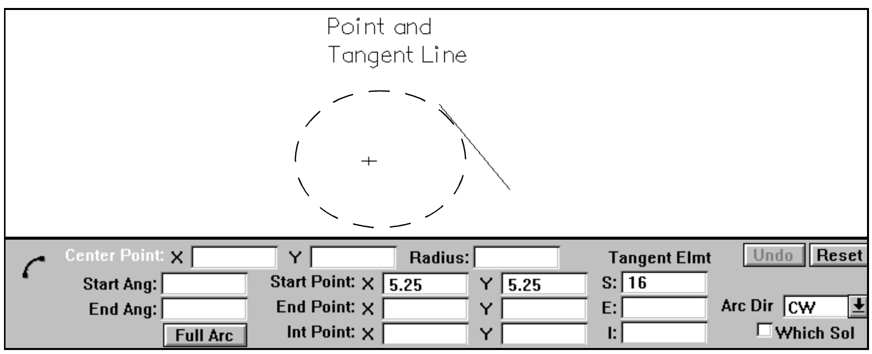


Figure 4-17 *Create an arc that is tangent to two lines and has any radius that you specify.*

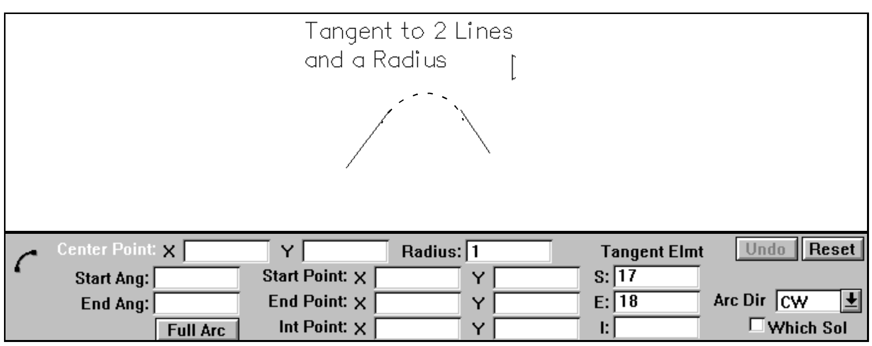


Figure 4-18
Create an arc that has a known center point and that is tangent to an arc.

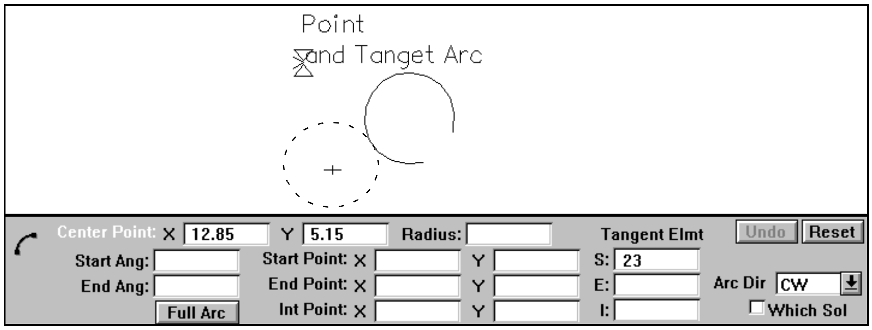


Figure 4-19
Create an arc with a start point, end point, and radius.

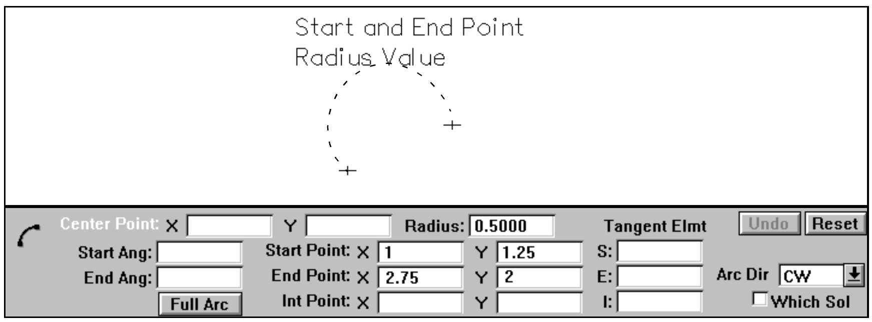


Figure 4-20
Create an arc with a known intermediate point, start point, and end point.

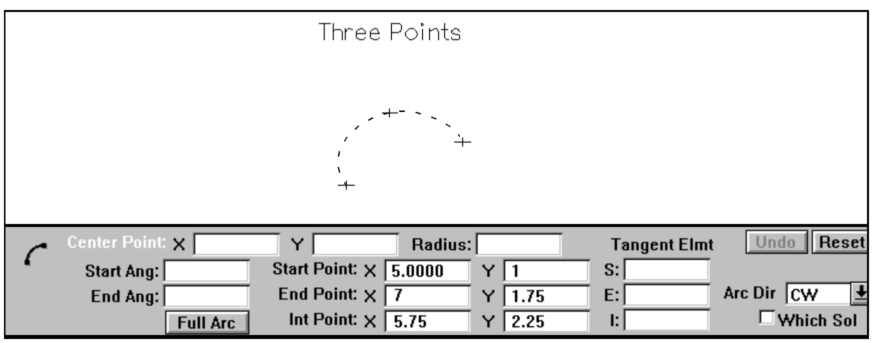


Figure 4-21
Create an arc that is tangent to three lines.

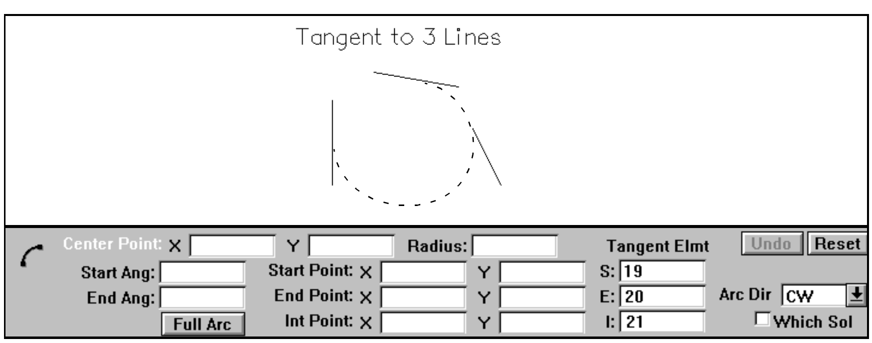
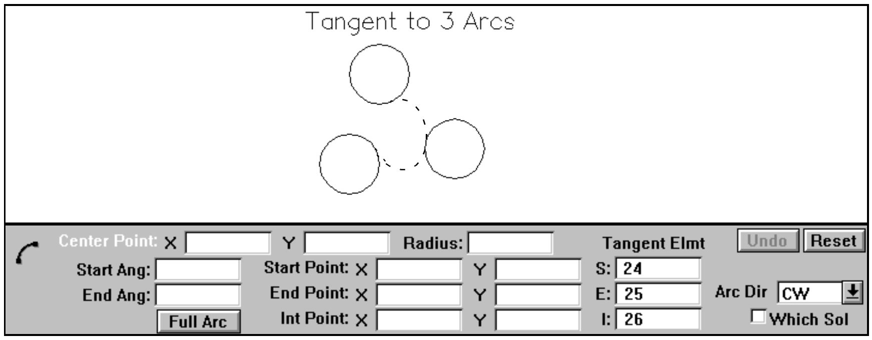
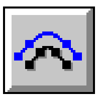


Figure 4-22
Create an arc that is tangent to three arcs.



Creating Wall Offsets

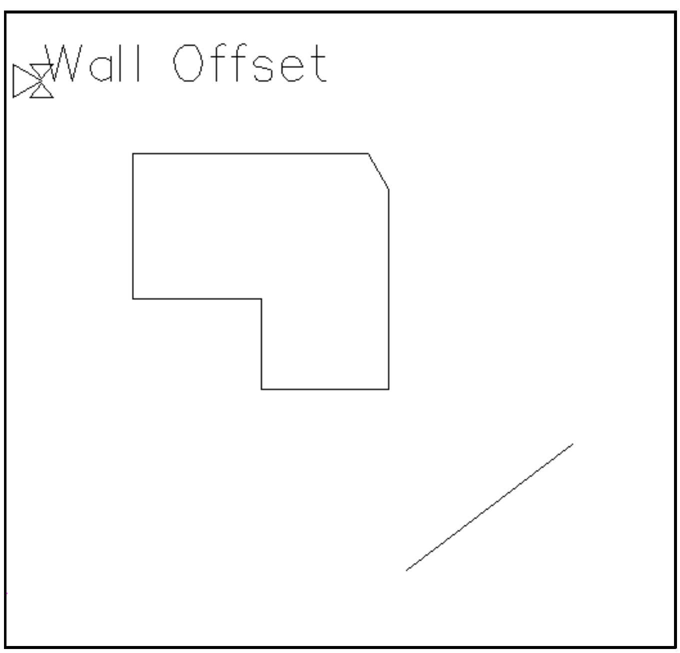


Model File: PMOFFSET.PM4

Use Wall Offset to create parallel geometry that is offset from the original geometry. Create offset geometry from an existing element, group of elements, or profile. The properties of the new geometry can match the existing geometry, or you can specify different properties with the Insert property bar. Perform these tasks to create a wall offset:

1. Open the model file **`PMOFFSET.PM4.`**

Figure 4-23
Open PMOFFSET.



2. Set the insert location:

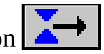
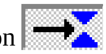
- Set the **Before** icon  or **After** icon  on the Insert property bar to indicate if the insert point is to be before or after the selected element, process step, tool, or layer.
- Select the **Element** icon  on the Insert property bar.
- Select the element to be before or after in the list view.
- Select the **With Step**  icon.
- Set the **Level**, **Prof Top**, **Clear**, and **Offset** input fields on the Insert property bar.

Figure 4-24
Set the Insert property bar.

3. Select **Create—Geometry**.
4. Select **Wall Offset** from the toolbox. The **Wall Offset** control panel is displayed.

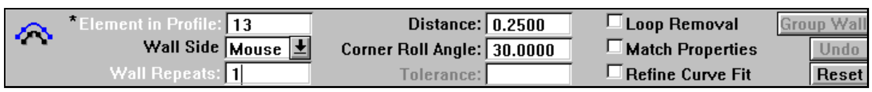
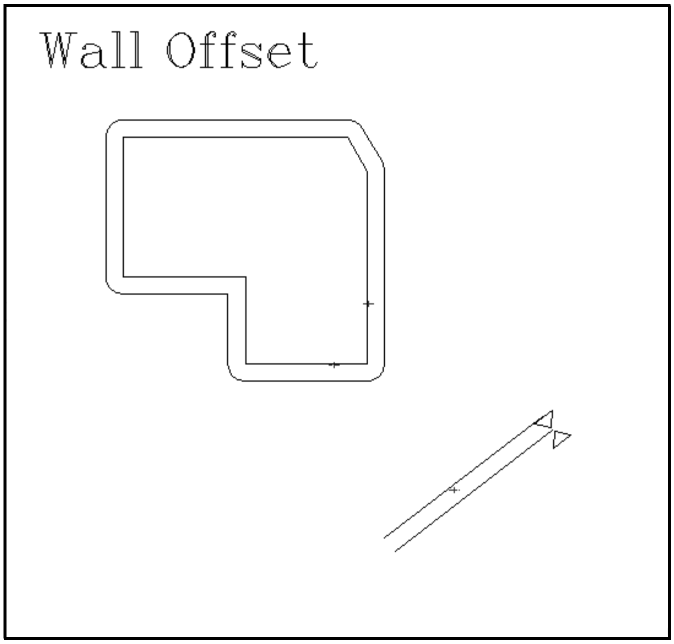


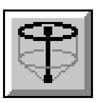
Figure 4-25
Set the Wall Side, Distance, and Wall Repeats.

5. Set the **Element in Profile** input field, and select a profile.
6. Set the following fields on the control panel as necessary.
   - Set the **Wall Side** input field.
   - Set the **Wall Repeats** input field to the desired number.
   - Set the **Distance** input field to the correct offset.
   - Set the **Corner Roll Angle** input field. If this value is less than the intersect angle, the offset will have a sharp corner. If this value is equal to or greater than the intersect angle, then the offset will have a rounded angle.
   - Turn on the **Refine Curve Fit** on/off switch, and enter an appropriate tolerance value, to remove colinear points from the part profile. This is useful when the part profile is a spline. Otherwise, turn it off.
   - Turn on the **Match Properties** on/off switch to force the wall offset geometry to use the same offset, level, clear, and Prof Top properties as the original geometry. When set to off, the wall offset geometry uses the offset, level, clear, and Prof Top properties set on the Insert property bar.

Figure 4-26
This is a result of creating wall offsets.



Creating Holes

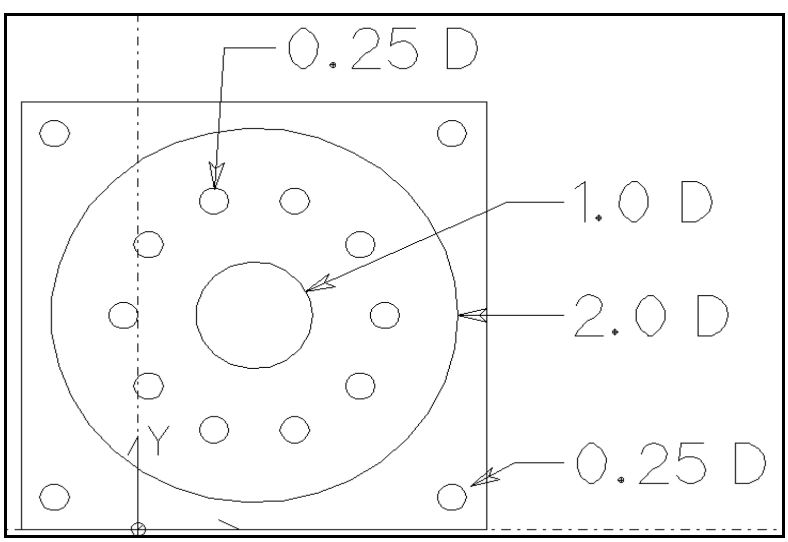


Model File: PMHOLES.PM4

Use Hole to create a hole-making operation using the active step and related properties. Hole enables you to create holes at selected locations, or in a radial or linear pattern on the part. Perform these tasks to create a hole:

1. Open the model file **`PMHOLES.PM4`**.

Figure 4-27
Open PMHOLES.



2. Set the insert location.

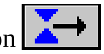
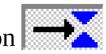
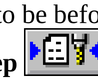
- Set the **Before** icon  or **After** icon  on the Insert property bar to indicate if the insert point is to be before or after the selected element, process step, tool, or layer.
- Select the **Element** icon  on the Insert property bar.
- Select the element to be before or after in the list view.
- Select the **With Step**  icon, and select a drill from the list view.
- Set the **Level, Prof Top**, **Clear** input fields on the Insert property bar.

Figure 4-28
Set the Insert property bar.

3. Select **Create—Geometry**.
4. Select **Hole** from the toolbox. The **Hole** control panel is displayed.
5. Set the **Tip Depth** or **Full Depth** input field to the depth of the hole.

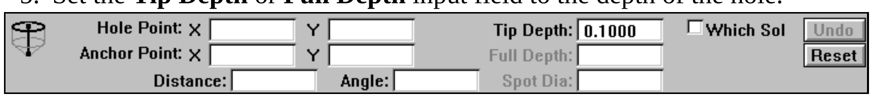


Figure 4-29
Set the values on the Hole control panel.

6. Locate the position of the hole by setting the **Hole Point** input fields or by setting the **Anchor Point**, **Distance**, and **Angle** input fields. Multiple hole operations can be performed at a single location.

   Note The hole size is determined by the properties of the tool used to create it.

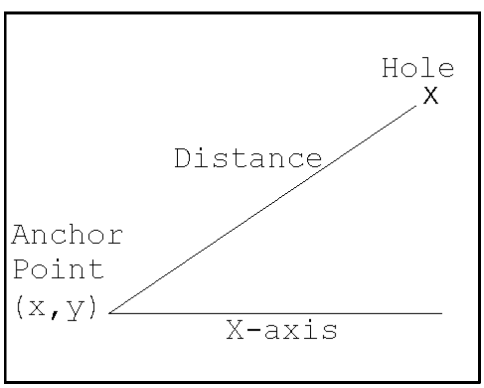
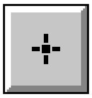


Figure 4-30
Use anchor points to define a hole.

7. These additional fields on the control panel can be set for further contol of the hole.

- Set the Spot Dia field if you are inserting with a spot drilling step.
- Turn on the **Which Sol** on/off switch if you want to choose between multiple possible solutions before accepting your geometry.

Creating Rapid Points

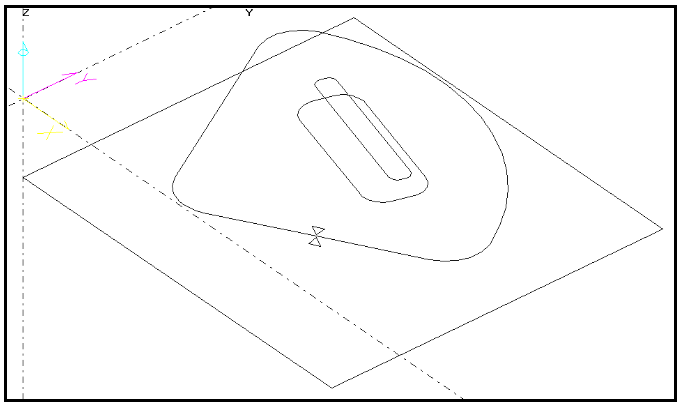


Model File: PMROUGH1.PM4

A rapid point is a three-dimensional location in space. Rapid points are often used to move the tool to a safe location at the start or end of the program or at tool changes. If you assign a rapid point to a step, the resulting point becomes a rapid-positioning move.

1. Open the model file **PMROUGH1.PM4**.

Figure 4-31 Open PMROUGH1.



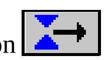
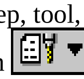
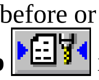
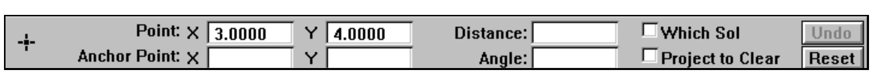
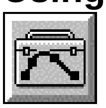
2. Set the insert location:
   - Set the **Before** icon  or **After** icon  on the Insert property bar to indicate if the insert point is to be before or after the selected element, process step, tool, or layer.
   - Select the **Step** icon  on the Insert property bar.
   - Select the step to be before or after in the list view.
   - Select the **With Step**  icon.
   - Select **Step #3** to make this a rapid move with a tool.
3. Select **Create—Geometry**.
4. Select **Point/Rapid** from the toolbox. The **Point/Rapid** control panel is displayed.

Figure 4-32 Set the values on the Point/ Rapid control panel.



5. Set the **Clear** input field on the Insert property bar.
6. Locate the position of the point by setting the **Point** input fields or by setting the **Anchor Point**, **Distance,** and **Angle** input fields.

Using the Profiles Toolbox



Use the Profiles toolbox modeling tools to create a continuous profile of line and arc elements. These modeling tools can solve relationships between elements that you cannot define completely with the available information, enabling you to maintain a continuous profile even if some elements on it are unsolved or pending. SmartCAM can have up to two pending elements at a time before it must solve them with the information you provide for a third element.

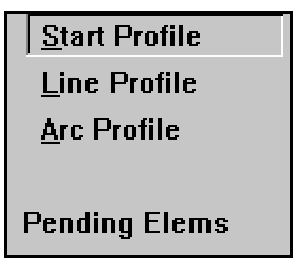
Information about pending elements is listed under Pending Elems on the workbench. SmartCAM tracks the information for pending elements, and as soon as it can solve for a pending element, it automatically does so and updates the model. This intelligent-solution capability speeds up the profiling process.

You can create open and closed profiles. An open profile has a separate start point and end point. You can use it to define the outline of a part or a feature. In a closed profile the start point and the end point are at the same coordinate location, thus creating a closed feature such as a cut-out.

There are two methods for creating a profile:

- Use Create—Profiles—Arc and Line Profiles to create the profile elements sequentially.
- Create each profile element independently. Use Group to place the elements in an active group, and then use Edit—Geo Edit—Profile Trim to create a profile from them.

Figure 4-33
Use the Profiles toolbox to create profiles.



Creating Start Profile Points

Use Start Profile to identify the start of the first element in a profile. If you do not specify a start profile point, SmartCAM uses the end point of the element occurring before the current insert position as the start of the profile. If you use the With Step option in the Insert property bar, the finished profile is a continuous toolpath.

Use values that you want as you perform the tasks in this procedure:

1. Open a model file.

2. Set the insert location:

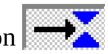
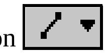
- Set the **After** icon  on the Insert property bar to indicate if the insert point is to be before or after the selected element, process step, tool, or layer.
- Select the **Element** icon  on the Insert property bar.
- Select the element **(13)** to be before or after in the list view.
- Select the **With Step** icon  and select a tool from the list view **(10)**.
- Set the **Level**, **Prof Top**, **Clear**, and **Offset** input fields.

Figure 4-34
Set the Insert property bar.

3. Select **Create—Profile**.
4. Select **Start Profile** from the toolbox. The **Start Profile** control panel is displayed.

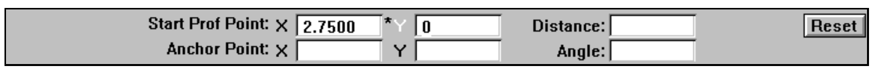


Figure 4-35
Set the values on the Profile Start control panel

5. Locate the start position by setting the **Start Prof Point** input fields, or by selecting the input field and using the snap icons. It is easiest to use snap pick mode and the Snap Endpoint icon to select the right end of the lower line. When the cursor turns into a snap pick mode cursor (short cross hairs), press the left mouse button.
6. Keep the model file open.

Creating Line Profiles

Use Line Profile to create a line that is tangent to or intersects the previous element in the profile. If the line is not completely defined, it is displayed as dashed. Use the Advance button to move to the next element.

Use the values you want as you perform the tasks in this procedure:

1. Continue from the start profile.
2. Select **Line Profile** from the toolbox. The **Line Profile** control panel is displayed.

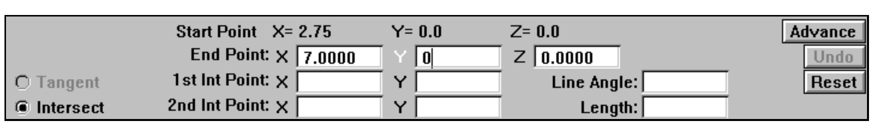


Figure 4-36
Set the values on the Line Profile control panel.

3. Locate the end position by setting the **End Point** input fields to **7, 0**, or you can define the line using an angle and a tangent element.
4. Select the **Advance** button. This is useful if you cannot fully define the line element.

Creating Arc Profiles

Use Arc Profile to create an arc that is tangent to or intersects the previous element in the profile. If the arc is not completely defined, it is displayed as dashed. Use the Advance button to move to the next element.

Use values that you want as you perform the tasks in this procedure:

1. Continue from the last line profile.
2. Select **Arc Profile** from the toolbox. The **Arc Profile** control panel is displayed.

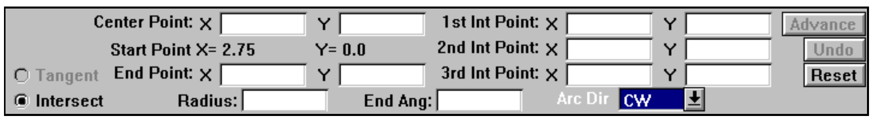


Figure 4-37
Set the values on the Arc Profile control panel.

3. Set these values on the control panel:
   - Select the**Tangent** option switch.
   - Set the **Arc Direction** selector switch to **CCW**.
   - Set the **Radius** input field to **.5 [12.7]**.
   - Set the **Center Point** input fields to **7 [177.8], 1.75 [44.45]**.
4. Select the **Advance** button and then define the next element.

Completing the Profile

To complete the profile, perform these tasks:

1. Select **Line Profile** from the toolbox.
2. Select the **End Point** input field, and snap to the endpoint, which is the right end of the upper horizontal line. The completed profile is displayed.

Challenge Project

Directions

Use the Profile feature to generate the part shown in Figure 4-38 by performing these tasks:

1. Set the start profile point at the origin.
2. Create the line and arc profiles in the clockwise direction.
3. If you want additional practice, create the profiles again, but use the counter-clockwise direction.

Note You need to use the **Advance** button due to the lack of information on the part print.

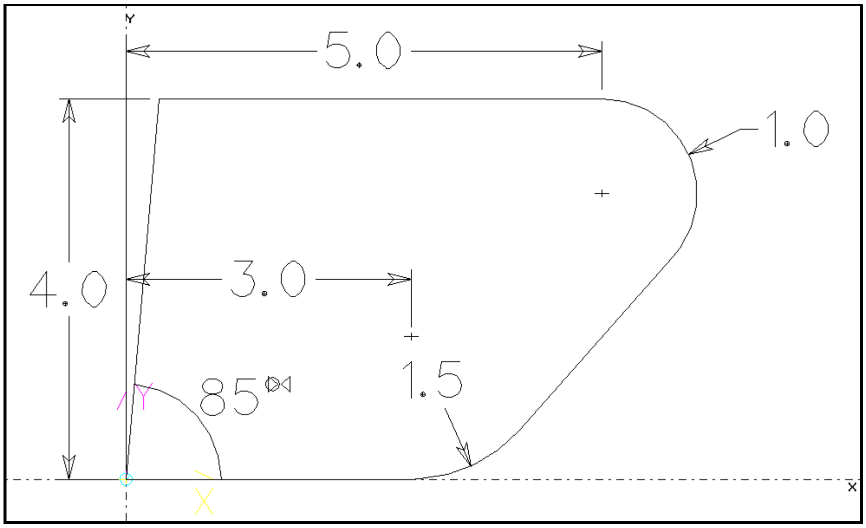


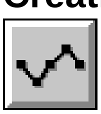
Figure 4-38
Use the Profile feature to generate this part.

Using the Curves Toolbox

Use the Curves toolbox modeling tools to create polyline, spline, ellipse, and helix elements.

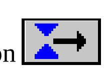
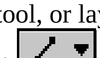
Figure 4-39
Use the Curves toolbox to create polylines, splines, ellipses, and helixes.

Creating Polylines



Use Polyline to create an element made up of a sequence of line segments. Use polylines when you need to create multiple straight line curves or stock boxes. Polylines help reduce the number of line segments in complex models.

Use values that you want as you perform these tasks to create a polyline:

1. Set the insert location:
   - Set the **Before** icon  or **After** icon  on the Insert property bar to indicate if the insert point is to be before or after the selected element, process step, tool, or layer.
   - Select the **Element** icon  on the Insert property bar.

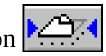
- Select the element to be before or after in the list view.
- Select the **On Layer** icon  or the **With Step** icon  on the Insert property bar.
- If you inserted on a layer, set the **Layer Number** input field on the Insert property bar. If you inserted on a step, set the step number in the list view.
- Set the **Level**, **Prof Top**, **Clear**, and **Offset** input fields on the Insert property bar.

Figure 4-40
Set the Insert property bar.

2. Select **Create—Curves**.
3. Select **Polyline** from the toolbox. The **Polyline** control panel is displayed.

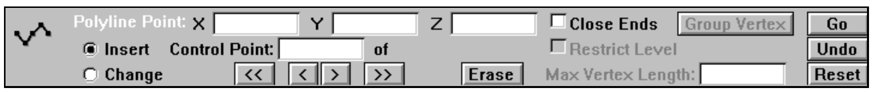
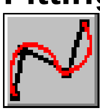


Figure 4-41
Set the values on the Polyline control panel.

4. Set the **Polyline Point** input fields for each of the points in the polyline.
5. Select the **Go** button.

Fitting a Polyline with a Polyarc



Model File: AFPARC.PM4

Use Polyarc Fit to transform the conventional point data of a polyline into a polyarc.

Perform these tasks to create a polyarc fit:

1. Set the insert location.
2. Select **Create—Curves—Polyarc Fit**.
3. Select **Polyarc Fit** from the toolbox. The **Polyarc Fit** control panel is displayed.

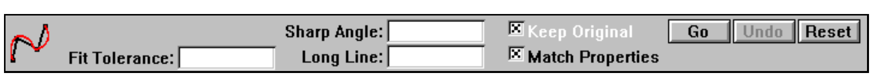
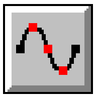


Figure 4-42
Set the values on the Polyarc Fit control panel.

4. Group the polylines that you want to fit.
5. Set the **Fit Tolerance** input field.
6. Set the **Sharp Angle** input field.
7. Set the **Long Line** input field to the length beyond which chords are not fit.
8. Turn off the **Keep Original** on/off switch to erase the original polylines after the new polyarcs are created.
9. Select the **Go** button. The resulting polyarcs are displayed in the Elements list as line or arc profiles.

Creating Splines



Use Spline to create a smooth element through multiple points. Splines are useful when you need a smooth flowing surface.

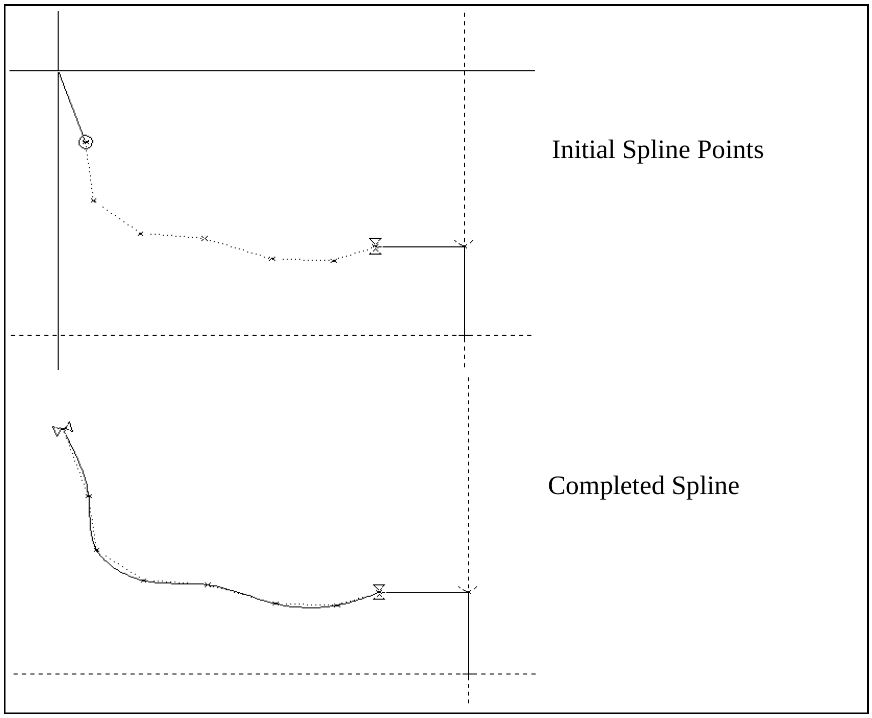
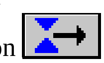
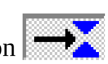
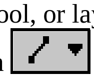
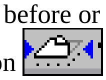


Figure 4-43
Compare initial spline points to a completed spline.

Perform these tasks to create a spline:

1. Set the insert location:

   - Set the Before icon  or After icon  on the Insert property bar to indicate if the insert point is to be before or after the selected element, process step, tool, or layer.
   - Select the **Element** icon  on the Insert property bar.
   - Select the element to be before or after in the list view.
   - Select the **On Layer** icon  or the **With Step** icon  on the Insert property bar.
   - If you inserted on a layer, set the **Layer Number** input field in the list view. If you inserted on a step, set the step number in the list view.
   - Set the **Level**, **Prof Top**, **Clear**, and **Offset** input fields on the Insert property bar.

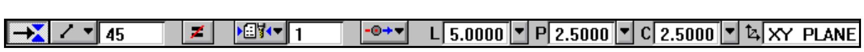


Figure 4-44
Set the Insert property bar.

2. Select **Create—Curves**.

3. Select **Spline** from the toolbox. The **Spline** control panel is displayed.

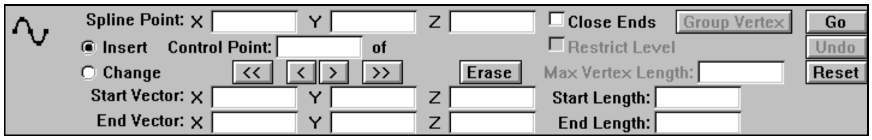
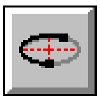


Figure 4-45 *Set values on the Spline control panel.*

4. Set the **Spline Point** input fields to the value for each of the points.
5. Select the **Go** button.

Creating Ellipses



Use the Ellipse control panel to create an ellipse. You need to indicate a center point, angle of inclination, primary axis length, secondary axis length, start point, start angle, and an end angle.

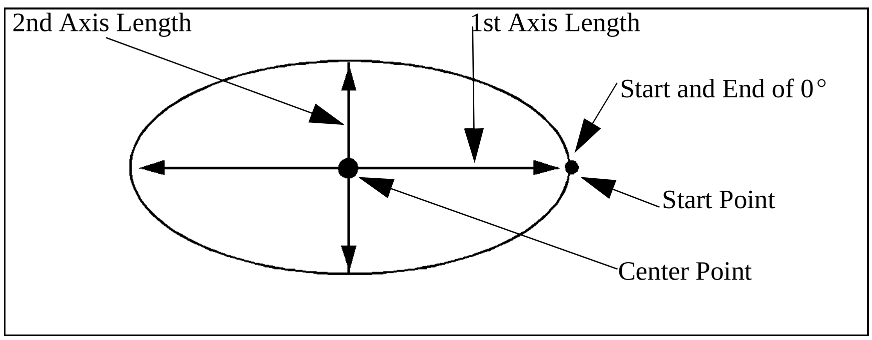


Figure 4-46 *Construct an ellipse from different locations.*

Once you set the insert location and elect to insert with a step or on a layer, perform these tasks:

1. Select **Create—Curves**.
2. Select **Ellipse** from the toolbox. The **Ellipse** control panel is displayed.

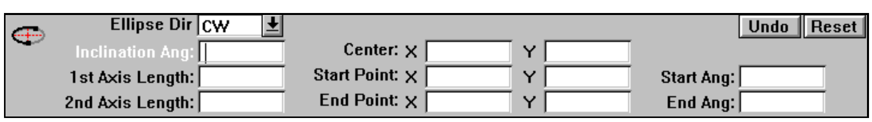
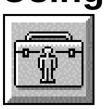


Figure 4-47 *Set the ellipse values.*

3. Select a clockwise (**CW**) or a counterclockwise (**CCW**) ellipse direction.
4. Specify the position of the center point.
5. Select the **Inclination Angle** input field. This is the angle of the primary axis of the ellipse, which is parallel to the X axis and has positive angles that are measured in the counterclockwise direction.
6. Set the **1st Axis Length** input field.
7. Set the **2nd Axis Length** input field.
8. Set the **Start Point** input fields.

9. Set the **End Point** input fields.
10. Set the **Start Angle** and **End Angle** input fields. To create a full ellipse, enter **0** for the start and end angles.
11. Enter a value in the **L** (level) input field on the Insert property bar to indicate the Z level on which to build the ellipse.

Using the User Elements Toolbox



Use the User Elements toolbox modeling tools to create user-specific commands, rectangles, and polygon elements.

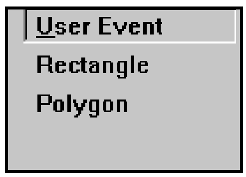
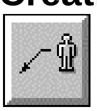


Figure 4-48 *Use the User Elements toolbox.*

Creating a User Event



Use the User Event control panel to insert a machine-control event at a specific point in the process model. These commands can include program stops, comments, calls to template file selections, and other machine functions. Perform these tasks to create a user event:

1. Set the insert location for the user event.
2. Select **Create—User Elmts**.
3. Select **User Event** from the toolbox. The **User Event** control panel is displayed.

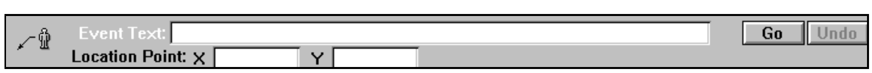
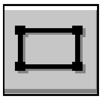


Figure 4-49 *Enter a command in the Event Text input field.*

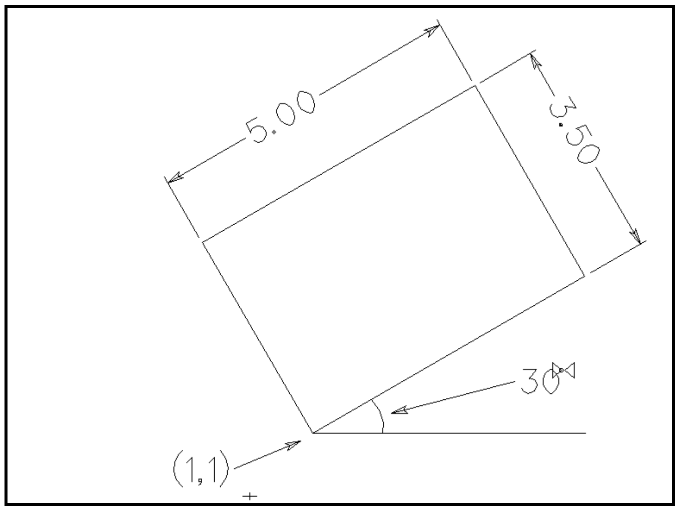
4. Enter the command that you want in the **Event Text** input field.
5. Set the **X** and **Y** coordinates of the **Location Point** for the point in the graphic view where the user event is displayed.
6. Select the **Go** button. The user event record is displayed in the list view.

Creating a Rectangle



Use the Rectangle control panel to create the rectangle shown by entering a length, width, corner position, and an angle. To generate a true rectangle with square corners, you must enter **0** in the Corner Radius input field.

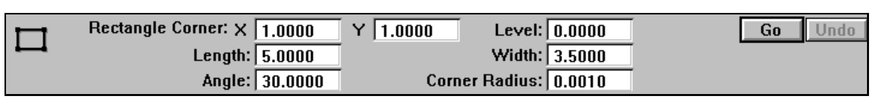
Figure 4-50
Draw this rectangle.



Perform these tasks to create the rectangle:

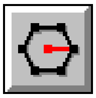
1. Set the insert location.
2. Select **Create—User Elmts**.
3. Select **Rectangle** from the toolbox. The **Rectangle** control panel is displayed.

Figure 4-51
Enter rectangle information.



4. Enter values in the **Rectangle Corner**, **Length** (along X axis), **Angle** (from X axis, positive is counterclockwise), **Width** (along Y axis), **Corner Radius**, and **Level** input fields. The **Level** sets the Z-height of the rectangle.
5. Select the **Go** button.

Creating a Polygon



Use the Polygon control panel to create a polygon with a specific number of sides.

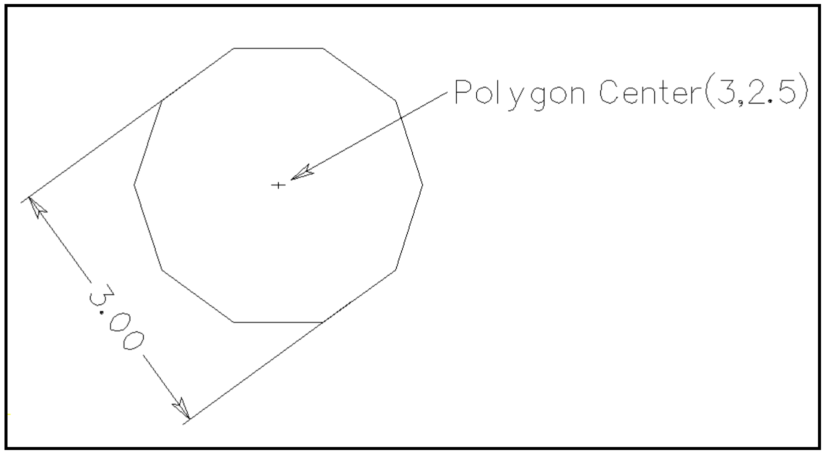


Figure 4-52
Draw this polygon.

Perform these tasks to create a polygon:

1. Set the insert location.
2. Select **Create—User Elmts**.
3. Select **Polygon** from the toolbox. The **Polygon** control panel is displayed.

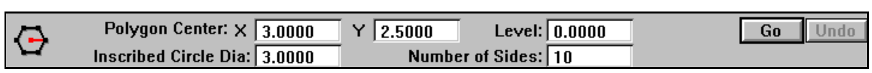
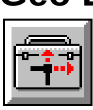


Figure 4-53
Enter polygon information.

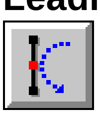
4. Enter the **Polygon Center**, **Inscribed Circle Dia**, **Number of Sides**, and **Level** input fields.
5. Select the **Go** button. The results are grouped automatically.

Geo Edit Toolbox



The Geo Edit toolbox contains modeling tools that enable you to change, add, or delete geometry elements.

Leading In/Out



Model File: PMLEAD.PM4

Use Lead In/Out to create lines or arcs to produce gradual tool-feed movement into and out of a profile. The start and end elements of the profile must be lines, arcs, polylines, or ellipses.

The lead is created on the same side as the profile offset. If there is no offset, the lead will be created on the right. SmartCAM automatically sequences the lead-in and lead-out moves relative to the profile.

Leading In/Out for an Open Profile

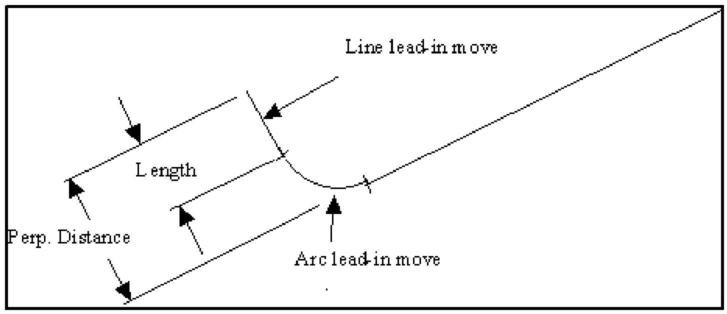


Figure 4-54 *Compare the line and arc lead-in moves.*

Perform these tasks to create a lead in or a lead out for an open profile:

1. Open the model file **PMLEAD.PM4**.

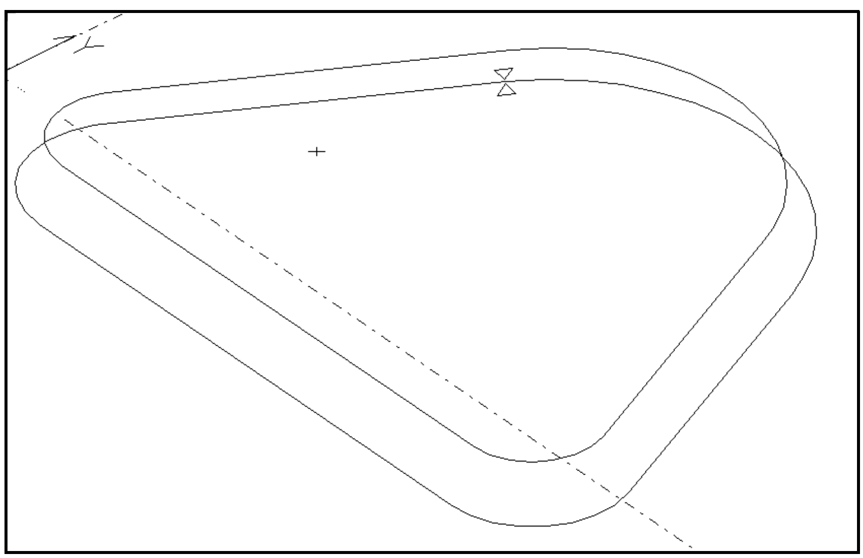


Figure 4-55 *Open PMLEAD.*

2. Select **Edit—Geo Edit**.
3. Select **Lead In/Out** from the toolbox. The **Lead In/Out** control panel is displayed.

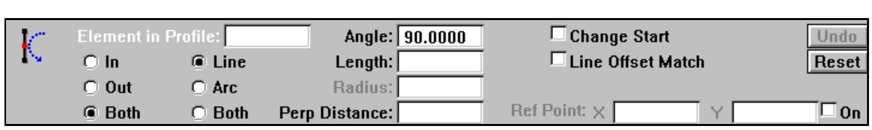


Figure 4-56 *Enter the Lead In/Out information.*

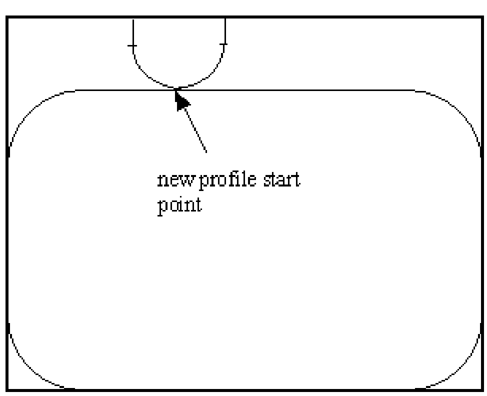
4. If the **Ref Point** on/off switch is turned on, set **Ref Point** to the point for the lead-in and lead-out moves to start and end.

5. If the **Ref Point** on/off switch is turned off, set these input fields:
   - Set **Angle** to the sweep angle of the arc.
   - Set **Length** to the length for lead-in and lead-out line moves. You can leave this field blank and specify a value in the Perp. Distance field instead.
   - Set **Radius** to a radius for lead-in and lead-out arc moves. You need to turn on **Arc** or **Both** (from the Line, Arc, Both column) to enable this field.
6. Select the **Element in Profile** input field and an element in the profile where you want the lead-in and lead-out moves to be done.
7. Turn on the **In**, **Out**, or **Both** option switch:
   - **In** places a lead-in move on the profile.
   - **Out** places a lead-out move on the profile.
   - **Both** places both a lead-in move and a lead-out move on the profile.
8. Turn on the **Line**, **Arc**, or **Both** option switch:
   - **Line** uses lines for the lead-in and lead-out moves.
   - **Arc** uses arcs for the lead-in and lead-out moves.
   - **Both** places a line before an arc lead-in move and after an arc lead-out move.
   - Set **Perp. Distance** to the perpendicular distance from the profile where the lead-in and lead-out moves will begin and end.
9. Turn on the **Change Start** on/off switch to reorder the elements in a profile before placing the lead-in and lead-out moves.

Leading In/Out for a Closed Profile

If you want to move the lead-in move somewhere other than the start/end point of a closed profile, use the Split function to create a new element point along the curve.

Figure 4-57 *Change the Lead In/Out of a closed profile.*



Perform these tasks to create a lead in or a lead out for a closed profile:

1. Select **Edit—Geo Edit**.
2. Select **Lead In/Out** from the toolbox. The **Lead In/Out** control panel is displayed.

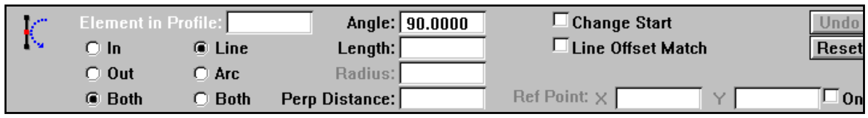


Figure 4-58 *Enter the Lead In/Out information.*

3. Select the element that should be first in the profile. This does not have to be the current start of the profile.
4. Make sure the **Ref Point** on/off switch is turned off.
5. Turn on the **Both** option switch to specify which moves to add.
6. Turn on the **Line** option switch to specify the type of move to add.
7. Set the **Length** input field.
8. Select the **Element in Profile** field.
9. Select the curve to add the Lead In/Out moves to.
10. Practice with other option switch settings and compare the results.

Points to Remember

- ❑ Identify the insert location and the properties for the geometry before creating new geometry.
- ❑ You do not need to know all the values on a control panel to create profiles. SmartCAM can solve for some data.
- ❑ Associating geometry with a step results in CNC code.
- ❑ Associating geometry with a layer does not result in CNC code.
- ❑ Use Wall Offset to create geometry that is parallel to existing geometry.
- ❑ Insert a machine-control event at a specific point in the process model with the User Event feature.

Viewing Element Data

Objectives

This lesson shows you how to find information about element coordinates and properties.

Overview

Use the Element Data feature to view information about the element coordinates and properties for each element in a model. Viewing element data does not affect the geometry database.

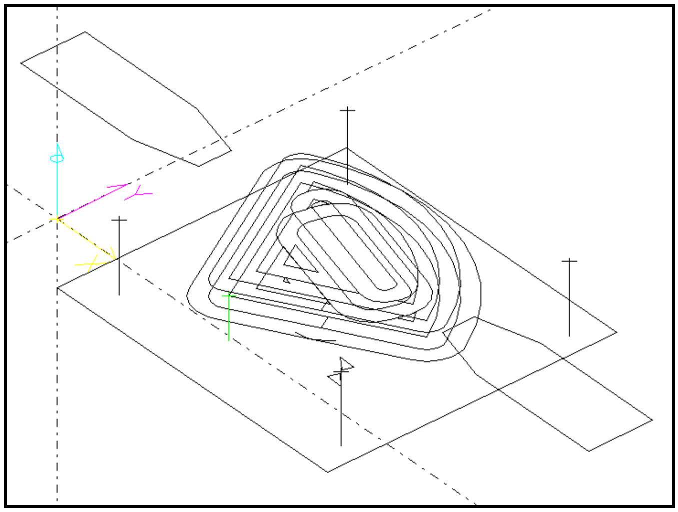
Using Element Data

Model File: PMVIEWS.PM4

Use the Element Data dialog box to view the modeling data for each element in a model. Hidden elements are not displayed. Perform these tasks to view visible modeling data:

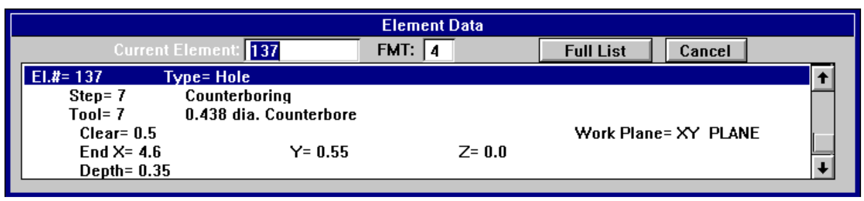
1. Open the model file **PMVIEWS.PM4**.

Figure 4-59
Open PMVIEWS.



2. Select **Utility—Element Data**, or press **F7**. The **Element Data** dialog box is displayed.

Figure 4-60
Set the values on the Element Data dialog box.



3. Use the input fields, list view, or graphic view to select an element.
4. Select the **Full List** button to display a full screen of the element data for the model.
5. Select the **Cancel** button when you are finished viewing the data.

Points to Remember

- ❑ You can view modeling data for each element in a model.
- ❑ Viewing element data does not affect the geometry database.
- ❑ Hidden data is not displayed in the Element Data dialog box.

Editing Geometry

Objectives

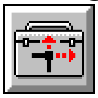
This lesson shows you how to perform these tasks:

- Trim and extend geometry.
- Trim a group of elements.
- Trim a profile.
- Explode an element.
- Blend lines and arcs
- Create a chamfer.
- Split a curve.
- Modify the shape of elements.
- Delete elements.

Overview

Use Trim/Extend to trim elements or extend disconnected elements to an intersection point. Elements to be trimmed or extended must be on the same work plane. Trim/Extend does not work on complex curves, such as splines and helixes.

Using the Geo Edit Toolbox



Use the modeling tools in the Geo Edit toolbox to change, add, or delete geometry elements.

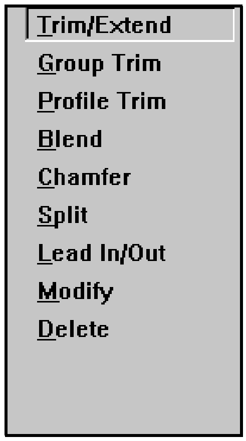
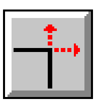


Figure 4-61
Use the options in the Geo Edit toolbox.

Trimming and Extending Geometry



Model File: PMTRIMEX.PM4

Use Trim/Extend to trim elements or to extend disconnected elements to an intersection point. Elements to be trimmed or extended must be on the same work plane. Perform these tasks to trim or extend geometry:

1. Open the model file **`PMTRIMEX.PM4`**.

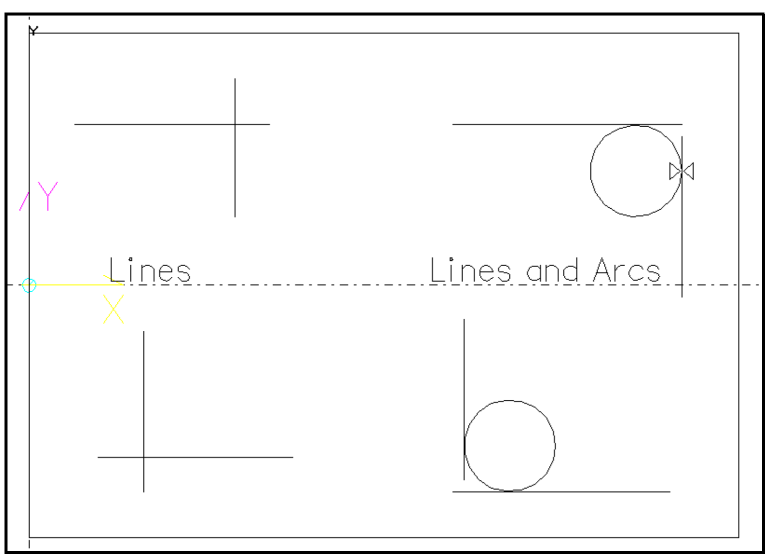


Figure 4-62
Open PMTRIMEX.

2. Select **Edit—Geo Edit**.

3. Select **Trim/Extend** from the toolbox. The **Trim/Extend** control panel is displayed.

Figure 4-63
Set the values on the Trim/ Extend control panel.

4. Set these values on the **Trim/Extend** control panel:
   - Turn on the **Which Segments** on/off switch to view all solutions for trimming and extending the elements you select.
   - Set the **1st Keep Side** selector switch to the side of the first curve to keep. Typically, this is set to **Mouse** to keep the portion of the curve you select.
   - Set the **2nd Keep Side** selector switch to the side of the second curve to keep. Typically, this is set to **Mouse** to keep the portion of the curve you select.
   - Select the **1st Element** input field.
   - Select the geometry to trim.
   - Select the **2nd Element** input field.

Figure 4-64
Trim the lines.

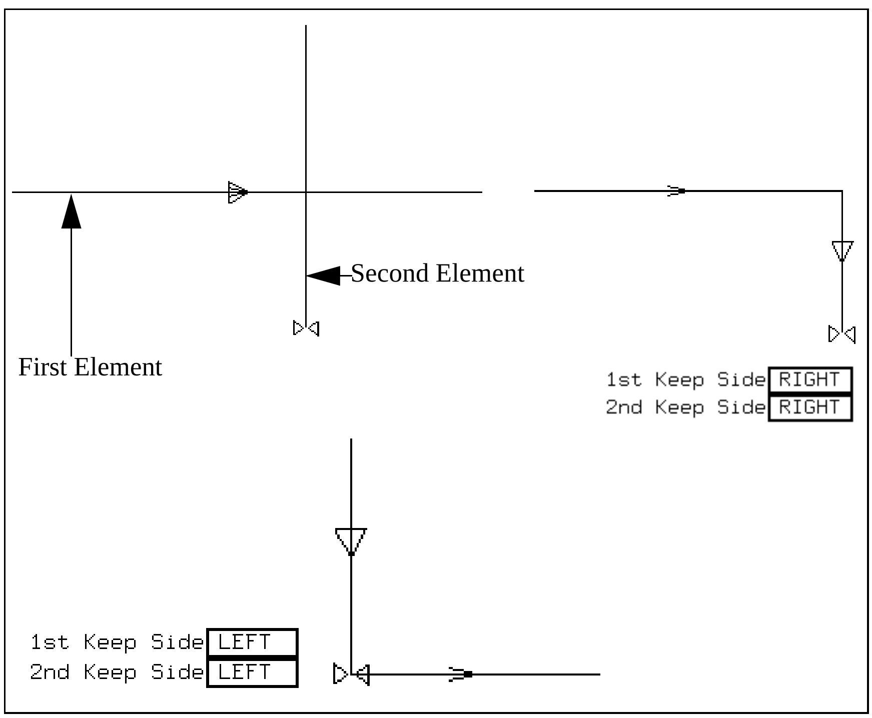
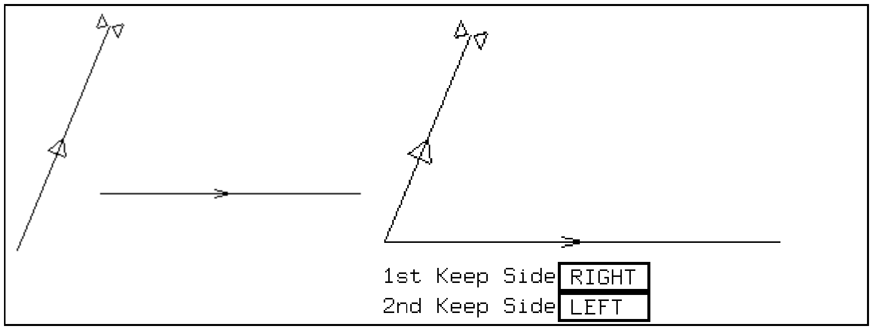
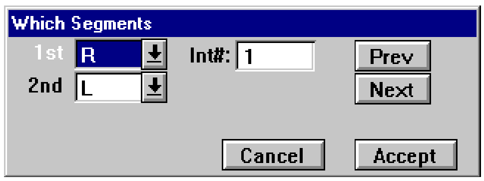


Figure 4-65
Extend the lines.



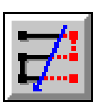
5. Select the geometry to trim. The **Which Segments** dialog box is displayed.

Figure 4-66
Specify the segments that you want to use.



6. Set these values on the **Which Segments** dialog box:
   - Select the **Previous** or **Next** buttons until the solution you want is displayed.
   - Select the **Accept** button when you have selected a solution. If you do not want any of the solutions, select the **Cancel** button.

Trimming a Group of Elements



Model File: PMTRIMGP.PM4

Use Group Trim to trim the active group to the intersections of a selected profile or element. You can trim a single element or a group consisting of lines, arcs, or polylines. Group Trim does not enable you to extend elements. Perform these tasks to trim a group of elements:

1. Open the model file **`PMTRIMGP.PM4`**.

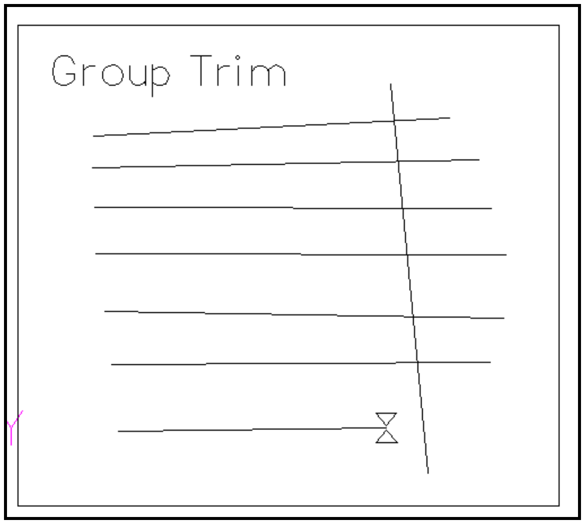


Figure 4-67
Open PMTRIMGP.

2. Group the elements to trim.
3. Select **Edit—Geo Edit**.
4. Select **Group Trim**. The **Group Trim** control panel is displayed.

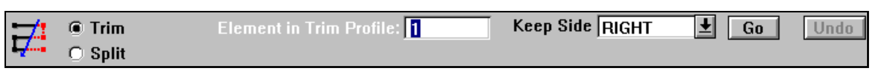


Figure 4-68
Set the values on the Group Trim control panel.

5. Set these values on the **Group Trim** control panel:
   - Select the element to trim to.
   - Turn on the **Trim** option switch.
   - Turn on the **Keep Side** selector switch to control which element parts are affected.
   - Select the **Element in Trim Profile** input field, and select an element with which to trim.
6. Select the **Go** button. If the results are not what you want, select the **Undo** button to remove the incorrect solution.

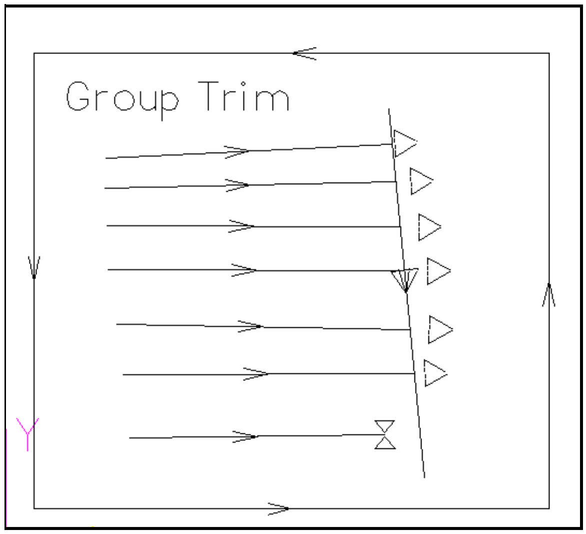
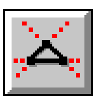


Figure 4-69
This is the result of trimming the group.

Trimming a Profile



Model File: PMTRIMPF.PM4

Use Profile Trim to intersect, trim or extend, and sequentially order groups that are on the same level and work plane. Profile Trim is useful for creating profiles from unconnected elements imported from a CAD system or created by planar cuts. Perform these tasks to trim a profile:

1. Open the model file **`PMTRIMPF.PM4`**.

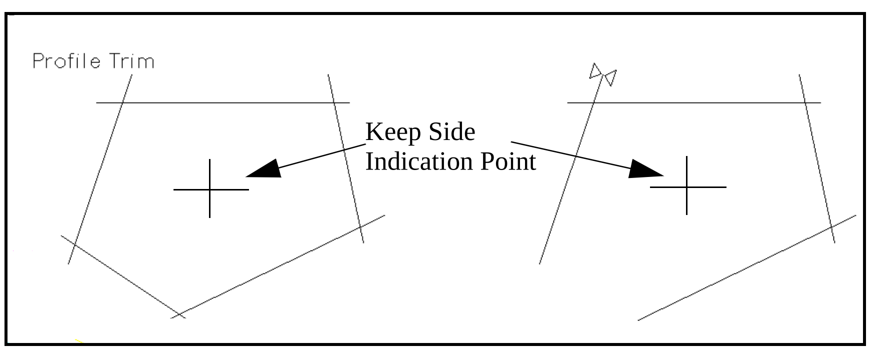


Figure 4-70
Open PMTRIMPF.

2. Group the geometry to change.
3. Select **Edit—Geo Edit**.
4. Select **Profile Trim** from the toolbox. The **Profile Trim** control panel is displayed.

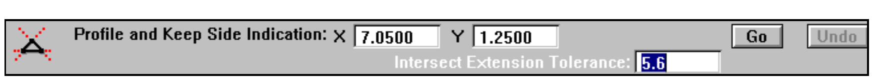


Figure 4-71
Set the Profile and Keep Side Indication input fields on the Profile Trim control panel.

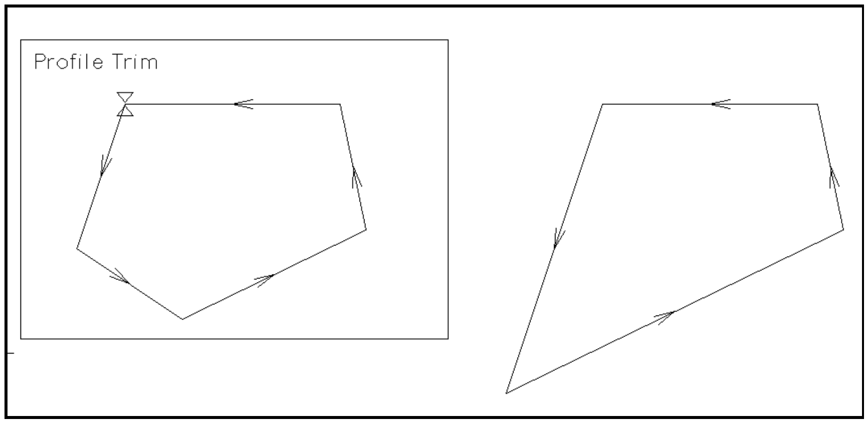
5. Set these values on the **Profile Trim** control panel:

- Set the **Profile and Keep Side Indication** input fields.
- Set the **Intersect Extension Tolerance** input field.

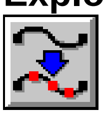
Note Identify the Profile and Keep Side Indication by either entering the coordinates in the X and Y fields or selecting a position in the graphics view.

6. Select the **Go** button. The profile is trimmed.

Figure 4-72 *This is a result of trimming the profile.*



Exploding an Element

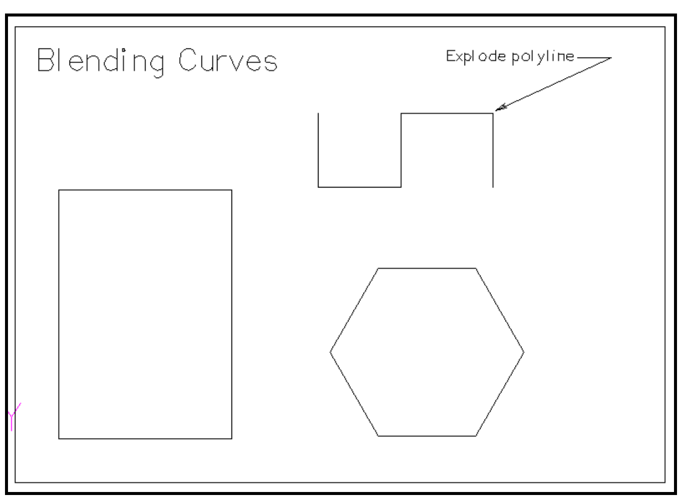


Model File: PMBLEND.PM4

Use Explode to convert elements in the active group into primitive elements (arcs, lines, or polylines). This lets you control the arc tangencies or polyline segments that represent curved portions of profiles.

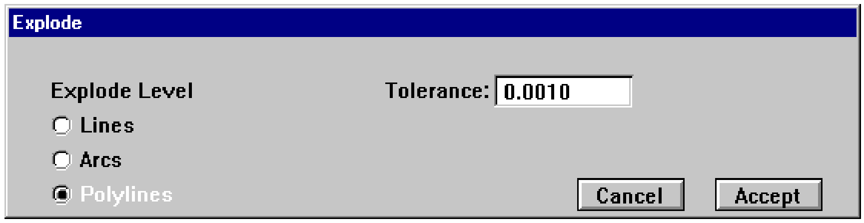
1. Open the model file **PMBLEND.PM4**.

Figure 4-73 *Open PMBLEND.*



2. Select **Group—New Group**, and select the elements to explode.
3. Select **Edit—Explode**. The **Explode** dialog box is displayed.

Figure 4-74
Set the values on the Explode dialog box.

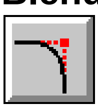


4. Turn on the **Polylines** option switch.
5. Set the **Tolerance** input field, which is the maximum chordal allowance for the resulting elements. The tighter the tolerance, the smaller the resulting elements.

   Note Once an element or group of elements is modified with Explode, you cannot change the new, primitive elements back to the original elements. Be sure you save your model and select the correct elements and explode level before completing the operation.

6. Select **Accept**.

Blending Lines and Arcs

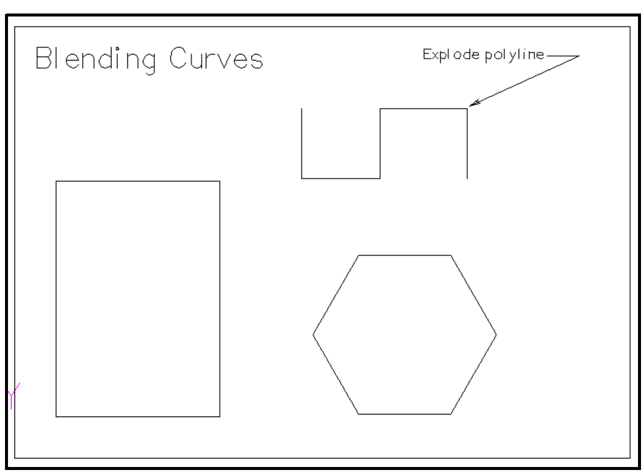


Model File: PMBLEND.PM4

Use Blend to insert an arc segment between two lines or arcs, and trim them to the arc tangent points. The lines and arcs to blend must be on the same work plane. The blended arc uses the same properties as the first selected element and is inserted immediately after it in the database. Perform these tasks to blend lines and arcs:

1. Open the model file **`PMBLEND.PM4`**.

Figure 4-75
Open PMBLEND.



2. Select **Edit—Geo Edit**.
3. Select **Blend** from the toolbox. The **Blend** control panel is displayed.

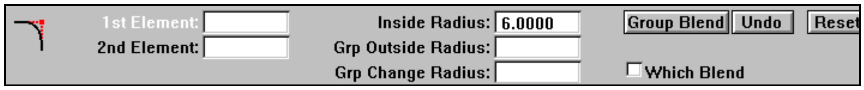


Figure 4-76
Set the values on the Blend control panel.

4. Select the **1st Element** input field, and select the first element or group the geometry to be blended with the Group Blend feature.
5. Set the following as necessary:
   - Set the **Inside Radius** input field to the blend radius value.
   - Set the **Grp Outside Radius** input field to the radius value to use for creating group blends when the tool offset is outside the corner. A value of **0** removes the blending arc between two elements.
   - Set the **Grp Change Radius** input field to specify a new radius for arcs if you are blending a group.
6. Either select the **2nd Element** input field and select the second element, or select the **Group Blend** button.

   The blend is executed.

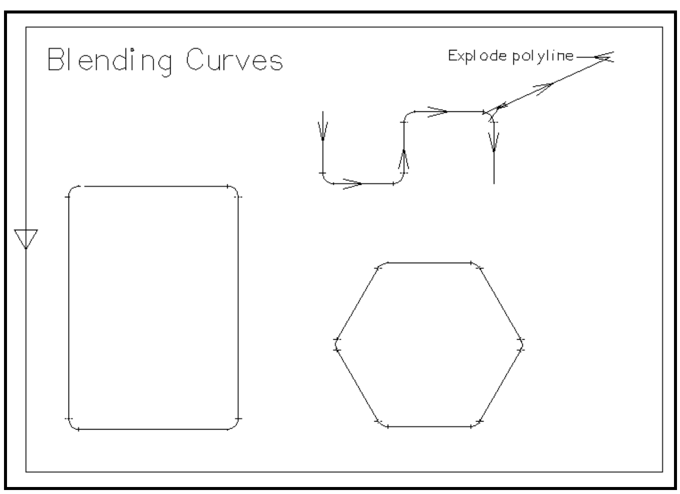
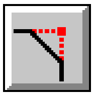


Figure 4-77
This is a result of blending the curves.

Creating a Chamfer



Model File: PMLINES.PM4

Use Chamfer to insert a line of a given size at an angle relative to two line, arc, elliptical, or polyline elements on the same work plane. The two elements do not have to connect, but the chamfer must reach between them. The Chamfer modeling tool also trims or extends the two existing elements to the intersection point of the chamfer. Perform these tasks to create a chamfer:

1. Open the model file **PMLINES.PM4**. You will chamfer the two lines in the first *Two Points* box.

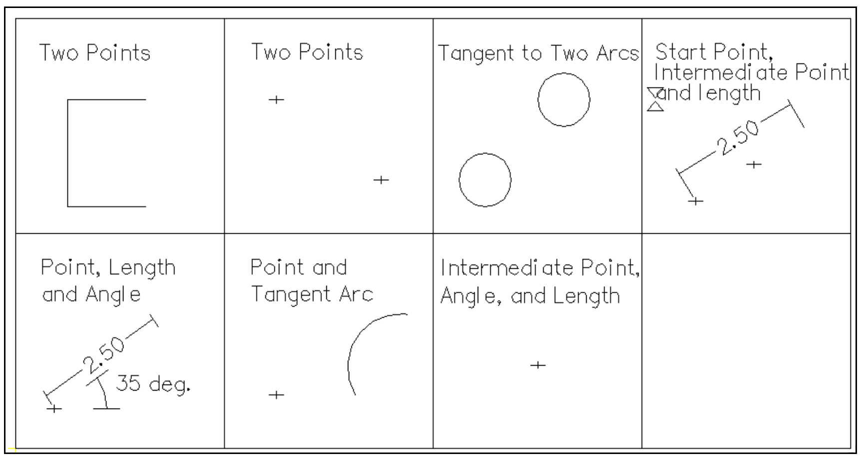


Figure 4-78
Open PMLINES.

2. Select **Edit—Geo Edit**.
3. Select **Chamfer** from the toolbox. The **Chamfer** control panel is displayed.

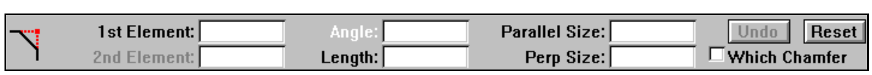
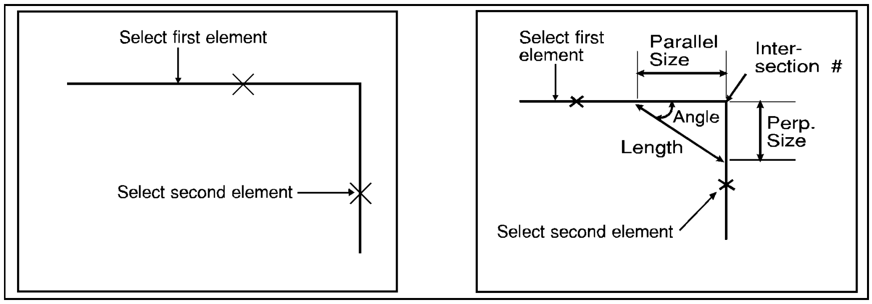


Figure 4-79
Set the values on the Chamfer control panel.

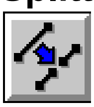
4. Set the **Angle** input field to the angle of the chamfer. The angle of the chamfer is measured from the first element. Positive angles are measured counterclockwise from the first element, and negative angles are measured clockwise from the first element.
5. Set the **Length**, **Parallel Size**, or **Perp Size** input fields.
   - **Length** specifies the length of the chamfer.
   - **Parallel Size** specifies the width of the chamfer. (If length and angle are set, SmartCAM can automatically calculate this value.)
   - **Perp. Size** specifies the height of the chamfer. (If length and angle are set, SmartCAM can automatically calculate this value.)
6. Select the **1st Element** input field, and select the first curve in the graphic view.

7. Select the **2nd Element** input field, and select the second curve in the graphic view.

Figure 4-80
Specify the solution that you want to use.



Splitting a Curve



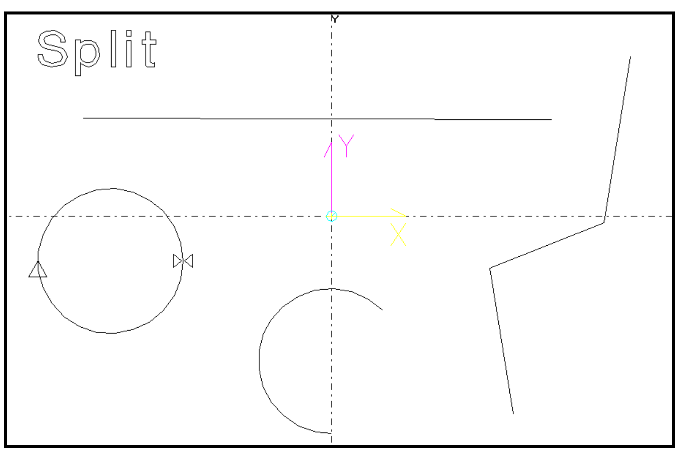
Model File: PMSPLIT.PM4

Use Split to divide an arc, line, ellipse, or polyline at any point along its length. You can extend or trim the elements at the split point to create a break or an overlap.

Split creates an additional element in the database. (Two elements appear in the database to replace the single element that was split.) SmartCAM keeps the new element in the correct sequence with the original properties. Perform these tasks to split a curve:

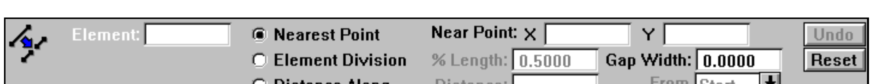
1. Open the model file **`PMSPLIT.PM4`**.

Figure 4-81
Open PMSPLIT.



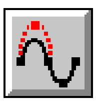
2. Select **Edit—Geo Edit**.
3. Select **Split** from the toolbox. The **Split** control panel is displayed.

Figure 4-82
Specify values on the Split control panel.



4. Set **Nearest Point**, **Element Division**, or **Distance Along**:
   - If you choose **Nearest Point**, enter the coordinates for the split point on the element in the **Near Point** input fields.
   - If you choose **Element Division**, enter the decimal equivalent for the percent of the element's length where the split should occur in the **% Length** input field. Also set the **From** selector switch to identify where to measure from to calculate **% Length**.
   - If you choose **Distance Along**, enter the distance along the element for the split to occur in the **Distance** input field. Also set the **From** selector switch to identify where to measure from to calculate **Distance**.
5. Set the **Element** input field and select an element to split.

Modifying the Shape of Elements



Model File: PMMODIFY.PM4

Use Modify to change the geometric information for a selected element. When you select Modify and an element, SmartCAM displays solution settings for that element in the control panel. Change the settings to create a different solution for the element.

You can modify these element types:

- Holes
- Lines
- User elements
- Points
- Helixes
- Subroutines
- Arcs
- Polylines
- Text (dimensions)
- Ellipses
- Splines

When you select an element to modify, the appropriate dialog box for the element is displayed.

1. Open the model file **`PMMODIFY.PM4`**.

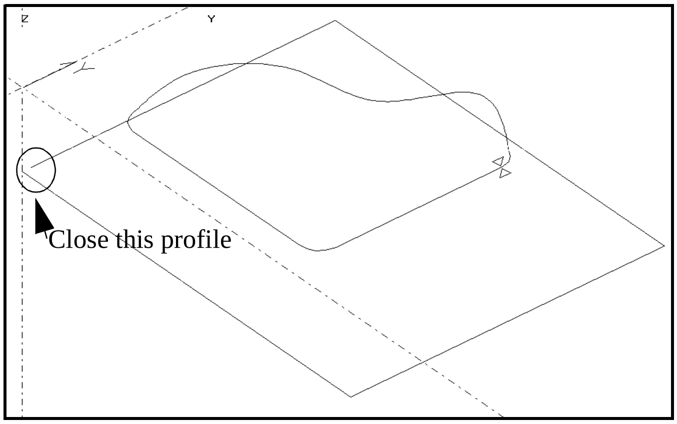


Figure 4-83
Open PMMODIFY.

2. Select **Edit—Geo Edit**.

3. Select **Modify** from the toolbox. The **Modify** control panel is displayed.

Figure 4-84
Open the Modify control panel.

4. Select the **Element to Modify** input field.

5. Select the element to modify. The element-appropriate dialog box, in this case the Modify Polyline dialog box, is displayed.

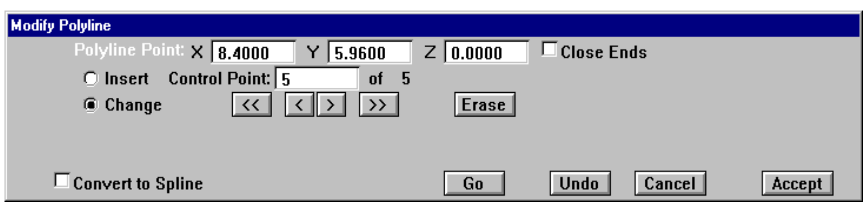
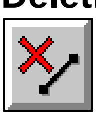


Figure 4-85
The Modify Polyline dialog box is displayed.

6. Make the necessary changes, and select the **Go** button.

7. Select the **Accept** button.

Deleting Elements



Model File: PMLINES.PM4

Use Delete to remove an element or a group of elements from the database. Undo enables you to recover the last element or group deleted. Once you close the Delete control panel, you cannot recover deleted items. Perform these tasks to delete an element:

1. Open the model file **`PMLINES.PM4`**

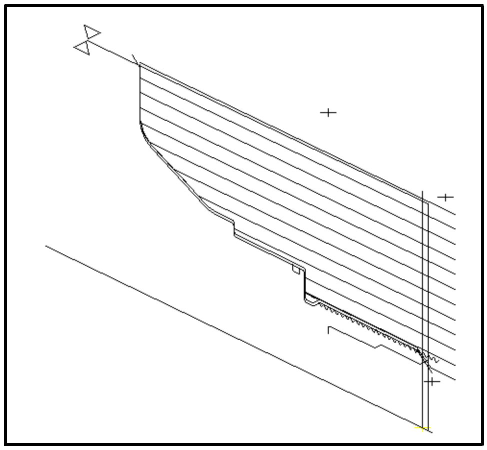


Figure 4-86
OpenPMLINES .

2. Select **Edit—Geo Edit**.
3. Select **Delete** from the toolbox. The **Delete** control panel is displayed.

Figure 4-87
Set the Delete control panel.

4. Select the **Element to Delete** input field.
5. Select the element to delete. The element is deleted.
6. Select the **Undo** button to restore the most recently deleted element.

   Note You cannot undo more than the most recently deleted element.

Points to Remember

- ❑ Elements to be trimmed or extended must be on the same work plane.
- ❑ Group Trim impacts only the active group. It does not extend an element.
- ❑ Split creates an additional element in the database.
- ❑ Only the last element or group deleted can be recovered with the Undo button.

Changing Properties and Attributes

Objectives

This lesson shows you how to perform these tasks:

- Change toolpath properties.
- Change layer properties.
- Change hole or point properties.

Overview

Use Property Change to change these properties for an element or set of elements:

- The clearance heights
- The step associated with the geometry
- The Z level
- The offset direction
- Whether a piece of geometry is drawn on a layer or with a step

Using the Property Chg Submenu

Use the Property Chg submenu to update modeling properties associated with existing elements.

Note You must have an active group of elements before using Property Change modeling tools. Use the Group tool palette to select an active group.

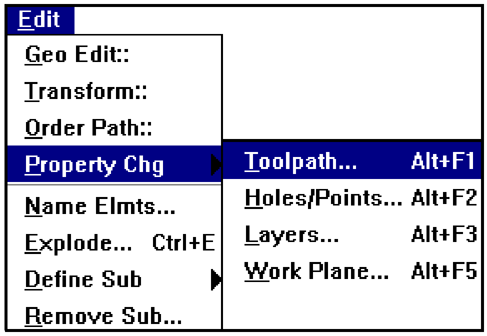
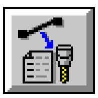


Figure 4-88
Learn the components of the Property Change submenu.

Changing a Toolpath Property



Model File: PMPRPCHG.PM4

Use Toolpath to change one or more toolpath properties for an active group of elements. Properties that you can change include step, tool offset, level,clear,and prof top. You must select an active group of elements before you can access the Toolpath Property Change dialog box. Perform these tasks to change a toolpath property:

1. Open the model file **PMPRPCHG.PM4**.

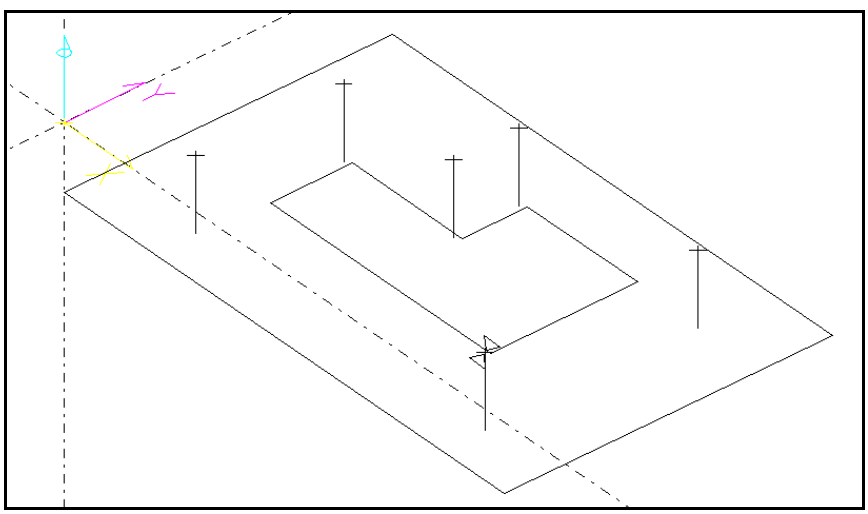
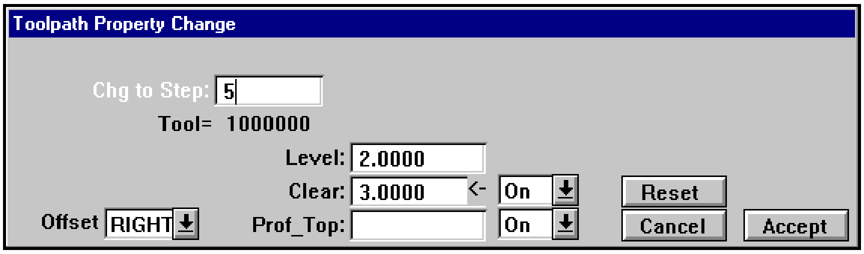


Figure 4-89
Open PMPRPCHG.

2. Group the geometry to change.
3. Select **Edit—Property Chg—Toolpath**. The **Toolpath Property Change** dialog box is displayed.

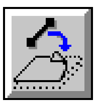
Figure 4-90
Set the values on the Toolpath Property Change dialog box.



4. Enter the changes on the dialog box. To change the **Offset**, **Clear**, and **Prof Top** values, you must change the **Offset** selector switch from **N/C** to another value.

5. Select the **Accept** button.

Changing a Layer Property

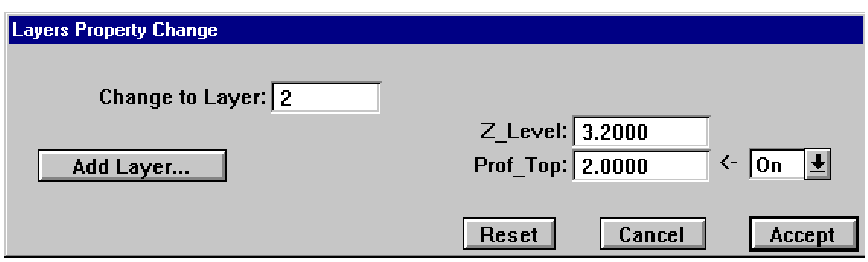


Use Layers to associate the active group of elements with a layer or to change layer assignments. If you associate elements with a layer, SmartCAM does not generate code for them. Place items such as clamps and part blanks on layers to check tool clearance of obstacles during Show Path. Perform these tasks to change a layer property:

1. Group the geometry that you want to change.

2. Select **Edit—Property Chg—Layers**. The **Layers Property Change** dialog box is displayed.

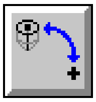
Figure 4-91
Set the values on the Layers Property Change dialog box.



3. Enter the changes on the dialog box. To change the **Prof Top** input field, you must change the **Prof Top <-** selector switch from **N/C** to another value.

4. Select the **Accept** button.

Changing a Hole or Point Property



Use Holes/Points to change the properties of hole or point elements in the active group. You can change the assigned step, level, clearance, or spot diameter, or you can change a group of points into holes. Perform these tasks to change a hole or a point property:

1. Select **Edit—Property Chg—Holes/Points**. The **Holes/Points Property Change** dialog box is displayed.

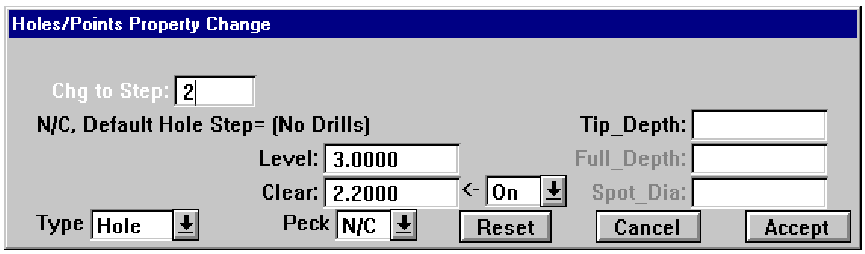
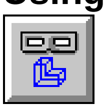


Figure 4-92 *Set the values on the Holes/ Points Property Change dialog box.*

2. Enter the changes on the dialog box. To change the **Type** selector switch and **Clear** input field, you must change the **Clear <-** selector switch from **N/C** to another value.
3. Select the **Accept** button.

Using Show/Mask



Model File: PMVIEWS.PM4

Use the Show/Mask dialog box, which is on the Utility menu, to show or mask geometry in a model file. You can select elements by layer, step, tool, or work plane. SmartCAM does not display, select, or code hidden or masked elements. Masked elements are not deleted from the database; they are just hidden.

Show/Mask is useful for selectively viewing only a portion of the model. This reduces the redraw time and the time required when using Show Path.

Perform these tasks to show and mask elements:

1. Open the model file **PMVIEWS.PM4**.

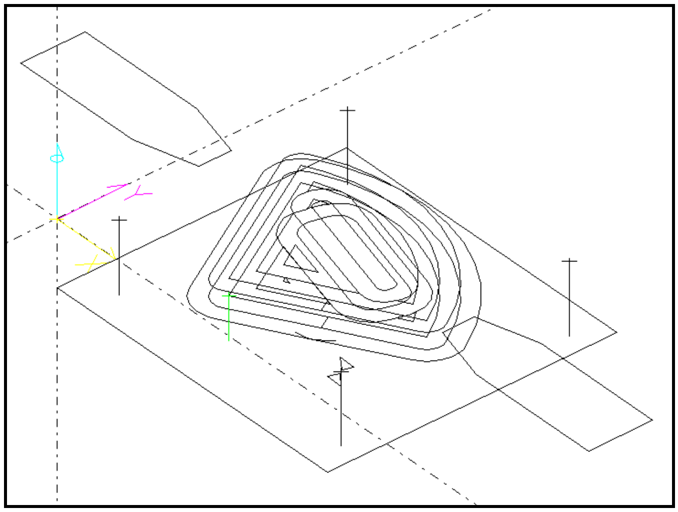


Figure 4-93
Open PMVIEWS.

2. Select **Utility—Show/Mask**. The **Show/Mask** dialog box is displayed.

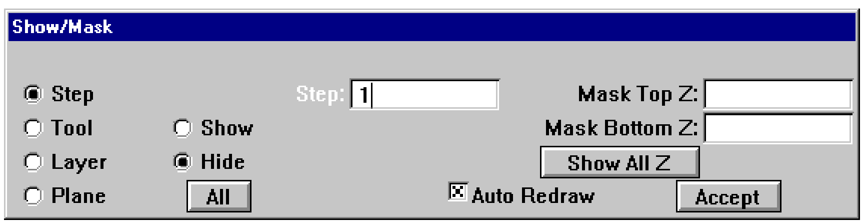


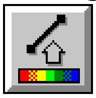
Figure 4-94
Set the values on the Show/ Mask dialog box.

3. Select the method to choose elements to show or mask (**Step**, **Tool**, **Layer**, or **Plane**).
4. Select either the **Show** or the **Hide** option.
5. Use the list view, the input field, or the graphic view to select the elements to show or mask.

   Note An H is displayed in the list view next to an element, step, or layer that is hidden. When the Show Mask dialog box is displayed, you can select the hidden element to redisplay that geometry. You can also select an element that is not hidden to automatically mask that geometry.

6. Select the **Accept** button. The geometry is displayed or hidden.
7. Press **F8** to redraw the screen, if necessary. Remember to use this key anytime you want to instantly identify what is masked.

Using Color Change



Model File: PMVIEWS.PM4

Use the Color Change dialog box, which is on the Utility menu, to change the color, layer style, or both for existing elements in the model. The colors available depend on your computer's graphic display card. This option affects only the display, not the code generation.

Perform these tasks to change the color:

1. Open the model file **`PMVIEWS.PM4`**.

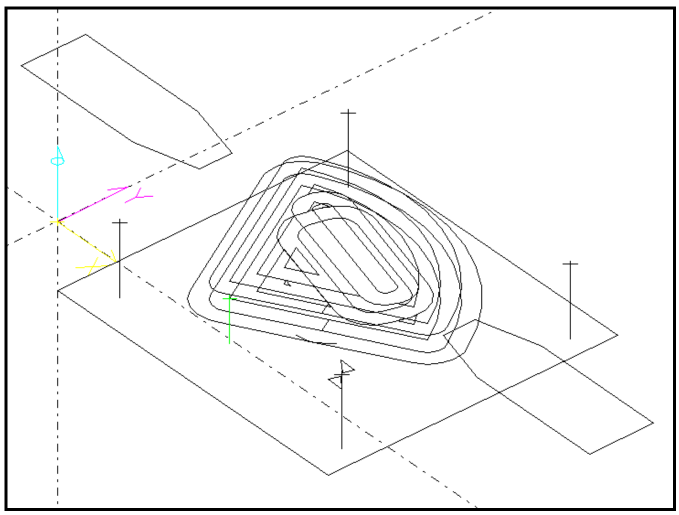


Figure 4-95 *Open PMVIEWS.*

2. Select **Utility—Color Change**. The **Color Change** dialog box is displayed.

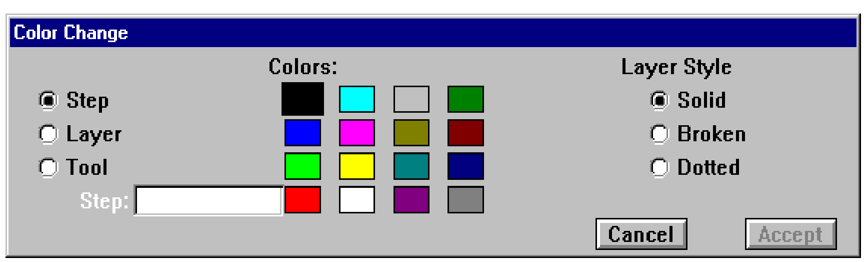


Figure 4-96 *Set the values on the Color Change dialog box.*

3. Select the **Step**, **Layer**, or **Tool** option to indicate what type of geometry to change.

   Note Color assignments for any steps that have numbers greater than 255 are not saved by SmartCAM following a session. Therefore, only color information for step numbers 255 or less is consistent between SmartCAM sessions.

4. Use the list view, input field, or graphic view to select elements to change.

5. Select the **Accept** button.

Points to Remember

- You must have an active group of elements before using Property Change modeling tools.
- You can change toolpath, layer, and hole/point properties.
- When you associate elements with a layer, code is not generated for them.
- The Color Change feature enables you to change the color of specified elements.
- Changing the color of elements affects only the display, not the code generation.

Using Show Path

Objectives

This lesson shows you how to perform these tasks:

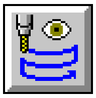
- Set the Show Path control panel.
- Start Show Path.
- Stop Show Path.

Overview

Show Path provides a representation of the toolpath before any parts are cut. Use Show Path to verify the sequence and toolpath of the model based on your machine's capabilities.

Note Show Path uses information stored in the custom tool graphics (CTG) file for each tool to perform the machining simulation in the graphic view. If you have not specified a CTG file in the tooling information, SmartCAM will display a standard tool based on your tooling selection.

Showing the Toolpath



Model File: PMSHPTH.PM4

Perform these tasks to show the toolpath:

1. Open the model file **`PMSHPTH.PM4`**.

2. Select **View—Get View**. The **Get View** dialog box is displayed.

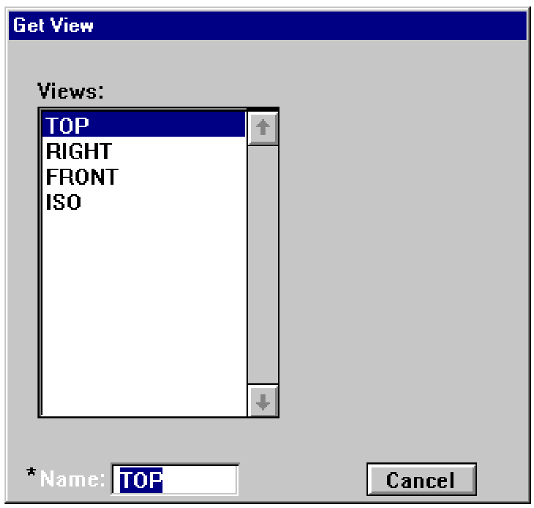


Figure 4-97
Select ISO from the Get View dialog box.

3. Select **ISO** from the **Get View** dialog box.

4. Select **View—Show Path**. The **Show Path** dialog box is displayed.

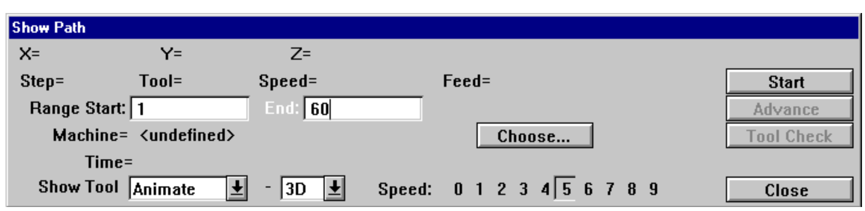


Figure 4-98
Set the values on the Show Path dialog box.

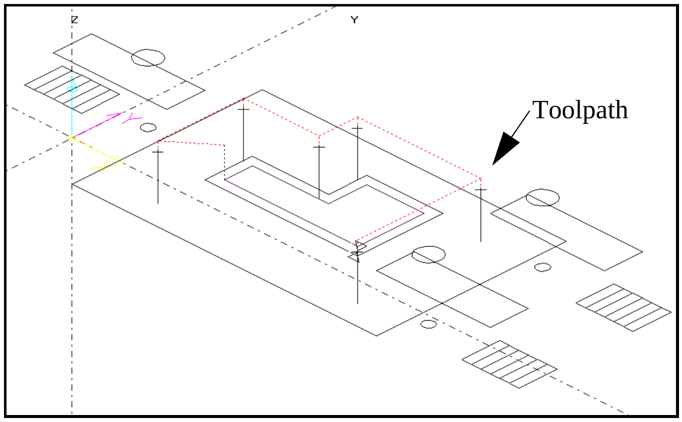
5. Set **Show Tool** to **Animate - 3D**, and set **Speed** to **5**.

6. Select the **Start** button. SmartCAM simulates the toolpath. You can see whether the geometry has the necessary sequence and manufacturing properties.

   Note You may change the simulation speed any time during Show Path by pressing the number keys 1-9.

7. To stop **Show Path**, press the Esc key.

Figure 4-99
The toolpath is displayed.



Points to Remember

- ❑ Show Path verifies the sequence and toolpath before you cut parts.
- ❑ SmartCAM displays a standard tool based on your tooling selection unless you specify a CTG file for the tooling information.
- ❑ Select the Start button on the Show Path dialog box to begin displaying toolpath.
- ❑ Select ESC to stop displaying toolpath.

Using Order Path

Objectives

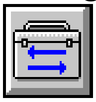
This lesson shows you how to perform these tasks:

- Chain geometry.
- Reverse the order of geometry.
- Move the profile start.
- Resequence curves.
- Optimize the order of hole operations.

Overview

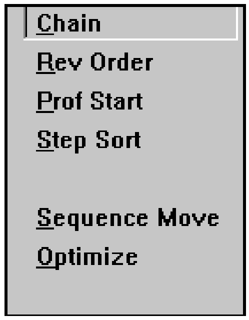
Use Order Path modeling tools to change the sequence and direction of the toolpath. You can verify that the toolpath is valid by using Order Path to order elements sequentially and in the same direction. If necessary, you can change the database sequence of the active group or sort it according to the assigned steps.

Using the Order Path Toolbox

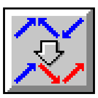


Open the Order Path toolbox by selecting the Order Path Toolbox icon or by selecting Order Path from the Edit menu. You can use the Order Path toolbox to change the order and sequence of the toolpath.

Figure 4-100
Open the Order Path toolbox.



Chaining Geometry

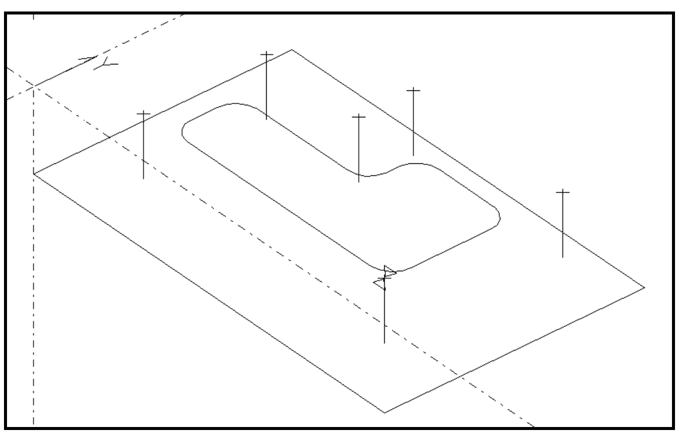


Model File: PMORDER.PM4

Use Chain to convert connecting elements into a contiguous profile of sequential elements so that the end point of one element is the start point of the next element. You can also join individual lines or polylines into one polyline element. Perform these tasks to chain geometry:

1. Open the model file **`PMORDER.PM4`**.

Figure 4-101
Open PMORDER.



2. Select **View—Show Path**. The **Show Path** control panel is displayed.
3. Select the **Start** button.
4. Select the **Close** button after you are finished viewing the toolpath.

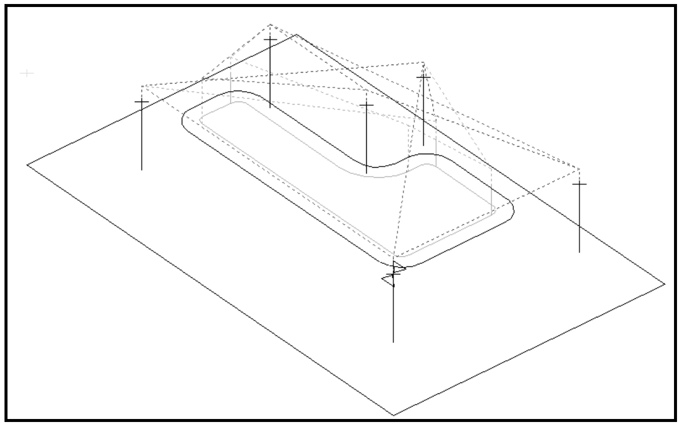


Figure 4-102
Notice the initial toolpath is out of order.

5. Select **Edit—Order Path**.

6. Select **Chain** from the toolbox. The **Chain** control panel is displayed.

Figure 4-103
Set the values on the Chain control panel.

7. Set the **Element in Profile to Chain** input field. Select any element of the profile to chain.

   Note The element you select determines the direction of the other elements. If the connected elements form a closed profile, the selected element is the starting element of the profile.

8. Set the **Chain**, **Polyline Join**, or **Both** option switch; otherwise, select the **Group Chain** button:

   ■ **Chain** links all elements together in one profile.

   ■ **Polyline Join** joins a series of lines or polylines into one polyline, decreasing the number of elements in the database.

   ■ **Both** creates a continuous polyline and a profile chain at the same time.

   ■ **Group Chain** chains selected groups.

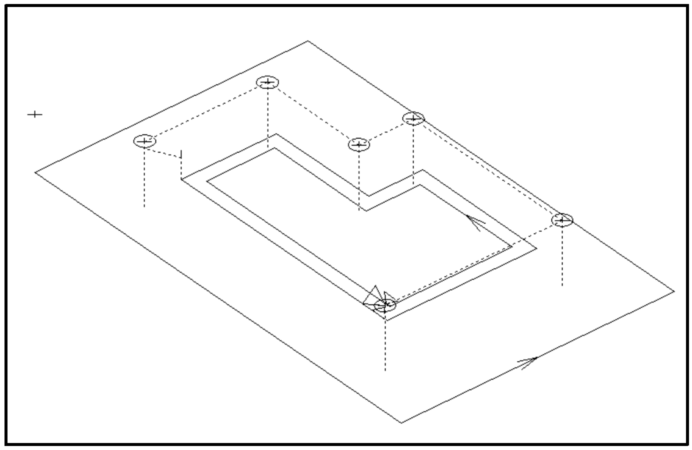
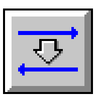


Figure 4-104
Notice that the final toolpath is in order and does not gouge the part.

Reversing the Order of Geometry



Model File: PMORDER.PM4

Use Rev Order to reverse the direction or database order of a selected element, profile, or group of elements. Perform these tasks to reverse the order of geometry:

1. Open the model file **PMORDER.PM4**.

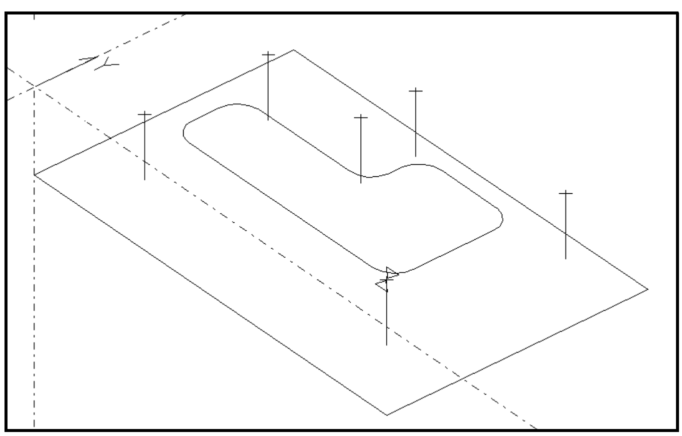


Figure 4-105
Open PMORDER.

2. Select **Edit—Order Path**.
3. Select **Rev Order** from the toolbox. The **Reverse Order** control panel is displayed.

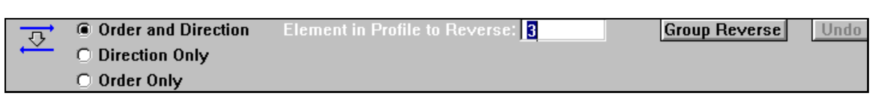


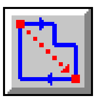
Figure 4-106
Specify values for the Reverse Order control panel.

4. Set one of the following:

- **Order and Direction** reverses the order of the elements or profiles relative to each other and the element direction of each individual element or profile.
- **Direction Only** reverses the element direction only, leaving individual elements or profiles in the same location in the database.
- **Order Only** reverses the order of profiles or individual elements without changing their direction.

5. Set the **Element in Profile to Reverse** input field. Select an element to reverse. To reverse an entire profile, select any element in the profile.
6. Select the **Group Reverse** button to perform the reverse order operation on the active group.

Moving Profile Start

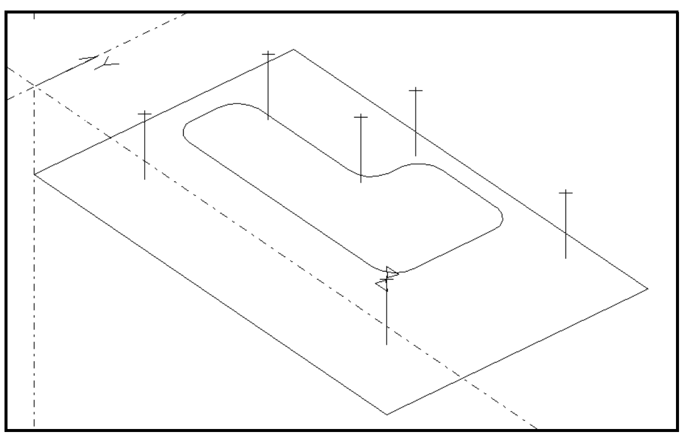


Model File: PMORDER.PM4

Use Prof Start to resequence a single profile so that the element you select is the first element. You can use Prof Start to position cut starts strategically at convenient locations. Perform these tasks to move the profile start:

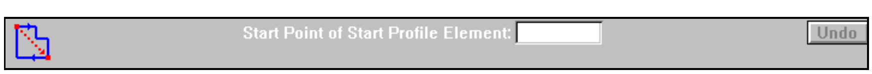
1. Open the model file **PMORDER.PM4**.

Figure 4-107 *Open PMORDER.*



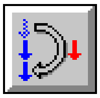
2. Select **Edit—Order Path**.
3. Select **Prof Start** from the toolbox. The **Prof Start** control panel is displayed.

Figure 4-108 *Specify values on the Prof Start control panel.*



4. Set the **Start Point of Start Profile Element** field to a start point.

Resequencing Curves



Model File: PMORDER.PM4

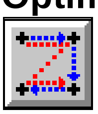
Use Sequence Move to change the sequence of the active group to a defined position. You can also rearrange the selected elements into the order in which they were added to the group. Use Sequence Move to place different machining events or elements into the most efficient order in the database. Perform these tasks to resequence curves:

1. Keep the **`PMORDER.PM4`** model file displayed.
2. Set the insert position where you want it in the database because Sequence Move places the newly sequenced curves at this location.
3. Select **Edit—Order Path**.
4. Select **Sequence Move** from the toolbox. The **Sequence Move** control panel is displayed.

Figure 4-109 *Specify values for the Sequence Move control panel.*

5. Group the geometry.
6. Set the **Move Group to Current Insert Position** switch to one of the following:
   - **By Existing Group Sequence** moves an active group of elements to a new location in the existing sequence.
   - **By Group Selection Sequence** moves an active group of elements to a new location in the order in which you select them.
7. Select the **Go** button to reorder the curves. Select the **Undo** button to undo the selection.

Optimizing the Order of Hole Operations



Model File: PMOPTIMZ.PM4

Use Optimize to create an efficient toolpath for your machine tool without changing the geometry or associated properties. For example, use Optimize to reduce machine travel and tool changes. You can also use this modeling tool to sort the active group by tool order for single or multiple parts.

Note Optimize resequences the database, so it is advisable to save the file before selecting the modeling tool. Also, do not exit the Optimize control panel until you are satisfied with the results. Use Show Path with Show Tool set to **Filled** for a more thorough verification process.

Perform these tasks to optimize the order of hole operations:

1. Open the model file **`PMOPTIMZ.PM4`**.

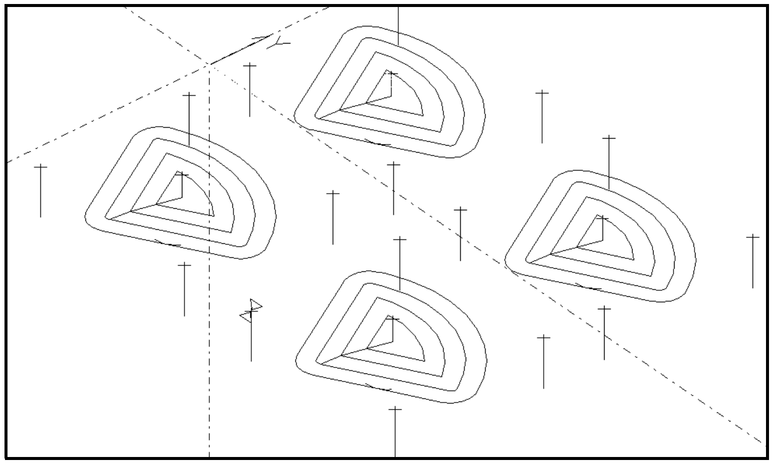


Figure 4-110
Open PMOPTIMZ.

2. Select **Edit—Order Path**.
3. Select **Optimize** from the toolbox. The **Optimize** control panel is displayed.

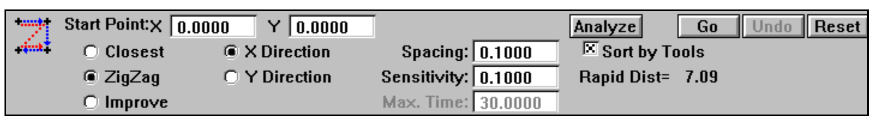


Figure 4-111
Specify values for the Optimize control panel.

4. Set the **Start Point** input fields to the starting location for the closest element or the starting corner of a zigzag pattern.
5. Turn on the **Closest**, **ZigZag**, or **Improve** switch:
   - **Closest** sequences toolpath elements according to the position of the closest geometry.
   - **ZigZag** creates a sequence of toolpath elements based on a zigzag pattern.
   - **Improve** determines the optimum sequence of toolpath elements based on the closest distance that SmartCAM can calculate within the time frame you specify in the Max. Time input field.
6. If you turn on **ZigZag**, turn on **X Direction** or **Y Direction** to determine the primary direction for a zigzag pattern.
7. Set the **Spacing** and **Sensitivity** input fields:
   - **Spacing** determines the maximum distance to search in the primary direction of a zigzag pattern.
   - **Sensitivity** determines the maximum division between zigzag passes in the secondary direction of the zigzag pattern. The value you enter is evaluated as a ratio of the spacing value. (For example, **0.5** would equal 50 percent of the spacing value.)

8. If you turn on **Improve**, set **Max. Time** to the maximum number of minutes to spend calculating the optimum sequence of toolpath elements.

Points to Remember

- ❑ Use Order Path to change the sequence and direction of the toolpath.
- ❑ Use Rev Order to reverse the direction or database order of a selected element.
- ❑ Use Sequence Move to place machining events or elements in the most efficient order in the database.
- ❑ When you use Optimize, save the file before selecting modeling tools.

Transforming Geometry

Objectives

This lesson shows you how to perform these tasks:

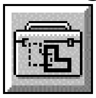
- Move geometry.
- Rotate geometry.
- Mirror geometry.
- Scale geometry.
- Show and mask geometry.

Overview

Use the Transform toolbox to change, or transform, the geometry of the active group. The Transform toolbox includes tools to move, rotate, scale, mirror, or copy a group.

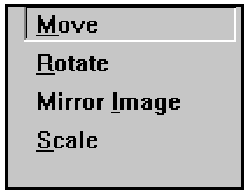
Note The Transform modeling tools are not available unless there is an active group.

Using the Transform Toolbox



Open the Transform toolbox by selecting the Transform Toolbox icon or by selecting Transform from the Edit menu. Use the modeling tools in the Transform toolbox to move, rotate, mirror, or scale geometry elements that are in the active group.

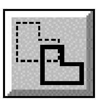
Figure 4-112 *Use the Transform toolbox.*



Creating Work Planes When Transforming

You can create a work plane when you rotate and mirror geometry by turning off the Suppress Plane on/off switch. The work plane you create is based on the current transformation.

Moving Geometry

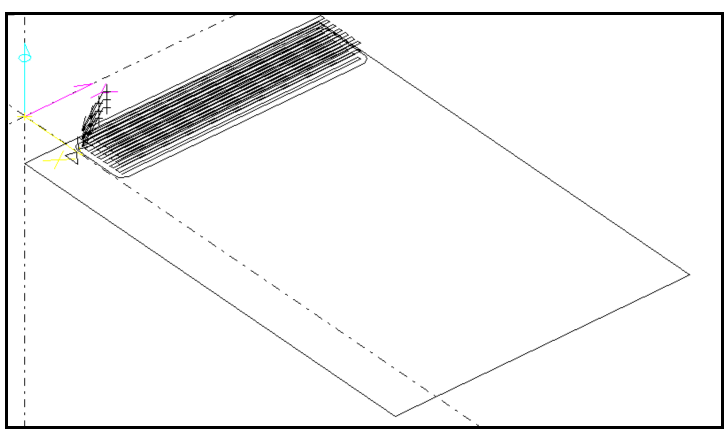


Model File: PMMOVE.PM4

Use Move to change the location of the active group or, with the Copy option, move one or more copies to new locations. You can also move an active group from the current work plane to a destination plane. Perform these tasks to move geometry:

1. Open the model file **PMMOVE.PM4**.

Figure 4-113 *Open PMMOVE.*



2. Group the elements to move.
3. Select **Edit—Transform**.
4. Select **Move** from the toolbox. The **Move** control panel is displayed.

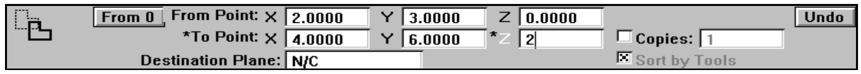
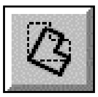


Figure 4-114 *Set the values on the Move control panel.*

5. Set the **From Point** input fields to the point from which to move the group.
6. Set the **To Point** input fields to the location to which you are moving the group.

Rotating Geometry



Model File: PMROTATE.PM4

Use Rotate to rotate an active group around a pivot axis on the active work plane. With the Sort by Tools option, elements are automatically sequenced and can be sorted so that all uses of each tool are grouped together to reduce tool changes. Perform these tasks to rotate geometry:

1. Open the model file **PMROTATE.PM4**.

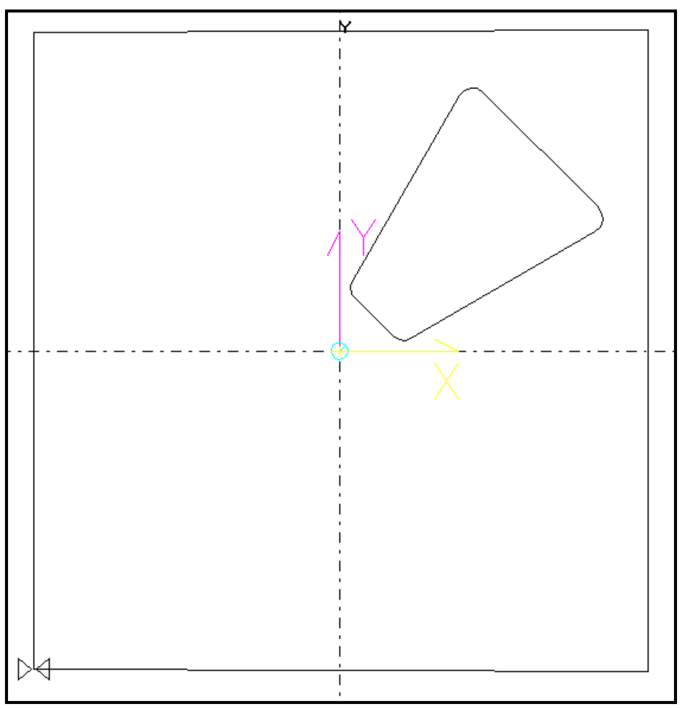


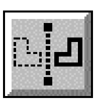
Figure 4-115 *Open PMROTATE.*

2. Group the elements to rotate.
3. Select **Edit—Transform**.
4. Select **Rotate** from the toolbox. The **Rotate** control panel is displayed.

Figure 4-116
Set the values on the Rotate control panel.

5. Turn on the **2D** or **3D** option switch:
   - **2D** pivots the active group around the specified point. The rotation is parallel to the active work plane.
   - **3D** rotates the active group around the specified three-dimensional axis.
6. Set the **Pivot Axis Point** or **Axis End Point** input fields to the location of the pivot point, as follows:
   - For **2D** rotation, this is the point around which the active group pivots.
   - For **3D** rotation, this is one end of the pivot axis. If the option switch is set to **3D**, identify the other end of the pivot axis.
7. Turn on the **Suppress Planes** on/off switch to limit creation of auto planes for two-dimensional operations and nonplanar elements for three-dimensional operations.
8. Set the **Rotation Angle** input field.
9. Select the **Go** button.

Mirroring Geometry



Model File: PMMIRROR.PM4

Use Mirror Image to create a reverse, or mirror image, of the elements in the active group. You can use this to create symmetrical parts or left- and right-handed versions of the same part.

Mirror Image reverses the toolpath direction and places the offsets on the proper side. You can maintain the same cut type for the image as for the original or you can reverse the cut type. Perform these tasks to mirror geometry:

1. Open the model file `PMMIRROR.PM4`.

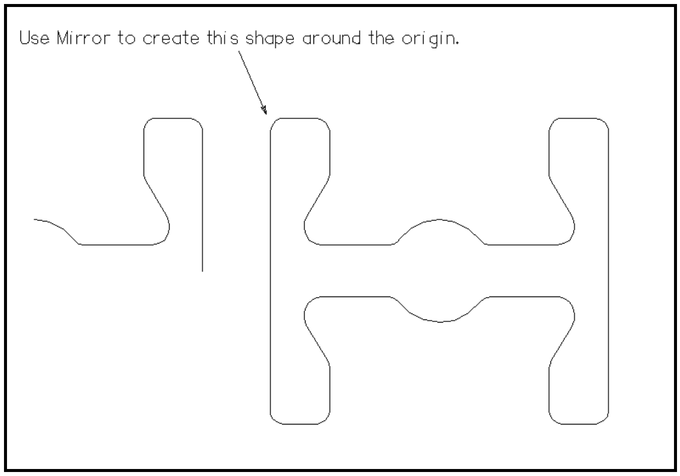


Figure 4-117
Open PMMIRROR.

2. Group the elements to mirror.
3. Select **Edit—Transform**.
4. Select **Mirror Image** from the toolbox. The **Mirror Image** control panel is displayed.

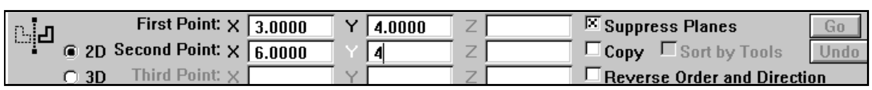


Figure 4-118
Set the values on the Mirror Image control panel.

5. Turn on the **2D** or **3D** option switch:
   - **2D** creates a mirror image across a line.
   - **3D** creates a mirror image across a plane.
6. Set the **First Point** input fields to the starting point of the line along which to mirror the image.
7. Set the **Second Point** input fields to the ending point of the line along which to mirror the image.
8. Turn on the **Copy** on/off switch on to create a copy of the original group of elements on the mirror side.
9. If **Copy** is on, turn off the **Sort by Tools** on/off switch to machine the elements in each copy independently of all other copies. Turn on this on/off switch to coordinate sequencing of the elements in all copies so that each tool completes its operation in all copies before moving to the next tool.
10. Turn on the **Reverse Order and Direction** on/off switch to create mirror-image elements that travel in the same direction as the original elements. Turn this switch off to create mirror-image elements that travel in the opposite direction.
11. Select the **Go** button.

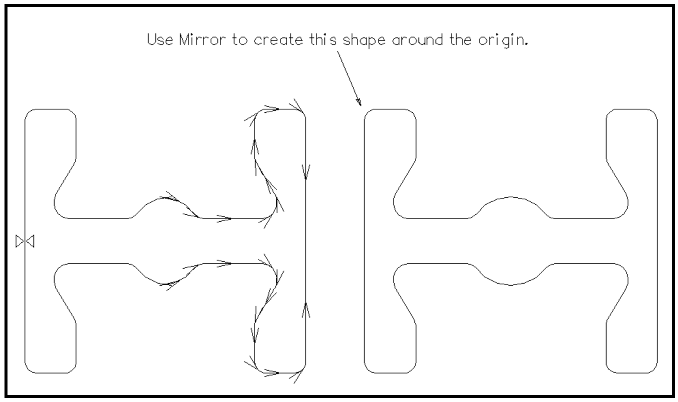
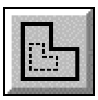


Figure 4-119
This is the result of mirroring the geometry.

Scaling Geometry



Model File: PMROTATE.PM4

Use Scale to increase or decrease the size of a selected group. You can change the size of the elements proportionally or distort them by using different values for the X, Y, or Z axes. Scale operates relative to the XY plane.

Elements that can be scaled include holes, arcs, lines, polylines, ellipses, splines, points, and user elements. If you scale a helix nonproportionally, SmartCAM uses the larger scale factor as a radius to maintain the circular shape. You cannot scale dimensioned text.

Scale is often used to allow for shrinkage or expansion factors and scale conversions. Scale changes the size of elements in the database and the corresponding machine code. Perform these tasks to scale geometry:

1. Open the model file **`PMROTATE.PM4`**.

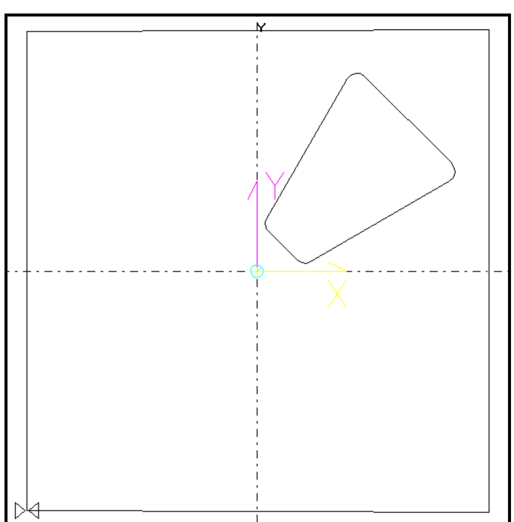


Figure 4-120
Open PMROTATE.

2. Group the elements to scale.
3. Select **Edit—Transform**.
4. Select **Scale** from the toolbox. The **Scale** control panel is displayed.

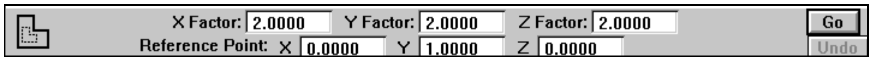


Figure 4-121 *Set the values on the Scale control panel.*

5. Set the following fields on the control panel:
   - Set the **X Factor** input field to the value by which to alter the size of the group of elements along the world X axis.
   - Set the **Y Factor** input field to the value by which to alter the size of the group of elements along the world Y axis.
   - Set the **Z Factor** input field to the value by which to alter the size of the group of elements along the world Z axis.
   - Set the **Reference Point** input fields to the value from which the scaling operation occurs. This is the only point in the scaled group that does not change.
6. Select the **Go** button.

Points to Remember

- ❑ Transforming tools are not available unless there is an active group.
- ❑ Use Move to change the location of the active group.
- ❑ Use Rotate to rotate an active group around a pivot axis on the active work plane.
- ❑ Mirror Image reverses the toolpath directions and places the offsets on the proper side. You can maintain the same cut type for the image as for the original or you can reverse the cut type.
- ❑ Use Scale to increase or decrease the size of a selected group.
- ❑ Use Scale for unit conversions.
- ❑ SmartCAM does not display masked elements, so you cannot select or code them.
- ❑ Masked elements are not deleted from the database.
- ❑ Press F8 to instantly identify masked elements.

Importing a CAD File

Objectives

This lesson shows you how to bring a CAD file into SmartCAM.

Overview

Import transfers the geometry contained in DXF, DWG, VDA-FS, and IGES files into a SmartCAM process model file (.pm4).

The quality and reliability of the data transfer improves when you select only the geometry you need and appropriate conversion options from the CAD system before importing the CAD file.

After importing a file, you can add or remove surfaces and apply machinin processes to generate code.

Using Import

A CNC process model is created using either inch or metric units, and consists of a .pm4 file and a .jof file. The .pm4 file holds the geometry while the .jof file specifies all the tooling and operation information. Every time you save a CNC process model, these files are saved as a pair.

The unit setting establishes the units for everything relating to the model, including the job tooling. SmartCAM does not support inch and metric units in the same model. When you are ready to import a file, determine the units of both the file being imported and the .jof file in the process model. How you import the geometry depends on whether these two file units are the same or different.

You can import in these ways:

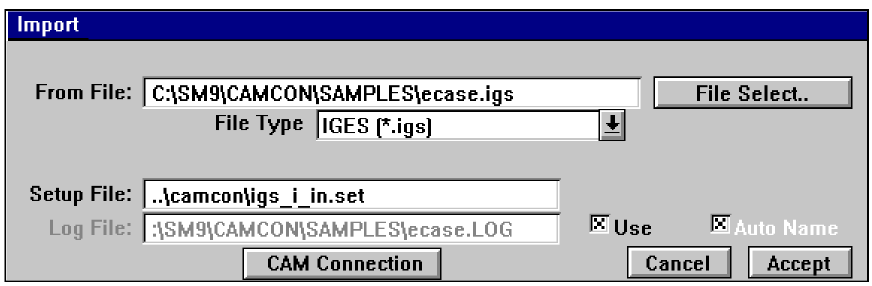
- Import into a model file that has the same units.
- Import into a model file that has different units, and adopt the units of the existing file.
- Import into a model file that has different units, and adopt the units of the incoming file.

Importing into a Model with the Same Units

Perform these tasks to import a CAD file into a new or existing model that has the same units:

1. If you are importing into a new file, set the units in the new file to match the units of the file to import. If you are importing into an existing file, proceed to step 2.
2. Select **File—Import**. The **Import** dialog box is displayed.

Figure 4-122
Open the Import dialog box.



3. Set the **File Type** selector switch.
4. Enter the path and name of the file to import in the **From File** input field, or use the **File Select** button to specify the path and name of the file.
5. Enter the name of the setup file used with the file type of the imported file in the **Setup File** input field. Generally, there is a separate setup file for each file type. In most cases, the setup file is automatically inserted when the file type is selected. You can also use a customized setup file.
6. Turn on the **Use** on/off switch if you want to create a log file.

   Note Leave this switch off unless you have problems importing the file because log files are typically very large.

7. Name the log file or have SmartCAM name it for you.
   - To name the log file, enter the name of the log file to create in the **Log File** input field.
   - To have SmartCAM name the log file, turn on the **Auto Name** on/off switch. (You must have turned on the **Use** on/off switch.)

8. Select **Accept** to load the file.

Importing and Adopting the Existing Model File's Units

Perform these tasks to import a CAD file into a new or existing model that has different units:

1. Open the model file that you want to import the CAD file into.
2. Complete steps 2 through 8 of *Importing into a Model with the Same Units,* on page 4-74.
3. Select the **Name Group** icon from the **Group** tool palette.
4. Select the **Result** group.
5. Select **Edit—Transform—Scale**.
6. Specify the appropriate scale factor.
   - To change from inch to metric, multiply by **25.4**.
   - To change from metric to inch, multiple by **.03937**.
7. After the import, assign tools and operations to the geometry.
8. Verify the model using **Show Path** or **Element Data**.

Importing and Adopting the Incoming Model File's Units

1. Open the model file that you want to import the CAD file into.
2. Complete steps 2 through 8 of *Importing into a Model with the Same Units,* on page 4-74.
3. Select **File—Load Job File**.
4. Use the **File Select** button, which opens the **Open** dialog box, and browse for the .jof file that you want to use to load your steps.

   Note When you load a `.jof` file, the units setting in the existing model is changed to the units of the newly loaded `.jof` file, regardless of the units set in Utility—System Units.
5. Select the **Name Group** icon from the **Group** tool palette.
6. Select the **Result** group.
7. Select **Edit—Transform—Scale**.
8. Specify the appropriate scale factor.
   - To change from inch to metric, multiply by **25.4**.
   - To change from metric to inch, multiple by **.03937**.

9. After the import, assign tools and operations to the geometry.
10. Verify the model using **Show Path** or **Element Data**.

Points to Remember

- The quality and reliability of the data transfer improves when you select only the geometry you need and appropriate conversion options from the CAD system before importing the CAD file.
- SmartCAM does not support inch and metric units in the same model.
- When you are ready to import a file, determine the units of both the file being imported and the .jof file in the process model. How you import the geometry depends on whether these two file units are the same or different.
- You can import in these ways:
  - Import into a model file that has the same units.
  - Import into a model file that has different units, and adopt the units of the existing file.
  - Import into a model file that has different units, and adopt the units of the incoming file.
- If you are importing into a new file, set the units in the new file to match the units of the file to import.

Exporting a SmartCAM File

Objectives

This lesson shows you how to convert the elements in an existing SmartCAM process model to a DXF (.dxf), DWG (.dwg), VDA-FS (.vda), or an IGES (.igs) file.

Overview

The quality and reliability of the data transfer improves when you select only the geometry you need and appropriate conversion options before exporting the SmartCAM process model.

Using Export

Perform these tasks to export a SmartCAM process model to a CAD file:

1. Select **File—Export**. The **Export** dialog box is displayed.

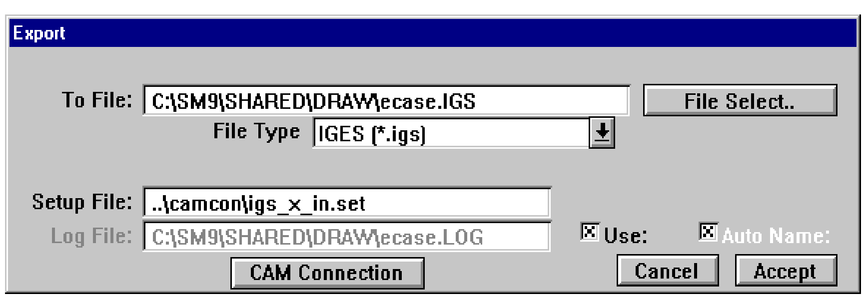


Figure 4-123 *Open the Export dialog box.*

2. Set the **File Type** selector switch.
3. Enter the path and name of the file to export in the **To File** input field, or use the **File Select** button to specify the path and name of the file.

4. Confirm the name of the setup file used with the file type of the exported file in the **Setup File** input field. Generally, there is a separate setup file for each file type. In most cases, the setup file is automatically inserted when the file type is selected. You can also create a customized setup file. If the selected file type does not require a setup file, this input field is dim.
5. Turn on the **Use** on/off switch if you want to create a log file.
6. Name the log file or have SmartCAM name it for you.
   - To name the log file, enter the name of the log file to create in the **Log File** input field.
   - To have SmartCAM name the log file, turn on the **Auto Name** on/off switch. (You must have turned on the **Use** on/off switch.)
7. Select **Accept** to export the file. When the export is complete, the window displays a *Done* message, and the status and export windows are automatically closed.

Points to Remember

- ❑ You can export SmartCAM process model files to a DXF (.dxf), DWG (.dwg), or an IGES (.igs) file.
- ❑ The quality and reliability of the data transfer improve when you select only the geometry you need and appropriate conversion options before exporting the SmartCAM process model.

Self-Test

Directions

Test your understanding of the concepts and procedures in this section by answering the following questions. The answers for each self-test are in *Appendix A* of this manual.

_______________ **1. What do you need to do before creating new geometry?**

a) Identify the insert location.
b) Identify the properties for the geometry.
c) both a and b
d) neither a nor b

_______________ **2. Associating geometry with a step will result in code.**

a) true
b) false

_______________ **3. Associating geometry with a layer will result in code.**

a) true
b) false

_______________ **4. How many elements in a model can you view modeling data for?**

a) 0
b) 1
c) 2
d) as many as you want

_______________ **5. Viewing element data affects the geometry database.**

a) true
b) false

______________ **6. Viewing element data affects the geometry database.**

a) true
b) false

______________ **7. Changing the color of elements:**

a) never affects code generation
b) rarely affects code generation
c) usually affects code generation
d) always affects code generation

______________ **8. Elements to be trimmed or extended should be on different work planes.**

a) true
b) false

______________ **9. How many groups does trimming by group impact?**

a) 0
b) 1
c) 2
d) as many as you want

______________ **10. How many deleted groups can be recovered with the Undo button?**

a) 0
b) 1
c) 2
d) as many as you want

______________ **11. Which feature verifies the sequence and toolpath before you cut parts?**

a) Order Path
b) Reverse Order
c) Sequence Move
d) Show Path

______________ **12. When you use Optimize, should you save the file before or after selecting modeling tools?**

a) before
b) after

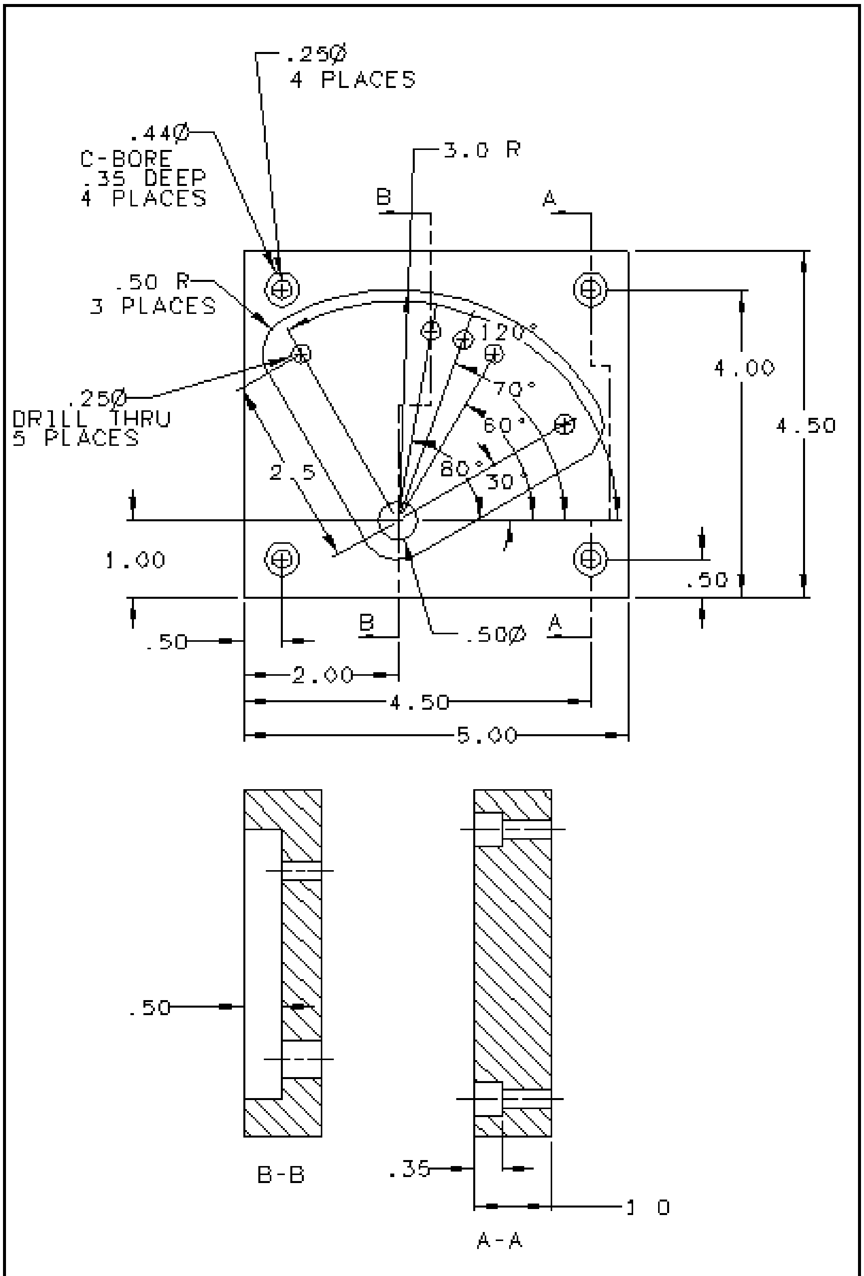
Challenge Project

Directions

Create a toolpath for the model in the part print.

Part Print

Figure 4-124 *Create a toolpath for the model.*



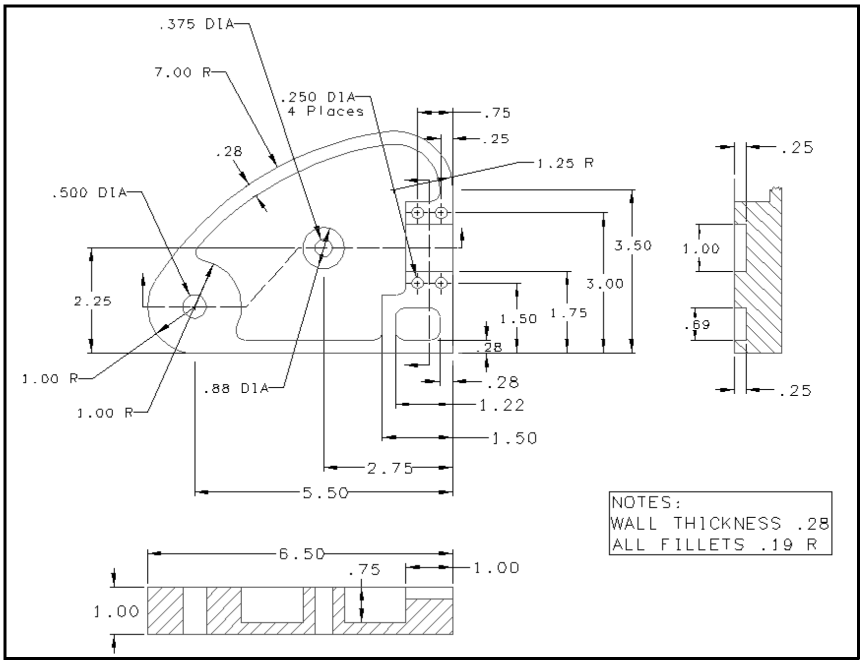
Challenge Project

Directions

Create a toolpath for the model in the part print.

Part Print

Figure 4-125 *Create a toolpath for the part.*



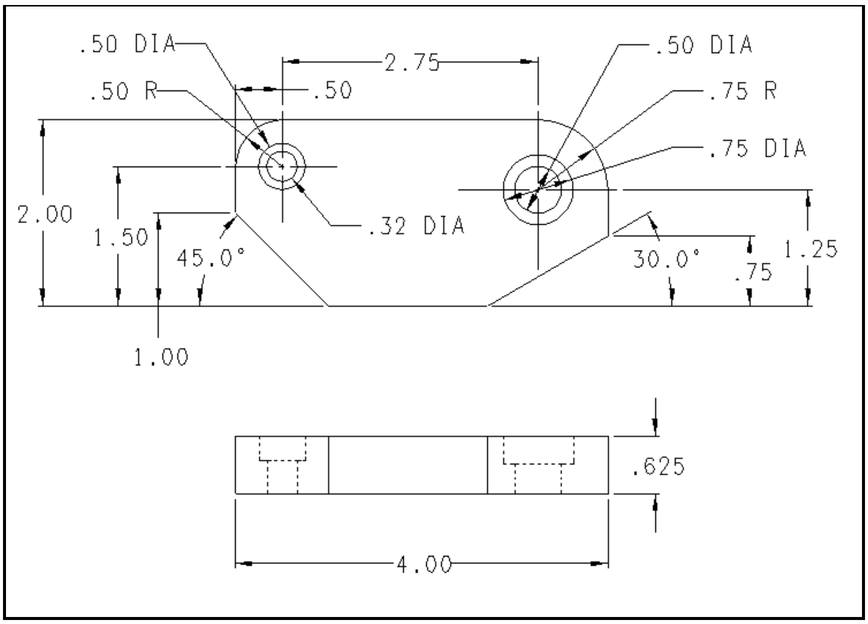
Challenge Project

Directions

Draw the geometry to make the toolpath for this part print. Fixture the part in a vise, then drill and counterbore the holes. Refixture the part using the drilled holes, then cut the outside profile of the part.

Sample Part Print

Figure 4-126 *Copy the part print.*



Procedure

1. Start a new .jof file.
2. Add four hole steps to the .jof file to make two holes. Ensure that each of the two holes has one drill step and one counterbore step.
3. Drill and counterbore the two holes.
4. Insert a user command, such as an Op Stop, to enable the part to be refixtured. Fixture the part using the two holes.
5. Add a milling step (End Mill) to the .jof file.
6. Mill the outside profile of the part.

Challenge Project

Take this opportunity to practice the skills that you learned.

Objectives

- Define the steps.
- Drill two holes.
- Mill an outside profile.
- Mill a slot through the center of the part.

Directions

Use the part print to draw the geometry to make the toolpath. Fixture the part in a vise, mill the slot, and drill the holes. Refixture the part using the drilled holes, then cut the outside profile of the part.

Part Print

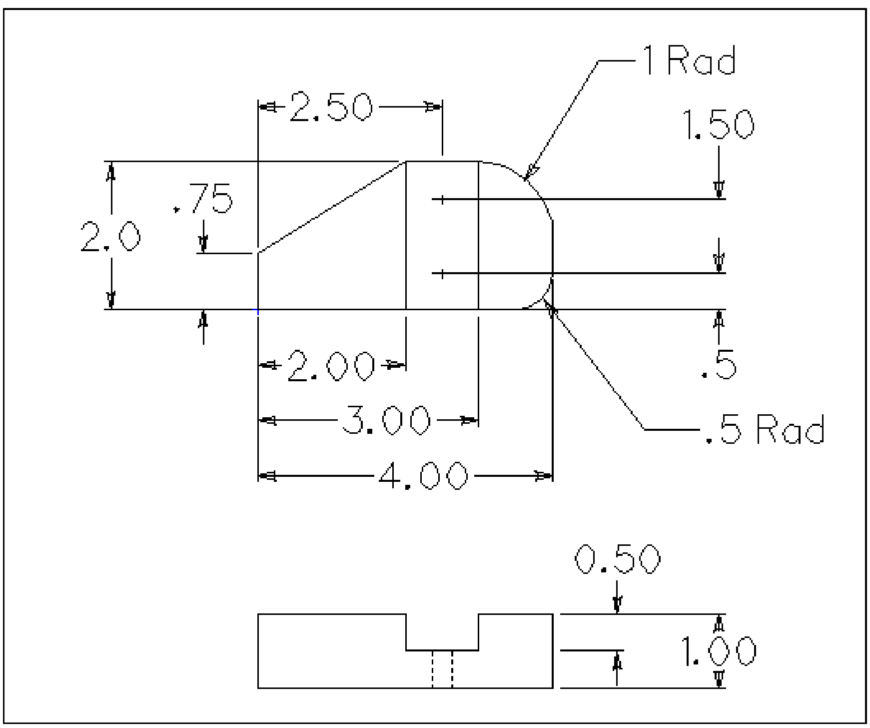


Figure 4-127
Copy the part print.

Procedure

1. Start a new .jof file:
   a. Select **File—New**.

b. Select **File—Planner** and then select the **Job Info** button.

c. Make sure the units are correctly set.

d. Select the **Accept** button and then select the **Close** button when you are finished.

2. Add a milling step to the Job Operation Planner:

a. Select **File—Planner**.

b. Select the **Process Step List** tab and then select the **Add** button.

c. Set **Op Category** to **Milling Operations**.

d. Set **Op Type** to **Rough Milling**.

e. Set **Tool Category** to **Milling Tools**.

f. Set **Tool Type** to **End Mill**.

g. Select the **Accept** button.

h. Set the **Process Step #** field.

i. Set the **Description** input field to **.75 [19.05] diameter end mill**.

j. Set **Tool Diameter** to .**75[19.05]**.

k. Set **Cut Length** to **1.25[31.75]**.

l. Select the **Operation** tab.

m. Set the **Speed** to **700 RPM**, **Primary Feed** (**IPM**) to **10[245],** and **Plunge Feed** to **5 IPM**.

n. Select the **Accept** button.

o. Select the **Close** button to save the information.

3. Cut the one-inch slot through the middle of the part for the first milling operation. Do this by drawing two lines (the lines are drawn longer so that the tool does not plunge into the material):

a. Select **Insert**–**With Step**, and then select the milling cutter you just created.

b. Set **Level** to **-.5[-12.7]**.

c. Set **Clear** to **.25[6.35]**.

d. Set **Prof Top** to **0**.

e. Set **Offset** to **Left**.

f. Select **Create—Geometry—Line**.

g. Set the **Start Point** field for the first line to **3[73.5], -.4[-9.8], -.5[-12.7]**.

h. Set the **End Point** field for the first line to **3[73.5], 2.4[58.8], -.5[-12.7]**.

i. Set the **Start Point** field for the second line to **2[50.8], 2.4[58.8], -.5[-12.7]**.

j. Set the **End Point** field for the second line to **2[50.8], -.4[-9.8], -.5[-12.7]**.

4. Add a hole step to the .jof file:

   a. Select **File—Planner**.

   b. Select the **Process Step** tab and select the **Add** button.

   c. Set **Op Category** to **Hole Operations**.

   d. Set **Op Type** to **Spot Drilling**.

   e. Set **Tool Category** to **Hole Tools**.

   f. Set **Tool Type** to **Spot Drill**.

   g. Set the **Process Step #** input field.

   h. Set the **Description** input field to **.25[6.35] diameter twist drill**.

   i. Set **Tool Diameter** to **.25[6.35]**.

   j. Set **Cutter Length** to **2.5[63.5]**.

   k. Select the **Operation** tab.

   l. Set **Spindle RPM** to **2000** and **Feedrates** (**IPM**) to **5[127]**.

   m. Select the **Accept** button.

   n. Select the **Close** button and save the information.

5. Drill the two holes in the bottom of the slot:

   a. Select **Insert—With Step**, then select the drill that you just created.

   b. Set **Level** to **-.5[-12.7]**.

   c. Set **Clear** to **.25[6.35]**.

   d. Select **Create—Geometry—Hole.**

   e. Set **Full Depth** to **.5[12.7]**.

   f. Set the drill holes to **2.5[63.5], .5[12.7]** and **2.5[63.5], 1.5[38.1]**.

6. Insert a user command, such as Op Stop, to enable the part to be refixtured. Fixture the part using the two holes:

   a. Select **Create—User Elmts—User Event**.

   b. Set **Text** to **M01**.

   c. Set the XY coordinates to **0,4[98]** and **level** to **2**.

   d. Select the **Go** button.

7. Cut the profile of the part with the .75[19.05] diameter end mill:

a. Select **Insert—With Step**, and select the **.75[19.05] end mill**.
b. Set **Level** to **-1[-25.4]**.
c. Set **Offset** to **Left**.
d. Set **Prof Top** to **0**.
e. Set **Clear** to **.25[6.35]**.
f. Select **Create—Profiles—Start Profile**.
g. Set the XY location to **0, 0**.
h. Select **Line Profile** from the toolbox.
i. Set the **End Point** field for the profile line to **0, .75[19.05]**.
j. Set **End Point** to **2[50.8], 2[50.8]**.
k. Set **End Point** to **3[73.5], 2[50.8]**.
l. Select **Arc Profile** from the toolbox.
m. Set the **Arc Direction** to **CW** and the **Radius** to **1[25.4]**.
n. Set the **Arc End Point** to **4[98], 1[25.4]**.
o. Toggle the **Other** switch in the **Which Solution** dialog box until the correct solution is shown, then select the **Accept** button.
p. Select **Line Profile** from the toolbox.
q. Set the **Line End Point** to **4[98], .5[12.7]**.
r. Select **Arc Profile** from the toolbox.
s. Set **Arc Direction** to **CW** and the radius to **.5[12.7]**.
t. Set **Relation** to **Tangent**.
u. Set **Arc End Point** to **3.5[88.9]**, **0**.
v. Select **Line Profile** from the toolbox.
w. Set **End Point** to **0, 0**.

5

Generating and Verifying Roughing Toolpath

Objectives

These lessons will show you how to perform Rough machining to generate toolpaths. You will then use Show Cut to verify the toolpaths.

Lessons for This Unit

- Generating Roughing Toolpath
- Verifying Roughing Toolpath

Generating Roughing Toolpath

Objectives

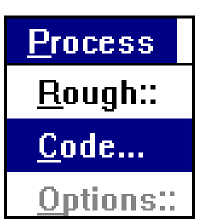
This lesson shows you how to perform these tasks:

- Pocket an area with islands.
- Pocket multiple areas.
- Cut an open profile.
- Face a part.
- Set spiral parameters.
- Set linear parameters.
- Profile a part.

Overview

Use the Process menu to generate a toolpath for roughing an area, to generate CNC code, and to access optional functions you have purchased.

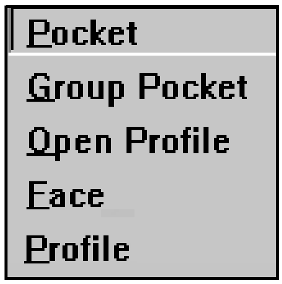
Figure 5-1
Use the Process menu.



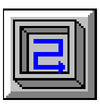
Using the Rough Toolbox

Use the Rough toolbox to create roughing toolpaths to remove a large volume of material using multiple cutting moves. The material to remove is defined by the XY position of the finish profile, combined with the Level and Profile Top (Z) position.

Figure 5-2
Create roughing toolpaths with the Rough toolbox.



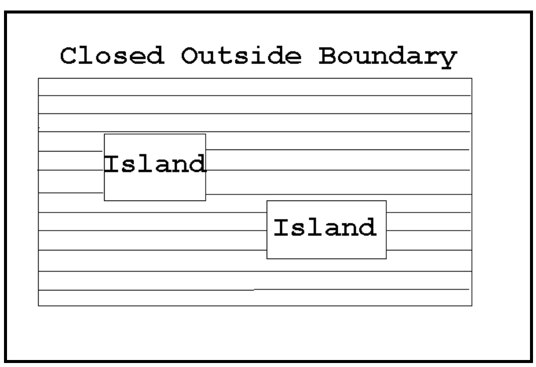
Pocketing



Model File: PMROUGH1.PM4

Use Pocket to create roughing profiles for the removal of material inside a single closed profile. Multiple islands can exist inside the profile.

Figure 5-3
Pocket a part.



The outside boundary defines the bottom edge of the pocket and has these characteristics:

- A closed profile
- A profile top value

Each island boundary defines the bottom edge of the island (where the island meets the pocket floor). To be recognized by SmartCAM, islands must be grouped. Islands have these characteristics:

- A closed profile
- A profile top value

Follow these steps to create a pocket:

1. Open the model file **`PMROUGH1.PM4`**.

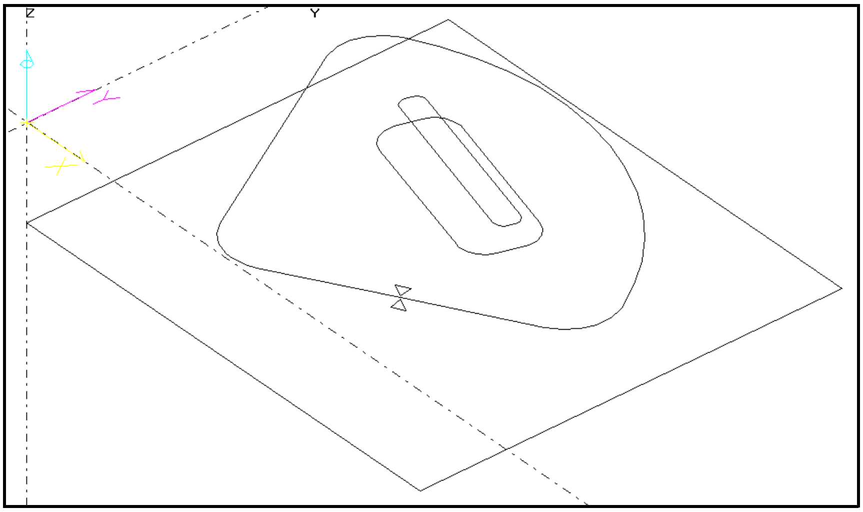
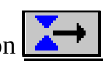


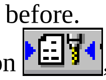
Figure 5-4
Open PMROUGH1.

2. Use the **Group Arrow** icon, or **Group** tool palette to identify an active group of islands and notches that SmartCAM should avoid. Each island profile must have a Prof_Top value to describe the island top.

3. Set the insert location:

   ■ Set the **Before** icon .

   ■ Select the **Step** icon , and select **Step #3**, which is the step for the new element to come before.

   ■ Set the **With Step** icon , and select **Step #2**.

   ■ Set the **Clear** input field.

4. Select **Process—Rough**.

5. Select **Pocket** from the toolbox. The **Pocket** control panel is displayed.

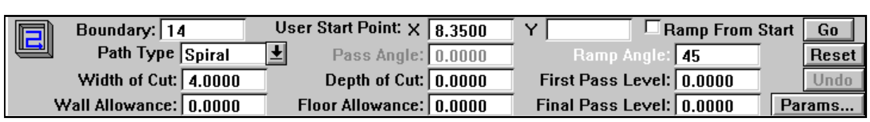


Figure 5-5
Set the values on the Pocket control panel.

6. Set the **Boundary** input field to any element in the boundary profile. The entire profile is selected.

   Note This must be a closed profile.

7. Set the **User Start Point** input fields to the coordinates for the beginning of the cut.

8. Set the **Path Type** selector switch to one of these options:

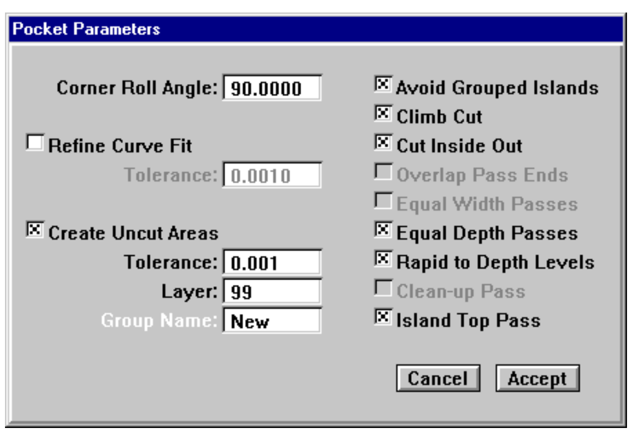
   ■ **Spiral** creates a spiral roughing operation.

- **Zig Zag** creates a zigzag roughing operation.
- **Linear** creates a linear roughing operation.

9. Set the remaining fields on the control panel:
   - Set the **Width of Cut** input field to the stepover distance.
   - Set the **Wall Allowance** input field to the amount of material to leave on the walls of the islands and pockets.
   - Set the **Pass Angle** input field to indicate the 2D cutting angle to use across the pocket.
   - Set the **Depth of Cut** input field to a positive value that represents the depth of cut.
   - Set the **Floor Allowance** input field to the amount of material to leave at the bottom of the pocket.
   - Set the **Ramp Angle** input field to the angle you want to cut on ramp moves.
   - Set the **First Pass Level** input field to a global value that represents the first level of pocketing.
   - Set the **Final Pass Level** input field to the depth of the bottom of the pocketing operation.
   - Turn on the **Ramp From Start** on/off switch to create ramp moves directly from the user start point location to the automatic start point of the toolpath. Turn off this switch to create a vertical plunge at the user start point. You must set a user start point to activate this option.

10. Select the **Params** button. The **Pocket Parameters** dialog box is displayed.

Figure 5-6
Set the values on the Pocket Parameters dialog box.

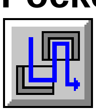


11. Set these fields on the Pocket Parameters dialog box:
    - Set **Corner Roll Angle** to the determining angle for corner rolling.
    - Turn on the **Refine Curve Fit** on/off switch to remove colinear points from the part profile. This is useful when the part profile is a spline. Otherwise, turn it off.

- Turn on the **Create Uncut Areas** on/off switch to create geometry for areas that the assigned roughing tool cannot cut.
- Turn on the **Avoid Grouped Islands** on/off switch to avoid an active group of islands or notches in the pocket.
- Turn on the **Climb Cut** on/off switch to create climb cuts. Turn off this switch to create conventional cuts.
- Turn on the **Cut Inside Out** on/off switch to begin the roughing operation in the center of the area and progressively step over toward the outside boundary.
- Turn on the **Overlap Pass Ends** on/off switch if you want the cutter to follow the profile up to the previous level after each cut before retracting.
- Turn on the **Equal Width Passes** on/off switch to make the width of all cutting passes the same.
- Turn on the **Equal Depth Passes** on/off switch to make the depth of all cutting passes the same.
- Turn on the **Rapid to Depth Levels** on/off switch to rapid-move the tool to the levels of the previous roughing passes.
- Turn on the **Clean-up Pass** on/off switch to have the tool perform a pass around the perimeter of the pocket.
- Turn on the **Island Top Pass** on/off switch to have the tool create a pass over all island tops at a depth without roughing the entire pocket at that depth. The next full-width pocket pass is performed at the next even depth of cut. Set this switch to Off to rough the entire pocket at the island top depth.

12. Select the **Accept** button.
13. Select the **Go** button on the **Pocket** control panel to start the operation.

Pocketing Multiple Areas



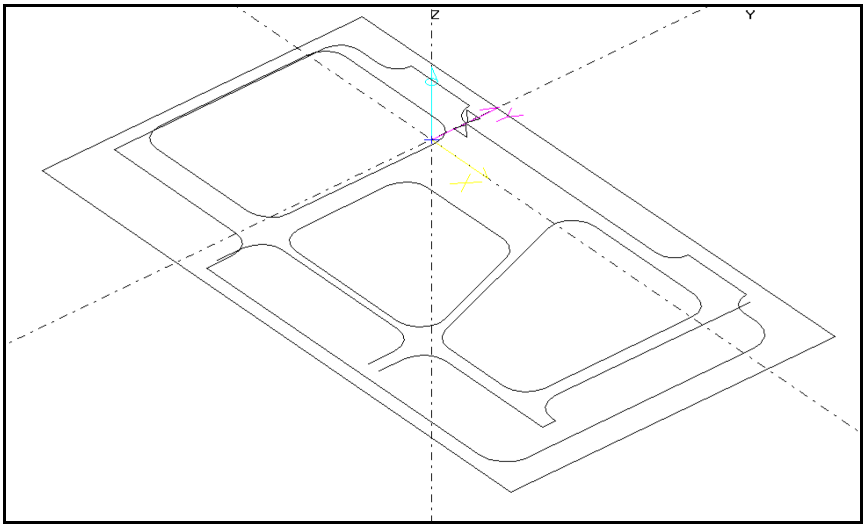
Model File: PMGRPOCK.PM4

Use Group Pocket to create roughing passes for a group of closed finish profiles. You cannot use island curves.

Perform these tasks to pocket multiple areas:

1. Open the model file **PMGRPOCK.PM4**.

Figure 5-7
Open PMGRPOCK.



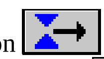
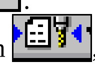
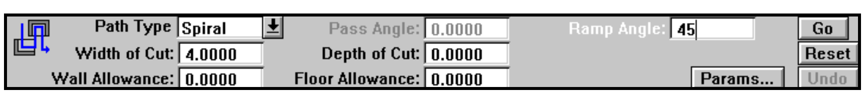
2. Set the insert location:
   - Set the **Before** icon .
   - Select the **With Step** icon ,
   - Select **Step #20**.
   - Set the **Clear** input field.
3. Group all closed pocket profiles.
4. Select **Process—Rough**.
5. Select **Group Pocket** from the toolbox. The **Group Pocket** control panel is displayed.

Figure 5-8
Set the values on the Group Pocket control panel.



6. Set the **Path Type** selector switch to one of the following:
   - **Spiral** creates a spiral roughing operation.
   - **Zig Zag** creates a zigzag roughing operation.
   - **Linear** creates a linear roughing operation.
7. Set these other input fields on the Group Pocket control panel as necessary:
   - Set the **Width of Cut** input field to the stepover distance.
   - Set the **Wall Allowance** input field to the amount of material to leave on the walls of the islands and pockets.
   - Set the **Pass Angle** input field to indicate the 2D cutting angle to use across the pocket.

- Set the **Depth of Cut** input field to a positive value that represents the depth of cut.
- Set the **Floor Allowance** input field to the amount of material to leave at the bottom of the pocket.
- Set the **Ramp Angle** input field to the angle you want to cut on ramp moves.

8. Select the **Params...** button. The **Group Pocket Parameters** dialog box is displayed.

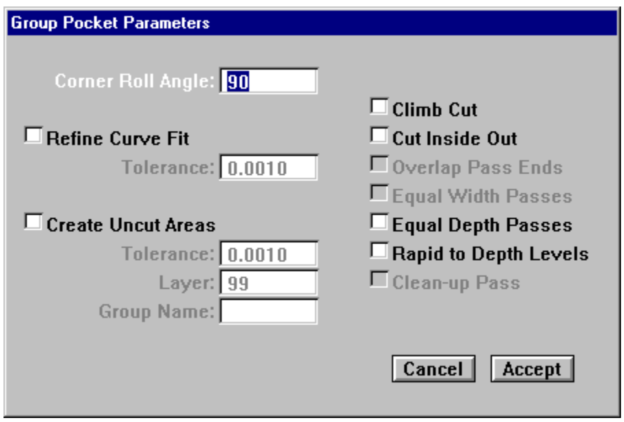


Figure 5-9
Set the values in the Group Pocket Parameters dialog box.

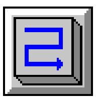
9. Set the following fields on the Group Pocket Parameters dialog box, as necessary:

- Set **Corner Roll Angle** to the determining angle for corner rolling.
- Turn on the **Refine Curve Fit** on/off switch to remove colinear points from the part profile. This is useful when the part profile is a spline. Otherwise, turn it off.
- Turn on the **Create Uncut Areas** on/off switch to create geometry for areas that the assigned roughing tool cannot cut.
- Turn on the **Climb Cut** on/off switch to create climb cuts. Turn off this switch to create conventional cuts.
- Turn on the **Cut Inside Out** on/off switch to begin the roughing operation in the center of the area and progressively step over toward the outside boundary.
- Turn on the **Overlap Pass Ends** on/off switch if you want the cutter to follow the profile up to the previous level after each cut before retracting.
- Turn on the **Equal Width Passes** on/off switch to make the width of all cutting passes the same.
- Turn on the **Equal Depth Passes** on/off switch to make the depth of all cutting passes the same.

- Turn on the **Rapid to Depth Levels** on/off switch to rapid-move the tool to the levels of the previous roughing passes.
- Turn on the **Clean-up Pass** on/off switch to have the tool perform a pass around the perimeter of the pocket.

10. Select the **Accept** button. The **Group Pocket Parameters** dialog box closes, and the values you entered are accepted.
11. Select the **Go** button to start the operation.

Facing a Part



Model File: PMFACE.PM4

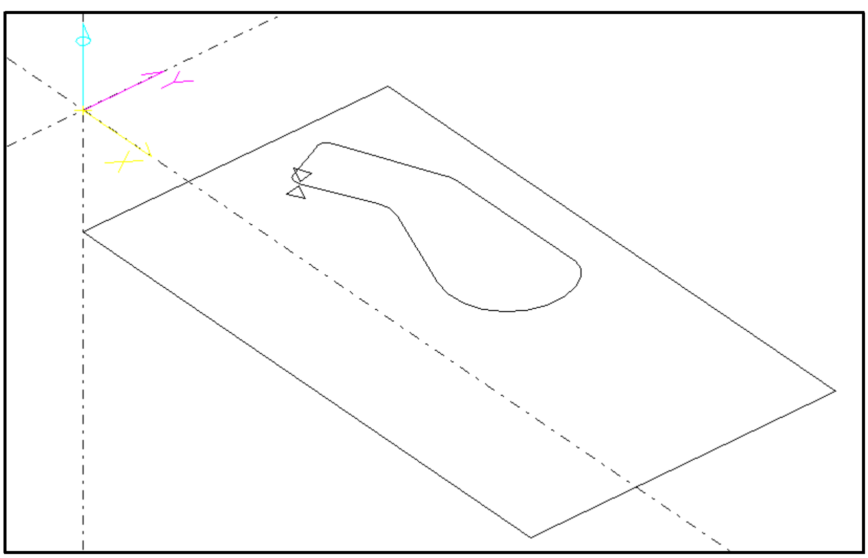
Use Face to create roughing profiles for the removal of material when you want the tool to overlap the outside of a profile boundary.

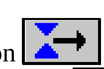
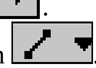
Use the Group toolbox to identify an active group of islands and notches that SmartCAM should avoid. Each island profile must have a Prof_Top value to describe the island top.

Perform these tasks to face a part:

1. Open the model file **PMFACE.PM4**.

Figure 5-10
Open PMFACE.



2. Set the insert location:
   - Set the **Before** icon .
   - Select the **Element** icon .
   - Select the **With Step** icon , and select step **32**.
   - Set the **Clear** input field.

3. Select **Process—Rough**.
4. Select **Face** from the toolbox. The **Face** control panel is displayed.

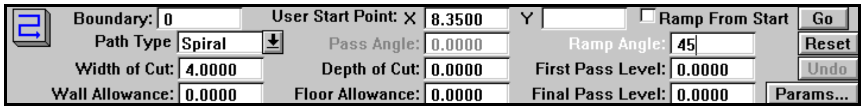


Figure 5-11
Set the values on the Face control panel.

5. Set the **Boundary** input field to any element in the boundary profile. The entire profile is selected.

   Note This must be a closed profile.

6. Set the **User Start Point** input fields to the coordinates for the beginning of the cut.
7. Set the **Path Type** selector switch to one of these options:
   - **Spiral** creates a spiral roughing operation.
   - **Zig Zag** creates a zigzag roughing operation.
   - **Linear** creates a linear roughing operation.
8. Set the remaining fields on the control panel:
   - Set the **Width of Cut** input field to the distance you want the cutter to step over for each pass. SmartCAM inserts a value that is calculated from the tool diameter.
   - Set the **Wall Allowance** input field to the amount of material to be left on any islands or notches for a finishing operation. A negative value is supported.
   - Set the **Pass Angle** input field to the angle at which you want angular roughing passes performed for linear and zigzag cutting.
   - Set the **Depth of Cut** input field to the amount of vertical material you want removed with each pass.
   - Set the **Floor Allowance** input field to the amount of material to leave at the bottom of the pocket.
   - Set the **Ramp Angle** input field to the angle you want to cut on ramp moves.
   - Set the **First Pass Level** input field to a global value that represents the first level of pocketing.
   - Set the **Final Pass Level** input field to the depth of the bottom of the pocketing operation.
   - Turn on the **Ramp From Start** on/off switch to create ramp moves directly from the user start point location to the automatic start point of the toolpath. Turn off this switch to create a vertical plunge at the user start point. You must set a user start point to activate this option.

9. Select the **Params** button. The **Face Parameters** dialog box is displayed.

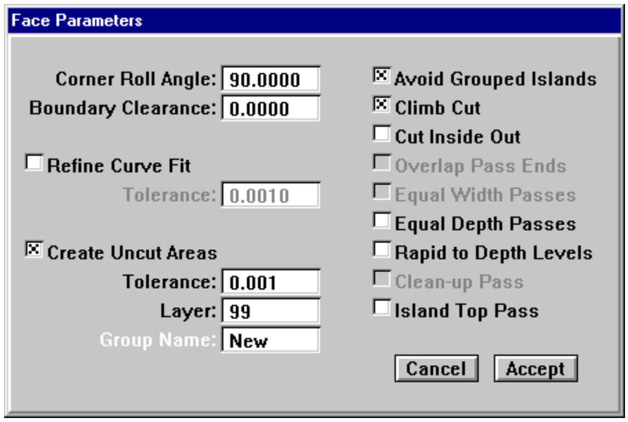


Figure 5-12 *Set the values on the Face Parameters dialog box.*

10. Set the following fields on the Face Parameters dialog box:

- Set **Corner Roll Angle** to the determining angle for corner rolling.
- Set the **Boundary Clearance** input field to the distance the tool is to cut beyond the face boundary. This input field accepts positive or negative numbers.

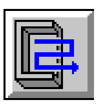
Note If you enter a value of 0, the center of the tool stops at a distance from the boundary profile that is equal to the radius of the tool.

- Turn on the **Refine Curve Fit** on/off switch to remove colinear points from the part profile. This is useful when the part profile is a spline. Otherwise, turn it off.
- Turn on the **Create Uncut Areas** on/off switch to create geometry for areas that the assigned roughing tool cannot cut.
- Turn on the **Avoid Grouped Islands** on/off switch to avoid an active group of islands or notches in the pocket.
- Turn on the **Climb Cut** on/off switch to create climb cuts. Turn off this switch to create conventional cuts.
- Turn on the **Cut Inside Out** on/off switch to begin the roughing operation in the center of the area and progressively step over toward the outside boundary.
- Turn on the **Overlap Pass Ends** on/off switch if you want the cutter to follow the profile up to the previous level after each cut before retracting.
- Turn on the **Equal Width Passes** on/off switch to make the width of all cutting passes the same.
- Turn on the **Equal Depth Passes** on/off switch to make the depth of all cutting passes the same.

- Turn on the **Rapid to Depth Levels** on/off switch to rapid-move the tool to the levels of the previous roughing passes.
- Turn on the **Clean-up Pass** on/off switch to have the tool perform a pass around the perimeter of the pocket.
- Turn on the **Island Top Pass** on/off switch to have the tool create a pass over all island tops at a depth without roughing the entire pocket at that depth. The next full-width pocket pass is performed at the next even depth of cut. Set this switch to Off to rough the entire pocket at the island top depth.

11. Select the **Accept** button. The **Face Parameters** dialog box closes, and the values you entered are accepted.
12. Select the **Go** button to start the operation.

Cutting an Open Profile



Model File: PMOPNPRF.PM4

Use Open Profile to cut areas that are partially enclosed but still have an open side. No islands can be used with Open Profile. A material boundary curve is drawn to show the edge of the material block to create a closed profile to define the volume of material to remove.

Perform these tasks to cut an open profile:

1. Open the model file **`PMOPNPRF.PM4`**.

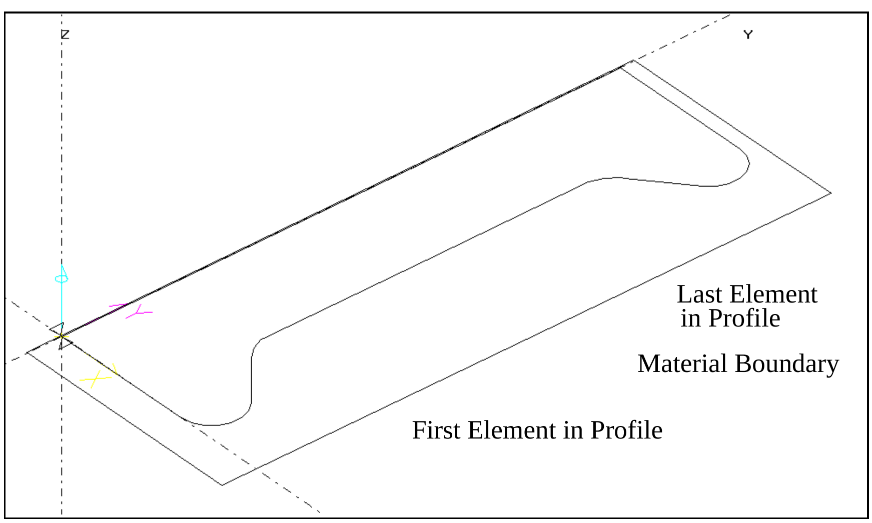
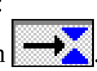
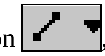


Figure 5-13
Open PMOPNPRF.

2. Set the insert location:
   - Set the **After** icon .

- Select the **Element** icon , and select the first element in the open profile.
- Select the **With Step** icon  and select the **step 12**.
- Set the **Clear** input field.

3. Select **Process—Rough**.
4. Select **Open Profile** from the toolbox. The **Open Profile** control panel is displayed.

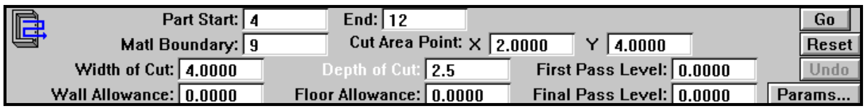


Figure 5-14 *Complete the Open Profile control panel.*

5. Set the following fields on the control panel:
   - Enter values for **Part Start** and **End**.
   - Set **Matl Boundary** to a material boundary.
   - Select **Cut Area Point** and select a point anywhere inside the profile.
   - Set the **Width of Cut** input field to the stepover distance.
   - Set the **Wall Allowance** input field to the amount of material to leave on the walls of the islands and pockets.
   - Set the **Depth of Cut** input field to a positive value that represents the depth of cut.
   - Set the **Floor Allowance** input field to the amount of material to leave at the bottom of the pocket.
   - Set the **First Pass Level** input field to a global value that represents the first level of pocketing.
   - Set the **Final Pass Level** input field to the depth of the bottom of the pocketing operation.
6. Select the **Params...** button. The **Open Profile Parameters** dialog box is displayed.

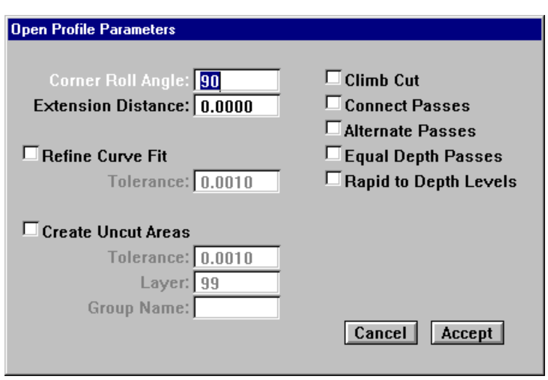


Figure 5-15 *Set the fields in the Open Profile Parameters dialog box.*

- Set **Corner Roll Angle** to the determining angle for corner rolling.
- Turn on the **Refine Curve Fit** on/off switch to remove colinear points from the part profile. This is useful when the part profile is a spline. Otherwise, turn it off.
- Turn on the **Create Uncut Areas** on/off switch to create geometry for areas that the assigned roughing tool cannot cut.
- Turn on the **Climb Cut** on/off switch to create climb cuts. Turn off this switch to create conventional cuts.
- Turn on the **Connect Passes** on/off switch to connect the end of each pass of the tool to the next pass. Turn off this switch to leave the tool passes disconnected, which causes the tool to retract and rapid to the next pass.
- Turn on the **Alternate Passes** on/off switch to alternate the direction of each pass, which places the start point of each pass near the end point of the pass that preceeded it.
- Turn on the **Equal Depth Passes** on/off switch to make the depth of all cutting passes the same.
- Turn on the **Rapid to Depth Levels** on/off switch to rapid-move the tool to the levels of the previous roughing passes.

7. Select the **Accept** button. The **Open Profile Parameters** dialog box closes, and the values you entered are accepted.
8. Select the **Go** button. The operation is performed.

Profiling

Model File: PMROUGH2.PM4

The primary use for this feature is to create toolpath from an existing profile. You identify a defining profile with the Profile Start and End fields, and you determine where the toolpath is created with the Offset Side field (relative to the direction of the profile). The amount of toolpath created with the original profile is determined by the value you enter in the Wall Stock field, which represents the distance between the first pass of the toolpath and the original profile.

Note Profiling toolpath starts a set distance—determined by the user-entered value in the Wall Stock field—from the generator profile and works *toward* the original (defining) profile.

The number of lateral passes SmartCAM creates with Profiling is determined by subtracting the Wall Allowance from the Wall Stock and dividing this adjusted Wall Stock value by the Width of Cut value. If, in the division, there is a decimal remainder, the number of passes is rounded up to the next whole number and the final, closest pass removes the smallest amount of material. For example, a Wall Stock of 1, a Width of Cut of .25, and a Wall Allowance of .1 will generate the following lateral behavior:

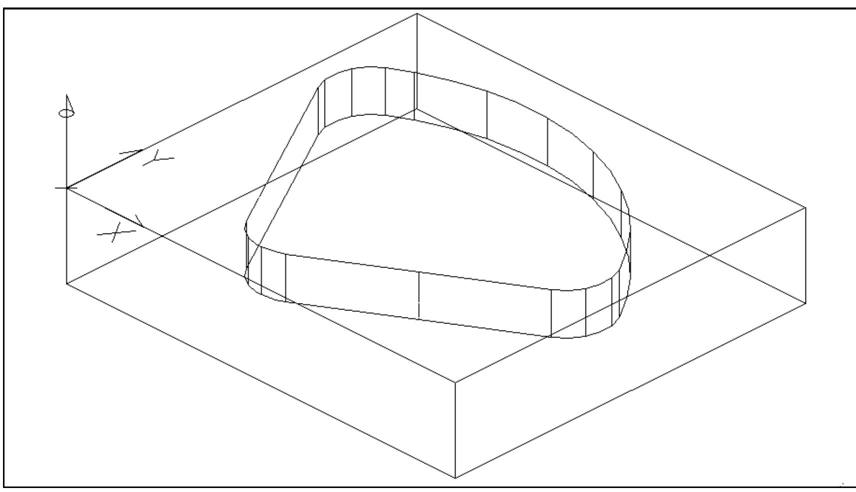
- three equal passes that step over exactly .25 each
- one pass (closest to the original profile) that steps over .15
- a wall allowance of .1

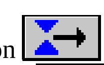
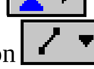
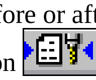
Additional toolpath attributes can be set using the Profile Parameters dialog box.

Use Profile to support multiple depth and width passes, width first and depth first progression, lead in/out moves, offset direction, and center-line cutting compensation.

1. Open the model file **PMROUGH2.PM4**

Figure 5-16
Open PMROUGH2.



2. Set the insert location:
   - Select the **Before** icon  or **After** icon .
   - Select the **Element** icon  or **Step** icon , and then select the element or step to be before or after.
   - Select the **With Step** icon , and select a roughing step from the list view, such as Step 2.
3. Select **Process—Rough**.
4. Select **Profile** from the **Rough** toolbox. The **Profile** control panel is displayed.

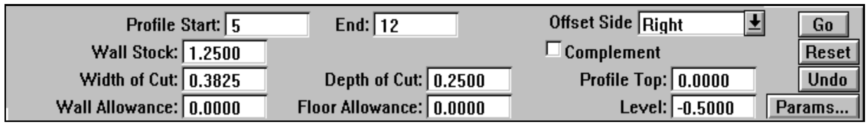


Figure 5-17 *Set the parameters on the Profile control panel.*

5. Set the input fields on the panel as necessary:

- Set the **Profile Start/End** parameters to define the part profile. The order in which you select elements determines the default toolpath direction for an open profile. The Complement on/off switch is available for a closed profile.
- Set the **Offset Side** selector switch to **Right** or **Left**. This controls where the toolpath is placed relative to the selected part profile.

Note The Offset Side is relative to profile direction, so verify that it is appropriate for the machining condition.

- Set the **Wall Stock** input field to the lateral thickness of the material relative to the part profile. The value of Wall Stock must be greater than or equal to that of Wall Allow, see Figure 5-18.
- Set the **Width of Cut** input field to the distance the cutter should move laterally between adjacent passes (step over distance).
- Set the **Wall Allowance** input field to the amount of material (skin) to leave in the part profile. Negative allowances are supported.

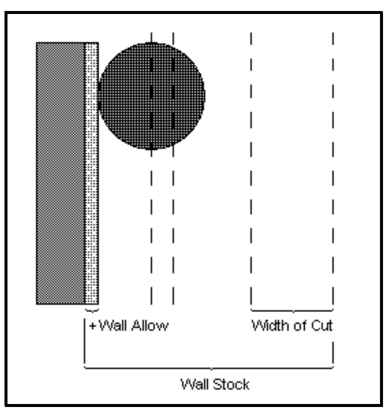


Figure 5-18 *Notice how the Wall Stock, Width of Cut, and Wall Allowowance values relate.*

- Set the **Depth of Cut** input field to the depth of cut for each pass. It must be greater than zero unless the vertical material thickness is zero, in which case the depth of cut can be zero.
- Set the **Floor Allowance** input field to the amount of material to leave on the floor of the part. When this value is zero, the final cut elevation is the same as the Level value. Negative allowances are supported.
- Turn on the **Complement** on/off switch to create profiling toolpath for the complementary geometry of a profile. Otherwise, turn it off to use the default selection order.

- Set the **Profile Top** input field to the top of the geometry. This value must be greater than or equal to that of Level.
- Set the **Level** input field to the depth for the deepest toolpath cut. The default value is the level of the Profile Start element.

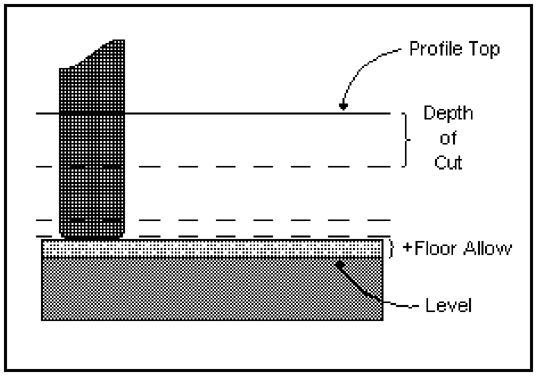


Figure 5-19 *Notice how the Profile Top, Depth of Cut, Floor Allowance, and Level relate.*

6. Select the **Params...** button. This opens a dialog for editing parameters that control various behaviors like lead in/out, plunge and retract movements.

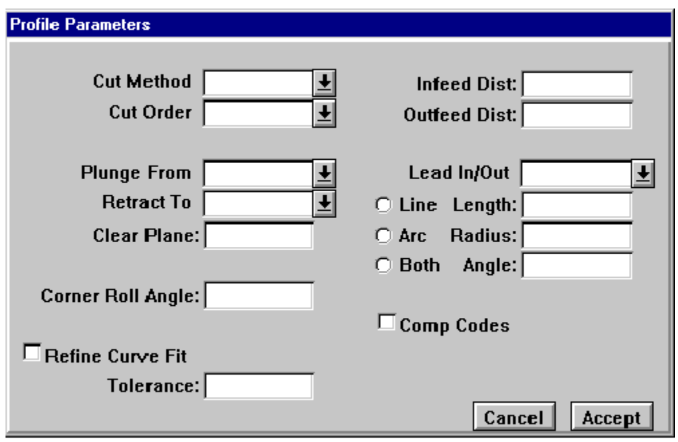


Figure 5-20 *Set the values on the Profile Parameters dialog box.*

7. Set the **Cut Method** selector switch to one of these methods:
   - **One Way** moves all elements in the toolpath the same direction.
   - **Alternate** moves every other element in the offset toolpath the direction opposite that of the adjacent offset toolpath element.
   - **Connect** alternates the direction of the elements in the offset toolpath and connects the end of one toolpath to the start of the next.

   Note Vertical connections are honored when Cut Order is set to Depth First.

8. Set the **Cut Order** selector switch to the method of toolpath progression.

9. Set the **Plunge From** selector switch to the place the plunging moves should come from:

   - **Profile Top** plunges from the Profile Top value on the Profiling control panel.
   - **Previous** plunges all first cut levels from the Profile Top and plunges succeeding cuts from the elevation of the previous cut level.
   - **None** plunges to the level of each element and rapids between cut levels. Position the tool out of cut when you use this option.

10. Set the other fields on the dialog box as necessary:

    - Set the **Retract To** selector switch to the place to retract the tool. If you select Clear Plan, specify in the Clear Plane input field the clearance value, which must have an absolute value greater than or equal to the Profile Top.
    - Set the **Corner Roll Angle** input field to the angle at which any corner having an interior angle less than it is traversed using a blending arc. This is useful for rolling around non-tangent intersections.
    - Turn on the **Comp Codes** on/off switch to insert cutter-compensation codes into the toolpath. This is disabled when the Cut Method selection is Connect.
    - Turn on the **Refine Curve Fit** on/off switch to use the Tolerance input field.
    - Set the **Tolerance** input field to the chordal tolerance to use to eliminate colinear points from the part profile. This is useful for processing splines.
    - Set the **Infeed Dist** input field to the distance to extend the starting end of the toolpath along its tangent, independent of any lead-in moves.
    - Set the **Outfeed Dist** input field to the distance to extend the trailing end of the toolpath along its tangent, independent of any lead-out moves.
    - Set the **Lead In/Out** selector swith to create lead-in and lead-out moves for engaging and disengaging the part profiling toolpath.

- Turn on one of these option buttons to represent the lead in/out movement: **Line**, **Arc**, **Both**.

  Turn on **Line** option switch, and set the **Length** input field to the length of any lead in/out line elements. The length must be greater than zero.

  Turn on the **Arc** option switch, and set the **Radius** input field to the radius of any lead in/out arc elements. The radius must be greater than zero.

  Turn on the **Both** option switch, and set the **Angle** input field to the approach and recession directions of the tool as it engages and disengages the profiling toolpath. When Line is selected, this angle is relative to the start/end tangent of the profiling toolpath. When Arc is selected, this angle determines the arc span. When Both is selected, this angle determines the arc span, and a line is prepended/appended to the start/end of any lead-in/lead-out element at zero degrees relative to the respective arc end tangent.

11. Select the **Accept** button.
12. Select the **Go** button on the **Profile** control panel.

Figure 5-21
*This is the result of profile roughing with the Offset Side set to **Right**.*

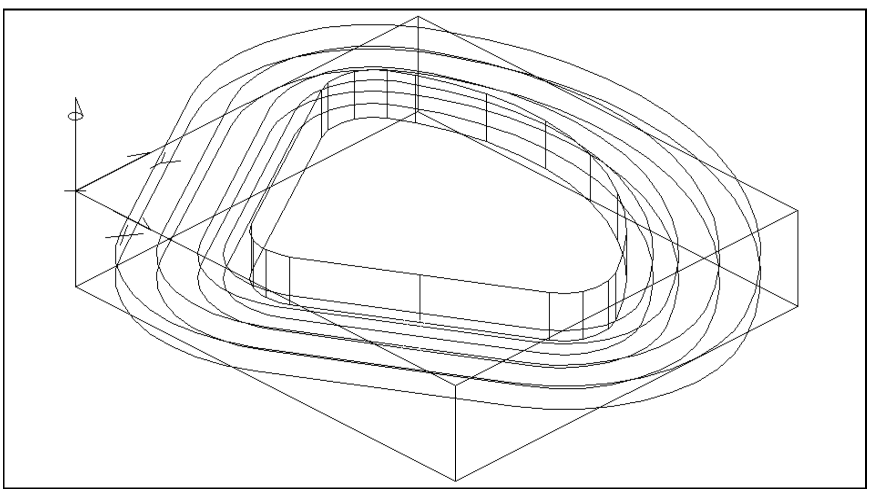
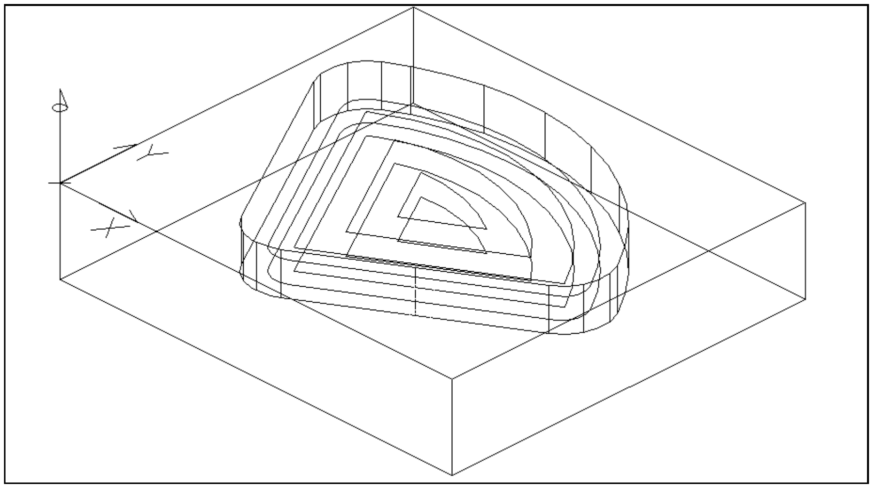


Figure 5-22
This is the result of profile roughing with the Offset Side set to ***Left****.*



Points to Remember

- ❑ Multiple islands can exist inside closed profiles that you use for pocketing.
- ❑ Use the Facing feature to create roughing profiles for removing material when you want the tool to overlap the outside of a profile boundary.
- ❑ You cannot have islands for Open Profile machining.

Verifying Roughing Toolpath

Objectives

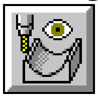
This lesson shows you how to perform these tasks:

- Create a stock box.
- Set the Show Cut dialog box.
- Start and stop Show Cut.

Overview

Use Show Cut to view the volumetric verification of the model. Use this tool to check the accuracy of the model for possible gouging, collision, or clearance problems during machining. You can verify all or part of the model and save the results in a file using Show Cut.

Using Show Cut

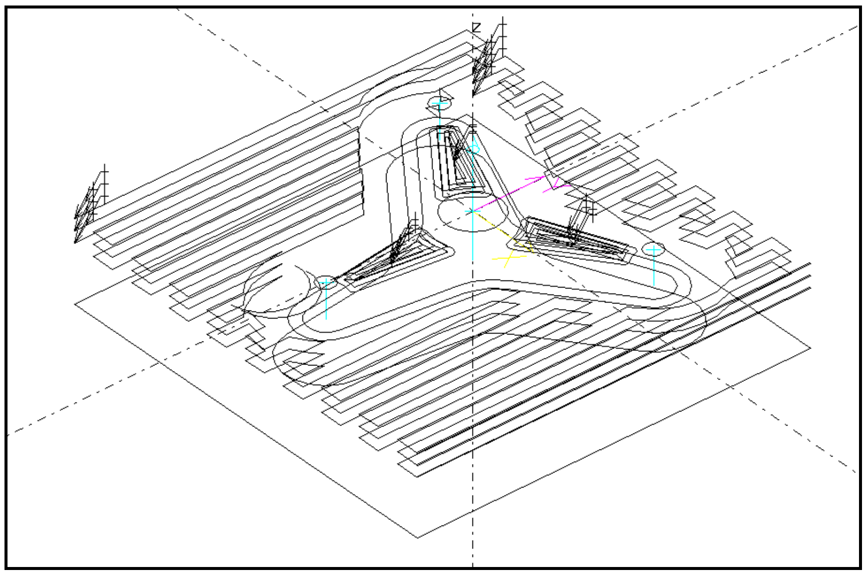


Model File: PMSHWCUT.PM4

Perform these tasks to use Show Cut.

1. Open the model file **PMSHWCUT.PM4**.

Figure 5-23
Open PMSHWCUT.

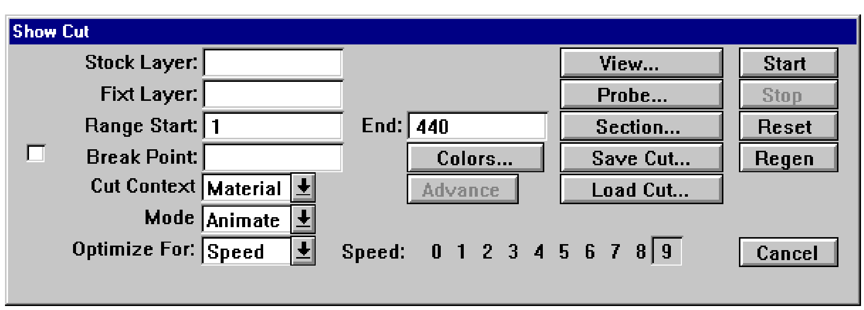


2. Select **View—Get View—ISO**, or dynamically rotate the part to any desired view.

3. Select **View—Full** to zoom the part to fill the screen, or use **View—Window** to zoom in on a particular feature of the part.

4. Select **View—Show Cut**. The **Show Cut** dialog box is displayed.

Figure 5-24
Set the values on the Show Cut dialog box.



5. Set the **Stock Layer** input field to the layer containing the geometry for the material stock. If the material is on multiple layers, enter the layer numbers, separated by commas; for example, enter **3,5** to indicate layers 3 and 5. You can also use a dash to indicate a range of layers; for example, enter **3-5** to indicate layers 3 through 5.

6. Set the **Fixt Layer** input field to the layer containing the geometry for the machine fixtures. If the material is on multiple layers, enter the layer numbers, separated by commas.
7. Set the **Range Start** input field to the first element in the range of elements to verify.
8. Set the **Speed** input field to a value between 0 and 9. The larger the number, the faster the graphics are displayed.
9. Set the **Cut Context** input field to **Step**. Each cut is displayed in the color corresponding to the step.
10. Select the **Start** button to start the simulation. Press the ESC key to stop the simulation.

Points to Remember

- ❑ Use Show Cut to verify roughing toolpath.
- ❑ You must create a stock box that matches the material out of which you will cut the part.

Self-Test

Directions

Test your understanding of the concepts and procedures in this section by answering the following questions. The answers for each self-test are in *Appendix A* of this manual.

______________ **1. More that one island can exist inside closed profiles that you use for pocketing.**

a) true
b) false

______________ **2. Which feature do you use to create roughing profiles for removing material when you want the tool to overlap the outside of a profile boundary?**

a) Pocketing
b) Facing
c) Contouring
d) Group Pocketing

______________ **3. How many islands can you have for Open Profile machining?**

a) 0
b) 1
c) 2
d) as many as you want

______________ **4. Which of these features do you use to verify roughing toolpath?**

a) Verify Toolpath
b) Roughing
c) Copy
d) Show Cut

______________ **5. How do you determine the size of the stock box to create?**

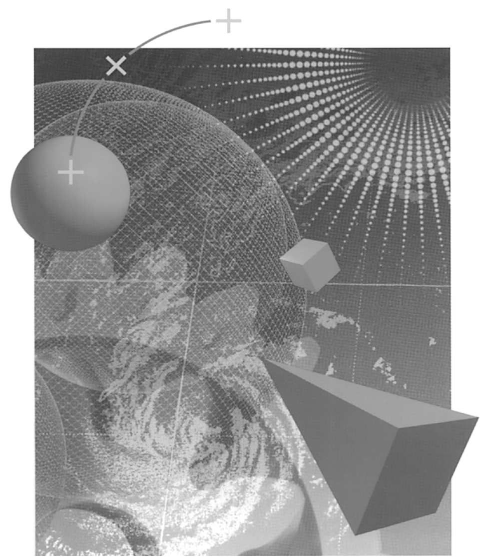
a) The stock box should match the finished part.
b) The stock box should match the material you use.
c) The stock box size is inconsequential.
d) The stock box should exceed your finished part size by .001 inches.

6

Generating Code

Overview

SmartCAM generates NC code for various CNC machine and controller operations using the Code feature.



Lessons for This Unit

- Generating Code

Generating Code

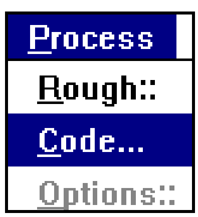
Objectives

This lesson shows you how to code a file.

Overview

Use the Code feature to generate NC code for the part and to estimate the total cycle time. SmartCAM generates code for all unmasked step-property elements in the database.

Figure 6-1
Use the Process menu to generate code.



Learning File Types for Code Generation

Code is generated from the process model as an ASCII text file (`.txt`) that can be viewed using any text editor. Edit Plus is the text editor that is shipped with SmartCAM. SmartCAM uses machine definition (`.smf`) and template (`.tmp`) files to generate code. You can find more information about .smf files and .tmp files in the *Code Generation Guide*, which is distributed online.

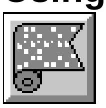
Machine Definition Files

Machine definition, or `.smf`, files contain information for the machine tool, including information on your machine tool configuration, the G and M codes it uses, and numeric formats. You must specify an **`.smf`** file for the tool.

Template Files

Template (**`.tmp`**) files contain variables and if /then statements that are used to format the code so that the controller can read it. You must specify a **`.tmp`** file for each `.smf` file that you define.

Using Code



Model File: PMCODE.PM4

Use Code to generate the CNC code for your part and estimate the total cycle time. SmartCAM generates code for all unmasked step property elements in the database. Code is generated as a text file that you can view and edit using a text editor.

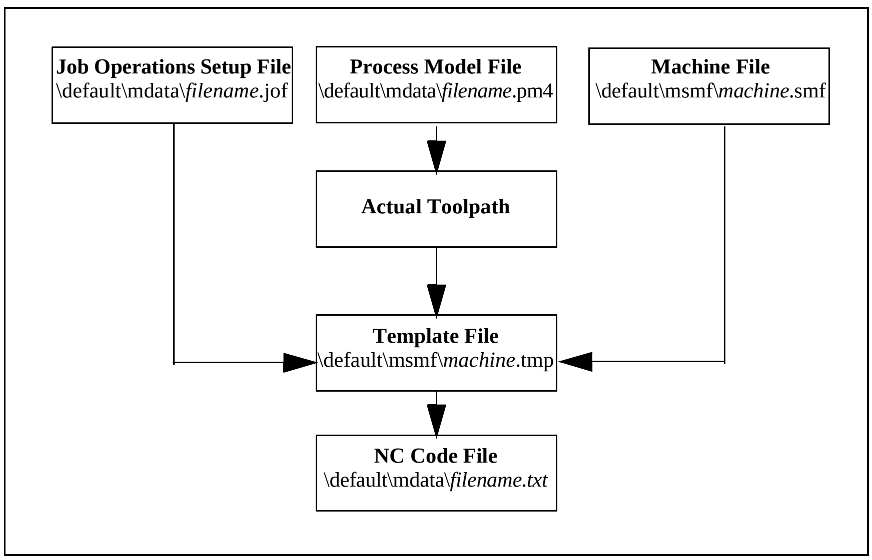


Figure 6-2
SmartCAM uses these files to generate code.

Perform these tasks to code a file:

1. Open the model file **PMCODE.PM4**.

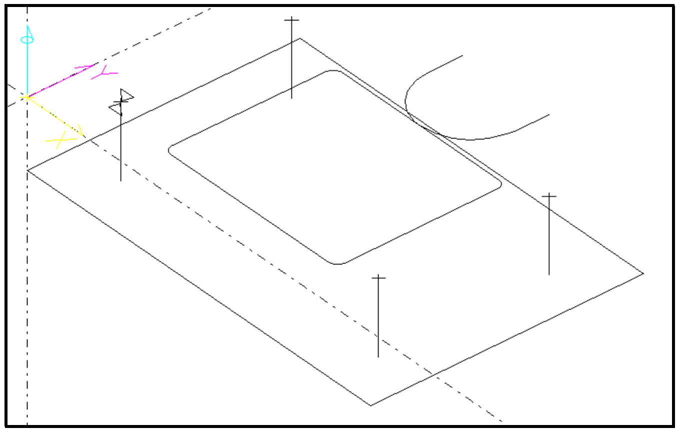


Figure 6-3
Open PMCODE.

2. Select **Process—Code**. The **Code** dialog box is displayed.

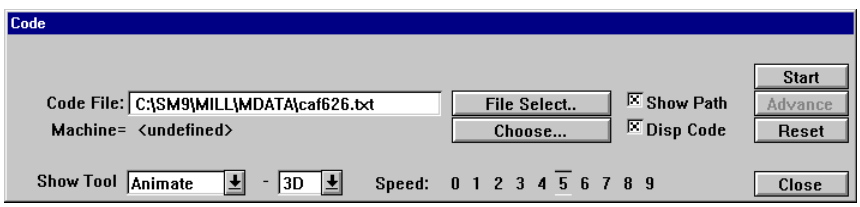


Figure 6-4
Set the values on the Code dialog box.

3. Set **Code File** to the name of the file in which you want SmartCAM to place the code, or choose the **File Select** button to access the **Save As** dialog box and locate the file.

4. Select the **Choose** button. The **Job Information** dialog box is displayed and is opened to the Machine page.

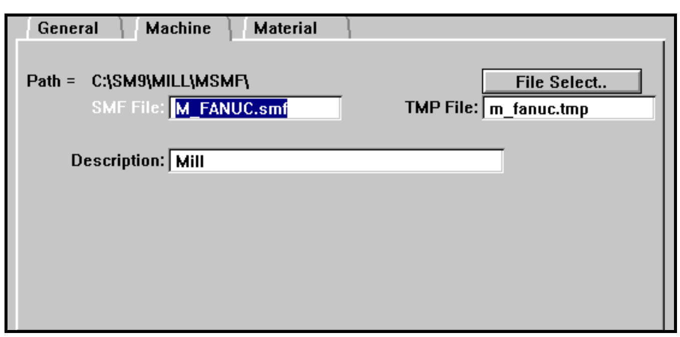


Figure 6-5
Set the .smf and the .tmp files.

5. Set the **Smf File** field to the machine (`.smf`) filename by selecting the input field and pressing the **File Select** button. The **Open** dialog box appears.

6. Perform the following actions on the **Open** dialog box:

   - Select a file.

- Select the **OK** button.

7. While the cursor is still in the SMF File input field, set the **Tmp File** input field to the template (`.tmp`) filename by performing a right mouse click. The cursor automatically advances to the TMP File input field and enters the name of the appropriate template file name in the field.

   If you need to use a different template file, press the File Select button when the cursor is in the TMP file field and repeat the selection method in step 6 .

8. Select the **Accept** button on the **Job Information** dialog box.

9. Set these values on the **Code** dialog box:

   - Turn on the **Show Path** on/off switch to see the toolpath simulation during code generation (optional).

     Note Code generation speed increases when this switch is turned off.

   - Turn on the **Disp Code** on/off switch to display each block of code as it is generated (optional).

     Note Code generation speed increases when this switch is turned off.

   - Set **Show Tool** to display variations of toolpath motion when **Show Path** is on, as follows:

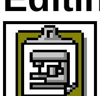
     - **Nibble** draws an image of the tool as it follows the toolpath on the model.
     - **Draw End** displays the outline of the tool at the start and end of each element.
     - **Animate** shows the tool's profile, location, and motion, leaving a path.
     - **Filled** draws the CTG shape of the tool and fills it with the tool's color.
     - **3D** displays the operation with three-dimensional graphics.
     - **Flat** displays the operation with two-dimensional graphics.

   - Set **Speed** to a value between 0 and 9. The larger the number, the faster the graphics are displayed, and the faster the code is processed.

10. Select the **Start** button to begin code processing.

    Note Select the **Start** button repeatedly to view the processing sequence one block at a time when **Speed** is set to **0**.

Editing an .smf File Using Machine Define



Perform these tasks to edit an `.smf` file using Machine Define:

1. Select the **Machine Define** icon in the SmartCAM Program Group. The **Machine Define** dialog box is displayed.

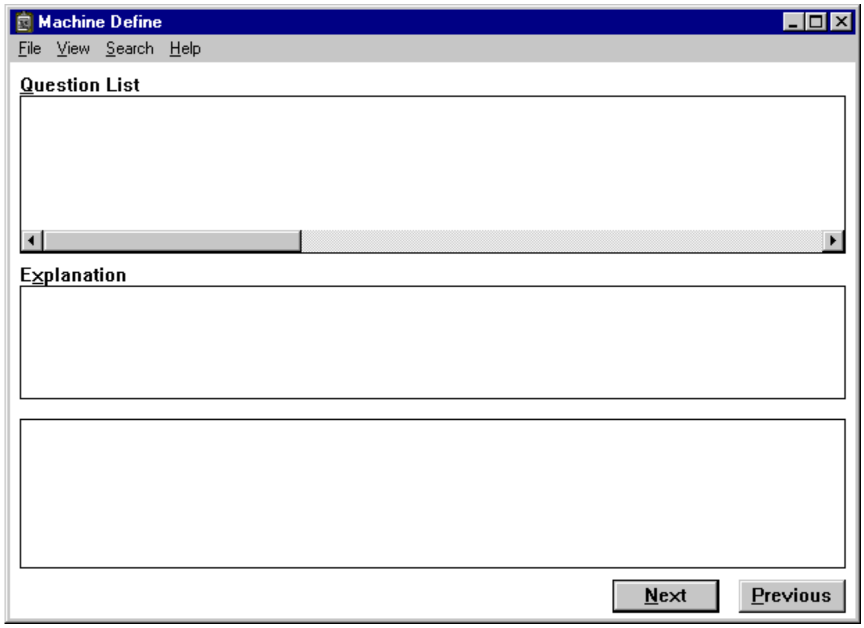


Figure 6-6
Set the values on the Machine Define dialog box.

2. Select **File—Open SMF**. The **Open SMF** dialog box is displayed.
3. Select the `M_DECKEL.SMF` file to open.

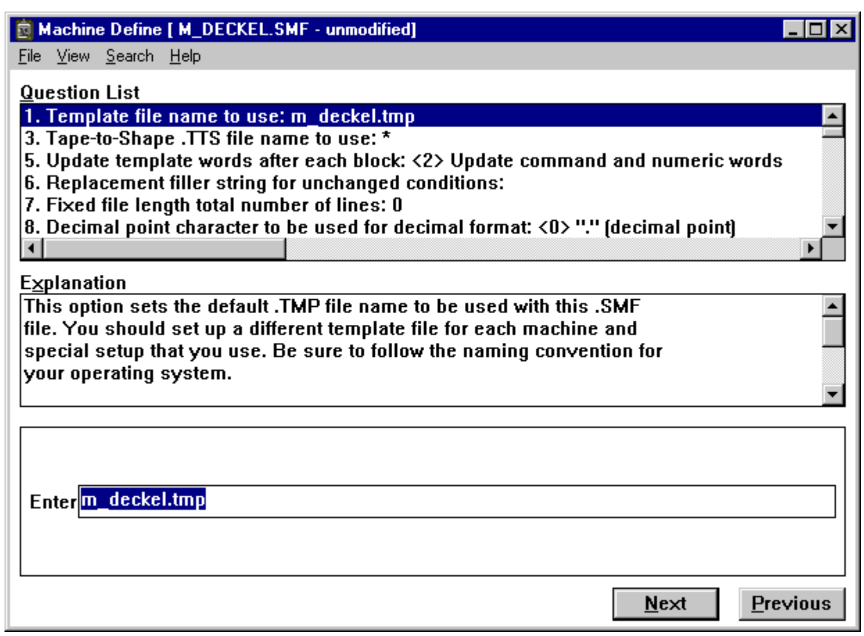


Figure 6-7
Select the question to change and enter the changes.

4. Select the question to change and enter the changes. You can use the **Search** menu to search for the question you want to change.

   Note Typically, you would select **File—Save** to save the file. However, do not save your changes now.

5. Select **File—Exit** to exit Machine Define.

   Note If you use Job Operations Planner data in your `.tmp` file, question 471 must be present and set accordingly in your `.smf` file. If this question is not present in your file (verify using Machine Define), choose the **File—Save As SMF V5** option to save your file as an updated `.smf` file. Saving as an updated `.smf` file generates question 471, which is available each time that you open `.smf` files in Machine Define.

Points to Remember

- SmartCAM generates code for all unmasked step-property elements in the database.
- Code is generated from the process model as a text file that can be viewed using any text editor. Edit Plus is the text editor that is shipped with SmartCAM.
- Machine Definition, or `.smf`, files contain information on the machine tool.
- Template (`.tmp`) files contain variables and if /then statements that are used to format the code so that the controller can read it.

Self-Test

Directions

Test your understanding of the concepts and procedures in this section by answering the following questions. The answers for each self-test are in *Appendix A* of this manual.

_______________ **1. .tmp files contain this information:**

a) geometry
b) code formatting information
c) tool information
d) machine definition questions and answers

_______________ **2. .smf files contain this information:**

a) machine tool
b) operations
c) part geometry
d) group

_______________ **3. Code is generated from the process model in this form:**

a) .doc
b) .txt
c) .tmp
d) .smf

______________ **4. SmartCAM generates code for all step-property elements in the database.**

a) true
b) false

______________ **5. SmartCAM ships this text editor:**

a) Production Milling
b) Material Librarian
c) Machine Define
d) Edit Plus

Exploring SmartCAM Advanced Milling

Exploring SmartCAM Advanced Milling

Welcome

Advanced Milling is the ideal solution for your fundamental 3-D production needs. It supports 2- through 3-axis machining, as well as 4- and 5-axis positioning. As the second tier in the SmartCAM family of milling applications, it offers the functionality you need today, plus an easy, long-term growth path for your future needs. Like all SmartCAM applications, it does more than reduce your programming time. It helps you improve your machining processes and move your products to market faster.

Units in This Exploration

- Using Profile Curves to Create Meshes
- Creating Meshes
- Editing Meshes
- Creating and Editing Elements from Solids
- Working with Surfaces
- Planar Cuts and Cavity Roughing

7

Using Profile Curves to Create Meshes

Objectives

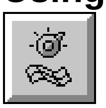
This unit shows you how to perform these tasks:

- Identify the types of profile curves you can use for building meshes.
- Check profile curves.

Overview

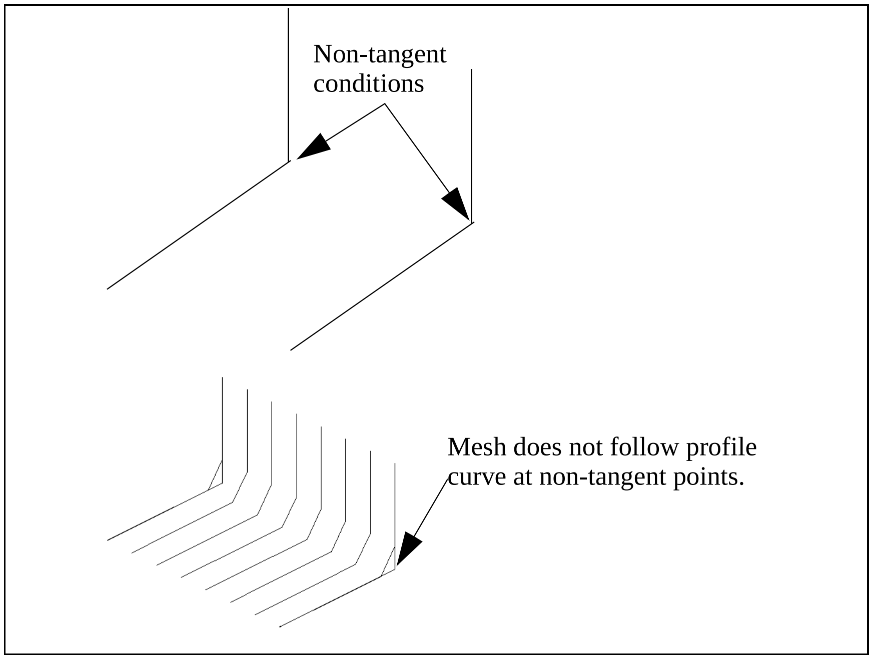
Profile curves can be a mixture of lines, splines, polylines, and arcs that have the same direction and are sequential. They are used to create mesh geometry. When you select the first and last element of a profile curve, all of the elements between them are selected as parts of the profile curve.

Using Tangent Profiles



The profile curves you select to create meshes should be tangent. Non-tangent conditions cause a warning message to be displayed. You can still build a mesh with non-tangent profiles; however, the mesh may not match the profile geometry.

Figure 7-1 *Notice the results of non-tangent profile curves.*



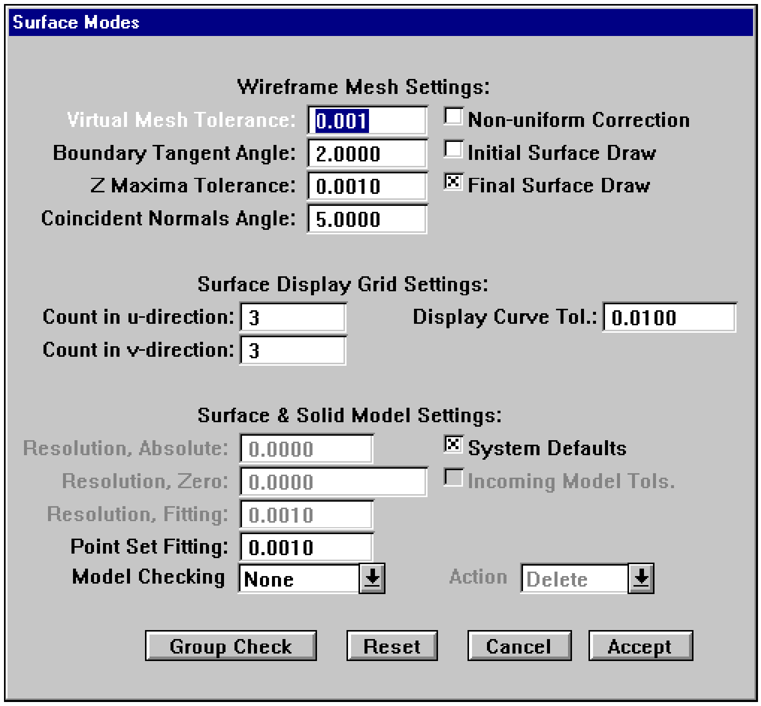
There is a tolerance value for tangency conditions. The boundary tangent angle defines the tangent condition, and it can be adjusted as needed. Angles or non-tangent conditions in a profile curve smaller than the boundary tangent angle are considered tangent. An angle or non-tangent condition larger than the boundary tangent angle causes a warning message to be displayed.

If you use polylines as profile curves, the boundary tangent angle is not used. Nearly all polylines contain angles that exceed the tangency angle.

To change the boundary tangent angle, perform these tasks:

1. Select **Utility—Surface Modes**. The **Surface Modes** dialog box is displayed.

Figure 7-2
Open the Surface Modes dialog box.

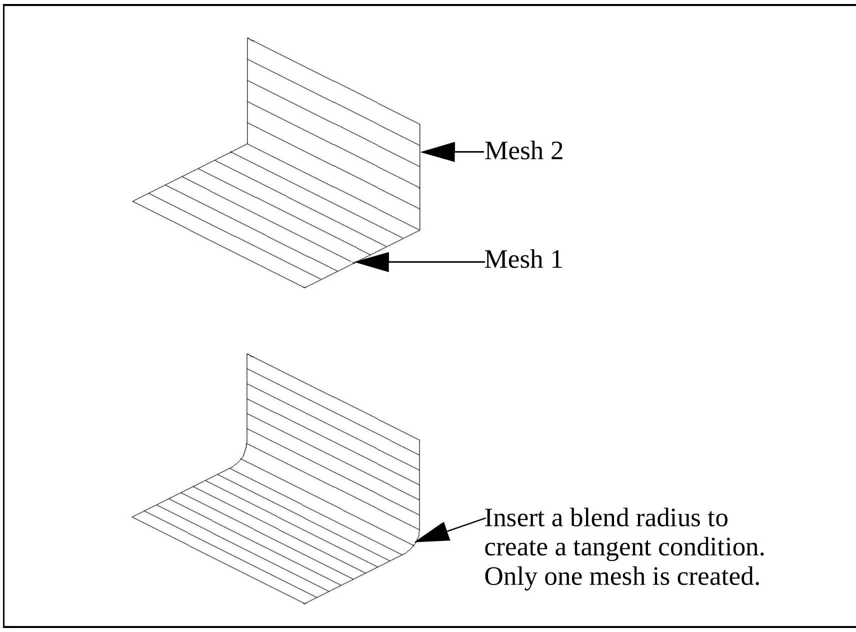


2. Set the **Boundary Tangent Angle** input field to the acceptable out-of-tangency condition that can exist on surface-defining profiles. The recommended setting is between 0 and 3 degrees.

Using Non-Tangent Profiles

You can edit profiles that are not tangent to become tangent. You can insert a simple blend radius at the non-tangent locations. To produce a sharp corner, create two or more meshes.

Figure 7-3
Notice the results of editing non-tangent profile curves.

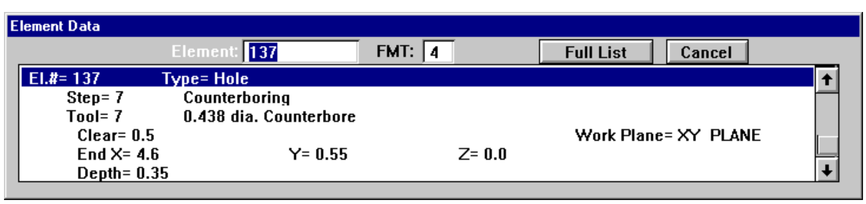


Checking the Type of Elements in Profile Curves

You can check the type of curve used in a profile using Element Data. You can use most types of curves for building meshes. Primitive curves such as lines and arcs work well. If you use a polyline as a profile curve, the mesh may not follow the profile at the sharp corners.

1. Select **Utility—Element Data**. The **Element Data** dialog box is displayed.

Figure 7-4
View element data with the Element Data dialog box.



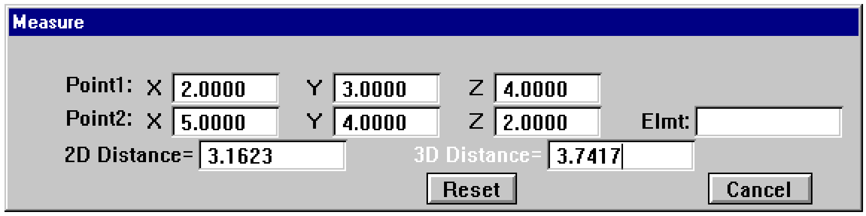
2. Select the element you want data for.

Checking for Gaps in Profile Curves

Make sure there are no gaps in a profile curve. If gaps are present, the mesh will not be created. To remove a gap, build two separate meshes or trim/extend the curves.

1. Select **Utility—Measure**. The **Measure** dialog box is displayed.

Figure 7-5
Set Point1 and Point2 on the Measure dialog box.



2. Select **Point1** and select the point from which you are measuring.
3. Select **Point2** and select the point to which you are measuring. The **3D Distance** input field displays the computed three-dimensional distance between the two points. It takes levels into account.

Grouping Profile Curves

Some meshes require that the profile curves be grouped. Use the Group Arrow icon or Group tool palette to group profile curves.

Points to Remember

- ❑ When you select the first and last element of the profile curve, all of the elements between are selected as parts of the profile curve.
- ❑ The profile curves you select to create meshes should be tangent.
- ❑ If polylines are used as profile curves, the boundary tangent angle is not used.
- ❑ Profiles that are not tangent can be edited to become tangent.
- ❑ No gaps should be present in a profile curve.

8

Creating Meshes

Objectives

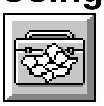
This unit shows you how to perform these tasks:

- Create a spun mesh from a profile curve.
- Create a translated mesh from two profile curves.
- Create a drafted mesh from two profile curves.
- Create a ruled mesh from two profile curves.
- Create a lofted mesh from multiple profile curves.
- Create a form patch mesh from four connected boundary profile curves.
- Create a coons patch mesh from two profile curves and cross-section profiles.
- Create planar cuts across an existing mesh.

Overview

You can create surface mesh profiles for primitive, swept, and sculpted mesh profiles. When you assign a mesh to a step, the mesh profiles represent toolpath to create the surface. When you assign a mesh to a layer, the mesh profiles represent a non-offset representation of the surface. Identifying the defining profiles for the surface enables you to select the appropriate modeling tool to use.

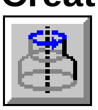
Using the Mesh Toolbox



Use the Mesh toolbox to access modeling tools for creating surface mesh profiles. Before generating a mesh, examine the profiles, making sure they meet the requirements for the type of surface.

Figure 8-1
Notice the tools in the Mesh toolbox.

Creating a Spun Mesh



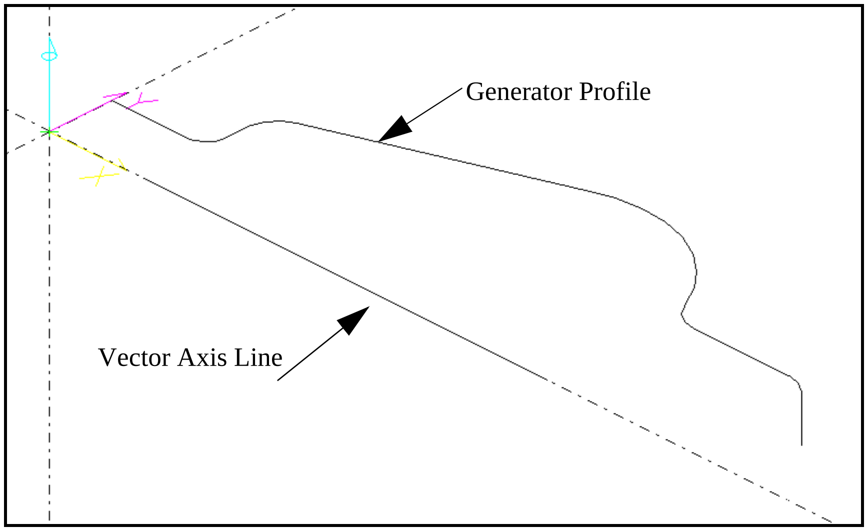
Model File: A3DSPUN.PM4

Use Spun Mesh to create mesh profiles by revolving a two-dimensional profile about an axis. The two-dimensional profile can consist of lines, arcs, splines, ellipses, or polylines. You must assign all profile elements to the same work plane. Use Spun Mesh to create a toolpath for symmetrical concave or convex surfaces. Use the Insert property bar to specify the properties and sequence for the new profiles.

Before generating a spun mesh, make sure that all the profile's elements connect and meet the Boundary Tangent Angle setting in the Surface Modes control panel.

1. Open the model file **A3DSPUN.PM4**.

Figure 8-2
Open A3DSPUN.



2. Select **Create—Mesh—Spun Mesh**. The **Spun Mesh** control panel is displayed.

Figure 8-3
Set the values on the Spun Mesh control panel.

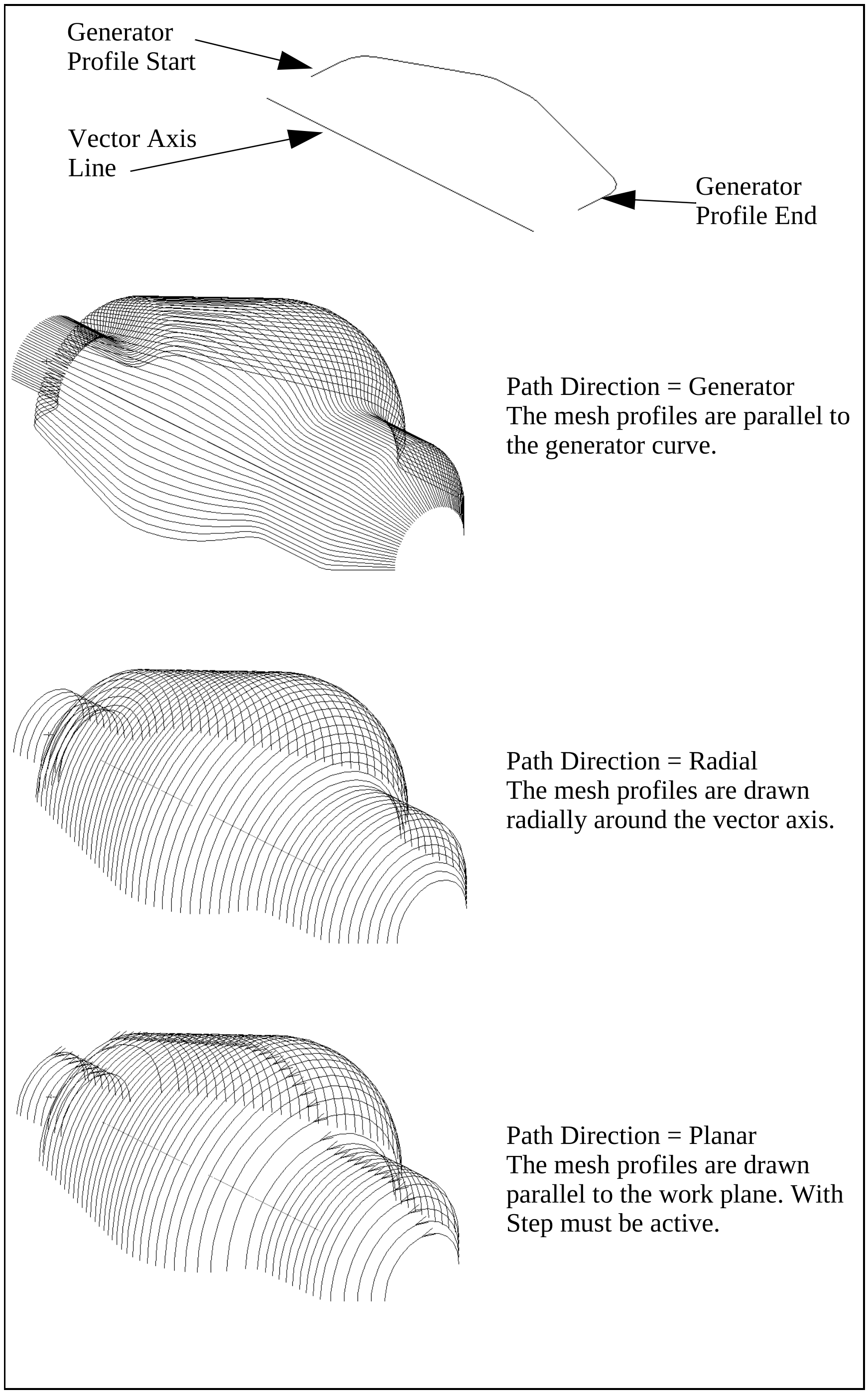
3. Identify the generator profile by setting the **Generator Profile Start** and **End** fields to the first and last elements of the generator profile. Set these fields by manually typing, selecting the element in the graphics view, or selecting from the list view.

4. Set the following fields on the control panel as necessary:

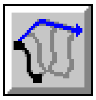
   - Select the **Vector Axis Line** input field and select a line that defines the axis of rotation as the axis line element to rotate the generator around.
   - Set the **Revolve Ang** input field to the number of degrees the generator profile is swept about the axis.
   - Set the **Path Dir** selector switch to **Generator**, **Radial** or **Planar**.
   - Set the **Planar Z** input field to the level at which the planar profiles should start (based on the active work plane's Z axis). This field is available only when **Path Dir** is set to **Planar**.

5. Set the **Spacing** selector switch to one of the following spacing methods if **Path Dir** is not set to **Planar**:
   - **Distance** specifies a fixed distance unit increment between mesh profiles.
   - **Count** specifies the number of mesh profiles. The spacing is determined by **Count** divided by **Distance**.
   - **cusp HGT** sets the spacing as the distance between passes, based on the acceptable cusp height (Spc Val) as measured on a 45-degree surface angle using the active tool.
   - **Junction** creates profiles at the start and end of the revolution angle or as radial profiles at each point along a new element that occurs on the generator profile. **Junction** is used only with geometry on a layer or with no tool offset.
6. Set the **Spc Val** input field to the spacing value to use with the **Distance**, **Count**, or **cusp HGT** spacing options. For planar path direction, enter the amount of spacing between planar profiles (based on the active work plane's Z axis).
7. Set the **Offset** selector switch to the direction of the toolpath offset, using one of these options:
   - **Pos** offsets the toolpath to the positive side.
   - **Zero** sets no offset.
   - **Neg** offsets the toolpath to the negative side. Use the surface normal or direction arrows to determine the correct side.
8. Set the **Finish Allow** input field to the amount to offset the layer from the defining profiles if a layer is active. Otherwise, set it to the amount to offset between the mesh profiles and the surface if a step is active. For steps, the finish allowance is added to the offset of the tool. This field is available only with positive or negative offsets.
9. Select the **Go** button.

Figure 8-4
Compare the different path directions for spun meshes.



Creating a Translated Mesh



Model File: A3DTRAN.PM4

Use Trans. Mesh to create mesh profiles by sweeping a generator profile along a director profile. The director curve is always perpendicular to the generator curve.

The generator and director profiles can consist of lines, arcs, splines, ellipses, or polylines. The elements for the generator and director profiles must be assigned to two separate work planes. For best results, make sure the work planes are perpendicular to each other and the generator and director curves lie flat in the planes.

1. Open the model file **A3DTRAN.PM4**.

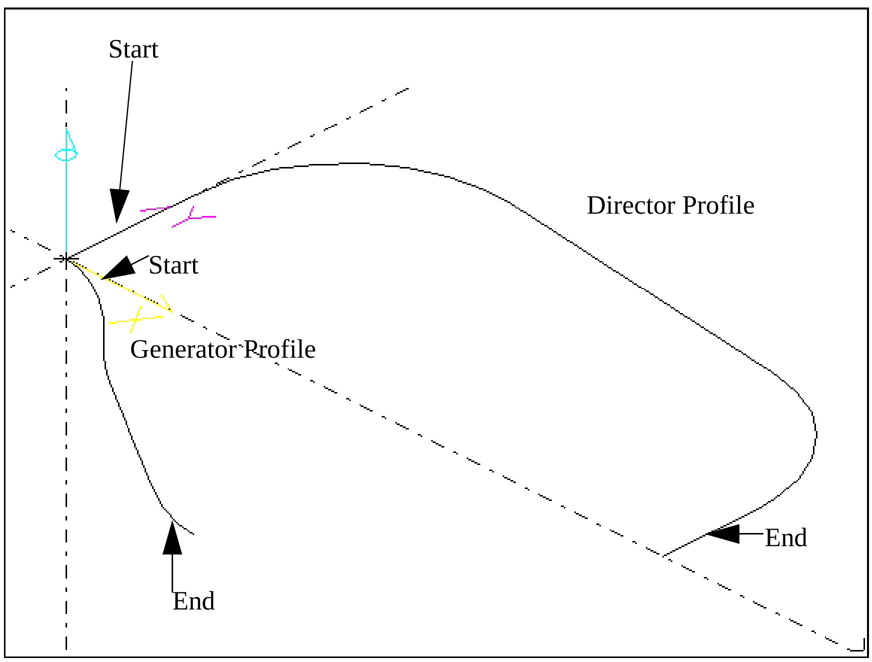


Figure 8-5
Open A3DTRAN.

2. Select **Create—Mesh—Trans. Mesh**. The **Trans. Mesh** control panel is displayed.

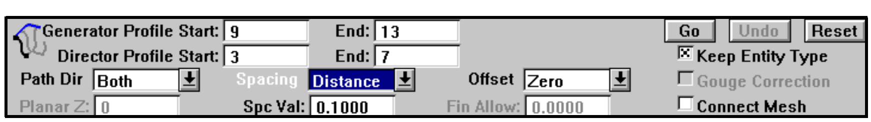


Figure 8-6
Set the values on the Trans. Mesh control panel.

3. Identify the generator profile by setting the **Generator Profile Start** and **End** fields to the first and last elements of the generator profile. Set these fields by manually typing, selecting the element in the graphics view, or selecting from the list view.

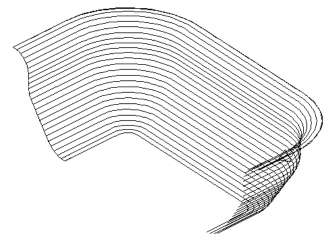
4. Identify the director profile by setting the **Director Profile Start** and **End** fields to the first and last elements of the director profile. Set these fields by manually typing, selecting the element in the graphics view, or selecting from the list view.

5. Set the remaining fields on the control panel as necessary.

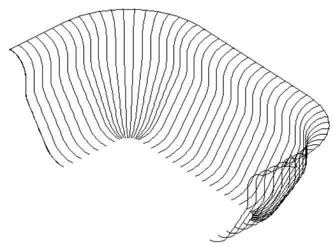
   - Set the **Path Dir** selector switch to the path direction.
   - Set the **Planar Z** input field to the level at which the planar profiles should start (based on the active work plane's Z axis). This field is available only when **Path Dir** is set to **Planar**.
   - Set the **Spacing** selector switch to a spacing method.
   - Set the **Spc Val** input field to a spacing value.
   - Set the **Offset** selector switch to the direction to offset the mesh.

6. Select the **Go** button.

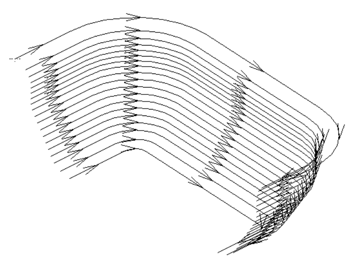
Figure 8-7
This is the result of creating a translated mesh.



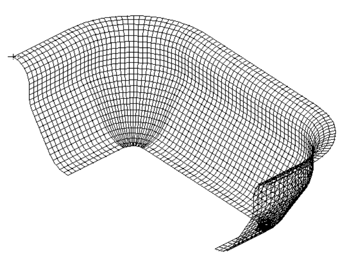
Path Direction—Director
The mesh profiles are parallel to the director profile.



Path Direction—Generator
The mesh profiles are parallel to the generator profile.

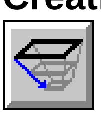


Path Direction—Planar
The mesh profiles are perpendicular to the tool axis.



Path Direction—Both
The mesh profiles are in both the director and the generator directions.

Creating a Draft Mesh



Model File: A3DDRFT.PM4

Use Draft Mesh to create mesh profiles by sweeping a generator profile along a three-dimensional director profile's path. The generator and director profiles can consist of lines, arcs, splines, ellipses, or polylines. For best results, all generator profile elements should be planar and assigned to one work plane that is parallel to the tool axis. The director profile can be nonplanar but must be assigned to one work plane. Use the Insert property bar to specify the properties and sequence for the new profiles.

With Draft Mesh, the director profile is a 3-D profile and the resulting mesh profiles are polylines. SmartCAM sweeps along the director profile, and so the resulting mesh profile is always perpendicular to the active tool plane.

Perform these tasks to create a draft mesh:

1. Open the model file **`A3DDRFT.PM4`**.

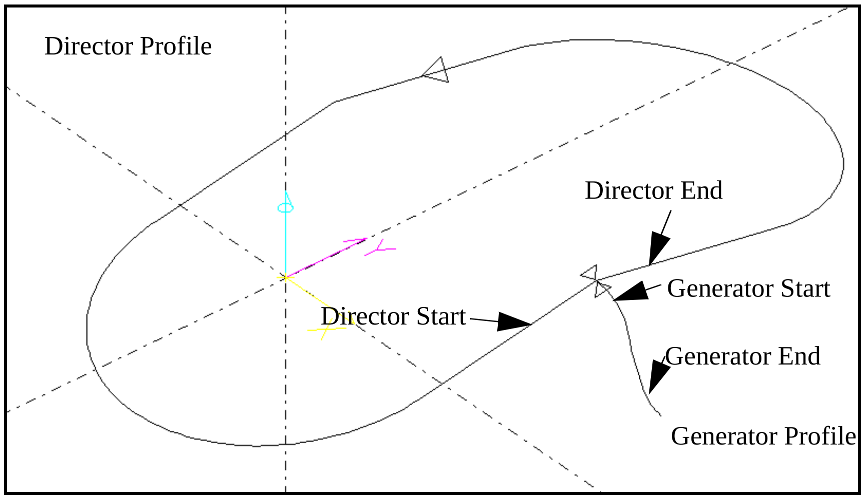


Figure 8-8
Open A3DDRFT.

2. Select **Create—Mesh—Draft Mesh**. The **Draft Mesh** control panel is displayed.

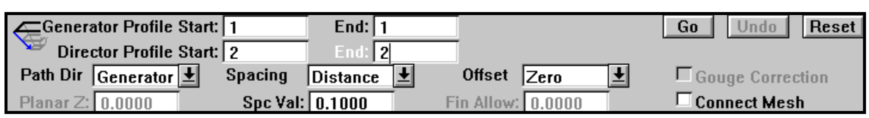


Figure 8-9
Set the values on the Draft Mesh control panel.

3. Identify the generator profile by setting the **Generator Profile Start** and **End** fields to the first and last elements of the generator profile. Set these fields by manually typing, selecting the element in the graphics view, or selecting from the list view.

4. Identify the director profile by setting the **Director Profile Start** and **End** fields to the first and last elements of the director profile. Set these fields by manually typing, selecting the element in the graphics view, or selecting from the list view.

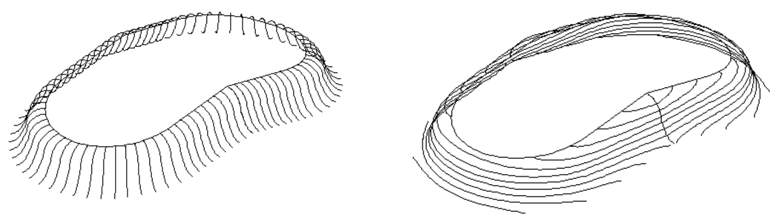
5. Set the remaining fields on the control panel as necessary:

   - Set the **Path Dir** selector switch to the path direction.
   - Set the **Planar Z** input field to the level at which the planar profiles should start (based on the active work plane's Z axis). This field is available only when **Path Dir** is set to **Planar**.
   - Set the **Spacing** selector switch to a spacing method.
   - Set the **Spc Val** input field to a spacing value.

- Set the **Offset** selector switch to the direction to offset the mesh.
- Set the **Finish Allow** input field to the amount to offset the layer from the defining profiles if a layer is active. Otherwise, set it to the amount to offset between the mesh profiles and the surface if a step is active. For steps, the finish allowance is added to the offset of the tool. This field is available only with positive or negative offsets.

6. Select the **Go** button.

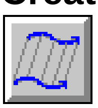
Figure 8-10
This is the result of creating a draft mesh.



Path Direction—Generator
The mesh profiles are parallel to the generator profile.

Path Direction—Planar
The mesh profiles are parallel to the tool plane of the active step.

Creating a Ruled Mesh



Model File: A3DRULE.PM4

Use Ruled Mesh to create mesh profiles between two profiles. Depending on the path direction, SmartCAM creates linear or curved profiles. Linear profiles move across the defining profiles, and curved profiles travel in the same direction as the defining profiles. The defining profiles can consist of lines, arcs, splines, points, ellipses, or polylines.

1. Open the model file **A3DRULE.PM4**.

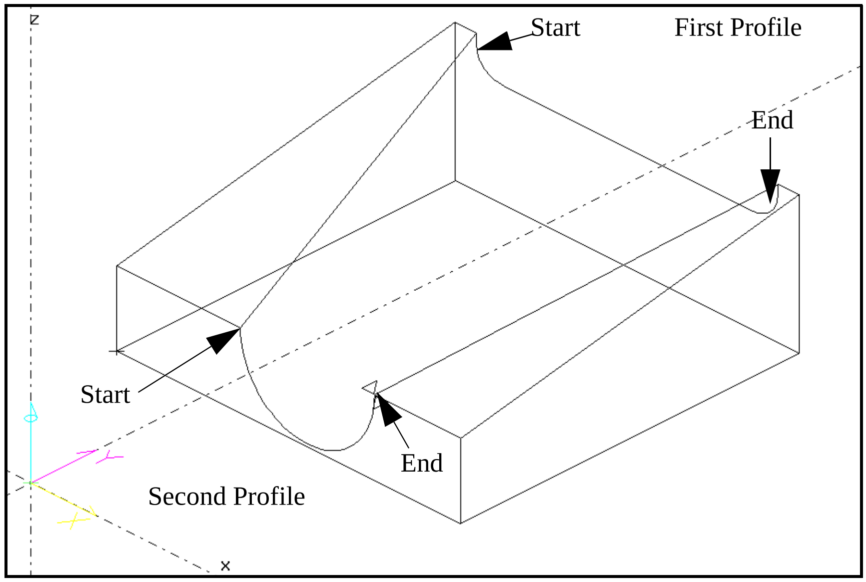


Figure 8-11 *Open A3DRULE.*

2. Select **Create—Mesh—Ruled Mesh**. The **Ruled Mesh** control panel is displayed.

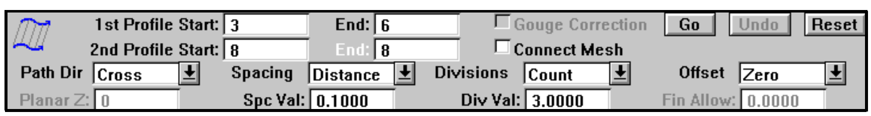


Figure 8-12 *Set the values on the Ruled Mesh control panel.*

3. Identify the first profile by setting the **1st Profile Start** and **End** fields to the first and last elements of the first profile. Set these fields by manually typing, selecting the element in the graphics view, or selecting from the list view.

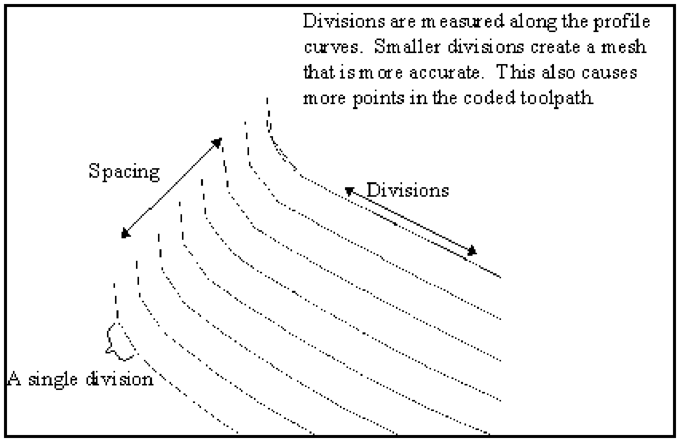
4. Identify the second profile by setting the **2nd Profile Start** and **End** fields to the first and last elements of the second profile. Set these fields by manually typing, selecting the element in the graphics view, or selecting from the list view.

5. Set the **Path Dir** selector switch to the path direction.

6. Set the following input fields on the control panel as necessary:

   - Set the **Planar Z** input field to the level at which the planar profiles should start (based on the active work plane's Z axis). This field is available only when **Path Dir** is set to **Planar**.
   - Set the **Spacing** selector switch to a spacing method.
   - Set the **Spc Val** input field to a spacing value.
   - Set the **Divisions** selector switch to the division method.
   - Set the **Div Val** input field to the value for the number of divisions.

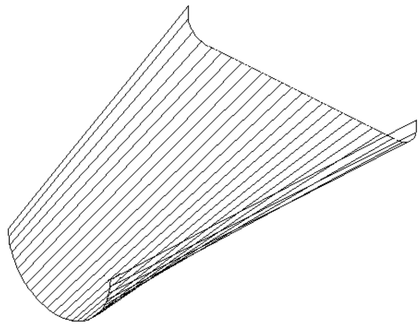
Figure 8-13
Notice the importance of divisions.



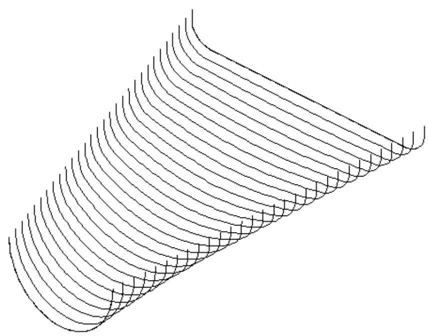
- Set the **Offset** selector switch to the direction to offset the mesh.
- If the Boundary Tangent Angle is 2 or less, you will get this error: “A Boundary Tangent Angle Comparison exceeds Surf Mode tolerance.” Select **OK** to continue.
- Set the **Finish Allow** input field to the amount to offset the layer from the defining profiles if a layer is active. Otherwise, set it to the amount to offset between the mesh profiles and the surface if a step is active. For steps, the finish allowance is added to the offset of the tool. This field is available only with positive or negative offsets.

7. Select the **Go** button.

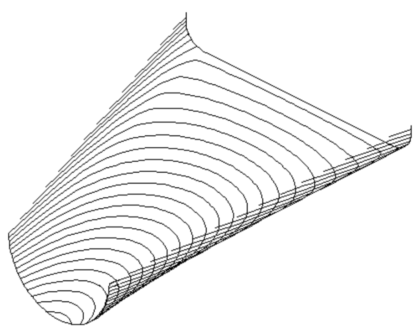
Figure 8-14
This is the result of creating a ruled mesh.



Path Direction—Cross
The mesh profiles go across the two profiles.

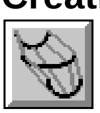


Path Direction—Along
The mesh profiles are parallel to the first and second profiles.



Path Direction—Planar
The mesh profiles are parallel to the tool plane of the active step.

Creating a Lofted Mesh



Model File: A3DLOFT.PM4

Use Lofted Mesh to create sculpted mesh profiles that travel over a series of sequential cross-section profiles. All cross-section profiles must be in the active group and travel in the same direction. Valid element types for the cross-section profiles include points, lines, arcs, splines, ellipses, and polylines. Use the Insert property bar to specify the properties and sequence for the new profiles.

Before you generate a lofted mesh, break sharp corners if necessary, and be sure that all the profile's elements connect.

Perform these tasks to create a lofted mesh:

1. Open the model file **A3DLOFT.PM4**.

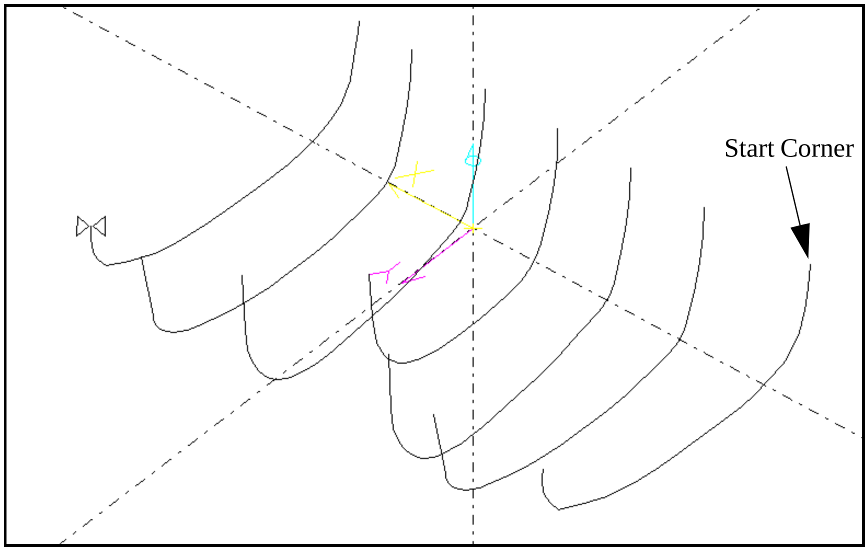


Figure 8-15 *Open A3DLOFT.*

2. Group all of the profile curves.
3. Select **Create—Mesh—Lofted Mesh**. The **Lofted Mesh** control panel is displayed.

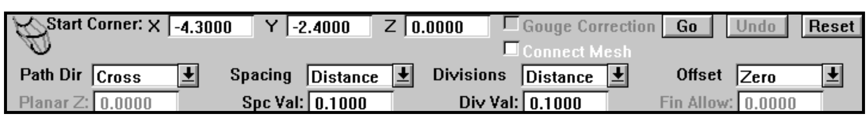
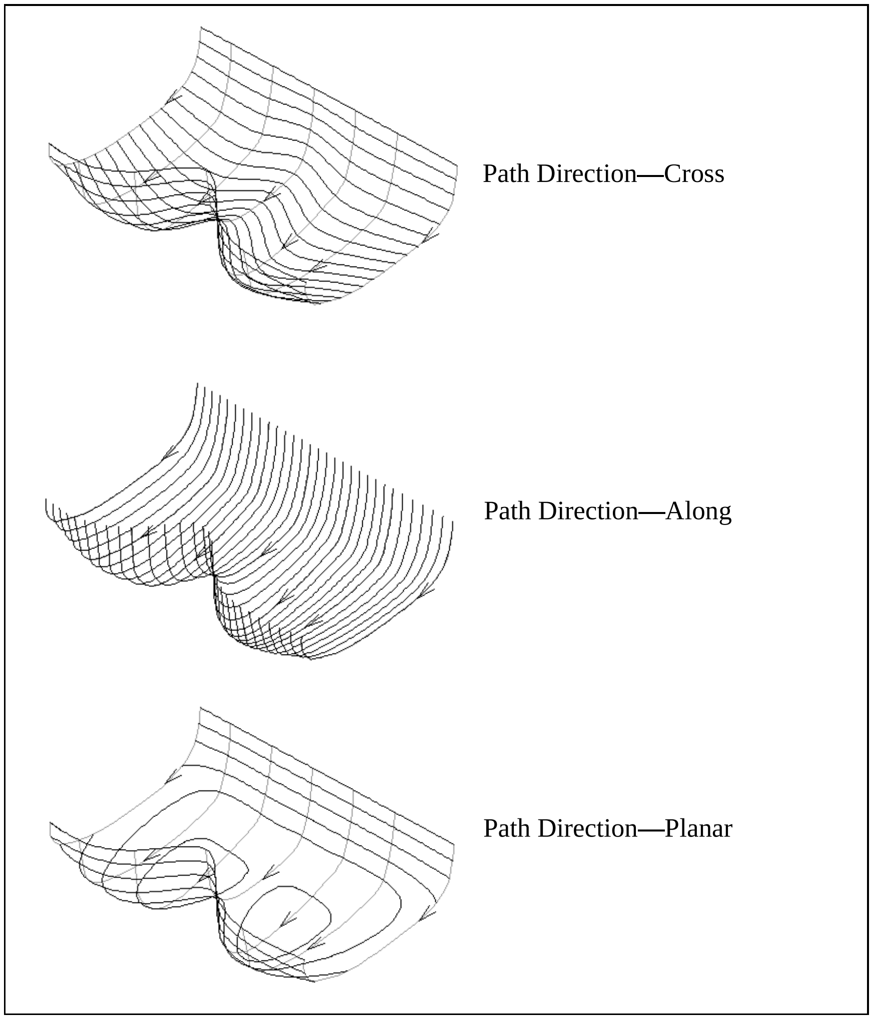


Figure 8-16 *Set the values on the Lofted Mesh control panel.*

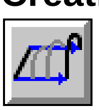
4. Set the following fields on the control panel as necessary:
   - Set the **Path Dir** selector switch to the path direction.
   - Set the **Planar Z** input field to the level at which the planar profiles should start (based on the active work plane's Z axis). This field is available only when **Path Dir** is set to **Planar**.
   - Set the **Spacing** selector switch to a spacing method.
   - Set the **Spc Val** input field to a spacing value.
   - Set the **Offset** selector switch to the direction to offset the mesh.
   - Set the **Divisions** selector switch to the division method.
   - Set the **Div Val** input field to the value for the number of divisions.
   - Set the **Finish Allow** input field to the amount to offset the layer from the defining profiles if a layer is active. Otherwise, set it to the amount to offset between the mesh profiles and the surface if a step is active. For steps, the finish allowance is added to the offset of the tool. This field is available only with positive or negative offsets.

5. Select the **Go** button.

Figure 8-17 *These are the results of creating a lofted mesh.*



Creating a Form Patch Mesh



Model File: A3DFRP.PM4

Use Form Patch to create sculpted mesh profiles by using four connected boundary profiles that define the surface area. The defining profiles for a Form Patch surface must be in the active group and can consist of lines, arcs, and polylines.

Perform these tasks to create a form patch mesh:

1. Open the model file **`A3DFRP.PM4`**.

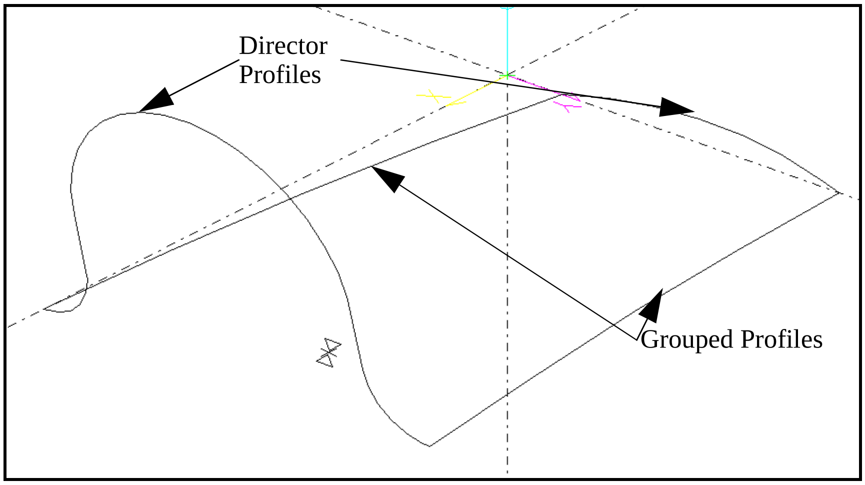


Figure 8-18
Open A3DFRP.

2. Group two of the profiles for the form patch. These are the defining profiles. The shape of the mesh changes slightly, depending on the profiles that are grouped.

3. Select **Create—Mesh—Form Patch**. The **Form Patch** control panel is displayed.

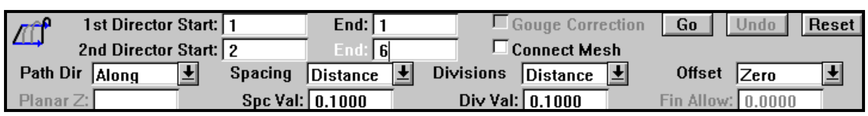


Figure 8-19
Set the values on the Form Patch control panel.

4. Identify the first director profile by setting the **1st Director Start** and **End** fields to the first and last elements of the first director profile. Set these fields by manually typing, selecting the element in the graphics view, or selecting from the list view.

5. Identify the director profile by setting the **2nd Director Start** and **End** fields to the first and last elements of the second director profile. Set these fields by manually typing, selecting the element in the graphics view, or selecting from the list view.

   Note For best results, opposing director profiles should run in the same direction. Use **Edit—Order Path** if they do not.

6. Set the **Path Dir** selector switch to the path direction, as follows:

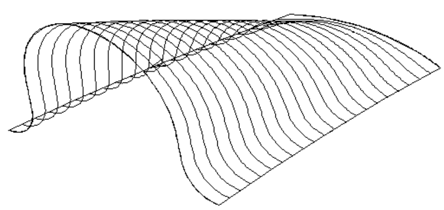
   - **Cross** creates mesh profiles across the director profiles.
   - **Along** creates mesh profiles parallel to the director profiles.
   - **Planar** creates mesh profiles perpendicular to the tool's axis (based on the active tool plane) at the levels you specify in the **Spc Val** and **Planar Z** input fields across the director profiles.

7. Set the following fields on the control panel as necessary:

   - Set the **Planar Z** input field to the level at which the planar profiles should start (based on the active work plane's Z axis). This field is available only when **Path Dir** is set to **Planar**.
   - Set the **Spacing** selector switch to a spacing method.
   - Set the **Spc Val** input field to a spacing value.
   - Set the **Offset** selector switch to the direction to offset the mesh.
   - Set the **Divisions** selector switch to specify the mode to use for calculating the mesh-profile segment length along each mesh profile.
   - Set the **Div Val** input field to the value for the number of divisions.
   - Set the **Finish Allow** input field to the amount to offset the layer from the defining profiles if a layer is active. Otherwise, set it to the amount to offset between the mesh profiles and the surface if a step is active. For steps, the finish allowance is added to the offset of the tool. This field is available only with positive or negative offsets.

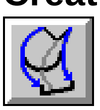
8. Select the **Go** button.

Figure 8-20 *This is the result of creating a form patch mesh.*



Path Direction—Along
The mesh profiles are parallel to the director curves.

Creating a Coons Patch Mesh



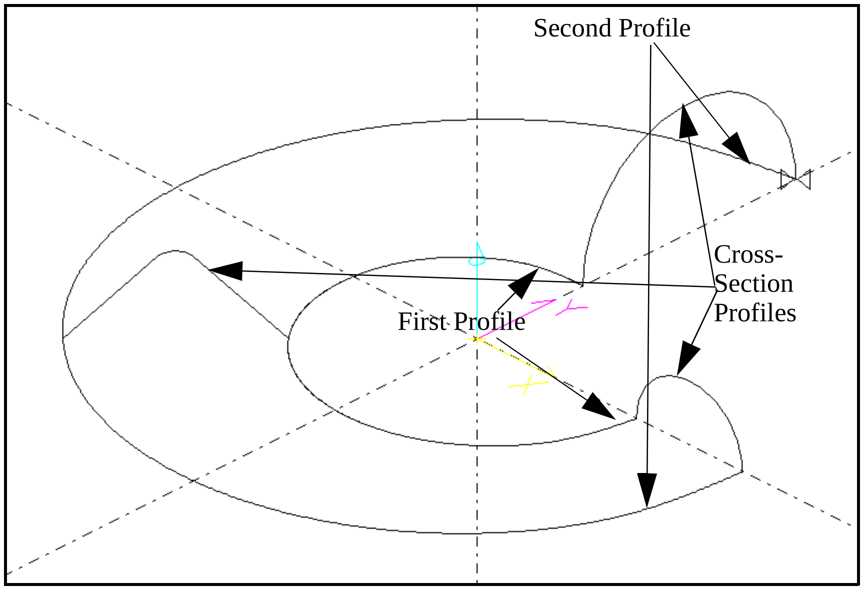
Model File: A3DCOON.PM4

Use Coons Mesh to create sculpted mesh profiles by using four connected boundary profiles that define the surface area and cross-section profiles that define the interior contour. The defining profiles for Coons Mesh must be in the active group and can consist of lines, arcs, and polylines.

Before generating a coons patch mesh, break sharp corners if necessary, and be sure that all the profile's elements connect. Toggle the By Patch switch on to relax the sharp corner requirement.

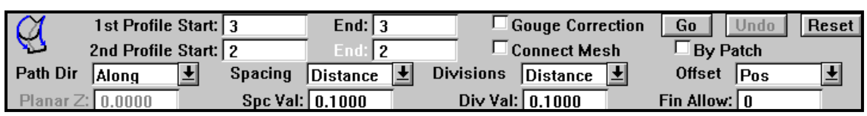
1. Open the model file **A3DCOON.PM4**.

Figure 8-21
Open A3DCOON.



2. Group the cross-section profiles.
3. Select **Create—Mesh—Coons Mesh**. The **Coons Mesh** control panel is displayed.

Figure 8-22
Set the values on the Coons Mesh control panel.

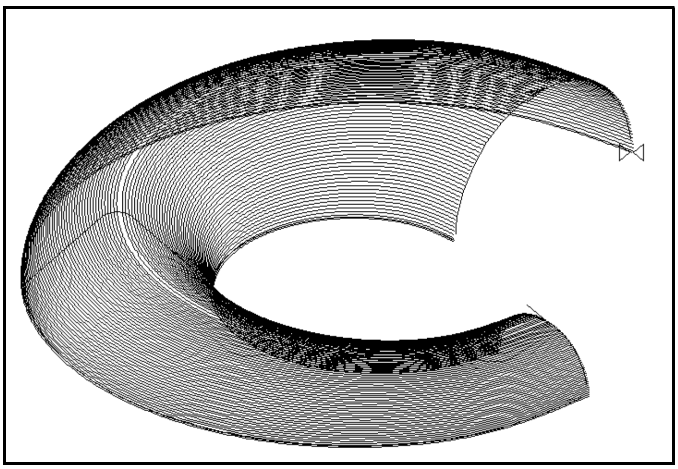


4. Identify the first profile by setting the **1st Profile Start** and **End** fields to the first and last elements of the first profile. Set these fields by manually typing, selecting the element in the graphics view, or selecting from the list view.
5. Identify the director profile by setting the **2nd Profile Start** and **End** fields to the first and last elements of the second profile. Set these fields by manually typing, selecting the element in the graphics view, or selecting from the list view.
6. Set the **Path Dir** selector switch to the path direction, using one of these methods:
   - **Cross** creates mesh profiles across the first and second profiles.
   - **Along** creates mesh profiles parallel to the first and second profiles.
   - **Planar** creates mesh profiles perpendicular to the tool’s axis.
7. Set the following fields on the control panel as necessary.
   - Set the **Planar Z** input field to the level at which the planar profiles should start (based on the active work plane's Z axis). This field is available only when **Path Dir** is set to **Planar**.

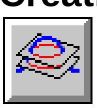
- Set the **Spacing** selector switch to a spacing method.
- Set the **Spc Val** input field to a spacing value.
- Set the **Offset** selector switch to the direction to offset the mesh.
- Set the **Divisions** selector switch to the mode to use for calculating the mesh-profile segment length along each mesh profile.
- Set the **Div Val** input field to the number of divisions.
- Set the **Finish Allow** input field to the amount to offset the layer from the defining profiles if a layer is active. Otherwise, set it to the amount to offset between the mesh profiles and the surface if a step is active. For steps, the finish allowance is added to the offset of the tool. This field is available only with positive or negative offsets.

8. Select the **Go** button.

Figure 8-23
This is the result of creating a coons patch mesh.



Creating Planar Cuts on an Existing Mesh



Model File: A3DPLNR.PM4

Use Planar Cuts to create planar profiles parallel to the active work plane at the Z levels you specify. The Planar Cuts feature operates on a selected range of existing mesh profiles. Use the Insert property bar to specify the properties and sequence for the planar cuts.

When you rough a cavity, you must have planar profiles before using the Cavity modeling tool. If you assign a step, it should match the step you plan to use with Cavity.

Each mesh-creation modeling tool has a planar option for path directions. That option performs the same function as the Planar Cuts modeling tool.

Before generating a planar cut, make sure there is an existing range of valid mesh profiles to use. These profiles should be sequential in the database, travel in the same direction, and be assigned to the same work plane.

1. Open the model file **A3DPLNR.PM4**.

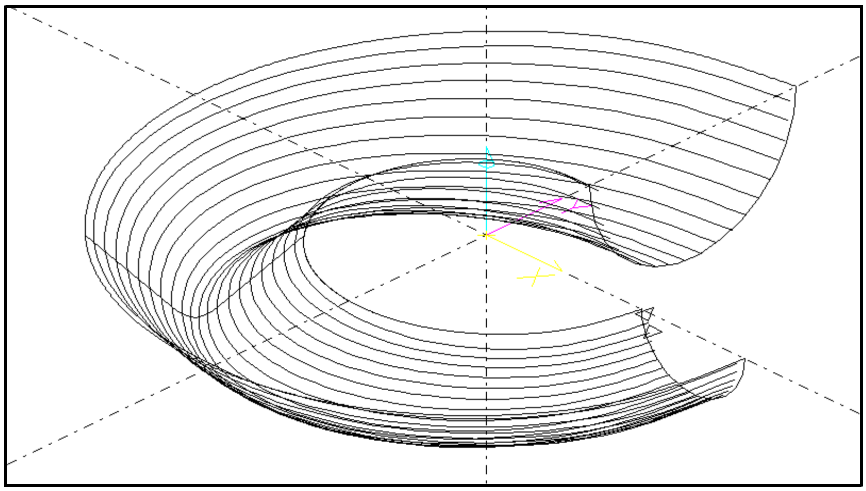


Figure 8-24
Open A3DPLNR.

2. Select **Create—Mesh—Planar Cuts**. The **Planar Cuts** control panel is displayed.

Figure 8-25
Set the values on the Planar Cuts control panel.

3. Identify the mesh by setting the **Mesh Start** and **End** fields to the first and last elements of the mesh to use for planar cuts. Set these fields by manually typing, selecting the element in the graphics view, or selecting from the list view.

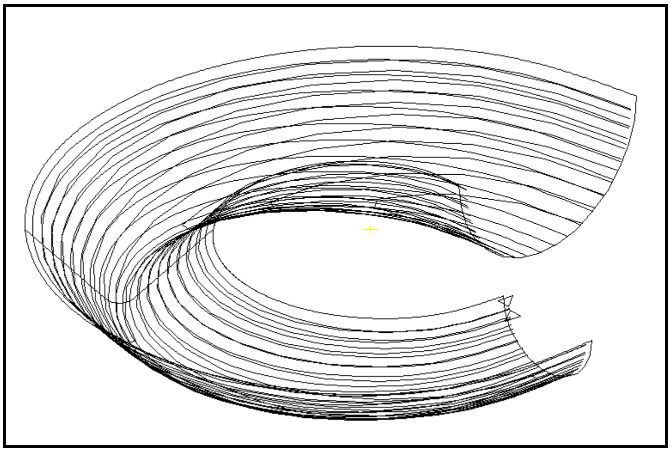
4. Set the following fields on the control panel as necessary:

   - Set the **Planar Z** input field, and enter the level at which the planar profiles should start (based on the active work plane's Z axis).
   - Set the **Spacing** input field to the distance between planar profiles.
   - Set the **Offset** selector switch to set the direction to offset the mesh.
   - Set the **Bgrnd Refine** selector switch to the mesh definition to use for determining the planar cuts polylines.
   - Set the **Bgrnd Val** input field to an appropriate value for the setting in the **Bgrnd Refine** input field.

- Set the **Finish Allow** input field to the amount to offset the layer from the defining profiles if a layer is active. Otherwise, set it to the amount to offset between the mesh profiles and the surface if a step is active. For steps, the finish allowance is added to the offset of the tool. This field is available only with positive or negative offsets.

5. Select the **Go** button.

Figure 8-26
This is the result of creating planar cuts on a mesh.



Points to Remember

- You can create surface mesh profiles for primitive, swept, and sculpted mesh profiles.
- Before you generate a mesh, examine the profiles. Make sure they meet the requirements for the type of surface.
- Before you generate a spun mesh, be sure that all the profile's elements connect and meet the Boundary Tangent Angle setting in the Surface Modes control panel.
- For translated meshes, the director curve is always perpendicular to the generator curve.
- For draft meshes, the director profile can be nonplanar but must be assigned to one work plane.
- For lofted meshes, all cross-section profiles must be in the active group and travel in the same direction.
- Before you generate a coons patch mesh, break sharp corners if necessary, and be sure that all the profile's elements connect.
- Before you generate a planar cut, be sure there is an existing range of valid mesh profiles to use. These profiles should be sequential in the database, travel in the same direction, and be assigned to the same work plane.

9

Editing Meshes

Objectives

This unit shows you how to perform these tasks:

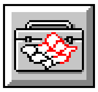
- Project elements to a mesh or other primitive shape.
- Intersect a mesh to another mesh or other primitive surface meshes.
- Trim a mesh to another mesh.
- Blend two meshes.
- Refine existing mesh profiles.
- Extend existing mesh profiles.
- Offset existing mesh profiles for a step.
- Connect existing mesh profiles.

Overview

Use Mesh Edit modeling tools to modify a mesh toolpath. You can project geometry on a mesh, create a blend toolpath between two mesh profiles, or extend the toolpath beyond the original mesh boundaries.

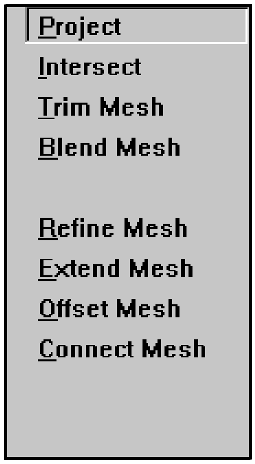
Where applicable, surface normal indicator arrows point to the top of the surfaces. Looking from the top, determine the right or left of an element.

Using the Mesh Edit Toolbox

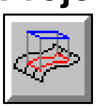


Open the Mesh Edit toolbox by selecting Edit—Mesh or by selecting the icon.

Figure 9-1
These are the tools in the Mesh Edit toolbox.



Projecting Elements to a Mesh or Other Primitive Shape

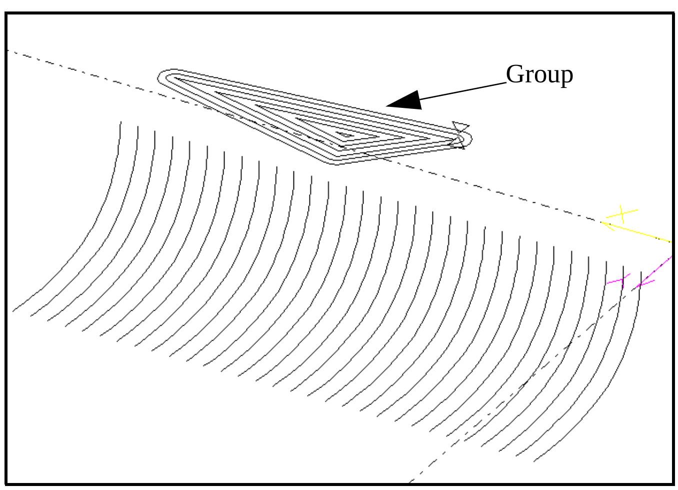


Model File: A3DPROJ.PM4

Use Project to project a copy of the active group onto a plane, cone, cylinder, sphere, or mesh. The resulting curves are polylines. Project is useful for creating toolpath or layer geometry that follows the shape of different surfaces.

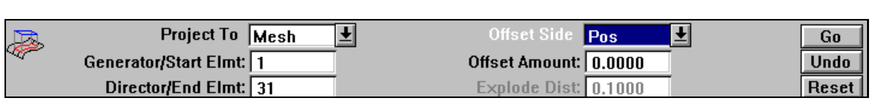
1. Open the model file **`A3DPROJ.PM4`**.

Figure 9-2
Open A3DPROJ.



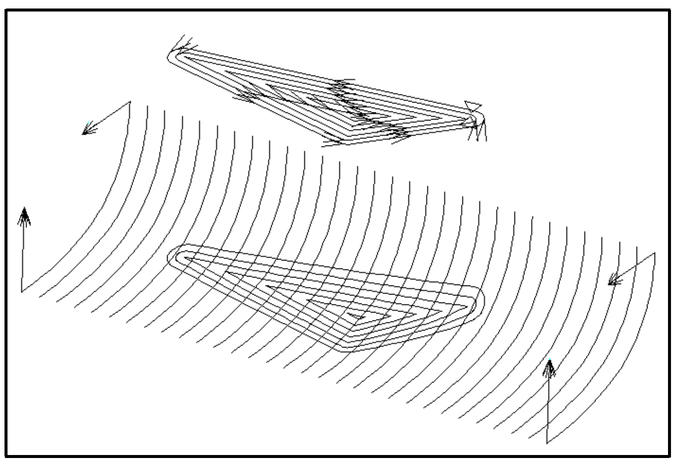
2. Group the elements to project.
3. Select **Edit—Mesh Edit—Project**. The **Project** control panel is displayed.

Figure 9-3
Set the values on the Project control panel.

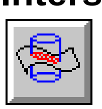


4. Set the **Project To** selector switch to **Mesh**.
5. Select the **Generator/Start Elmt** input field, and select the starting element on the generator profile. This is the object to project to.
6. Select the **Director/End Elmt** input field, and select the ending element on the director profile.
7. Set the **Offset Side** selector switch, and select the side of the surface to offset the projection.
8. Set the **Offset Amount** input field to the distance you want the projection offset from the surface. This field is active only when **Offset Side** is set to **Pos** or **Neg**.
9. Set the **Explode Dist** input field to the distance to use when dividing the active group of elements before projecting them. This choice is only available for **Cone/Cyl**, **Sphere**, or **Plane**.
10. Select the **Go** button. The projection is completed.

Figure 9-4
This is the result of the projection.



Intersecting a Mesh with a Mesh or Primitive Surface

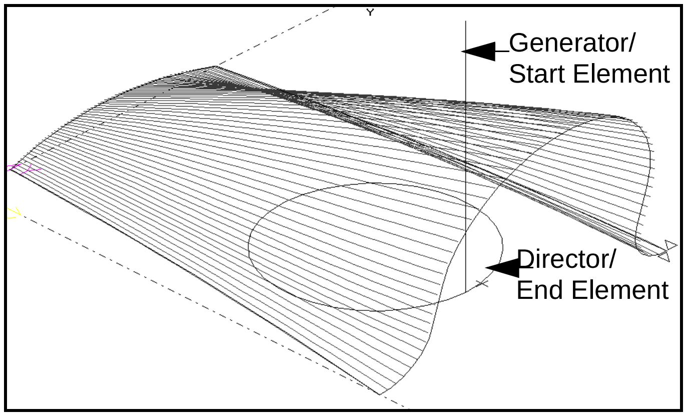


Model File: A3DCYL.PM4

Use Intersect to trim the active group where a plane, cone, cylinder, sphere, or mesh intersects it. Select which side of the intersection to keep and whether you want to leave an offset. Use this modeling tool to create toolpath or geometry that conforms to the shape of a mesh.

1. Open the model file **A3DCYL.PM4**.

Figure 9-5
Open A3DCYL.



2. Group the mesh profile to intersect.
3. Select **Edit—Mesh Edit—Intersect**. The **Intersect** control panel is displayed.

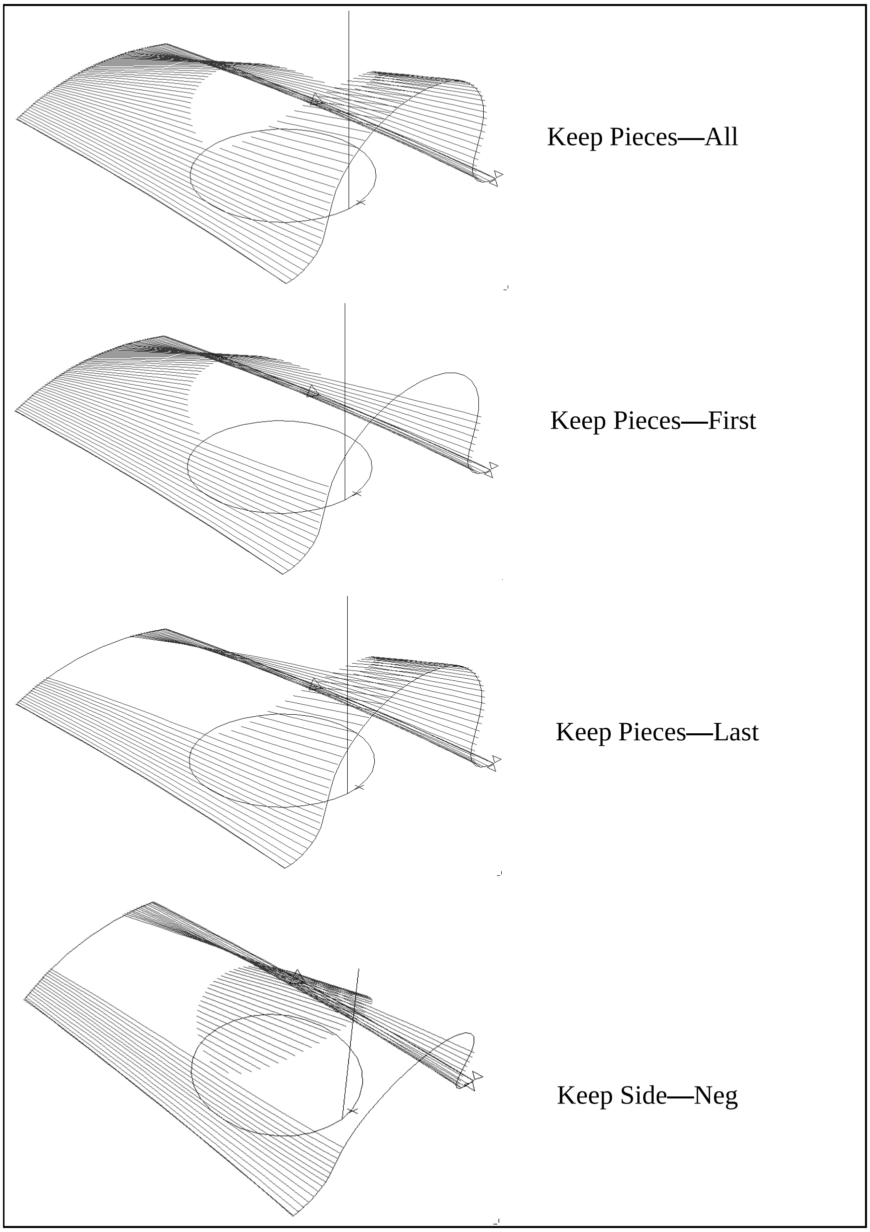
Figure 9-6
Set the values on the Intersect control panel.

4. Set the **Intersect To** selector switch to the surface type to which you want the profiles to intersect. For this lesson, select **Cone/Cyl**.
5. Select the **Generator/Elmt Mesh Start** input field, and select the start element of the generator profile in the list view or graphic view. Depending on the surface type, the element is as follows:
   - **Plane** uses the first boundary line.
   - **Cone/Cyl** uses the first generator line.
   - **Sphere** uses the arc that defines the shape of the intersection.
   - **Mesh** uses the starting element in the mesh range.
6. Select the **Director/Elmt Mesh End** input field, and select the ending element of the director profile in the list view or graphic view. Depending on the surface type, the element is as follows:
   - **Plane** uses the second boundary line.
   - **Cone/Cyl** uses the arc that defines the diameter.
   - **Sphere** uses the arc that defines the area of the intersection.
   - **Mesh** uses the ending element in the mesh range.
7. Select the **Keep Side** selector switch, and select which side of the active group you want SmartCAM to keep when trimming the intersection.
8. Set the **Keep Pieces** selector switch when the mesh intersects the active group in more than one location, as follows:

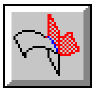
- **All** keeps the intersecting pieces of the active group on the side you specify.
- **First** keeps the first segment of the active group, based on element direction.
- **Last** keeps the last segment of the active group, based on element direction.

9. Select the **Go** button.

Figure 9-7
This is the result of intersecting a mesh with a mesh or primitive surface.



Trimming a Mesh to Another Mesh



Model File: A3DTRIM.PM4

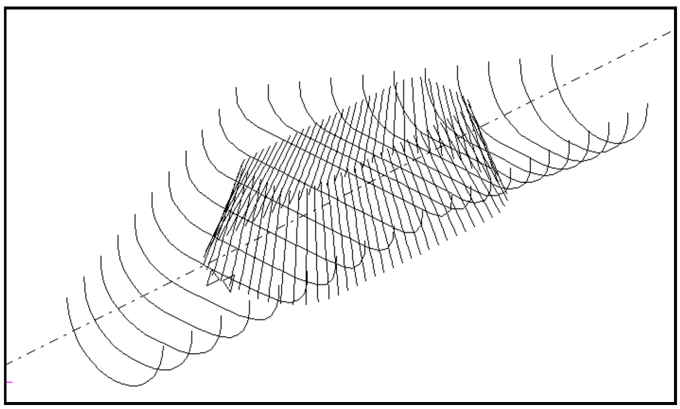
Use Trim Mesh to trim two ranges of either toolpath or layer geometry at their intersection or to remove unnecessary geometry that occurs where two ranges of elements intersect. Specify which side of the intersection SmartCAM should keep for both mesh profiles. You can also use Trim Mesh to create a profile along the points of intersection.

Mesh profiles can be assigned to either a step or a layer.

Perform these tasks to trim a mesh to another mesh:

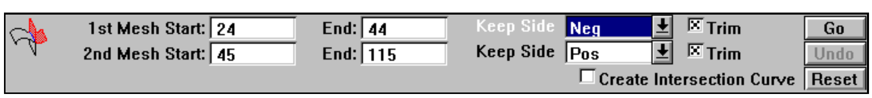
1. Open the model file **A3DTRIM.PM4**.

Figure 9-8
Open A3DTRIM.



2. Select **Edit—Mesh Edit—Trim Mesh**. The **Trim Mesh** control panel is displayed.

Figure 9-9
Set the values on the Trim Mesh control panel.



3. Identify the first mesh by setting the **1st Mesh Start** and **End** fields to the first and last elements of the first range of mesh profiles. Set these fields by manually typing, selecting the element in the graphics view, or selecting from the list view.

4. Identify the director profile by setting the **2nd Mesh Start** and **End** fields to the first and last elements of the second range of mesh profiles. Set these fields by manually typing, selecting the element in the graphics view, or selecting from the list view.

5. Set the **Keep Side** selector switch to one of the following options to indicate the side of the first range of mesh profiles you want to keep:

   - **Pos** keeps the positive part of the first range of elements.
   - **Neg** keeps the negative part of the first range of elements.

6. Turn on the **Create Intersection Curve** on/off switch to create a profile where the two mesh profiles intersect.
7. Select the **Go** button. The mesh is trimmed.

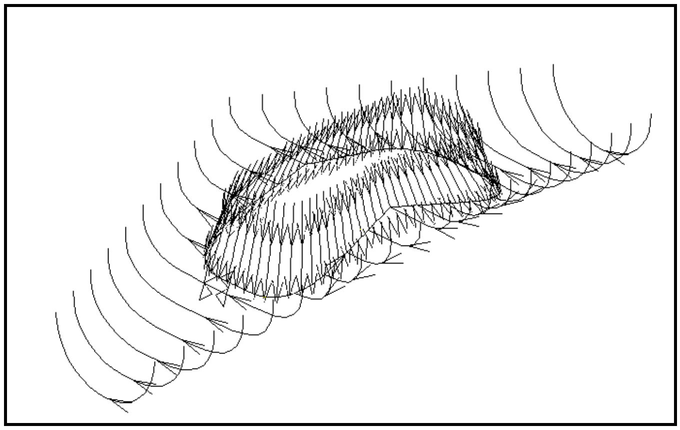
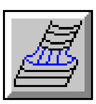


Figure 9-10 *This is the result of a trimmed mesh.*

Blending Two Meshes



Model File: A3DBLEND.PM4

Use Blend Mesh to create a constant or variable radius blend between two ranges of profiles using the properties and sequence you specify with the Insert property bar. The two ranges of profiles you select must be close enough for the blend radius to intersect them. Use Blend Mesh to create a curved toolpath that moves between two ranges of elements.

Use Blend Mesh to create a polyline mesh blend between two selected ranges of disconnected mesh elements. The two ranges must intersect with the blend size you identify. Each selected range must be in ascending or descending order.

1. Open the model file **`A3DBLEND.PM4`**.

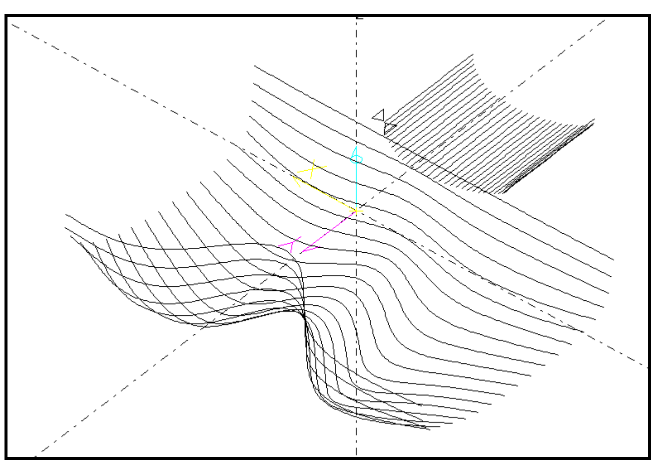
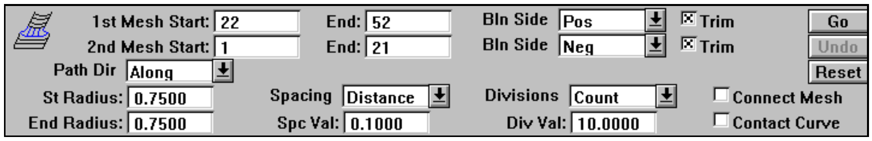


Figure 9-11 *Open A3DBLEND.*

2. Select **Edit—Mesh Edit—Blend**. The **Blend** control panel is displayed.

Figure 9-12
Set the values on the Blend control panel.



3. Identify the first mesh by setting the **1st Mesh Start** and **End** fields to the first and last elements of the first range of mesh profiles. Set these fields by manually typing, selecting the element in the graphics view, or selecting from the list view.

   Note You might need to use **View—Window** to identify specific elements. This selection also places the location of the start radius for variable radius blends closest to this element.

4. Identify the director profile by setting the **2nd Mesh Start** and **End** fields to the first and last elements of the second range of mesh profiles. Set these fields by manually typing, selecting the element in the graphics view, or selecting from the list view.

5. Set the **Bln Side** selector switches to **Pos** or **Neg**, to indicate on which side of the each range of mesh profiles to position the center of the blend arcs. The red blend arrows should point toward the center of the blend radius.

6. Turn on the **Trim** on/off switch to trim the second mesh along the intersection.

7. Set the **St Radius** input field to **.75** [**19.05**], which is the beginning blend radius value. Enter the same start and end radius values to generate a constant radius blend. Use different values to produce a variable radius blend. SmartCAM places this radius at the **1st Mesh Start** element.

8. Set the **End Radius** input field to **.75** [**19.05**], which is the ending blend radius value.

9. Set the **Path Dir** selector switch to one of the following:

   - **Cross** creates mesh profiles that run perpendicular to the direction of the blend's arcs.
   - **Along** creates mesh profiles that run parallel to the direction of the blend's arcs.

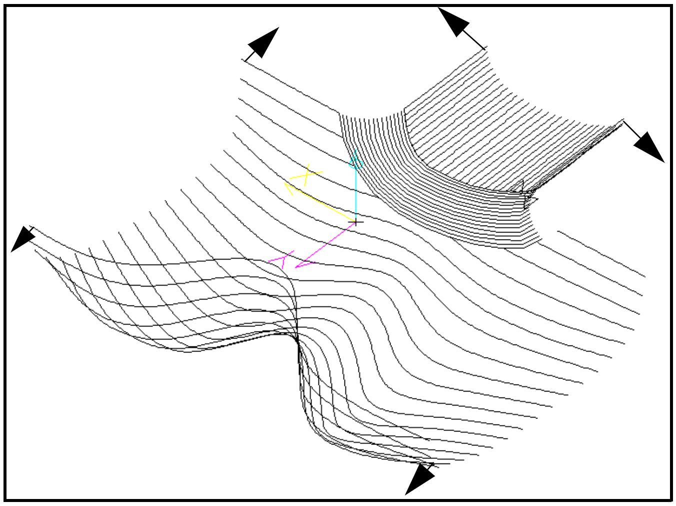
10. Set the **Spacing** selector switch to one of the following:

    - **Distance** specifies the distance between parallel mesh profiles.
    - **Count** specifies the number of mesh profiles.

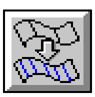
11. Set the **Spc Val** input field to the distance or count values.

12. Select the **Go** button. The meshes are blended.

Figure 9-13
This is the result of blending the meshes.



Refining Existing Mesh Profiles



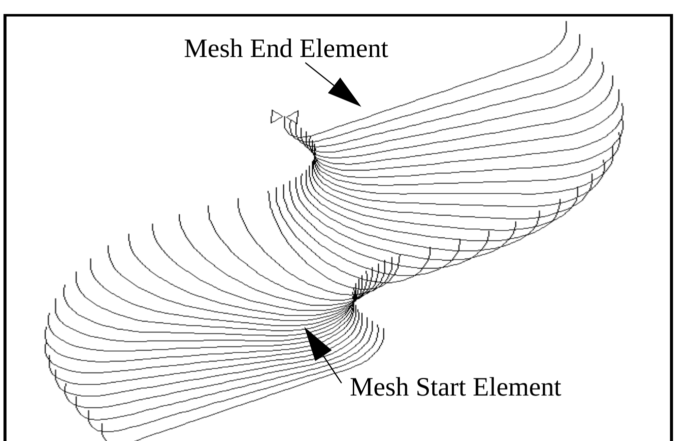
Model File: A3DREFN.PM4

Use Refine Mesh to change the controlling point count, spacing, divisions, and cut direction on a range of mesh profiles. You can choose to remove or keep the original range of profiles. Use Refine Mesh to modify mesh toolpath polylines for other modeling tools such as Blend Mesh or to improve mesh quality.

The geometry you use must be disconnected, point in the same direction, and be sequential in the database. SmartCAM creates polylines from the range of elements you specify. If those elements are lines and arcs, the Virtual Mesh Tolerance setting in the Surface Modes control panel controls the number of polyline segments that result.

1. Open the model file **A3DREFN.PM4**.

Figure 9-14
Open A3DREFN.



2. Select **Edit—Mesh Edit—Refine**. The **Refine** control panel is displayed.

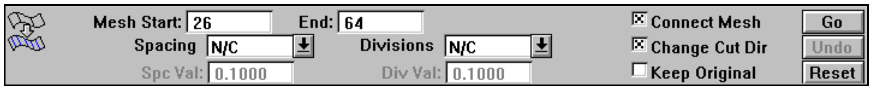
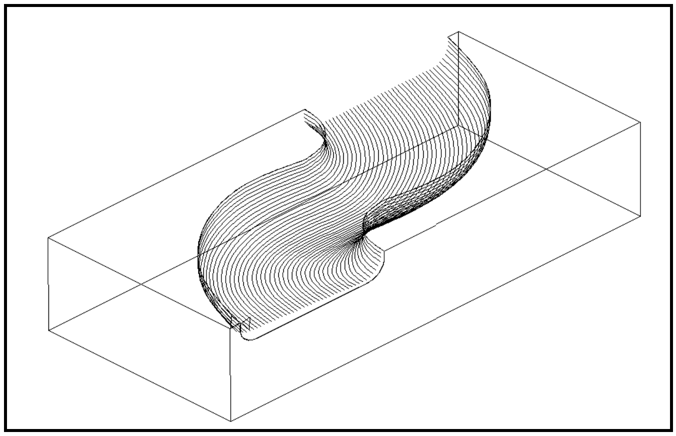


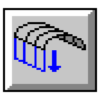
Figure 9-15 *Set the values on the Refine control panel.*

3. Identify the mesh by setting the **Mesh Start** and **End** fields to the first and last elements of the range of mesh profiles to refine. Set these fields by manually typing, selecting the element in the graphics view, or selecting from the list view.
4. Turn on the **Change Cut Dir** on/off switch to invert the direction of the mesh element.
5. Turn on the **Keep Original** on/off switch if you want to keep the original model geometry also.
6. Set the **Spacing** selector switch to one of the following:
   - **N/C** specifies the original spacing.
   - **Distance** specifies spacing determined by the distance between parallel mesh profiles.
   - **Count** specifies spacing determined by the number of passes.
   - **cusp HGT** specifies spacing determined by the allowable height of cusp based on tool geometry and a standard 45-degree mesh angle. Enter a value smaller than the corner radius of the tool.
7. Set the **Divisions** selector switch to specify the mode to use for calculating the surface-profile segment length, as follows:
   - **N/C** keeps the original divisions.
   - **Distance** specifies the length of each mesh or profile’s polyline segments.
   - **Count** specifies the number of segments for each mesh or profile’s polyline.
   - **Mid Tol** specifies a tolerance band on both sides of the mathematical mesh that the polyline segments must fall within.
8. Turn on the **Connect Mesh** on/off switch to place an element between the ends of the passes, joining the elements into one zigzag pattern.
9. Select the **Go** button. The mesh is refined.

Figure 9-16
This mesh was refined to change the cut direction.



Extending Existing Mesh Profiles



Model File: A3DEXTND.PM4

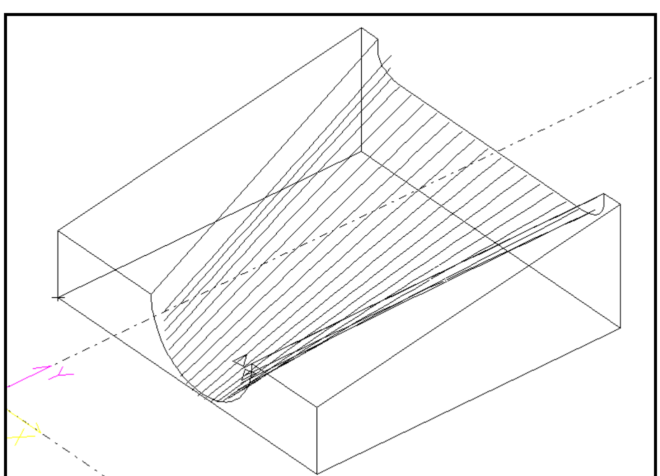
Use Extend Mesh to extend a selected range of mesh profiles beyond the mesh's defining boundaries. The properties and sequence position for the selected profiles remain the same. Use Extend Mesh to extend mesh profiles for operations such as Planar Cuts and Trim Mesh or to extend the toolpath beyond a material boundary.

Extend Mesh either lengthens a selected range of mesh profiles perpendicular to the first and last mesh profile or extends the ends of the profiles. The properties and sequence for the selected range remain the same.

The range of profiles you select must be disconnected before you use Extend Mesh. Use Connect Mesh to disconnect profiles.

1. Open the model file **`A3DEXTND.PM4`**.

Figure 9-17
Open A3DEXTND.



2. Select **Edit—Mesh Edit—Extend**. The **Extend Mesh** control panel is displayed.

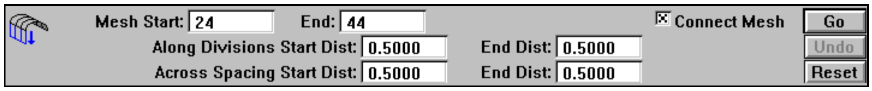


Figure 9-18 *Set the values in the Extend Mesh control panel.*

3. Identify the mesh by setting the **Mesh Start** and **End** fields to the first and last elements of the range of mesh profiles to extend. Set these fields by manually typing, selecting the element in the graphics view, or selecting from the list view.

4. Set the following fields on the control panel as necessary:

   - To extend along the profiles, set the **Along Divisions Start Dist** input field to the distance to add to the start of the mesh profiles, and set the **End Dist** input field to the distance to add to the end of the mesh profiles.
   - To extend across the profiles, set the **Across Spacing Start Dist** input field to the distance to add to the beginning of the mesh cross-sections or width of the mesh, and set the **End Dist** input field to the distance to add to the end of the mesh cross-sections or width of the mesh.
   - Turn on the **Connect Mesh** on/off switch to place an element between the ends of the passes, joining the elements into one zigzag pattern.

5. Select the **Go** button.

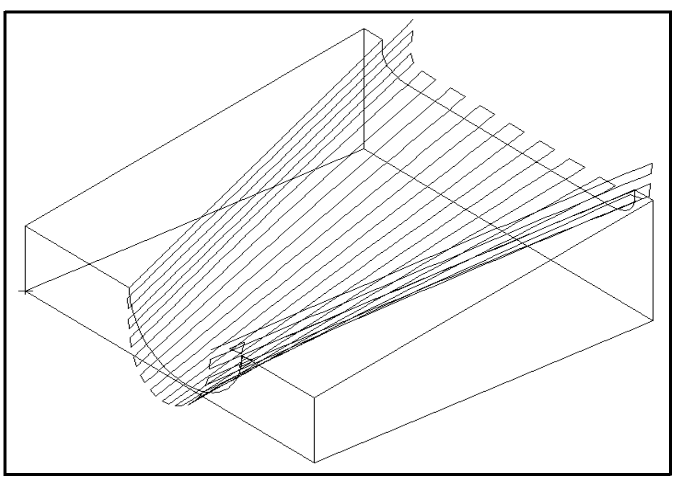
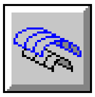


Figure 9-19 *This is a result of extending the mesh.*

Offsetting Existing Mesh Profiles for a Step



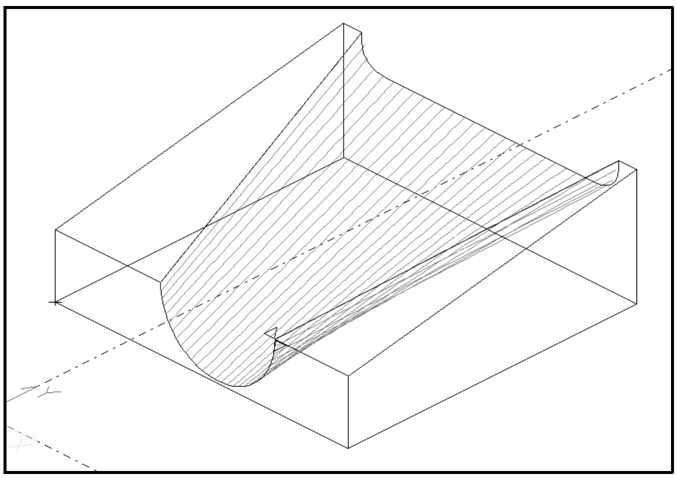
Model File: A3DOFFS.PM4

Use Offset Mesh to offset the range of mesh profiles you select by using the assigned tool's geometry and the mesh normal orientation. The offset calculation is based on the tool type, resulting in the positions for the bottom center of the tool on the offset toolpath. Offset Mesh provides a full offset, which includes a finish amount, or a partial offset for just the finish amount. You can also use the Offset option within a mesh modeling tool to create an offset toolpath.

Offset Mesh requires a sequential range of disconnected mesh profiles. Mesh profiles can be assigned to either a step or a layer.

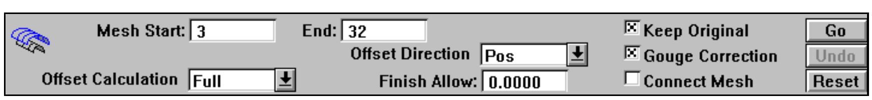
1. Open the model file **A3DOFFS.PM4**.

Figure 9-20 *Open A3DOFFS.*



2. Select **Edit—Mesh Edit—Offset**. The **Offset** control panel is displayed.

Figure 9-21 *Set the values on the Offset control panel.*



3. Identify the mesh by setting the **Mesh Start** and **End** fields to the first and last elements of the range of mesh profiles. Set these fields by manually typing, selecting the element in the graphics view, or selecting from the list view.

4. Set the **Offset Calculation** selector switch to **Full**.

5. Set the **Offset Direction** selector switch to one of the following:

   - **Pos** offsets to the positive side of the mesh profiles.
   - **Neg** offsets to the negative side of the mesh profiles. Use the direction arrows to make the selection.

6. Turn on the **Connect Mesh** on/off switch to place an element between the ends of the passes, joining the elements into one zigzag pattern.
7. Turn on the **Gouge Correction** on/off switch to check and correct for toolpath reversal, reducing the occurrence of surface violation by the tool.
8. Select the **Go** button.
9. Look at the model from the Front view.

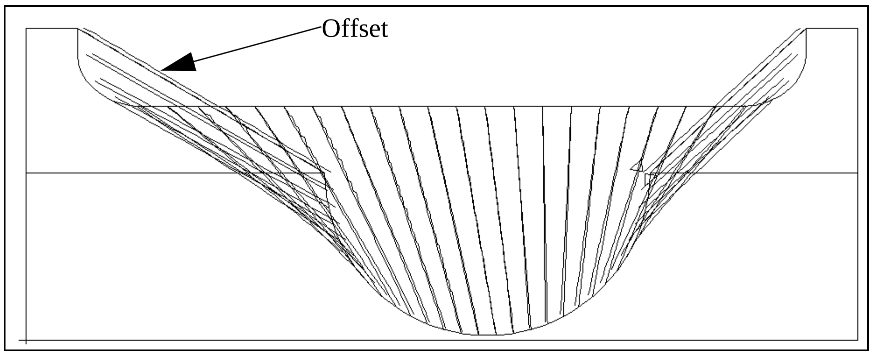
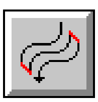


Figure 9-22
This is a result of offsetting the mesh.

Connecting Existing Mesh Profiles



Model File: A3DREFN.PM4

Use Connect Mesh to connect individual mesh profiles in the active group into a continuous zigzag profile by inserting a line element between the end of one profile and the start of the adjacent profile. You can also disconnect profiles, which results in elements traveling in the same direction and sequentially in the database.

1. Open the model file **`A3DREFN.PM4`**.

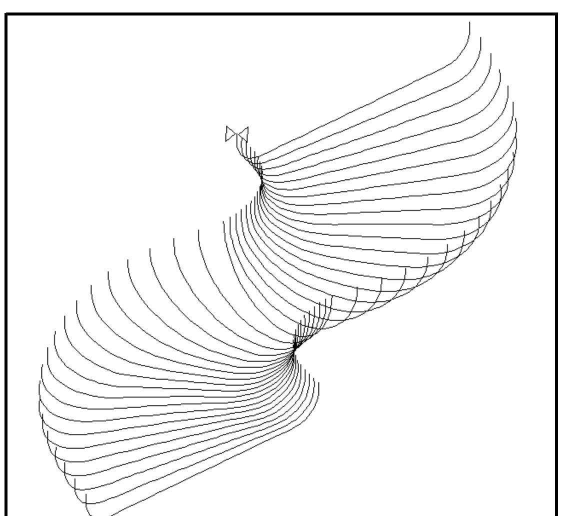


Figure 9-23
Open A3DREFN.

2. Group all the mesh profiles to connect or disconnect.

3. Select **Edit—Mesh Edit—Connect**. The **Connect** control panel is displayed.

Figure 9-24
Set the values on the Connect control panel.

4. Set the **Maximum Connection Distance** input field to the maximum distance that separates the start and end of adjacent profiles. Any profiles separated by more than this amount are not connected. This setting applies only when you connect profiles.

5. Turn on the **Disconnect Mesh** on/off switch if you want to disconnect previously connected mesh profiles.

6. Select the **Go** button.

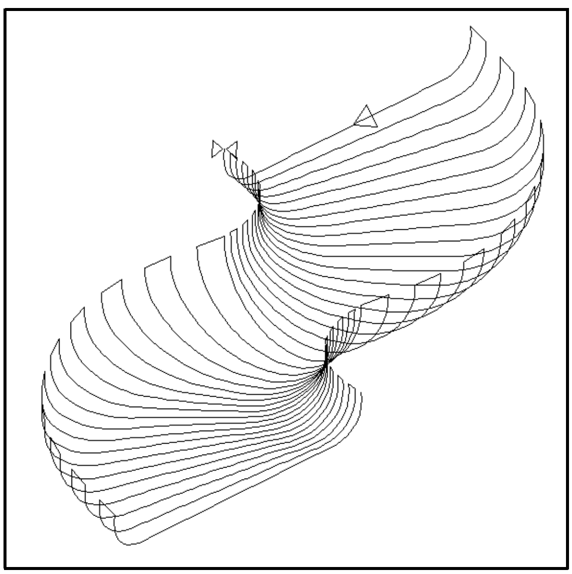


Figure 9-25
This is the result of connecting the meshes.

Points to Remember

- Project is useful for creating toolpath or layer geometry that follows the shape of different surfaces.
- When you trim, specify which side of the intersection SmartCAM should keep for both mesh profiles.
- To refine existing meshes, you must use geometry that is disconnected, points in the same direction, and is sequential in the database.
- When you extend existing mesh profiles, the properties and sequence for the selected range remain the same.
- Offset Mesh requires a sequential range of disconnected mesh profiles.

- You can disconnect profiles, which results in elements traveling in the same direction and sequentially in the database.
- When you blend meshes, the two ranges of profiles you select must be close enough for the blend radius to intersect them.

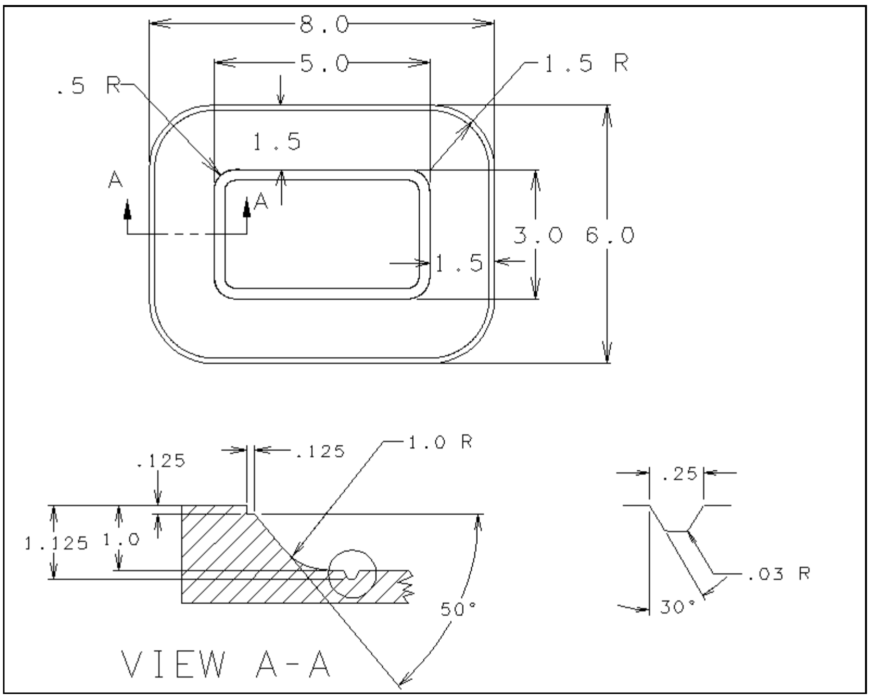
Challenge Project

Directions

Use the part print to draw the geometry to make the toolpath. Create two meshes. One mesh is the contour for the side wall. The second mesh is the bottom groove in the part. Cut the other area of the cavity, using pocketing or simple profile curves. This is one of many different methods of creating and machining this part.

Part Print

Figure 9-26
Draw this part.



Procedure

1. Study the part print and determine the tools needed to machine the part, from the following list:
   - Ball nosed end mill for the wall contour
   - .0625[1.531] ball nosed end mill for the groove on the bottom of the part
   - .75[19.05] end mill for cutting other areas

2. Draw a material box for the part on a layer (Layer #1). The box can have a Z level of **-1.75[42.875]** and a profile top of **0**.
3. Using the XY work plane, draw the outer edge of the cavity. This can be drawn as a profile with the following parameters:
   - With Step #3
   - Level -.125[-3.0625]
   - Profile Top 0
   - Clear .25[6.35]
   - Offset left
4. Draw the profile counterclockwise to produce a climb cut.
5. Draw the top edge of the contoured wall on a layer (Layer #2). The level should be **-.125[3.0625]**.
6. Draw the profile of the contoured wall on a layer (Layer #3) using the XZ work plane.
7. Create a translated mesh with the following parameters:
   - With Step #1
   - Clear .25[6.35]
   - Offset To the inside
   - Generator curve Contour of the profile
   - Director curve Top edge of the wall
8. Draw the top edge of the bottom groove on a layer (Layer #4). The level should be **-1.0[-24.5]**. Use the XY work plane.
9. Draw the profile of the groove on a layer (Layer #5) using the XZ work plane.
10. Create a translated mesh with the following parameters:
    - With Step #2
    - Clear .25[6.35]
    - Offset To the inside
    - Generator curve Profile of the groove
    - Director curve Top edge of the groove

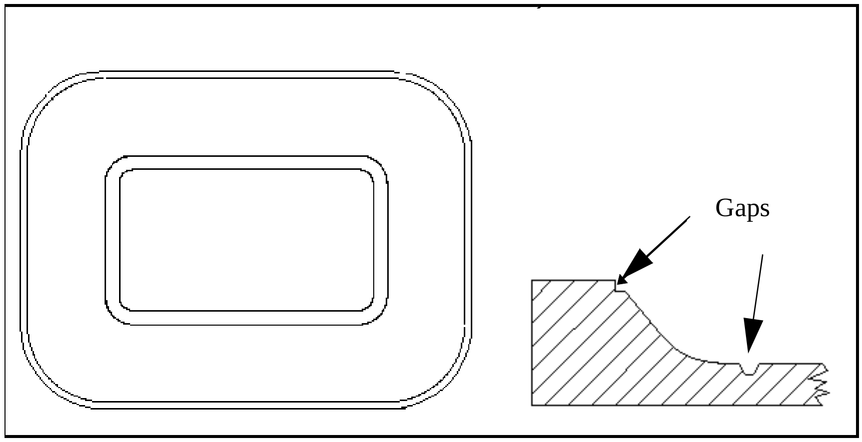


Figure 9-27
The mesh is blended.

10

Creating and Editing Elements from Solids

Objectives

This unit shows you how to perform these tasks:

- Import a CAD file.
- Export a SmartCAM file.
- Create boundary profiles from a solid.
- Create slice profiles from a solid.
- Create hole elements from a solid.

Overview

Use the From Solids function to obtain information from an ACIS solid body to create SmartCAM elements. You can create two- or three-dimensional profiles from the surface boundaries that define a solid. You can also create hole elements from the cylinders in the solid and cross-section profiles from the intersection of a plane and a solid. You can generate the elements on a layer or with a step.

Importing a CAD File

Objectives

This lesson shows you how to bring a CAD file into SmartCAM Advanced Milling.

Overview

Import transfers the geometry contained in ACIS, ATS, DXF, DWG, and IGES files into a SmartCAM process model file (.pm4).

If the import is for a surface and solids file type, SmartCAM displays an Import Status dialog box during the import showing the number of surfaces, number of surfaces processed, number of solids found, and the name of the log file. The log file saves detailed information about the import process. The Import Status dialog box is not displayed for CAM Connection File Types.

The quality and reliability of the data transfer improves when you select only the geometry you need and appropriate conversion options from the CAD system before importing the CAD file.

After importing a file, you can add or remove surfaces and apply machining processes to generate code.

Using Import

A CNC process model is created using either inch or metric units, and consists of a .pm4 file and a .jof file. The .pm4 file holds the geometry while the .jof file specifies all the tooling and operation information. Every time you save a CNC process model, these files are saved as a pair.

The unit setting establishes the units for everything relating to the model, including the job tooling. SmartCAM does not support Inch and metric units in the same model. When you are ready to import a file, determine the units of both the file being imported and the .jof file in the process model. How you import the geometry depends on whether these two file units are the same or different. You can import in these ways:

- Import into a model file that has the same units.
- Import into a model file that has different units, and adopt the units of the existing file.
- Import into a model file that has different units, and adopt the units of the incoming file.

Importing into a Model with the Same Units

Perform these tasks to import a CAD file into a new or existing model that has the same units:

1. If you are importing into a new file, set the units in the new file to match the units of the file to import. If you are importing into an existing file, proceed to step 2.
2. Select **File—Import**. The **Import** dialog box is displayed.

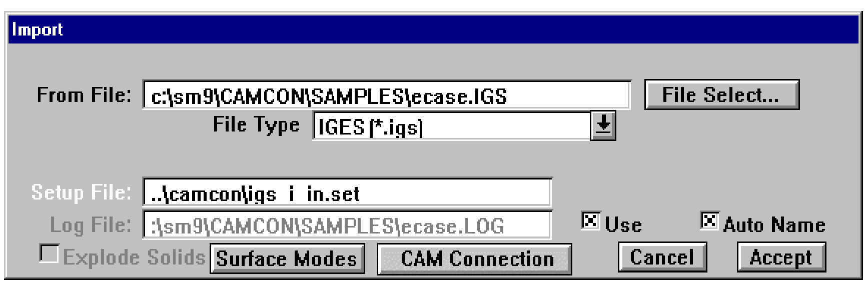
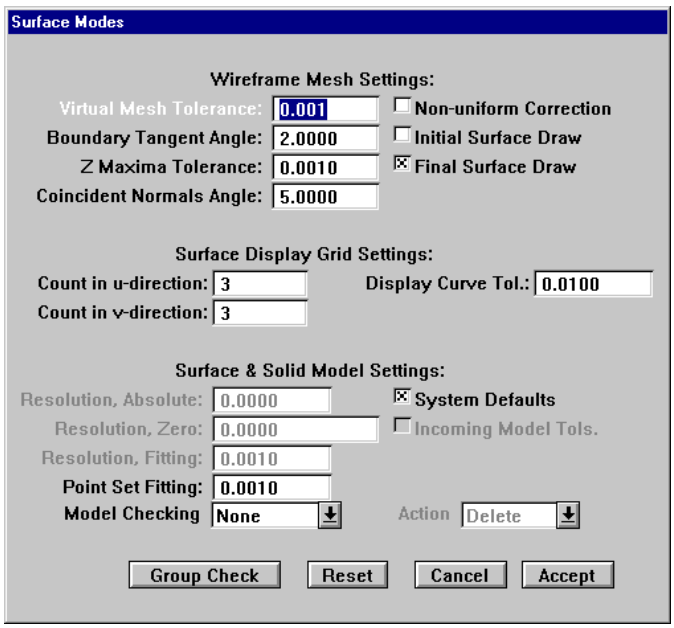


Figure 10-1
Open the Import dialog box.

3. Set the **File Type** selector switch.
4. Enter the path and name of the file to import into the **From File** input field, or use the **File Select** button to specify the path and name of the file.
5. Enter the name of the setup file used with the file type of the imported file in the **Setup File** input field. Generally, there is a separate setup file for each file type. In most cases, the setup file is automatically inserted when the file type is selected. You can also use a customized setup file.
6. Turn on the **Use** on/off switch if you want to create a log file.

7. Name the log file or have SmartCAM name it for you.
   - To name the log file, enter the name of the log file to create in the **Log File** input field.
   - To have SmartCAM name the log file, turn on the **Auto Name** on/off switch. (You must have turned on the **Use** on/off switch.)
8. If you are importing an ACIS file, set the **Explode Solids** switch to the desired setting.
9. Select the **Surface Modes** button. The **Surface Modes** dialog box is displayed.

Figure 10-2
Open the Surface Modes dialog box.



10. Set the **Count in u-direction**, **Count in v-direction**, **Display Curve Tol**, and/or the **Incoming Model Tols** fields. Import operates faster if you set the Display Grid Settings to low numbers (for example, 0 or 1) and set the Display Curve Tolerance to a loose value (for example, 0.010 inches or 0.25 millimeters).
11. Select the **Accept** button. The **Import** dialog box is reactivated.
12. Select **Accept** to load the file.

Importing and Adopting the Existing Model File's Units

Perform these tasks to import a CAD file into a new or existing model that has different units:

1. Open the model file that you want to import the CAD file into.
2. Complete steps 2 through 12 of *Importing into a Model with the Same Units* on page 10-4.
3. Select the **Name Group** icon from the Group tool palette.
4. Select the **Result** group.
5. Select **Edit—Transform—Scale**.
6. Specify the appropriate scale factor.
   - To change from inch to metric, multiply by 25.4.
   - To change from metric to inch, divide 1 by 25.4.
7. After the import, assign tools and operations to the geometry.
8. Verify the model using Show Path or Element Data.

Importing and Adopting the Incoming Model File's Units

1. Open the model file that you want to import the CAD file into.
2. Complete steps 2 through 12 of see *Importing into a Model with the Same Units,* on page 10-4.
3. Select **File—Load Job File**.
4. Use the **File Select** button, which opens the **Open** dialog box, and browse for the .jof file that you want to use to load your steps.

   Note When you load a .jof file, the units setting in the existing model is changed to the units of the newly loaded .jof file, regardless of the units set in Utility—System Units.
5. Select the **Name Group** icon from the Group tool palette.
6. Select the **Result** group.
7. Select **Edit—Transform—Scale**.
8. Specify the appropriate scale factor.
   - To change from inch to metric, multiply by 25.4.
   - To change from metric to inch, divide 1 by 25.4.
9. After the import, assign tools and operations to the geometry.
10. Verify the model using Show Path or Element Data.

Exporting a SmartCAM File

Objectives

This lesson shows you how to convert the elements in an existing SmartCAM process model to an ACIS text (.sat), ACIS binary (.sab), DXF (.dxf), DWG (.dwg), or IGES (.igs) file.

Overview

Solids or surfaces in the process model that were preserved from a previous ACIS import are inserted (unchanged) into the exported file.

If the export is for a surface and solids File Type, SmartCAM displays an Export Status dialog box during the export to record information. The Export Status dialog shows the total number of surfaces, the number of the surfaces already processed, and the number of solids found. The Export Status dialog box also provides the path and name of the log file. This Status dialog box is not displayed for CAM Connection File Types.

Note When you export a process model containing solids to an ACIS file, you may get more surfaces than you expect. This can happen when you have deleted or extracted solid-related surfaces because these surfaces are still stored internally in SmartCAM (even though they no longer appear on the screen). You can work around this by using Extract to remove surfaces from their parent solids to export them.

Using Export

Perform these tasks to export a SmartCAM process model to a CAD file:

1. Select **File—Export**. The **Export** dialog box is displayed.

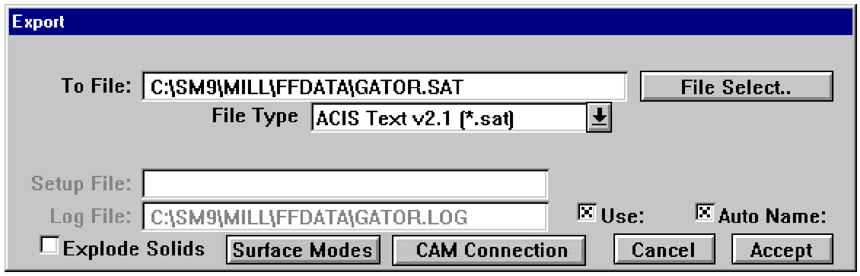


Figure 10-3
Open the Export dialog box.

2. Set the **File Type** selector switch.
3. Enter the path and name of the file to export to the **To File** input field, or use the **File Select** button to specify the path and name of the file.
4. Confirm the name of the setup file used with the file type of the exported file in the **Setup File** input field. Generally, there is a separate setup file for each file type. In most cases, the setup file is automatically inserted when the file type is selected. You can also create a customized setup file. If the selected file type does not require a setup file, this input field is dim.
5. Turn on the **Use** on/off switch if you want to create a log file.
6. Name the log file or have SmartCAM name it for you.
   - To name the log file, enter the name of the log file to create in the **Log File** input field.
   - To have SmartCAM name the log file, turn on the **Auto Name** on/off switch. (You must have turned on the **Use** on/off switch.)
7. Select **Accept** to export the file. When the export is complete, the window displays a *Done* message and the status and export windows are automatically closed.

Creating Boundary Profiles

Objectives

This lesson shows you how to create boundary profiles.

Overview

Set the insertion point and the insert parameters before using the From Solids toolbox. Use action buttons to control which profile is created and where the location of the profile starts.

Note When you insert with a step, SmartCAM offsets to the opposite side of the underlying solid that defines the part; it does not look at the Offset input field.

Using Boundary

Model File: FFMSOLID.PM4

1. Open the model file **FFMSOLID.PM4**.

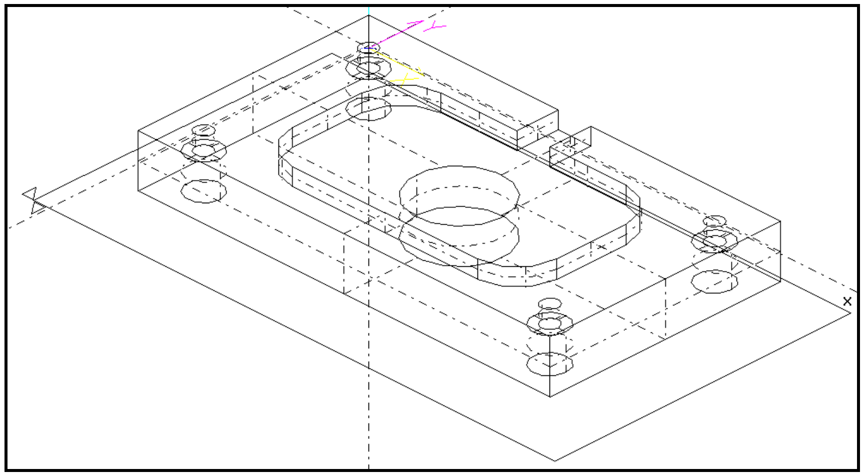


Figure 10-4
Open FFMSOLID.

2. Set the insert location and parameters.
3. Select **Create—From Solid**.
4. Select **Boundary** from the toolbox. The **Boundary** control panel is displayed.

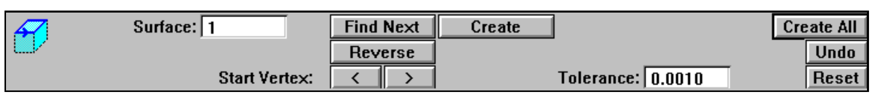
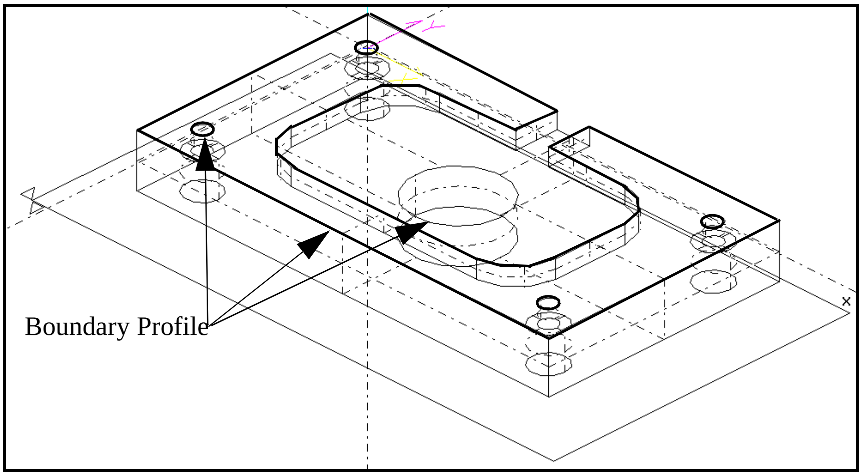


Figure 10-5
Set the values on the Boundary control panel.

5. Select the **Surface** input field.
6. Select the surface to use from the list view or the graphic view.
7. Use these action buttons to control how the profile is created:
   - < moves the start point of the profile to the previous vertex.
   - > moves the start point of the profile to the next vertex.
   - **Reverse** reverses the direction of the profile.
   - **Find Next** identifies the next possible profile on the selected surface.
8. Select the **Create** button to generate the indicated profile, or select **Create All** to create all of the possible boundary profiles on the selected surface.

Figure 10-6
This is the result of creating all boundary profiles.



Creating Slice Profiles

Objectives

This lesson shows you how to create slice profiles.

Overview

Use Slice to create profiles on the intersection of a plane with an ACIS solid body. You can use the resulting profiles to create finish or roughing toolpath for cavities. You can also use slice profiles for dimensioning or developing cross sections. The profiles are placed in the Result group when they are created. This function does not change the existing surfaces or solid body.

Slice profiles are generated at specified Z levels through a solid. You can use the resulting profile to generate finish or roughing toolpath for cavities. Use only solid bodies to generate slice profiles. Do not use surface editing tools to extract the surfaces before creating the profiles.

Using Slice

Model File: FFMSOLID.PM4

1. Open the model file **FFMSOLID.PM4**.

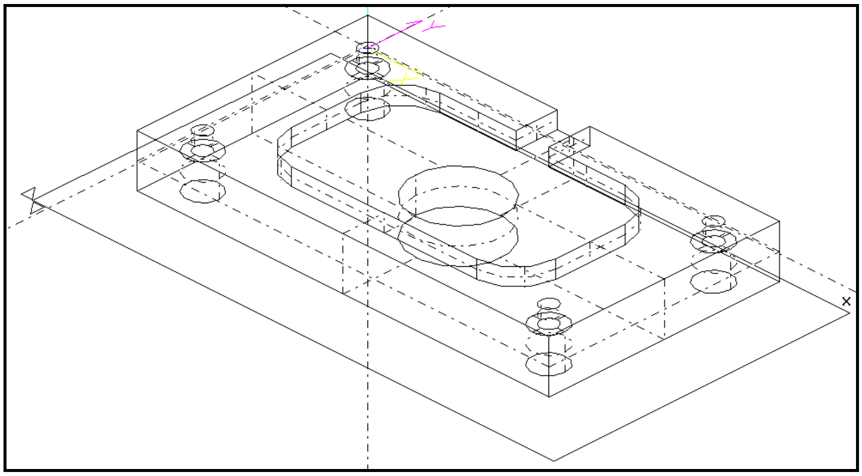


Figure 10-7
Open FFMSOLID.

2. Select **Create—From Solid**.
3. Select **Slice** from the toolbox. The **Slice** control panel is displayed.

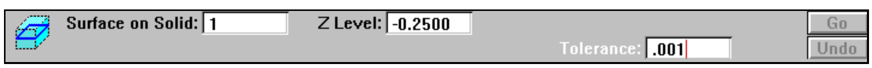


Figure 10-8
Set the values on the Slice control panel.

4. Select the **Surface on Solid** input field.
5. Select the surface to work with.
6. Indicate the **Z level** for the creation of the slice profile.
7. Select the **Go** button.

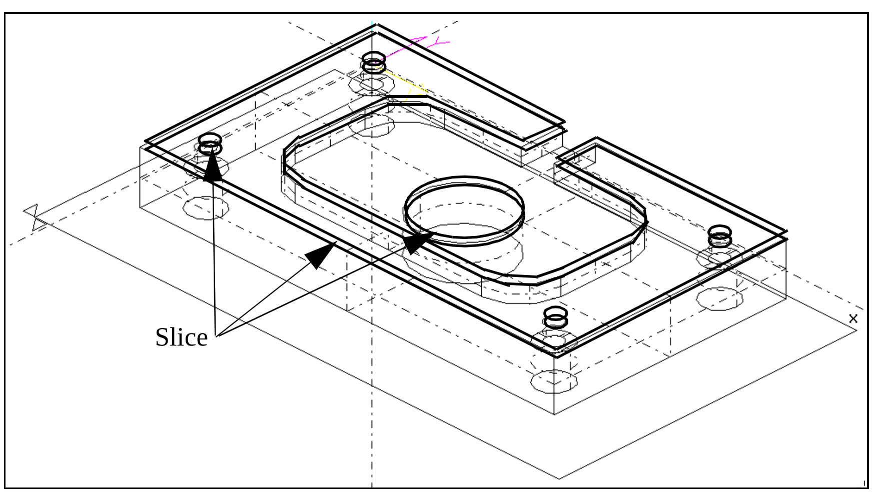


Figure 10-9
This is the result of creating slices.

Creating Hole Elements

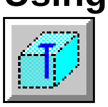
Objectives

This lesson shows you how to create hole elements.

Overview

Use Creating Hole Elements to create hole geometry from the cylinders that are contained in a solid body. The Hole toolbox provides search capabilities to enable you to quickly find the cylinder needed to create a hole. You can base the search on a selected cylinder's diameter, a specific diameter value, split cylinders, or by matching the Z axis of the active tool plane. The depth of the hole is set to the depth of the cylinder used. If the hole element is created with a drill as the active tool, the depth is the full depth of the drill.

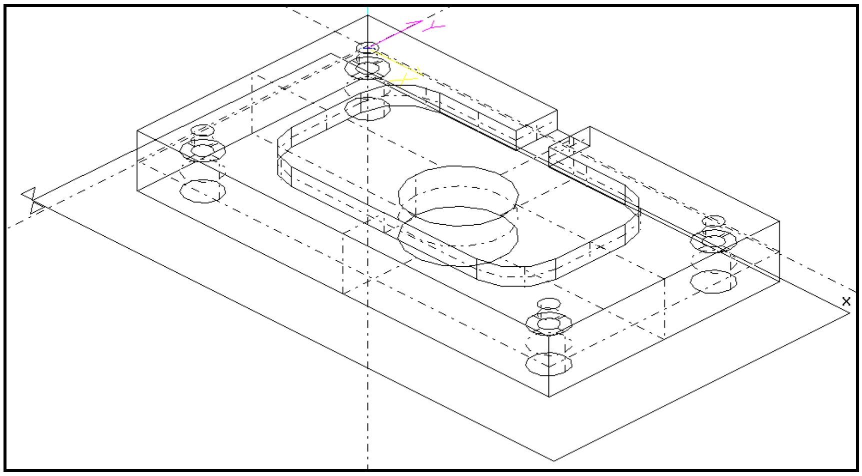
Using Hole



Model File: FFMSOLID.PM4

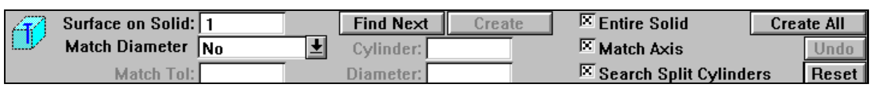
1. Open the model file **FFMSOLID.PM4**.

Figure 10-10
Open FFMSOLID.



2. Select **Create—From Solid.**
3. Select **Hole** from the toolbox. The **Hole** control panel is displayed.

Figure 10-11
Set the values on the Hole control panel.



4. Select the **Surface on Solid** input field.
5. Select the surface to work with in the solid.
6. Use the **Entire Solid** on/off switch to search the entire solid or the selected surface.
7. Use the **Match Axis** on/off switch to limit the search to cylinders that are parallel to the Z axis of the current tool plane (or the current work plane if you are inserting on a layer).
8. Use the **Search Split Cylinders** on/off switch to search for cylinders that are comprised of two or more entities (such as half cylinders).

9. Use these input fields to control the scope of the search:

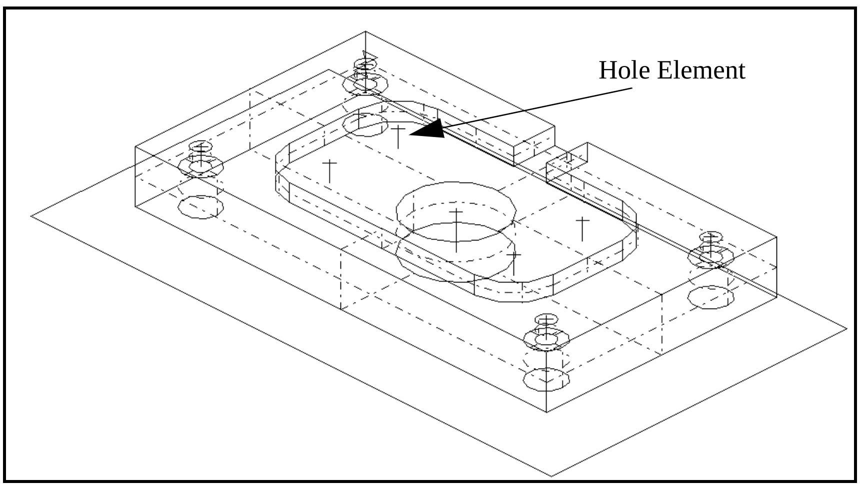
   ■ **Match Diameter**

   - **No** causes the search to ignore the diameters of the cylinders.
   - **By Example** limits the search to cylinders that match the number you indicated in the **Cylinder** input field.
   - **By Value** uses the value that you specified in the **Diameter** input field.

   ■ **Match Tol** indicates the permissible tolerance for the diameter of the cylinder to deviate from the search criteria.

10. Use the **Find Next** button to select the correct cylinder.
11. Use **Create** or **Create All** to create the hole element.

Figure 10-12
This is a result of creating hole elements.



Blending Mesh Representations of Surfaces

Objectives

This lesson shows you how blend meshes that are generated from surfaces.

Overview

You must generate meshes to blend surfaces. After the blend is created, you can convert the meshes back to surfaces.

The blend mesh can be a constant or variable radius. The meshes used to generate the blend must be close enough to allow the blend to touch both meshes.

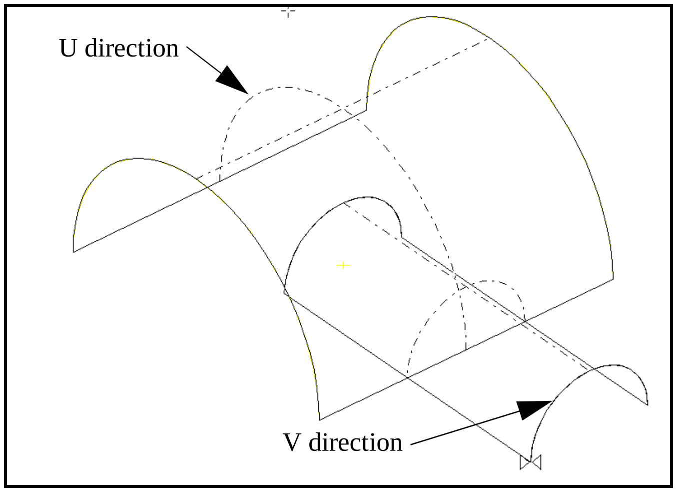
Using Blend

Model File: FFMBLND.PM4

Perform these tasks to blend surfaces:

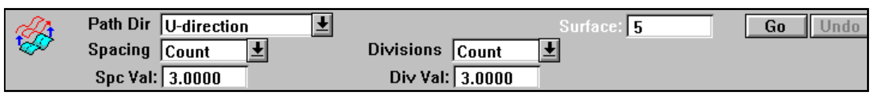
1. Open the model file **`FFMBLND.PM4`**.

Figure 10-13 Open FFMBLND.



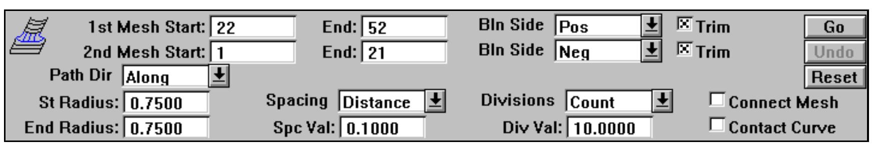
2. Select **Surface Edit—Generate Mesh**, and convert both surfaces to meshes.

Figure 10-14 Set the values on the Generate Mesh control panel.



3. Select **Edit—Mesh Edit—Blend**. The **Blend** control panel is displayed.

Figure 10-15 Set the values on the Blend control panel.



4. Identify the first mesh by setting the **1st Mesh Start** and **End** fields to the first and last elements of the first range of mesh profiles. Set these fields by manually typing, selecting the element in the graphics view, or selecting from the list view.

   Note You might need to use **View—Window** to identify specific elements. This selection also places the location of the start radius for variable radius blends closest to this element.

5. Identify the director profile by setting the **2nd Mesh Start** and **End** fields to the first and last elements of the second range of mesh profiles. Set these fields by manually typing, selecting the element in the graphics view, or selecting from the list view.

6. Set the **Bln Side** selector switches to **Pos** or **Neg**, to indicate on which side of the each range of mesh profiles to position the center of the blend arcs. The red blend arrows should point toward the center of the blend radius.

7. Turn on the **Trim** on/off switch to trim the second mesh along the intersection.

8. Set the **St Radius** input field to the beginning blend radius value. Enter the same start and end radius values to generate a constant radius blend. Use different values to produce a variable radius blend. SmartCAM places this radius at the **1st Mesh Start** element.
9. Set the **End Radius** input field to the ending blend radius value.
10. Set the **Path Dir** selector switch to one of the following:
    - **Cross** creates mesh profiles that run perpendicular to the direction of the blend's arcs.
    - **Along** creates mesh profiles that run parallel to the direction of the blend's arcs.
11. Set the **Spacing** selector switch to one of the following:
    - **Distance** specifies the distance between parallel mesh profiles.
    - **Count** specifies the number of mesh profiles.
12. Set the **Spc Val** input field to the distance or count values.
13. Select the **Go** button.

Extracting Surfaces

Objectives

This lesson shows you how to extract surfaces.

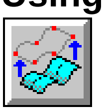
Overview

Use Extract to explode surfaces from solids before using the Transform tools to move, rotate, mirror, or scale a surface. You can also use it to extract the constituent elements from a surface or group of surfaces. If the surface was created in SmartCAM, you can extract the construction geometry and display grid. If the surface is trimmed (imported from an ACIS or IGES file), you can extract base surface edges and trimming curves.

You can keep the original surface as part of the graphic view, hide it on a layer, or delete it as SmartCAM performs the extract.

Select an active group of elements before using Extract.

Using Extract



Model File: FFMEXTSF.PM4

1. Open the model file **FFMEXTSF.PM4**.

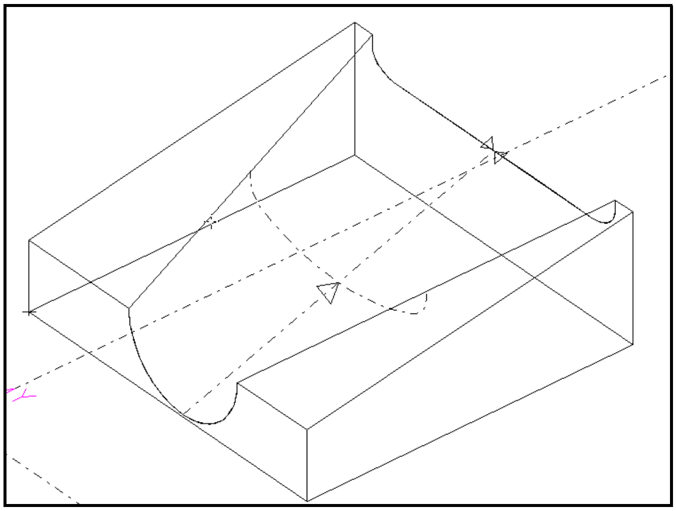


Figure 10-16
Open FFMEXTSF.

2. Group the surfaces that you want to extract.

3. Select **Edit—Surface Edit—Extract**. The **Extract** control panel is displayed.

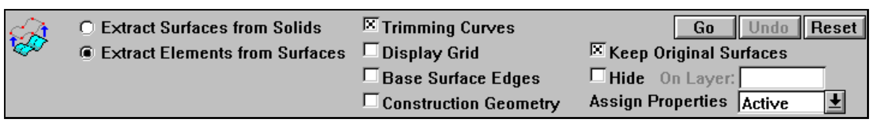


Figure 10-17
Set the values on the Extract control panel.

4. Turn on the **Extract Surfaces from Solids** option button.

5. Select the **Go** button.

6. Group the surfaces that you want to extract.

7. Select **Edit—Surface Edit—Extract**. The **Extract** control panel is displayed.

8. Turn on the **Extract Elements from Surfaces** option switch, and set the following fields on the control panel:

   - Turn on the **Construction geometry** on/off switch to extract the construction geometry from surfaces created with SmartCAM modeling tools. You cannot extract construction geometry from an imported ACIS or IGES surface.

   - Turn on the **Base surface edges** on/off switch to extract the edges of the base surface for trimmed surfaces in the active group. SmartCAM uses the **Display Curve Tolerance** setting in the **Surface Modes** dialog box to create the resulting elements.

   - Turn on the **Trimming curves** on/off switch to extract the trimming curves from the surfaces in the active group.

- Turn on the **Display grid** on/off switch to extract the display grid curves in the U and V directions. SmartCAM will not extract the center U and center V curves.
- Turn on the **Keep Original Surfaces** on/off switch to keep the original surfaces in addition to the extracted elements. Toggle this switch off to delete the original surfaces when you extract elements. If you extract elements from an imported trimmed surface, it may not be possible to re-create the surface exactly from the extracted elements. In this case, keep the original surface.
- Turn on the **Hide** on/off switch to move the original surfaces to another layer and hide them when you extract elements. Toggle this switch off to leave the original elements visible in the graphic view. The **Hide** switch is dimmed if the **Keep Original Surfaces** switch is on.

9. Select the **Go** button.

Points to Remember

- ❑ When you create elements from solids and insert with a step, SmartCAM offsets to the opposite side of the underlying solid that defines the part. It does not look at the Offset input field.
- ❑ Creating slice profiles does not change the existing surfaces or solid body.
- ❑ Do not use surface editing tools to extract the surfaces before creating the slice profiles.
- ❑ The depth of the hole is set to the depth of the cylinder used unless the hole element is created with a drill as the active tool, then the depth is the full depth of the drill.
- ❑ The meshes that you blend must be close enough to allow the blend to touch both meshes.
- ❑ Use Extract to explode surfaces from solids, extract the constituent elements from a surface or group of surfaces, and extract the construction geometry and display grid.

11

Working with Surfaces

Objectives

This unit shows you how to perform these tasks:

- Work with surfaces.
- View a shaded part.
- Change surface properties.
- Work with surface and display modes.

Overview

There are many options added to standard Production Milling forms to enable you to work with surfaces in Advanced Milling.

Viewing a Shaded Surface Part

Objectives

This lesson shows you how to view a shaded surface part by rendering the image.

Overview

Use Render Image to view a shaded representation of the surface elements in a process model. You can view all surfaces or a specified group of surfaces. Render Image does not display wireframe elements, material stock, fixtures, or hidden elements or layers.

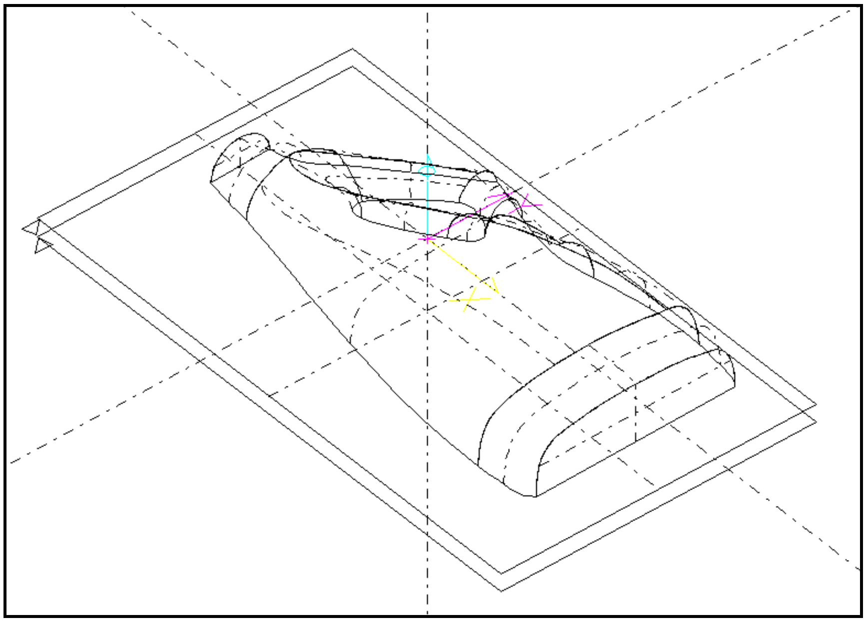
Rendering an Image

Model File: FFMBOTIN.PM4

Perform these tasks to render an image:

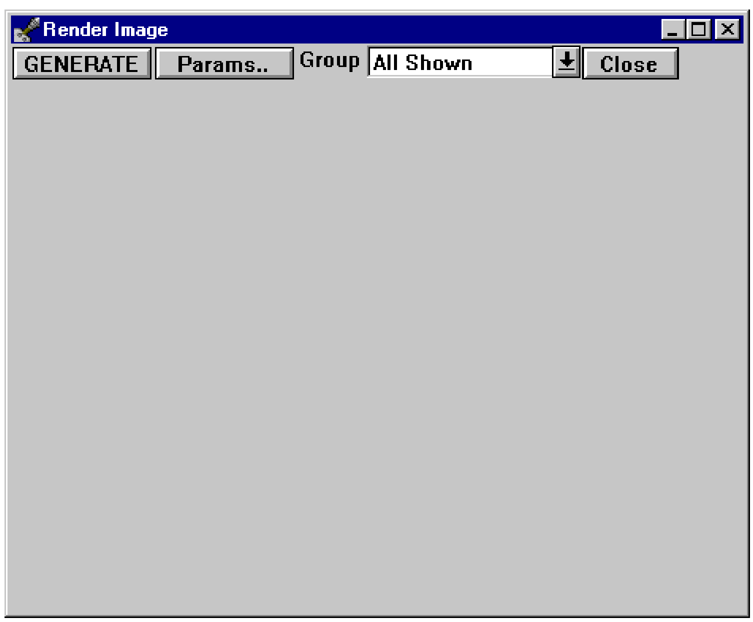
1. Open the model file **`FFMBOTIN.PM4`**.

Figure 11-1
Open FFMBOTIN.



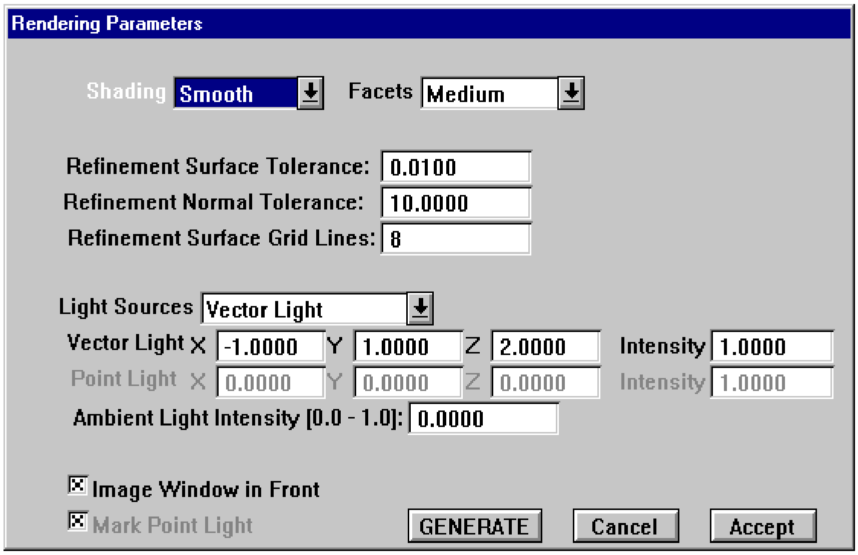
2. Select **View—Render Image**. The **Render Image** dialog box is displayed.

Figure 11-2
Select a button on the Render Image dialog box.



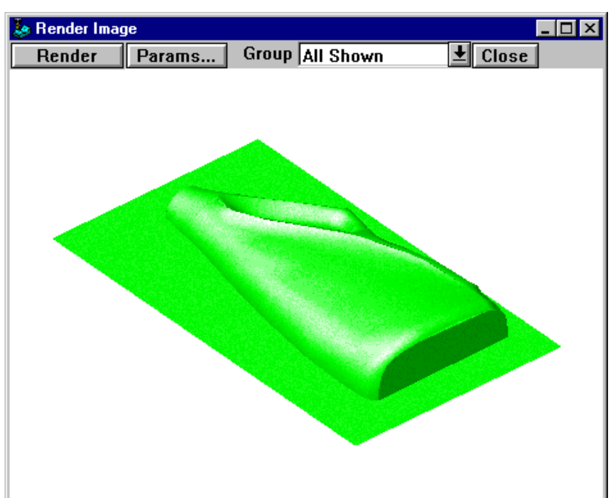
3. Select the **Params** button. The **Rendering Parameters** dialog box is displayed.

Figure 11-3
Set the values on the Rendering Parameters dialog box.



4. Set **Shading** to **Smooth**.
5. Set **Facets** to the desired smoothness.
6. Select the **Light Sources**: **Vector Light, Point Light**, or **Vector and Point**.
7. Enter the coordinates for the light source that you chose and the corresponding intensity values. If you selected **Point Light**, enable the **Mark Point Light** check box and mark its location on the rendered image.
8. Set the **Ambient Light Intensity**.
9. Select the **Accept** button.
10. Select the **GENERATE** button from the **Render Image** dialog box.

Figure 11-4
This is the result of rendering the image.



Changing the Properties of Surfaces

Objectives

This lesson shows you how to change the properties of surfaces.

Overview

Use Property Chg—Surfaces to update modeling properties associated with existing elements and to change settings that control the visual display of surfaces in the active group. Surface grid settings do not affect the internal surface structure of a surface, only the visual representation. You can also set these values by using Surface Modes when you create or import new surfaces. Use the Display Modes dialog box to control the color of the surface display grids.

Note You must have an active group of elements before using Property Change modeling tools. Use the Group Arrow icon or the Group tool palette to select an active group.

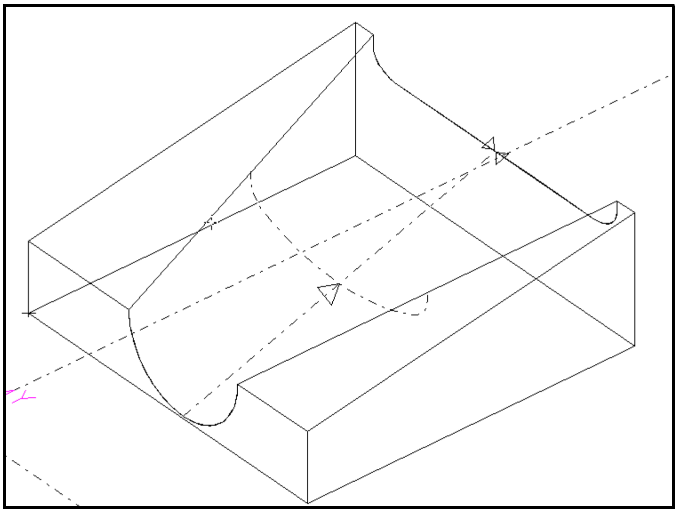
Using Property Change

Model File: FFMEXTSF.PM4

Perform these tasks to change surface properties:

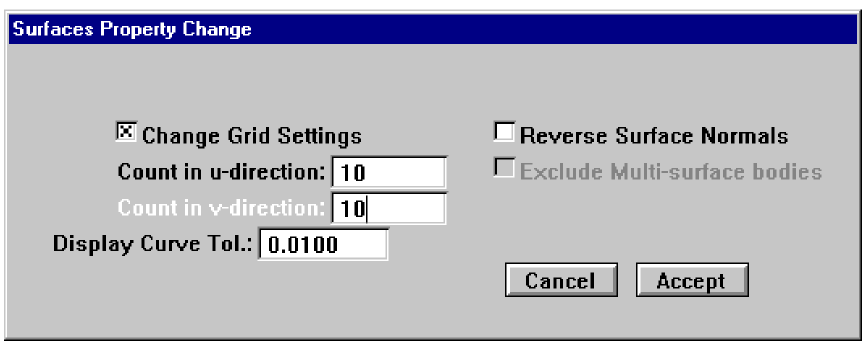
1. Open the model file **FFMEXTSF.PM4**.

Figure 11-5
Open FFMEXTSF.



2. Group the elements whose properties you want to change.
3. Select **Edit—Property Chg—Surfaces**. The **Surfaces Property Change** dialog box is displayed.

Figure 11-6
Set the values on the Surfaces Property Change dialog box.



4. Turn on the **Change Grid Settings** on/off switch to activate the grid settings input fields.
5. Enter the number of isoparametric lines for SmartCAM to use for the display grid in the surface's U direction in **Count in u-direction**. The graphic view display of the surface shows the insert markers in the U direction.
6. Enter the number of isoparametric lines for SmartCAM to use for the display grid in the surface's V direction in **Count in v-direction**.
7. Enter the maximum that the display grid curves can deviate from the exact surface in **Display Curve Tol**. (Tolerance). The tolerance is calculated from the chord height of each grid curve polyline segment.

8. Turn on the **Reverse Surface Normals** on/off switch to reverse the direction of the surface normals for surface elements in the active group.

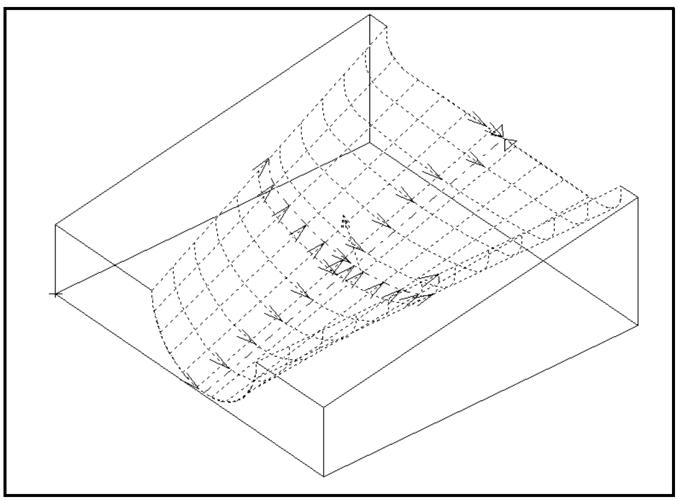
   Note Surface normals are used by surface machining operations to indicate the orientation of the surface. To display surface normals, set the Count in u-direction and Count in v-direction to a number greater than 0.

9. Determine whether to **Exclude Multi-surface bodies**:

   - **On** excludes any surface that is part of a multiple-surface body (such as a solid) from the Reverse Surface Normals action.
   - **Off** includes all surfaces in the active group in the Reverse Surface Normals action. If there are surfaces in the active group that are part of a multi-surface body, the surface normals for all surfaces in the body will change, even those that are not in the active group.

10. Select **Accept** to save your selections.

Figure 11-7
This is a result of changing surface properties.



Working with Surface and Display Modes

Objectives

This lesson will show you how to perform these tasks:

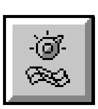
- Set surface modes.
- Set display modes.

Overview

Use Surface Modes to control settings that SmartCAM uses for creating mesh toolpath and for displaying surfaces.

Use Display Modes to control how SmartCAM displays your model in the graphic view.

Setting Surface Modes



The surface element settings affect only the visual representation of surfaces in SmartCAM FreeForm Machining, not the internal surface structure. You can also set these values for editing existing surfaces using Edit—Property Chg—Surfaces.

Perform these tasks to change surface modes:

1. Select **Utility—Surface Modes**. The **Surface Modes** dialog box is displayed.

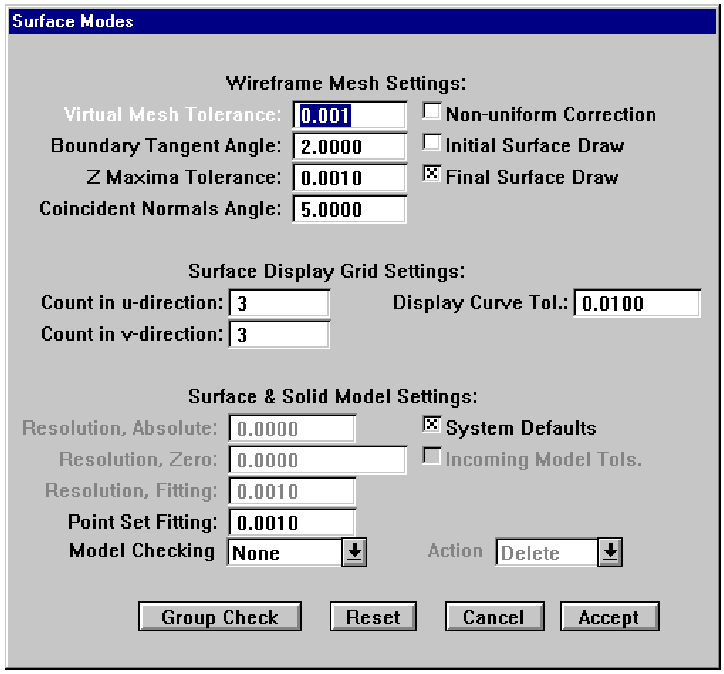


Figure 11-8
Set the values on the Surface Modes dialog box.

2. Set the **Virtual Mesh Tolerance** input field by entering the tolerance band to use when converting primitive elements or polylines of uneven count into acceptable polylines.

3. Set the **Boundary Tangent Angle** input field by entering the acceptable out-of-tangency condition that can exist on surface-defining profiles. The recommended setting is between 0 and 3 degrees.

4. Set the **Z Maxima Tolerance** input field by entering the allowable distance that can be removed below the maximum point of a convex toolpath.

5. Set the **Coincident Normals Angle** input field by entering the size of a conical tolerance band that tests for a change in the relationship between the tool normal and surface normal. Advanced Milling makes this comparison to determine the direction for the tool during offset calculations.

6. Turn on the **Non-uniform Correction** on/off switch to use alternate settings to determine gouge elements. This function is not used in normal operation.

7. Turn on the **Initial Surface Draw** on/off switch to display the initial surface (non-offset) mesh before generating the final offset surface toolpath. When it is off, only the final surface mesh is drawn.

   Note Use the Initial Surface Draw option to check Spacing and Divisions settings. If they do not appear acceptable when SmartCAM displays the initial mesh, press ESC to stop the processing and change the settings.

8. Turn on the **Final Surface Draw** on/off switch to draw the final surface mesh and update the list view. Turn it off to speed up execution of macros that create multiple meshes.

9. Set the **Count in u-direction** input field by entering the number of isoparametric lines for SmartCAM to use for the display grid in the surface's U-direction. The graphic view display of the surface shows the insert markers in the U-direction.

10. Set the **Count in v-direction** input field by entering the number of isoparametric lines for SmartCAM to use for the display grid in the surface's V-direction.

    Note If the count for the isoparametric lines is set to 0 for the U-direction and V-direction, surface normal arrows do not display.

11. Set the **Display Curve Tol**. (tolerance) input field by entering the maximum amount that the display grid curves can deviate from the exact surface. The tolerance is calculated from the chord height of each grid curve polyline segment.

12. Set these Surface and Model Settings only when you turn off the **System Defaults** on/off switch (The **Model Checking** selector switch to specify the level of checking to use when you create, import, or machine surfaces and surface edges.)

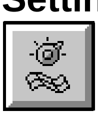
    Note These Surface and Solid Model Settings should be adjusted when you change model units.

    - Set the **Resolution, Absolute** input field to the minimum amount that two points can be apart without SmartCAM considering them the same point.
    - Set the **Resolution, Zero** input field to the maximum amount that a number can be away from zero without SmartCAM considering that number to be zero.
    - Set the **Resolution, Fitting** input field to the maximum distance that a point can be from a curve and a curve can be from a surface for interpolation.
    - Turn on the **Incoming Modeling Tols** on/off switch to set the active model tolerances to the values in an incoming mode; otherwise, turn it off.

13. Set the **Point Set Fitting** input field by entering the tolerance to use for fitting NURBS curves and surfaces through a set of points. The tolerance is calculated by measuring the distance from one of the points to the exact point on the surface along a normal vector. This setting is used for creating surfaces with the Draft Surface, Coons Surface, and From Mesh tools.

14. Select the **Accept** button to save the settings for the next time you use a mesh modeling tool.

Setting Display Modes



Perform these tasks to work with display modes:

1. Select **Utility—Display Modes**. The **Display Modes** dialog box is displayed.

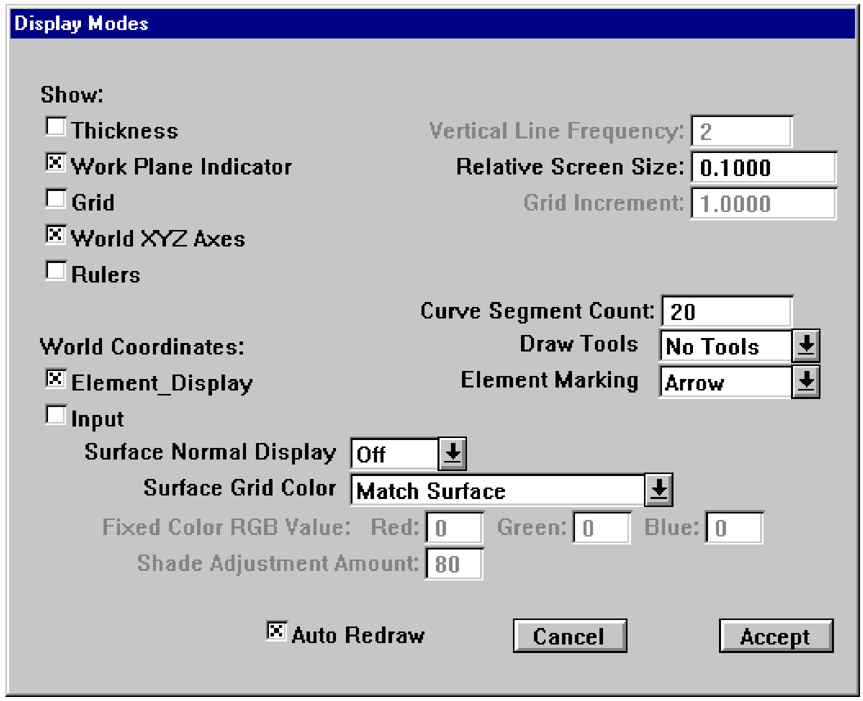


Figure 11-9
Set the values on the Display Modes dialog box.

2. Select the **Surface Normal Display** selector switch to choose the display of surface normal vector indicators. Surface normal indicators appear as small arrows near the center of each surface element, indicating the orientation of the surface. To display Surface Normals, the Display Grid Setting counts in the Surface Modes dialog box must be set to a number greater than 0. Select from: **Off**, **All**, **Group**.

   Note You can temporarily display the surface normal vector indicators for all surfaces by selecting the Normals Draw icon. The indicators disappear when you redraw the graphic view. Selecting the icon does not change the Display Modes settings.

3. Select the **Surface Grid Color** selector switch, and select the method to use for setting the color of the display grid for SmartCAM Advanced Milling surfaces: **Match Surface**, **Shade Darker**, **Shade Lighter**, **Fixed Color RGB**, **Fixed Color Dark**, or **Fixed Color Light**.

4. Set the **Fixed Color RGB Value** input field to create the color for the fixed-color RGB display of Advanced Milling surfaces. For information about setting RGB values, see the *SmartCAM Customization Manual*.

5. Set the **Shade Adjustment Amount** input field to adjust the color shade when using the Shade Darker or Shade Lighter option for displaying SmartCAM Advanced Milling surfaces. This value is added to each of the fixed color RGB values if you are using the Shade Lighter option; it is subtracted from each of the fixed color RGB values if you are using the Shade Darker option. Valid values are between 1 and 100.

Points to Remember

- ❑ Render Image does not display wireframe elements, material stock, fixtures, or hidden elements or layers.
- ❑ Surface grid settings do not affect the internal surface structure of a surface, only the visual representation.
- ❑ You must have an active group of elements before using Property Change modeling tools.
- ❑ The surface element settings affect only the visual representation of surfaces in SmartCAM Advanced Production Milling, not the internal surface structure.

12 Using Planar Cuts and Cavity Roughing

Objectives

This unit shows you how to perform these tasks:

- Prepare to perform cavity roughing.
- Create planar cuts across meshes.
- Rough a cavity using planar cuts.

Overview

Use Planar Cuts to generate the planar profiles necessary for a cavity-roughing operation. The orientation of the planar cuts is based on the active work plane, with the spacing set by the Spacing and Planar Z settings. The planar cuts then must be cleaned up using Geo Edit tools such as Split and Trim/Extend. You need to do cleanup where planar cuts from adjacent meshes overlap or intersect. Then, join the planar cuts together to create one large planar mesh across the meshes. Use Cavity Rough to generate roughing cuts using the large planar mesh.

Using the Mesh Toolbox

Use the Mesh toolbox to create surface mesh profiles. When assigned to a step, the mesh profiles represent toolpath to create the surface. When assigned to a layer, the mesh profiles represent a non-offset representation of the surface. Identifying the defining profiles for the surface enables you to select the correct modeling tool to use. Before you generate a mesh, examine the profiles to ensure that they meet the requirements for the type of surface they correspond to.

Figure 12-1 *The Mesh toolbox contains tools to create mesh profiles.*

Preparing to Perform Cavity Roughing

Model File: A3DCAVTY.PM4

Before you begin to rough a cavity, it is important to set up your insert options correctly. Perform these tasks to set your insert options on the Insert property bar:

1. Open the model file **`A3DCAVTY.PM4`**.

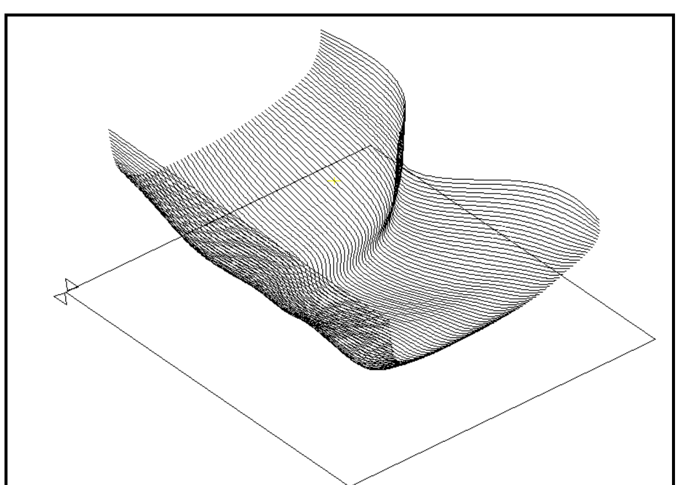
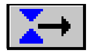
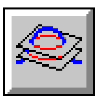


Figure 12-2 *Open A3DCAVTY.*

2. Set the **Before** icon .
3. Select the **Step** icon .
4. Specify the insert location using one of these methods:
   - Select #20 from the list view.
   - Select the magenta finish mesh from the graphic view.
5. Select the **With Step** icon .
6. Insert with **step #22**.
7. Set **C** (Clear) to **.75** [**19.05**].
8. Set **P** (Profile Top) to **.25** [**6.35**].

Creating Planar Cuts across Meshes



Use Planar Cuts to create planar profiles parallel to the active work plane at the Z levels you specify. Planar Cuts operate on a selected range of existing mesh profiles. Create Planar cuts across each of the meshes in the model using the same settings in the Planar Cuts control panel.

Sometimes it is easier to generate and clean up the planar cuts if they are generated from untrimmed meshes.

1. Keep the model file **`A3DCAVTY.PM4`** open.
2. Select **Create—Mesh**, and select **Planar Cuts** from the toolbox. The **Planar Cuts** control panel is displayed.

Figure 12-3 *Set the values on the Planar Cuts control panel.*

3. Set the following fields on the control panel:
   - Select the **Mesh Start** input field, and select the starting element of the range of mesh profiles from the list view.
   - Select the **Mesh End** input field, and select the ending element of the range of mesh profiles from the list view.
   - Set the **Planar Z** input field to **-.125** [**-3.175**], which is the level at which the planar profiles should start (based on the active work plane's Z axis).
   - Set the **Spacing** input field to the distance between the planar profiles, which is .**325** [**8.255**].
   - Set the **Bgrnd Refine** selector switch to **Count**.
   - Set the **Bgrnd Val** input field to **40**.
4. Set the **Offset** selector switch to the direction of the toolpath offset, as follows:

- **Pos** offsets the toolpath to the positive side.
- **Zero** results in no offset.
- **Neg** offsets the toolpath to the negative side. Use the offset arrows to determine the correct side.

5. Set the **Fin Allow** input field to **.03** [**.762**].

Note If a layer is active, you must set **Fin Amt** to a finish amount to offset the layer from the defining profiles. Enter the amount of space between the surface profiles and the surface. If a tool is active, the finish amount is added to the offset of the tool.

6. Select the **Go** button.

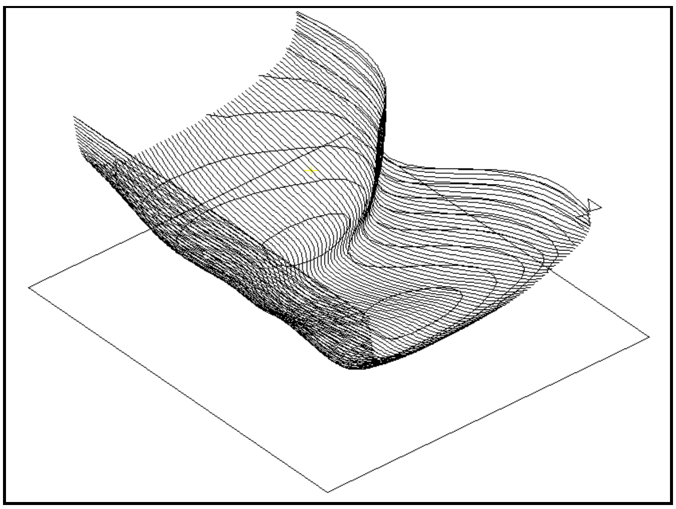


Figure 12-4
This is the result of planar cuts.

Using the Rough Toolbox

Use the Rough toolbox to create roughing toolpath to remove a lot of material using multiple cutting moves. The material to be removed is defined by the XY position of the finished profile, level, and profile top (Z) position.

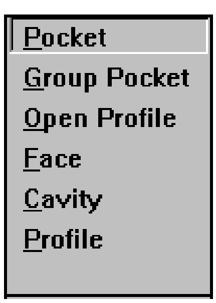
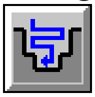


Figure 12-5
Specify tools from the Rough toolbox.

Roughing a Cavity



Use Cavity to rough an opening in preparation for mesh profile finishing cuts created by Planar Cuts. One roughing pass is made for each planar cut level. To rough a cavity, perform these steps:

1. Keep the model file **A3DCAVTY.PM4** open.
2. Group the planar cut profiles that you just created.
3. Select **Process—Rough—Cavity**. The **Cavity** control panel is displayed.

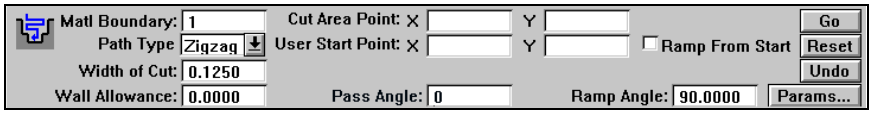
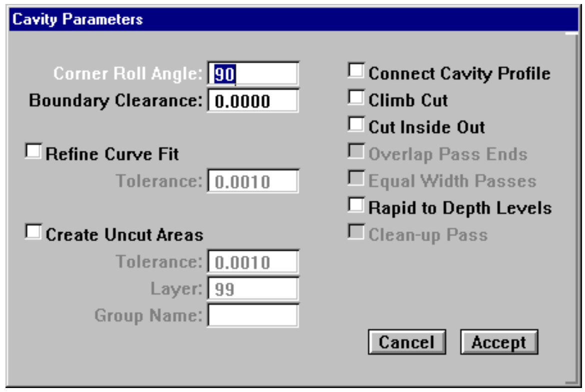


Figure 12-6
Set the values on the Cavity control panel.

4. Select **Matl Boundary**, and select the green stock geometry. Each planar cut is extended to meet the material boundary and form closed areas to machine. The material boundary must be a closed profile.
5. Set the following fields on the Cavity control panel as necessary:
   - Set **Cut Area Point** to the specific location within the boundary containing the deepest Z level for cutting to begin. This location indicates the inside of the cavity. This is best selected from a top view by digitizing a point in one of the two closed profiles at the bottom of the cavity.
   - Set **User Start Point** to the coordinates for the beginning of the cut. (This is best selected from a top view.) If you enter a start point, SmartCAM will create a move directly from the user start point location to the automatic start point of the toolpath.
   - Set **Path Type** to **Spiral** to create a spiral roughing operation.
   - Set **Width of Cut** to **.4** [**10.16**], which is the maximum stepover distance for each pass (generally less than the width of the cutting tool). The default is half the active tool diameter.
   - Set the **Wall Allowance** input field to the amount of material to leave on the walls of the islands and pockets.
   - Set **Pass Angle** to the angular orientation of roughing passes for linear and zigzag cutting.
   - Set **Ramp Angle** to the angle in degrees for tool entry if neither **Ramp From Start** nor **User Start Point** is to be used. A 90-degree angle results in a vertical plunge.
   - Turn on the **Ramp From Start** on/off switch to create ramp moves directly from the user start point location to the automatic start point of the toolpath. Otherwise, turn it off to create a vertical plunge at the user start point.

6. Select the **Params...** button. The **Cavity Parameters** dialog box is displayed.

__Figure 12-7__
Set the values on the Cavity Parameters dialog box.



7. Set the following fields on the Cavity Parameters dialog box as necessary:

■ Set **Corner Roll Angle** to the determining angle for corner rolling.

■ Set **Boundary Clearance** to **.375** [**9.525**], which is the distance the tool is to cut beyond the material boundary. If you enter **0**, the center of the tool will stop at the material boundary profile.

■ Turn on the **Refine Curve Fit** on/off switch to remove colinear points from the part profile. This is useful when the part profile is a spline. Otherwise, turn it off.

■ Turn on the **Create Uncut Areas** on/off switch to create geometry for areas that the assigned roughing tool cannot cut. You can use this geometry to create a toolpath to finish the roughing with a different tool.

■ Turn on the **Connect Cavity Profile** on/off switch.

Note It is important to turn this on if the roughing tool is the same as the tool used for the planar cuts. This resequences each roughing pass to occur before the planar cut at the same level and connects the roughing passes into a single, continuous profile.

■ Turn on the **Climb Cut** on/off switch to create climb cuts; otherwise, turn it off for conventional cuts.

■ Turn on the **Cut Inside Out** on/off switch to begin the roughing operation in the center of the area and progressively step over toward the outside boundary.

■ Turn on the **Overlap Pass Ends** on/off switch if you want the cutter to follow the profile up to the previous level after each cut before retracting.

■ Turn on the **Equal Width Passes** on/off switch to make the width of all cutting passes the same.

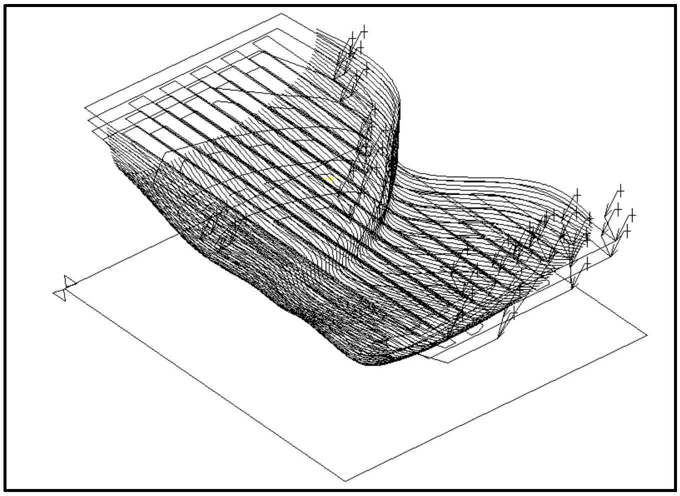
■ Turn on the **Rapid to Depth Levels** on/off switch to rapid-move the tool to the levels of the previous roughing passes.

- Turn on the **Clean-up Pass** on/off switch to have the tool perform a pass around the perimeter of the pocket.

8. Select **Accept** for SmartCAM to use the values you set and to close the **Cavity Parameters** dialog box.

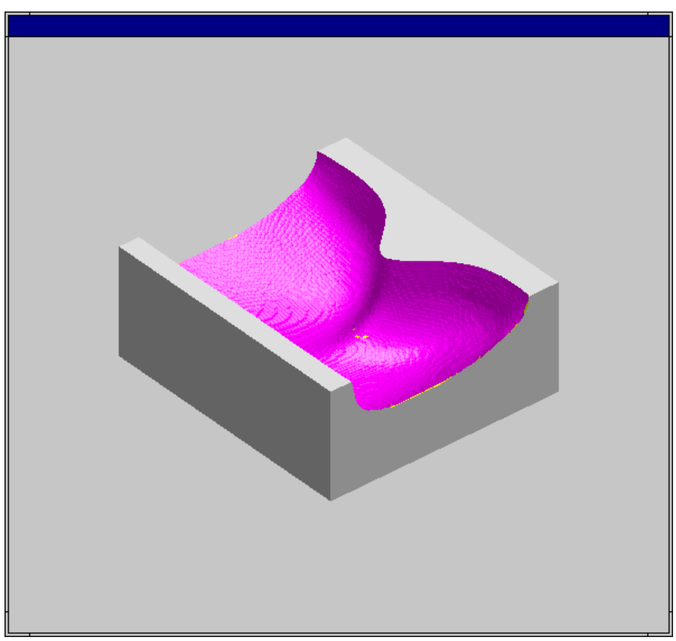
9. Select the **Go** button. The operation is performed.

Figure 12-8
This is the result of cavity roughing.



10. Use Show Path or Show Cut to verify the toolpath.

Figure 12-9
This is the image as it appears in Show Cut.



Points to Remember

- Before you generate a mesh, examine the profiles to ensure that they meet the requirements for the type of surface they correspond to.
- Sometimes it is easier to generate and clean up the planar cuts if they are generated from untrimmed meshes.
- Before you perform cavity rough operations using planar cuts, you must clean up and chain your planar cuts.
- Each level of planar cuts should be a series of profiles. (There should be no gaps, all curves should be numbered sequentially, and all curves should point in the same direction.)
- One roughing pass is made for each planar cut level.

Self-Test

Directions

Test your understanding of the concepts and procedures in this section by answering the following questions. The answers for each self-test are in *Appendix A* of this manual.

______________ **1. Selecting the first and last element of the profile curve selects only those elements and the most central element.**

a) true
b) false

______________ **2. When polylines are used as profile curves, the boundary tangent angle is used.**

a) true
b) false

______________ **3. Can non-tangent profiles be edited to become tangent?**

a) never
b) rarely
c) usually
d) always

______________ **4. How many gaps can be present in a profile curve?**

a) 0
b) 1
c) 2
d) as many as you want

_______________ **5. How many of the profile's elements should connect to make a surface mesh profile?**

a) 0
b) 1
c) 2
d) as many as you want

_______________ **6. For translated meshes, is the director curve parallel to the generator curve?**

a) never
b) rarely
c) usually
d) always

_______________ **7. When you generate a coons patch mesh, how many sharp corners can you have?**

a) 0
b) 1
c) 2
d) as many as you want

_______________ **8. When you extend existing mesh profiles, do the properties and sequence for the selected range change?**

a) The properties remain the same, but the sequence changes.
b) The properties change, but the sequence remains the same.
c) Both the properties and the sequence remain the same.
d) Both the properties and the sequence change.

_______________ **9. When you insert a solid element with a step, how does SmartCAM determine the offset?**

a) SmartCAM offsets to the same side of the underlying solid that defines the part.
b) SmartCAM looks at the Offset input field.
c) SmartCAM offsets to the opposite side of the underlying solid that defines the part.
d) SmartCAM uses a default value of .001, which you cannot control.

_______________ **10. What does creating slice profiles change?**

a) existing surfaces
b) solid bodies
c) both a and b
d) neither a nor b

_______________ **11. Use surface editing tools to extract the surfaces:**

a) before creating profiles
b) after creating profiles
c) half-way through creating profiles
d) never

______________ **12. If a hole element is created with a drill as the active tool, how is the depth determined?**

a) The full length of the drill becomes the hole depth.
b) The cylinder depth becomes the hole depth.
c) The stock box depth becomes the hole depth.
d) The tolerance value is added to the stock box depth.

______________ **13. How close should the meshes that you blend be?**

a) The meshes should overlap each other completely.
b) The meshes should be tangent to each other.
c) The meshes should almost touch each other.
d) The meshes should be intersected by the radius of the blend.

______________ **14. Surface grid settings affect the visual representation.**

a) true
b) false

______________ **15. Is it easier to generate and clean up the planar cuts if they are generated from untrimmed meshes?**

a) never
b) rarely
c) sometimes
d) usually

______________ **16. How many roughing passes are made for each planar cut level?**

a) 0
b) 1
c) 2
d) as many as you want

Challenge Project

Directions

Use the part print to create meshes, planar cuts on those meshes, and roughing cuts.

Part Print

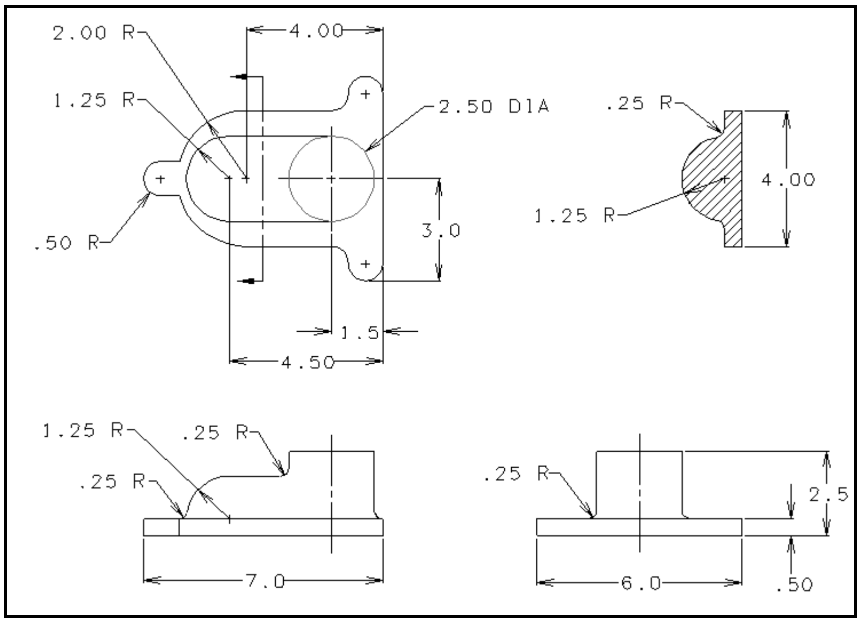


Figure 12-10
Create meshes, finish cuts, and roughing cuts.

Exploring SmartCAM FreeForm Machining

Exploring SmartCAM FreeForm Machining

Welcome

FreeForm machining is the most powerful milling solution in the SmartCAM family of applications. It provides multiple-surface and solids-based machining for a full range of parts—from 2-axis work to complex 3-axis molds, dies, and prototypes. It is the fastest gouge-free machining system available. Like all SmartCAM applications, it does more than reduce your programming time. It helps you improve your machining processes and move your products to market faster.

Units in This Exploration

- Using Profile Curves to Create Surfaces
- Editing Surfaces
- Machining Surfaces

13

Using Profile Curves to Create Surfaces

Objectives

This unit shows you how to perform these tasks:

- Use profiles.
- Create surfaces from profiles.

Overview

Profile curves can be a combination of lines, splines, polylines, or arcs that point in the same direction and are sequential. You can use profile curves to create surface geometry.

Using Profiles

Objectives

This lesson shows you how to perform these tasks:

- Check profile curves for tangency.
- Check the types of elements in profile curves.
- Check for gaps in profile curves.
- Group profile curves.

Overview

Profile curves can be a combination of lines, splines, polylines, or arcs that point in the same direction and are sequential. When you select the first and last element of the profile curve, all the elements between these curves are selected as part of the profile curve. Use profile curves to create surface geometry.

Checking Profile Curves for Tangency

If the elements in the profile are not within 3.14 degrees of tangency, use Edit—Geo Edit—Blend or use Edit—Geo Edit—Split to blend or split the profile at the non-tangent point. You can use the minimum blend radius of .001 to smooth the profiles.

Figure 13-1 *Notice the results of non-tangent profile curves.*

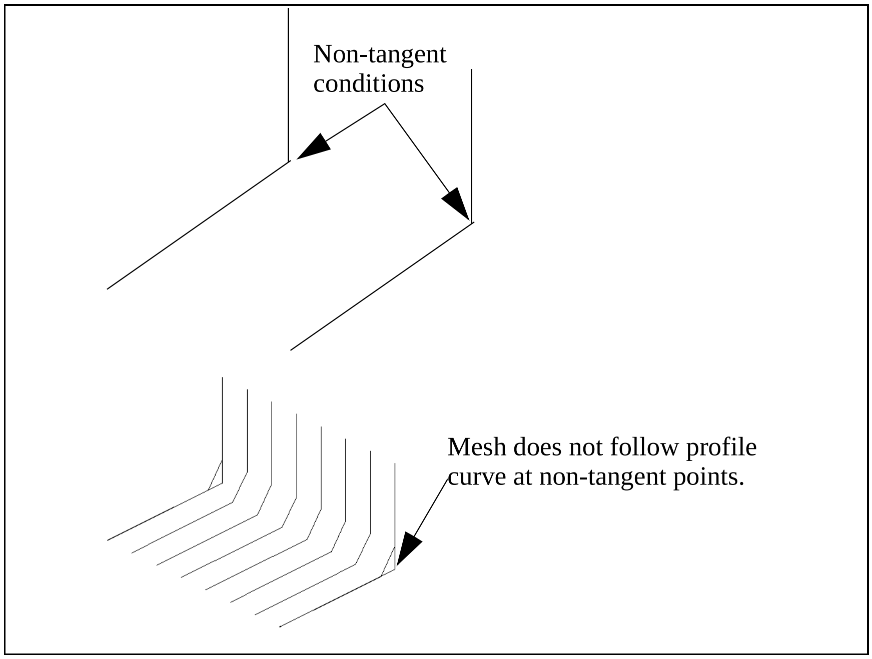
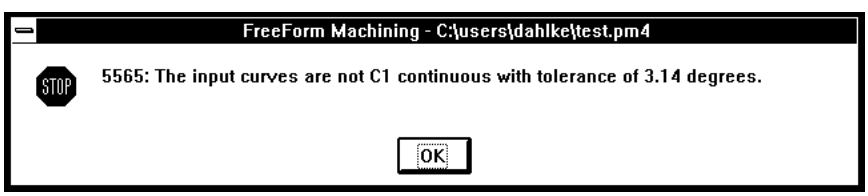
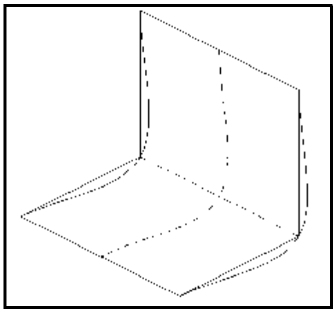


Figure 13-2 *Non-tangent profiles that are constructed from lines and arcs will cause this error message.*

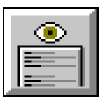


If polylines are used as profile curves, the surface will be constructed. The surface, however, does not exactly follow the polyline; it passes through the points of the polyline.

Figure 13-3
Use polylines as a profile curve.



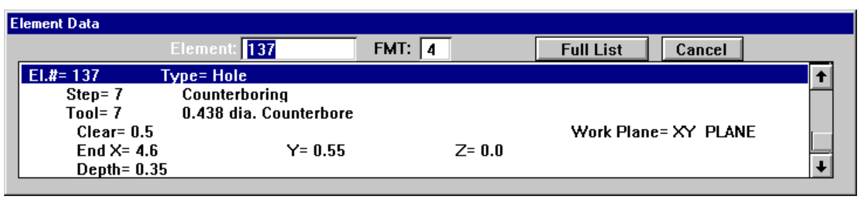
Checking the Type of Elements in Profile Curves



Perform these tasks to check the type of elements in profile curves:

1. Select **Utility—Element Data**. The **Element Data** dialog box is displayed.

Figure 13-4
Check for element types.



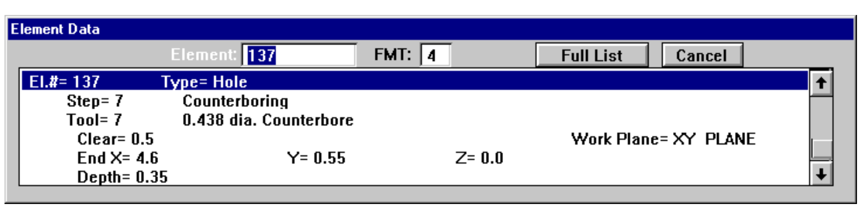
2. Select the element you want data for. Primitive curves such as lines and arcs work well. Polylines also work well, but the surface does not exactly match the shape of the polyline.

Checking for Gaps in Profile Curves

Perform these tasks to check for gaps in profile curves.

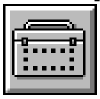
1. Select **Utility—Element Data**. The **Element Data** dialog box is displayed.

Figure 13-5
Check for gaps.



2. Select the element that you want data about. Even the smallest gaps can cause the surface to not be created. If small gaps are present, build two separate meshes, edit the two profile curves to make a continuous profile, or check the start and end points of the elements to ensure that they are the same.

Grouping Profile Curves



Many surfaces require that the profile curves be grouped. Use the Group toolbox to group profile curves.

Points to Remember

- If the elements in the profile are not within 3.14 degrees of tangency, use Edit—Geo Edit—Blend or use Edit—Geo Edit—Split to blend or split the profile at the non-tangent point.
- If polylines are used as profile curves, the surface passes through the points of the polyline.
- Many surface creating methods require that the profile curves be grouped.

Creating Surfaces

Objectives

This lesson shows you how to perform these tasks:

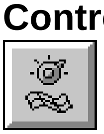
- Control surface construction.
- Create a spun surface from a profile curve.
- Create a translated surface from two profile curves.
- Create a draft surface from two profile curves.
- Create a ruled surface from two profile curves.
- Create a lofted surface from multiple profile curves.
- Create a coons patch surface from two profile curves and cross-section profiles.
- Create a surface from mesh profiles.
- Shade a surfaced part.

Overview

By using different combinations of profile curves, you can create a variety of different surfaces.

Note You must insert on a layer (not with a step) to create a surface.

Controlling Surface Construction



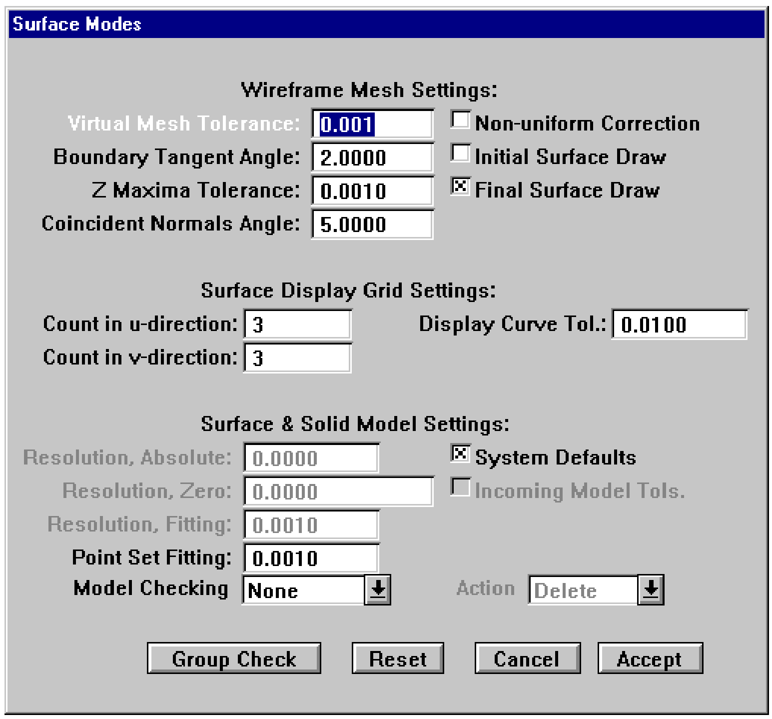
Use Surface Modes to control settings used for creating and editing a mesh toolpath, and for displaying surfaces. The Count in U-direction and Count in V-direction settings control how many display lines are shown per surface. Higher values display more lines but take longer to redraw.

All of the surface element settings except for Point Set Fitting affect only the visual representation of surfaces in FreeForm Machining, not the internal surface structure. You can also set these values for editing existing surfaces using Edit—Property Chg—Surfaces.

Use Display Curve Tol to set the maximum that the display grid curves can deviate from the exact surface. The tolerance is calculated from the chord height of each grid curve polyline segment.

1. Select the **Utility—Surface Modes**. The **Surface Modes** dialog box is displayed.

Figure 13-6
Set the values on the Surface Modes dialog box.



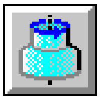
2. Set the **Count in u-direction** input field to a positive number of isoparametric lines for SmartCAM to use for the display grid in the surface's U direction. The graphic view display of the surface shows the insert markers in the U direction.

3. Set the **Count in v-direction** input field to a positive number of isoparametric lines for SmartCAM to use for the display grid in the surface's V direction.

4. Set the **Display Curve Tol** input field to the maximum amount that the display grid curves can deviate from the exact surface. The tolerance is calculated from the chord height of each grid curve polyline segment.

5. Select the **Accept** button.

Creating a Spun Surface



Model File: A3DSPUN.PM4

Use Spun Surface to create a surface by rotating a two-dimensional generator curve around an axis vector line. The generator curve must be a profile consisting of lines, arcs, splines, ellipses, or polylines. Before you create a spun surface, ensure that all the generator profile elements connect.

1. Open the model file **A3DSPUN.PM4**.

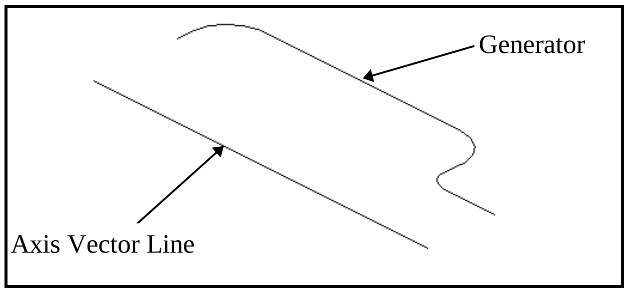
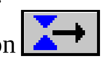
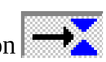
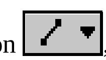
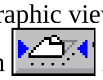


Figure 13-7 *Open A3DSPUN.*

2. Set the insert location.
   - Set the **Before** icon  or **After** icon  to indicate if the insert point is to be before or after the selected element, process step, tool, or layer.
   - Select the **Element** icon , and select the element to be before or after in the list view or graphic view.
   - Select the **On Layer** icon  icon.
3. Select **Create—Surface—Spun Surface**. The **Spun Surface** control panel is displayed.

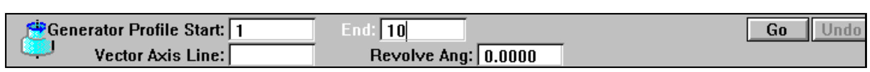


Figure 13-8 *Set the values on the Spun Surface control panel.*

4. Select the **Generator Profile Start** input field, and select the element for the start of the generator profile.
5. Select the **End** input field, and select the element for the end of the generator profile.
6. Select the **Vector Axis Line** input field, and select the line element to define the axis that the generator rotates around.
7. Set the **Revolve Ang** input field to the number of degrees for the generator profile to rotate around the vector axis line.
8. Select the **Go** button.

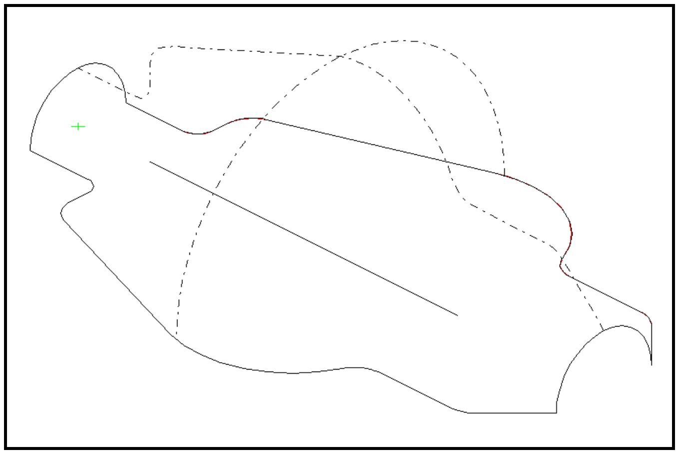
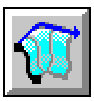


Figure 13-9
This is the result of creating a spun surface.

Creating a Translated Surface



Model File: A3DTRAN.PM4

Use Trans. (Translated) Surface to create a surface by sweeping a generator profile along a director curve. The generator profile maintains a perpendicular orientation to the tangent vector for each segment as it moves along the director curve.

The generator profile can consist of lines, arcs, splines, ellipses, points, or polylines. You must assign all elements of the profile to the same work plane and make sure that they run in the same direction. The generator must be on a work plane that is perpendicular to the director's first segment tangent vector.

The director curve can consist of lines, arcs, splines, points, ellipses, or polylines and it must be a profile.

1. Open the model file **`A3DTRAN.PM4`**.

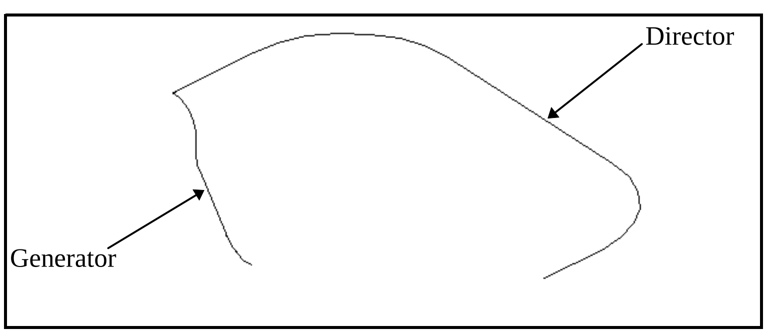
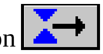
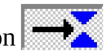
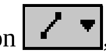


Figure 13-10
Open A3DTRAN.

2. Set the insert location.

- Set the **Before** icon  or **After** icon  to indicate if the insert point is to be before or after the selected element, process step, tool, or layer.
- Select the **Element** icon , and select the element to be before or after in the list view or graphic view.
- Select the **On Layer** icon  icon.

3. Select **Create—Surface—Trans. Surface**. The **Trans. Surface** control panel is displayed.

Figure 13-11 *Set the values on the Trans. Surface control panel.*

4. Specify the generator profile using the **Generator Prof Start** and **End** fields.

   Note For best results, the generator profile should be located at the start of the director curve.

5. Specify the director profile using the **Director Prof Start** and **End** fields.
6. Select the **Go** button.

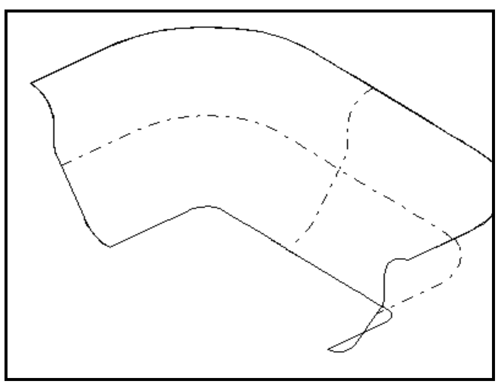
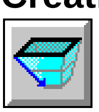


Figure 13-12 *This is the result of translating a surface.*

Creating a Draft Surface



Model File: FFMDRFT.PM4

Use Draft Surface to create a surface by sweeping a two-dimensional generator profile along a two- or three-dimensional director curve. The generator rotates as it sweeps so that its projection remains perpendicular to the two-dimensional projection of the director curve into the active tool plane. Draft Surface uses the value in the Point Set Fitting field of the Surface Modes dialog box to control the tolerance for creating the surface.

The generator profile can consist of lines, arcs, splines, ellipses, or polylines. All elements of the profile must be assigned to the same work plane and run in the same direction.

The director curve must be a profile consisting of lines, arcs, splines, ellipses, points, or polylines.

Perform these tasks to create a draft surface:

1. Open the model file **FFMDRFT.PM4**.

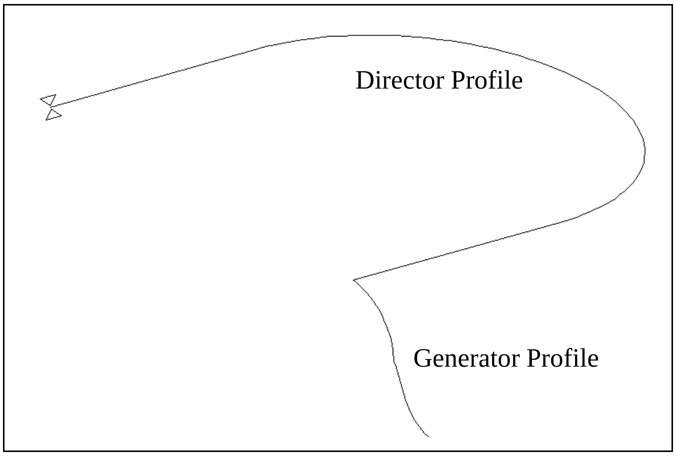
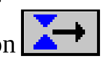
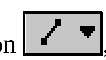
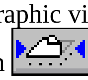


Figure 13-13 *Open FFMDRFT.*

2. Set the insert location.

- Set the **Before** icon  or **After** icon  to indicate if the insert point is to be before or after the selected element, process step, tool, or layer.
- Select the **Element** icon , and select the element to be before or after in the list view or graphic view.
- Select the **On Layer** icon  icon.

3. Select **Create—Surface—Draft Surface**. The **Draft Surface** control panel is displayed.

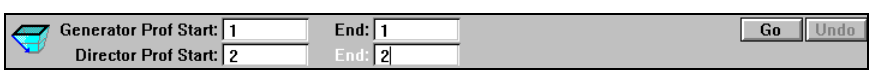


Figure 13-14 *Set the values on the Draft Surface control panel.*

4. Specify the generator profile using the **Generator Prof Start** and **End** fields.

   Note For best results, the generator profile should be located at the start of the director curve.

5. Specify the director profile using the **Director Prof Start** and **End** fields.
6. Select the **Go** button.

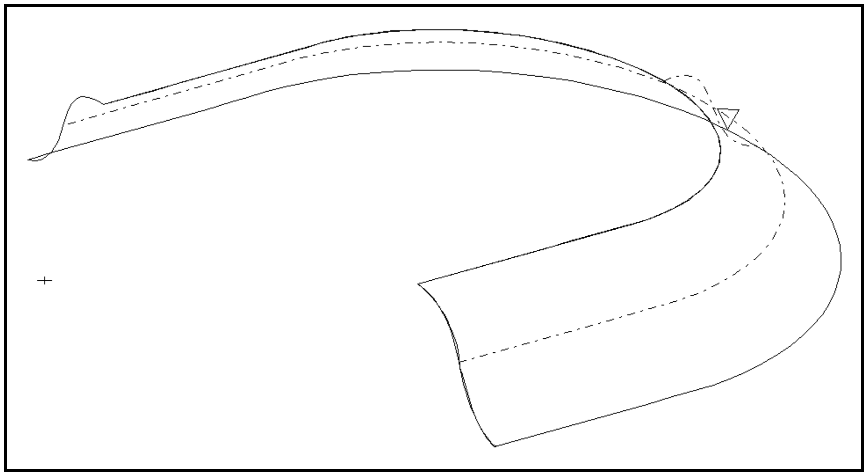
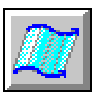


Figure 13-15
This is the result of creating a draft surface.

Creating a Ruled Surface



Model File: A3DRULE.PM4

Use Ruled Surface to create a surface that is defined by straight lines drawn between two profiles. The profiles can consist of lines, arcs, splines, points, ellipses, or polylines.

1. Open the model file **A3DRULE.PM4**.

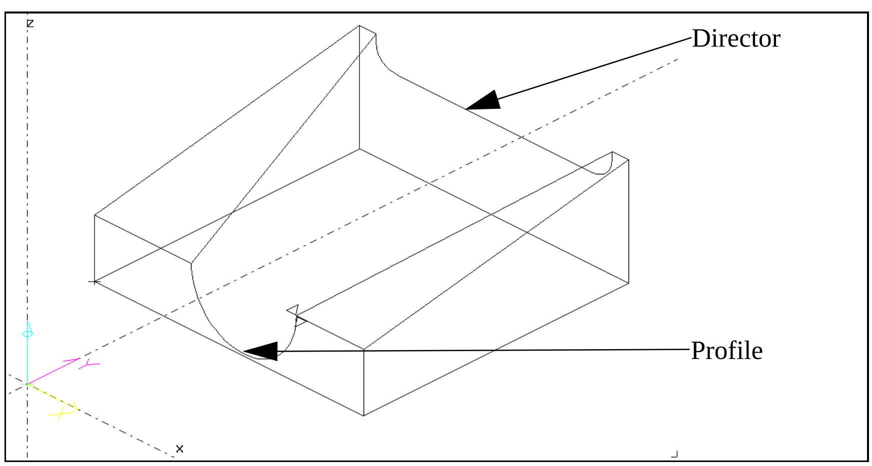
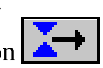
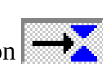
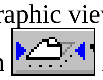


Figure 13-16
Open A3DRULE.

2. Set the insert location.

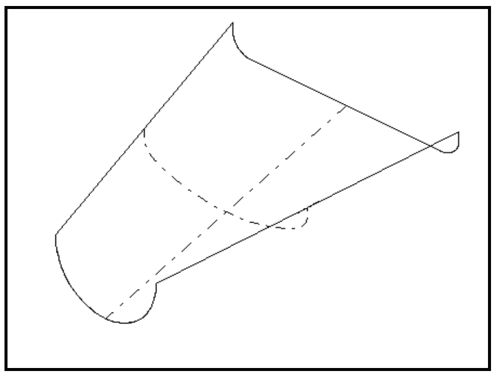
- Set the **Before** icon  or **After** icon  to indicate if the insert point is to be before or after the selected element, process step, tool, or layer.
- Select the **Element** icon , and select the element to be before or after in the list view or graphic view.
- Select the **On Layer** icon  icon.

3. Select **Create—Surface—Ruled Surface**. The **Ruled Surface** control panel is displayed.

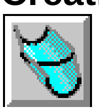
__Figure 13-17__
Set the values on the Ruled Surface control panel.

4. Specify the first profile using the **1st Profile Start** and **End** fields.
5. Specify the second profile using the **2nd Profile Start** and **End** fields.
6. Select the **Go** button.

__Figure 13-18__
This is the result of creating a ruled surface.



Creating a Lofted Surface



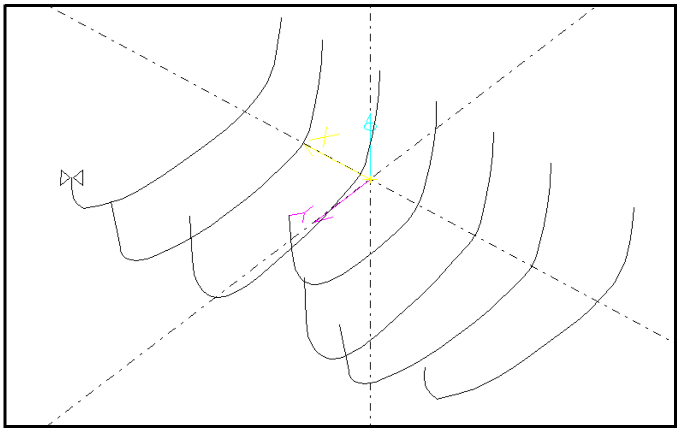
Model File: A3DLOFT.PM4

Use Lofted Surface to create a surface that is defined by being blended through a series of curves. The surface is based on a series of cubic splines equally spaced along the sections. The curves must be sequential profiles that run in the same direction. The profiles can consist of lines, arcs, points, or polylines.

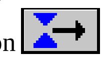
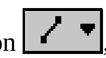
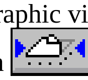
Note You must group the elements that represent the cross-sections of the surface that you are creating.

1. Open the model file **`A3DLOFT.PM4`**.

Figure 13-19
Open A3DLOFT.

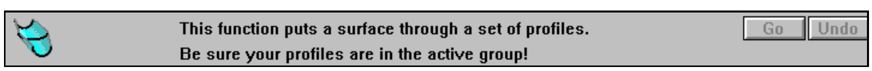


2. Set the insert location.

- Set the **Before** icon  or **After** icon  to indicate if the insert point is to be before or after the selected element, process step, tool, or layer.
- Select the **Element** icon , and select the element to be before or after in the list view or graphic view.
- Select the **On Layer** icon  icon.

3. Select **Create—Surface—Lofted Surface**. The **Lofted Surface** control panel is displayed.

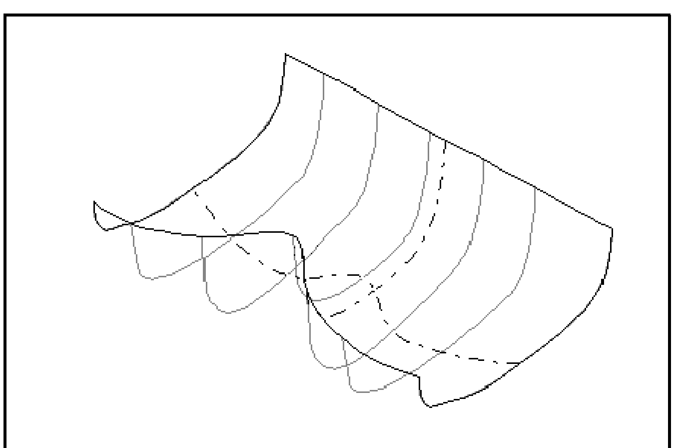
Figure 13-20
Set the values on the Lofted Surface control panel.



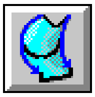
4. Make sure that all profiles are sequential in the database and run in the same direction.

5. Select the **Go** button.

Figure 13-21
This is the result of using Lofted Surface.



Creating a Coons Patch Surface



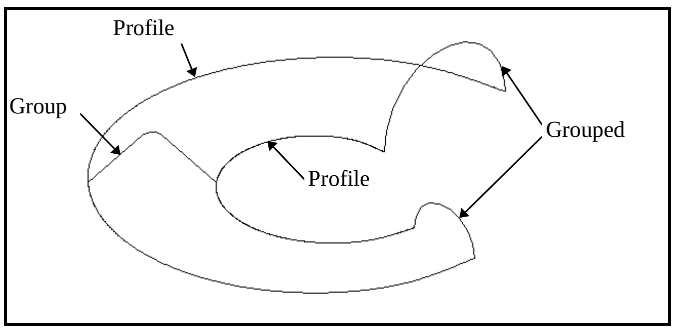
Model File: A3DCOON.PM4

Use Coons Surface to create a composite surface defined by a starting profile, an ending profile, and any number of interior profiles that define interior patch boundaries. All defining curves must be profiles that consist of lines, arcs, or polylines. Continuity between composite patches will be contiguous (CO). The continuity is specified in the Coons Surface control panel.

Coons extends to all of the outer boundaries of the outer profile. Most surface creation tools allow you to match two boundaries while Coons extends to all outer boundaries.

1. Open the model file **`A3DCOON.PM4`**.

Figure 13-22 Open A3DCOON.



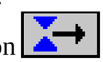
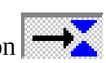
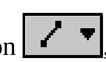
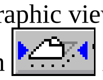
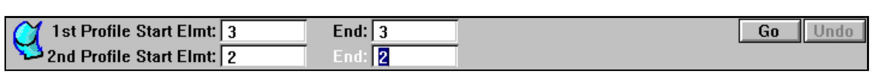
2. Set the insert location.
   - Set the **Before** icon  or **After** icon  to indicate if the insert point is to be before or after the selected element, process step, tool, or layer.
   - Select the **Element** icon , and select the element to be before or after in the list view or graphic view.
   - Select the **On Layer** icon  icon.
3. Group the cross-sectional profiles in the active group.
4. Select **Create—Surface—Coons Surface**. The **Coons Surface** control panel is displayed.

Figure 13-23 Set the values on the Coons Surface control panel.



5. Specify the first defining profile using the **1st Profile Start Elmt** and **End** input fields.
6. Select the second defining profile using the **2nd Profile Start Elmt** and **End** input fields.
7. Select the **Go** button.

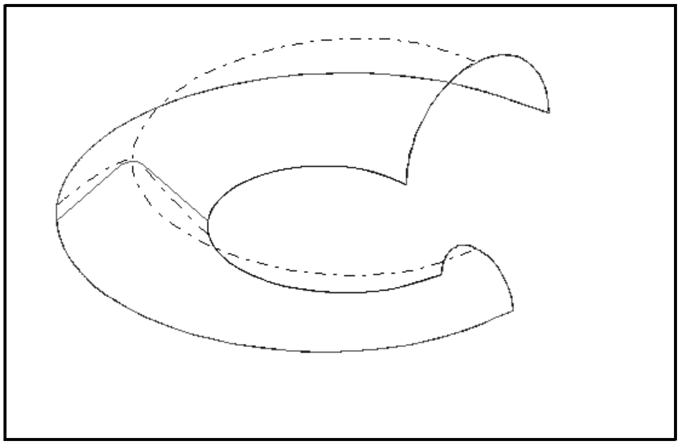
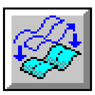


Figure 13-24
This is the result of creating a coons patch surface.

Creating a Surface From a Mesh



Model File: FFMMSHSF.PM4

Use From Mesh to create a surface from an existing polyline mesh. SmartCAM refines the mesh to a uniform point set and fits the surface through it. From Mesh uses the value in the Point Set Fitting field of the Surface Modes dialog box to control the tolerance for creating the surface.

1. Open the model file **`FFMMSHSF.PM4`**.

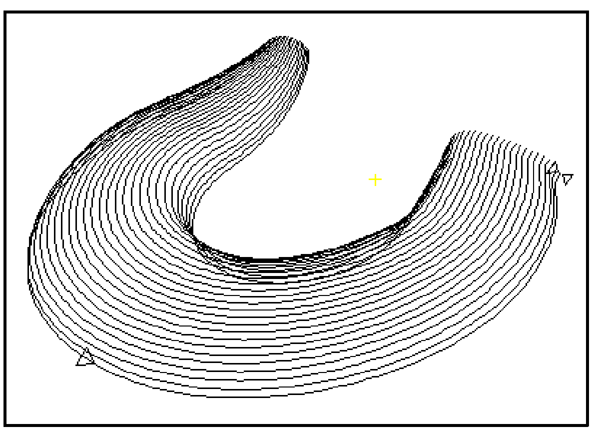
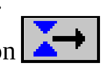
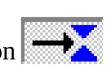
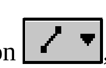
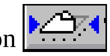


Figure 13-25
Open FFMMSHSF.

2. Set the insert location.

- Set the **Before** icon  or **After** icon  to indicate if the insert point is to be before or after the selected element, process step, tool, or layer.
- Select the **Element** icon , and select the element to be before or after in the list view or graphic view.

- Select the **On Layer** icon  icon.

3. Select **Create—Surface—From Mesh**. The **From Mesh** control panel is displayed.

Figure 13-26 *Set the values on the From Mesh control panel.*

4. Specify the polyline mesh using the **Mesh Start** and **End** input fields.
5. Select the **Go** button.

Points to Remember

- ❑ Higher u and v values display more lines but take longer to redraw.
- ❑ Surfaces are created on a layer.
- ❑ All of the surface element settings except Point Set Fitting affect only the visual representation of surfaces in FreeForm Machining, not the internal surface structure.
- ❑ Before you create a spun surface, ensure that all the generator profile elements connect.
- ❑ Use Trans. (Translated) Surface to create a surface by sweeping a generator profile along a director curve.
- ❑ Use Draft Surface to create a surface by sweeping a two-dimensional generator profile along a two- or three-dimensional director curve.
- ❑ Use Ruled Surface to create a surface that is defined by straight lines drawn between two profiles.
- ❑ Use Lofted Surface to create a surface that is defined by being blended through a series of curves.
- ❑ Use Coons Surface to create a composite surface defined by a starting profile, an ending profile, and any number of interior profiles that define interior patch boundaries.
- ❑ Use From Mesh to create a surface from an existing polyline mesh.

14

Editing Surfaces

Objectives

This unit shows you how to perform these tasks:

- Split surfaces.
- Blend surfaces.
- Extend surfaces.
- Extract surfaces and elements.
- Punch surfaces.
- Convert a surface to a mesh.
- Convert a mesh to a surface.

Overview

Use the Surface Edit toolbox to edit, extend, split, and remove surface elements; extract surface elements from imported solids; and generate uniform polyline mesh from existing surface elements.

Splitting Surfaces

Objectives

This lesson shows you how to split surfaces.

Overview

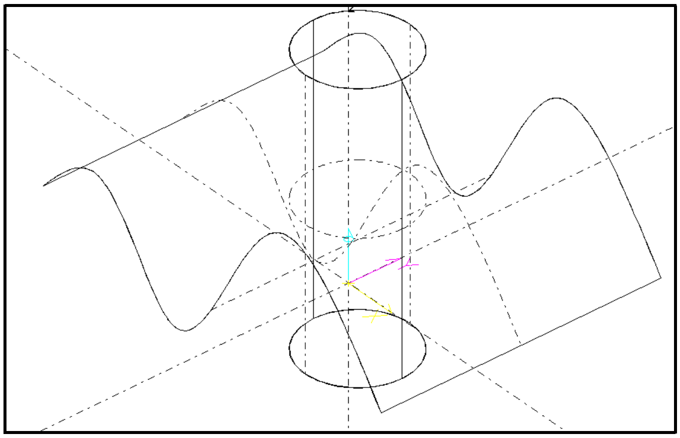
Use Split Surface to divide a surface element or group, or to create polyline curves at the intersection of two surfaces or groups of surfaces. The surface to be split must intersect another surface. Surfaces cannot be split to an intersecting curve or a tangent surface. Surfaces that are edited or split can be an element or a group.

Using Split Surface

Model File: FFMSPSRF.PM4

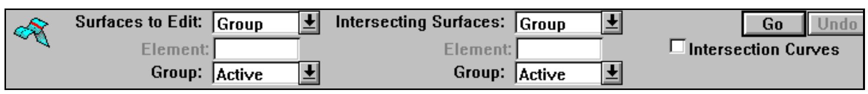
1. Open the model file **FFMSPSRF.PM4**.

Figure 14-1
Open FFMSPSRF.



2. Select **Edit—Surface Edit—Split Surface**. The **Split Surface** control panel is displayed.

Figure 14-2
Set the values on the Split Surface control panel.



3. Set the **Surfaces to Edit** selector switch to **Group**.
4. Set the **Group** selector switch to the group to split.
5. Set the **Intersecting Surfaces** selector switch to **Group**.
6. Set the **Group** selector switch to the group for the intersecting surfaces.
7. Set the intersecting surface **Element** input field. This input field is dim unless the intersecting surfaces selection is Element.
8. Turn off the **Intersection Curves** on/off switch.
9. Select the **Go** button to perform the split. If the result is not the desired split, select the **Undo** button to undo the action.

Blending Surfaces

Objectives

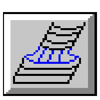
This lesson shows you how to blend surfaces.

Overview

You must convert surfaces to meshes to blend surfaces. After the blend is created, the meshes can be converted back to surfaces.

The blend mesh can be a constant or variable radius. The meshes used to generate the blend must be close enough to allow the blend to touch both meshes.

Blending a Surface



Model File: FFMBLND.PM4

Perform these tasks to blend surfaces:

1. Open the model file **FFMBLND.PM4**.

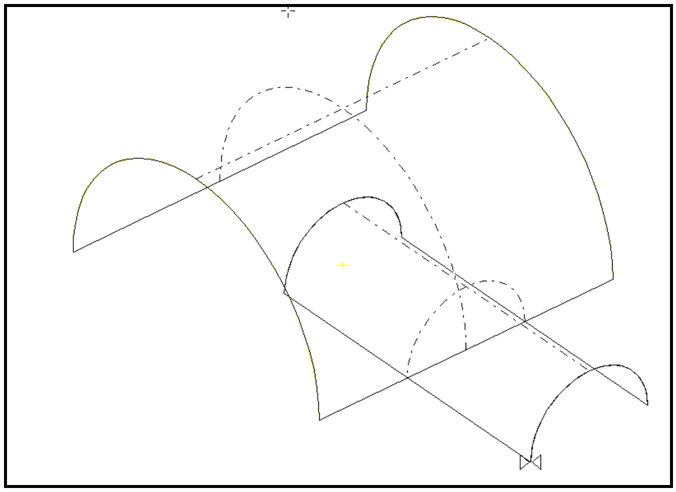


Figure 14-3
Open FFMBLND.

2. Select **Edit—Surface Edit—Generate Mesh**, and convert the surfaces to meshes.

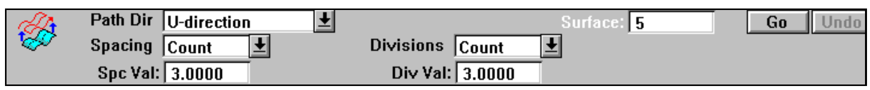


Figure 14-4
Set the values on the Generate Mesh control panel.

3. Select **Edit—Mesh Edit—Blend**. The **Blend** control panel is displayed.

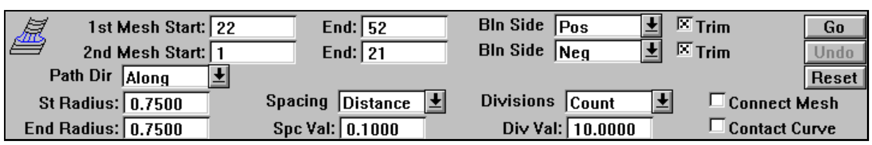


Figure 14-5
Set the values on the Blend control panel.

4. Identify the first mesh by setting the **1st Mesh Start** and **End** fields to the first and last elements of the first range of mesh profiles. Set these fields by manually typing, selecting the element in the graphics view, or selecting from the list view.

   Note You might need to use **View—Window** to identify specific elements. This selection also places the location of the start radius for variable radius blends closest to this element.

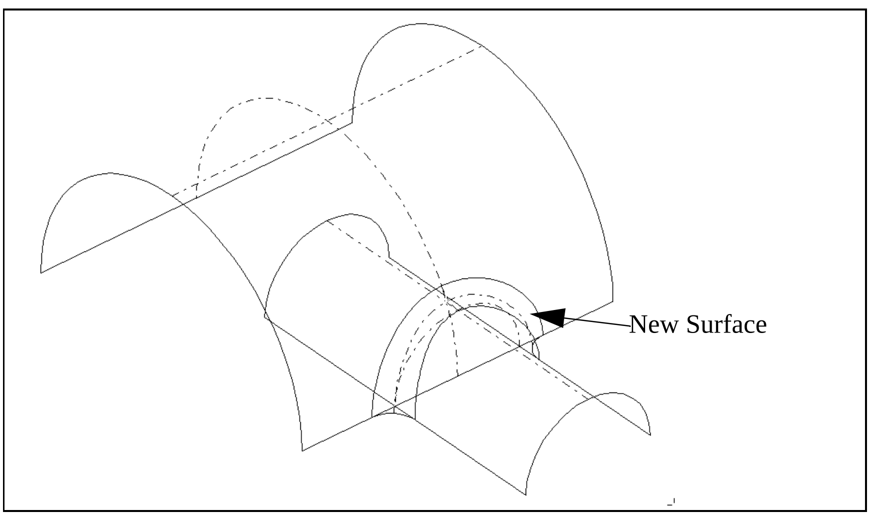
5. Identify the director profile by setting the **2nd Mesh Start** and **End** fields to the first and last elements of the second range of mesh profiles. Set these fields by manually typing, selecting the element in the graphics view, or selecting from the list view.

6. Set the **Bln Side** selector switches to **Pos** or **Neg** to indicate on which side of each range of mesh profiles to position the center of the blend arcs. The red blend arrows should point toward the center of the blend radius.

7. Turn on the **2nd Mesh Trim** on/off switch to trim the second mesh along the intersection.

8. Set the **St Radius** input field to the beginning blend radius value. Enter the same start and end radius values to generate a constant radius blend. Use different values to produce a variable radius blend. SmartCAM places this radius at the **1st Mesh Start** element.
9. Set the **End Radius** input field to the ending blend radius value.
10. Set the **Path Dir** selector switch to one of the following:
    - **Cross** creates mesh profiles that run perpendicular to the direction of the blend's arcs.
    - **Along** creates mesh profiles that run parallel to the direction of the blend's arcs.
11. Set the **Spacing** selector switch to one of the following:
    - **Distance** specifies the distance between parallel mesh profiles.
    - **Count** specifies the number of mesh profiles.
12. Set the **Spc Val** input field to the distance or count values.
13. Select the **Go** button.
14. Select **Create—Surface—From Mesh**, and convert the blended mesh to a surface.

Figure 14-6
This is the result of blending the surfaces.



Extending and Extracting Surfaces

Objectives

This lesson shows you how to perform these tasks:

- Extend surfaces.
- Extract surfaces.

Overview

Use Extend Surface to extend an existing surface from an edge boundary. The resulting extension is a new NURBS surface that shares the edge boundary with the original surface element. Extend Surface uses the current insert position and properties to create the new surface element.

You can create the surface extension with a contiguous (C0) or tangent (C1) continuity to the existing surface.

Using Extend Surface

Model File: FFMEXTSF.PM4

1. Open the model file **`FFMEXTSF.PM4`**.

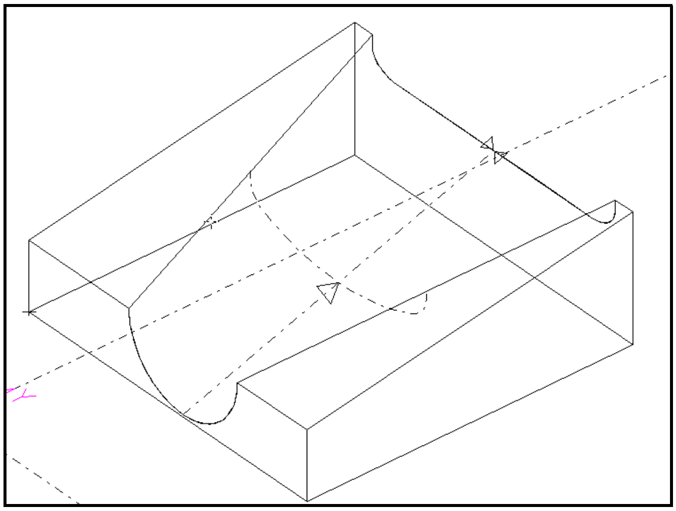


Figure 14-7
Open FFMEXTSF.

2. Select **Edit—Surface Edit—Extend Surface**. The **Extend Surface** control panel is displayed.

Figure 14-8
Set the values on the Extend Surface control panel.

3. Set the **Surface To Edit** input field to the surface element to use for creating the extension.

4. Set the **Continuity** option switch to **Contiguous** or **Tangent**, depending on the type of continuity that you want the surface extension to have.

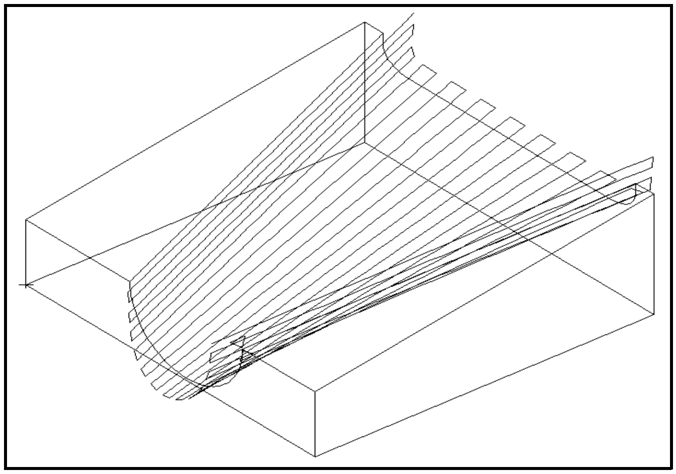
5. Specify the **Start** or **End** boundary vector to change if you set **Continuity** to **Tangent**.

6. To change the direction of the extension, use the **Rotation** arrow buttons (< or >) on the control panel, or select **Vector** to open the **Vector Input** dialog box and set the desired vector information.

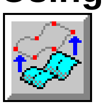
7. To change the length of the extension, use the **Length** + and **-** buttons on the control panel, or select **Vector** to open the **Vector Input** dialog box and set the desired length.

8. Select the **Go** button to create the surface extension. If the new surface is not correct, select the **Undo** button to remove it, then select the **Redraw** button to redisplay the boundary vectors.

Figure 14-9
This is a result of extending the mesh.



Using Extract



Use Extract to explode surfaces from solids before you use the Transform tools to move, rotate, mirror, or scale a surface. You can also use it to extract the constituent elements from a surface or group of surfaces. If the surface was created in SmartCAM, you can extract the construction geometry and display grid. If the surface is trimmed (imported from an ACIS or IGES file), you can extract base surface edges and trimming curves.

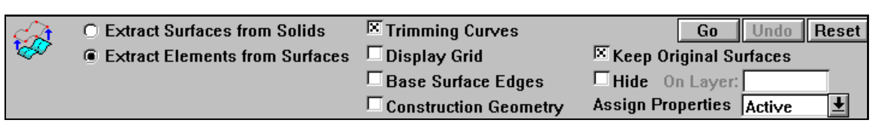
You can keep the original surface as part of the graphic view, hide it on a layer, or delete it as SmartCAM performs the extract.

Select an active group of elements before using Extract.

This is the basic procedure for extracting elements from solids:

1. Group the surfaces that you want to extract.
2. Select **Edit—Surface Edit—Extract**. The **Extract** control panel is displayed.

Figure 14-10
Set the values on the Extract control panel.



3. Turn on the **Extract Surfaces from Solids** option switch.
4. Select the **Go** button.

This is the basic procedure for extracting elements from a surface:

1. Group the surfaces that you want to extract.
2. Select **Edit—Surface Edit—Extract**. The **Extract** control panel is displayed, see Figure 14-10.
3. Turn on the **Extract Elements from Surfaces** on/off switch.
4. Turn on the **Construction geometry** on/off switch to extract the construction geometry from surfaces created with SmartCAM modeling tools. You cannot extract construction geometry from an imported ACIS or IGES surface.
5. Turn on the **Base Surface Edges** on/off switch to extract the edges of the base surface for trimmed surfaces in the active group. SmartCAM uses the **Display Curve Tolerance** setting in the **Surface Modes** dialog box to create the resulting elements.
6. Turn on the **Trimming Curves** on/off switch to extract the trimming curves from the surfaces in the active group.
7. Turn on the **Display Grid** on/off switch to extract the display grid curves in the u and v directions. SmartCAM will not extract the center u and center v curves.
8. Turn on the **Keep Original Surfaces** on/off switch to keep the original surfaces in addition to the extracted elements. Turn off this switch to delete the original surfaces when you extract elements.

   Note If you extract elements from an imported trimmed surface, it may not be possible to re-create the surface exactly from the extracted elements. In this case, keep the original surface.

9. Turn on the **Hide** on/off switch to move the original surfaces to another layer and hide them when you extract elements. Toggle this switch off to leave the original elements visible in the graphic view. The **Hide** switch is dimmed if the **Keep Original Surfaces** switch is off.
10. Select the **Go** button.

Punching Surfaces

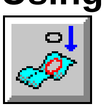
Objectives

This lesson shows you how to punch surfaces.

Overview

Use Punch Surface to remove a piece of an existing surface, based on its intersection with a volume defined by a punch profile and punch distance. The surface to edit must be an isolated surface and not part of a solid.

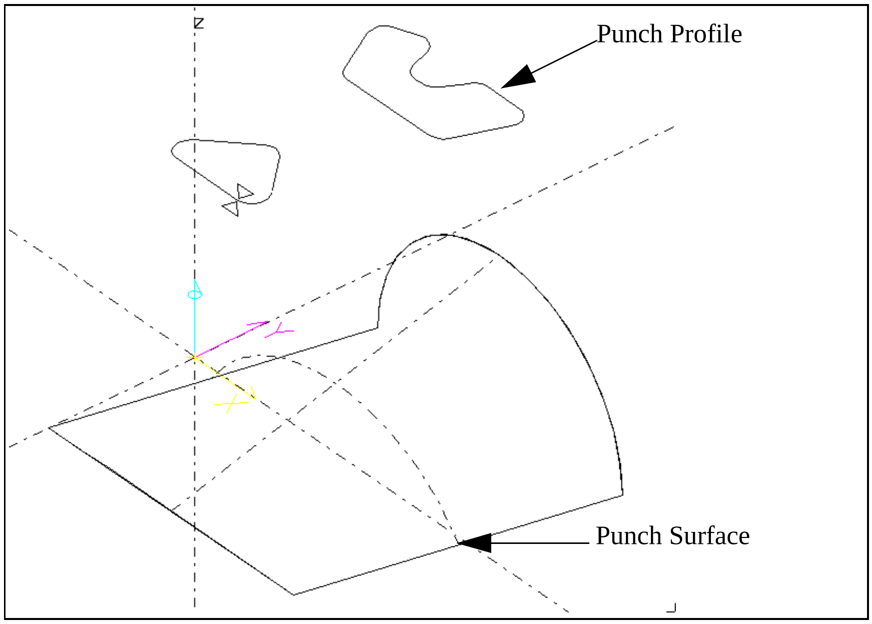
Using Punch Surface



Model File: FFMPNCH.PM4

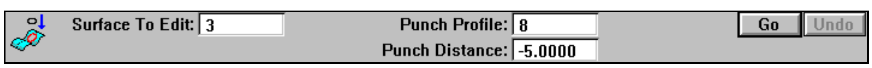
1. Open the model file **`FFMPNCH.PM4`**.

Figure 14-11
Open FFMPNCH.



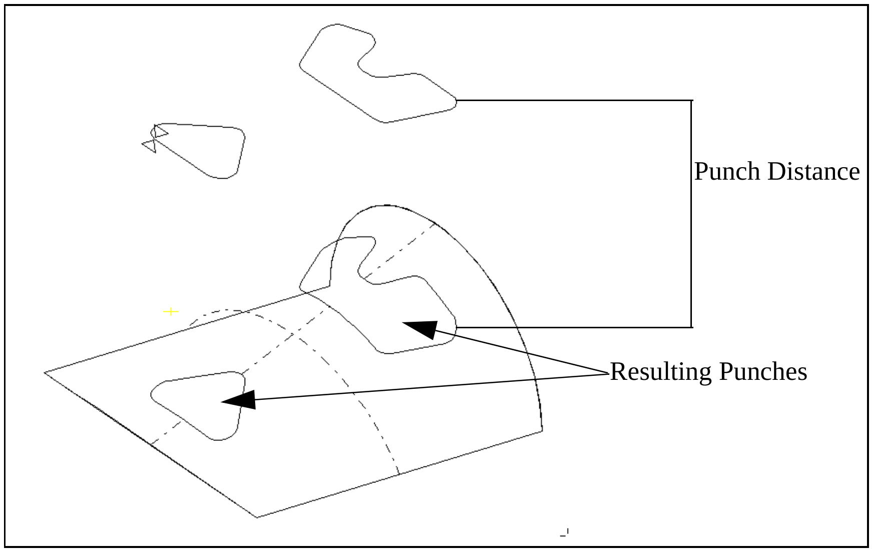
2. Select **Edit—Surface Edit—Punch Surface**. The **Punch Surface** control panel is displayed.

Figure 14-12
Set the values on the Punch Surface control panel.



3. Set the **Surface To Edit** input field to enter the surface to punch.
4. Set the **Punch Profile** input field to the element in the profile to use for the punch, or select the profile from the graphic view. The punch profile must be a closed profile that is constructed only of line and arc elements, and it must lie on a single work plane.
5. Set the **Punch Distance** input field to the distance (along the Z axis of the active work plane) from the level of punch profile to the level at which the punch will terminate. To perform a downward punch, enter a negative value in this input field.
6. Select the **Go** button. The surface is punched.

Figure 14-13
This is the result of the punch.



Working with Meshes and Surfaces

Objectives

This lesson shows you how to perform these tasks:

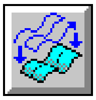
- Create a surface from a mesh.
- Generate a mesh from a surface.

Overview

Use From Mesh to create a surface from an existing mesh. Element types for the mesh can include polylines, lines, arcs, ellipses, and splines. The mesh elements must not be connected, and all profiles must flow in the same direction. If the mesh is not uniform, FreeForm Machining will refine the mesh to a uniform point set and fit the surface through it.

Use Generate Mesh to generate an exact uniform polyline mesh that you can use to add mesh fillets, correct imported design surfaces, and produce intersection curves. The points of the mesh coincide with the surface from which it is generated. The mesh assumes the active insert properties for step or layer geometry and for work plane.

Creating a Surface from a Mesh

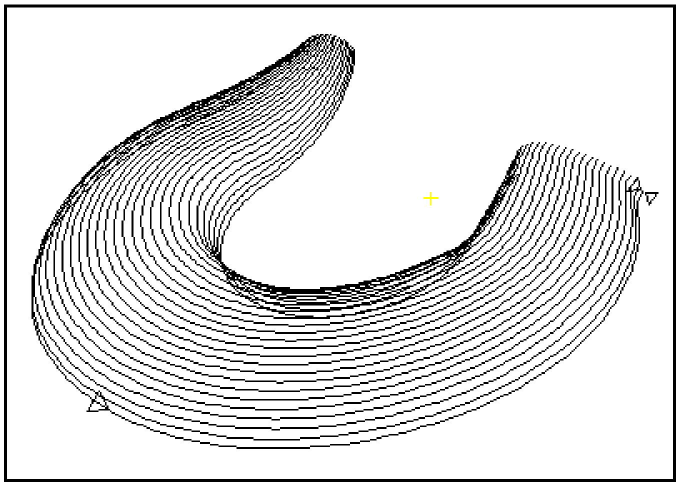


Model File: FFMMSHSF.PM4

From Mesh uses the value in the Point Set Fitting field of the Surface Modes dialog box to control the tolerance for creating the surface.

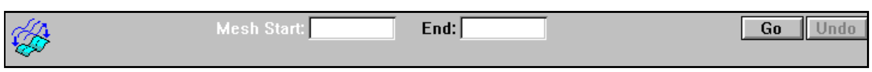
1. Open the model file **`FFMMSHSF.PM4`**.

Figure 14-14
Open FFMMSHSF.



2. Select **Create—Surface—From Mesh**. The **From Mesh** control panel is displayed.

Figure 14-15
Set the values on the From Mesh control panel.



3. Select **Mesh Start Elmt**, and select the first element of the range of elements that makes up the polyline mesh.
4. Select **Mesh End Elmt**, and select the last element of the range of elements that makes up the polyline mesh.
5. Select the **Go** button.

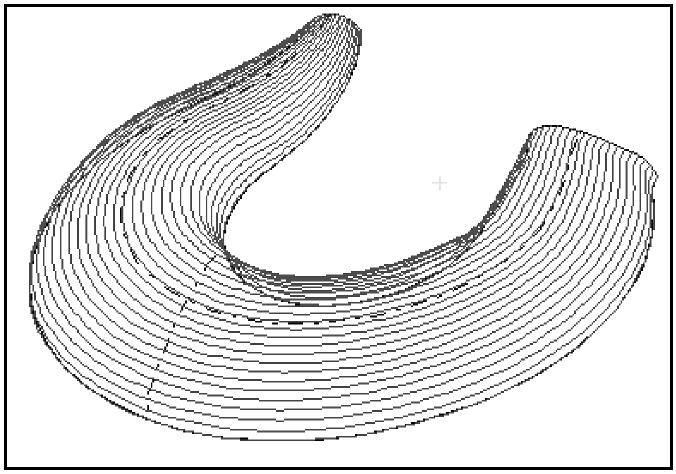
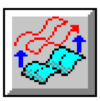


Figure 14-16
This is a result of creating a surface from a mesh.

Generating a Mesh from a Surface



Perform these tasks to generate a mesh from a surface.

1. Select **Edit—Surface Edit—Generate Mesh**. The **Generate Mesh** control panel is displayed.

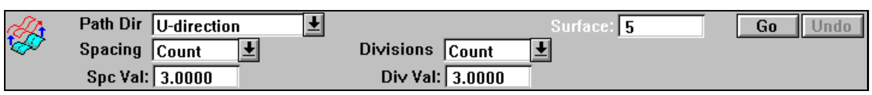


Figure 14-17
Set the values on the Generate Mesh control panel.

2. Set **Surface** to the element number of the surface to use for generating the mesh, or select the surface element from the graphic view or the list view.
3. Set **Path Dir** to choose the direction for toolpath elements, as follows:
   - **U-direction** creates elements in the u direction. The graphic view display of the surface shows the insert markers in the u direction.
   - **V-direction** creates elements in the v direction.
4. Set **Spacing** to the method for determining the number of polylines, as follows:
   - **Distance** creates polylines at a specified maximum distance apart.
   - **Count** creates a specified number of polylines an equal distance apart.
   - **Mid Tol** creates polylines with a specified maximum chord height tolerance.

5. Set **Spc Val** to enter the value for the spacing of polylines as follows:
   - **Distance** sets the maximum allowable distance between polylines.
   - **Count** sets the number of polylines to be created in equal spacing across the surface.
   - **Mid Tol** sets the maximum chord height tolerance between mesh curves.
6. Set **Divisions** to the method for determining the number of segments in the polylines, as follows:
   - **Distance** creates polylines using the specified maximum segment length.
   - **Count** creates a specified number of segments in each polyline at an equal distance apart.
   - **Mid Tol** creates segments using the specified maximum chord height tolerance.
7. Set **Div Val** to the value for creating the polyline segments, as follows:
   - **Distance** sets the maximum allowable length of each segment.
   - **Count** sets the number of segments to be created in equal spacing in each polyline.
   - **Mid Tol** sets the maximum chord height tolerance to use for determining the length of each segment.
8. Select the **Go** button.

Points to Remember

- When you split surfaces, the surface to split must intersect another surface. Surfaces cannot be split to an intersecting curve or a tangent surface.
- The meshes used to generate the blend must be close enough to allow the blend to touch both meshes.
- Use Extend Surface to extend an existing surface from a single edge boundary.
- Use Extract to explode surfaces from solids, extract the constituent elements from a surface or group of surfaces, and extract the construction geometry and display grid.
- When you punch surfaces, the surface to punch must be an isolated surface and not part of a solid.
- You can generate a mesh from a surface and a surface from a mesh.

15

Machining Surfaces

Objectives

This unit shows you how to perform these tasks:

- Create a straight toolpath across multiple surfaces.
- Produce projected cuts on multiple surfaces.
- Create Z-level roughing.
- Produce roughing cuts on multiple surfaces.
- Create flow line cuts across a single surface.
- Create contour cuts across multiple surfaces.

Overview

Use the Surface Machine toolbox to create a straight toolpath across multiple surfaces, project a toolpath across multiple surfaces, generate roughing cuts over a set of surfaces, convert toolpath to a series of roughing cuts, or create a toolpath that follows the U- or V-direction of a single surface. Be sure to set Insert properties to With Step and use the proper P (Profile Top) and C (Clear) values in the Insert property bar.

Creating a Straight Toolpath Across Multiple Surfaces

Objectives

This lesson shows you how to create a toolpath that follows a straight line across multiple surfaces.

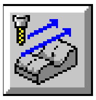
Overview

Use Straight to create a toolpath that follows a straight line across multiple surfaces. The straight toolpath is gouge free for all supported tools, regardless of the part topology. The number of surfaces that the toolpath can cross is limited only by available memory.

The Straight modeling tool emulates a copy mill, giving a clear definition of surface intersections, consistent stepover, and direction of cut that allows the least amount of polishing to finish the part.

Straight relies on data from the Insert property bar and the active step. Values for the work plane, Z clear, and profile top are set on the Insert property bar. Tool definition and operation parameters (for example, stepover and tolerance) are set by the current step.

Using Straight



Model File: FFMGATOR.PM4

1. Open the model file **`FFMGATOR.PM4`**.

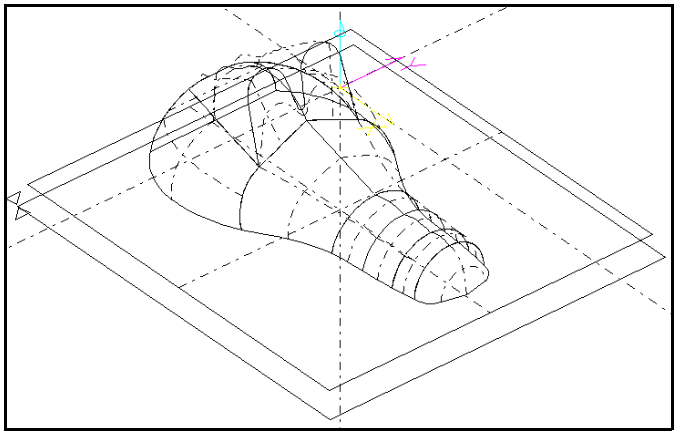


Figure 15-1
Open FFMGATOR.

2. Group the surfaces to be cut.
3. Make sure that surface normals are properly oriented.
4. Select **Process—Surface Machine—Straight**. The **Straight** control panel is displayed.

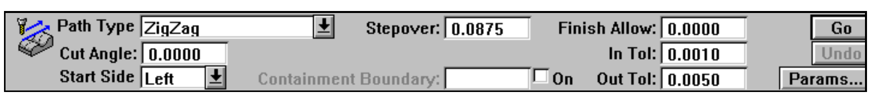


Figure 15-2
Set the values on the Straight control panel.

5. Set the following parameters on the control panel:
   - Set the **Path Type** to **ZigZag**.
   - Set the **Finish Allow** input field to the thickness of the material to be left on the part after the machining operation. For certain applications, such as die making, you can enter a negative number. The absolute value of the negative finish amount must be less than or equal to the tool corner radius.
   - Set the **In Tol** input field to the maximum allowable distance for any point on the toolpath to deviate toward the part surface. Use any value greater than or equal to 0. The sum of the **In Tol** and **Out Tol** fields must be at least 0.00001.
   - Set the **Out Tol** input field to the maximum allowable distance for any point on the toolpath to deviate away from the part surface. Use any value greater than or equal to 0. The sum of the **In Tol** and **Out Tol** fields must be at least 0.00001.

- Set the **Cut Angle** input field to the cutting angle of the straight line cuts measured about the Z axis, starting from the positive X-axis direction (counterclockwise). Use any value from -360 to 360 degrees.
- Set the **Start Side** selector switch to the side of the cut angle to start the cuts, either **Left** or **Right**. The side is determined from the origin of the angle indicator, looking in the direction of the cut angle.
- Set the **Stepover** input field to the distance between consecutive toolpath passes. The minimum value you can use is 0.0001 inch. If you enter a smaller number, FreeForm Machining will use the minimum value.

6. Select the **Params...** for access to additional controls for the straight machining process. The **Straight Parameters** dialog box is displayed.

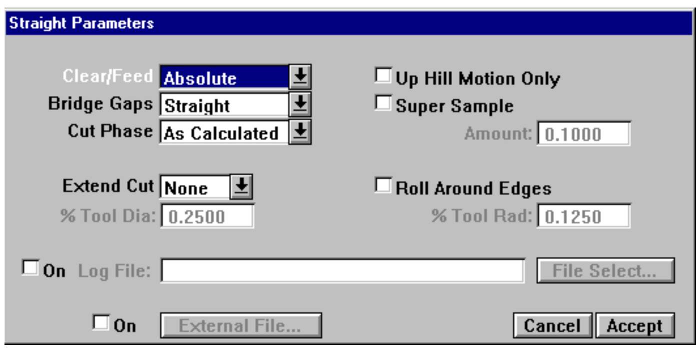


Figure 15-3
Set fields as necessary on the Straight Parameters dialog box.

7. Set the fields, as needed.
8. Select the **Accept** button and the values you entered on the dialog box are stored by SmartCAM. They will be used to control the Straight Surface Machining process.
9. Select the **Go** button.

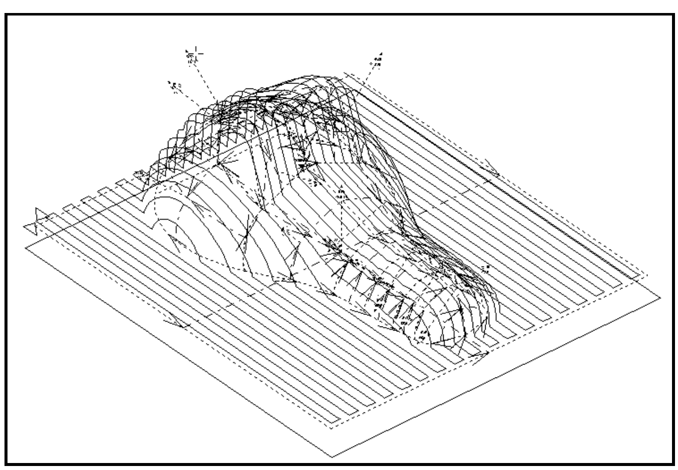


Figure 15-4
This is a result of creating a straight toolpath across multiple surfaces.

Producing Projected Cuts on Multiple Surfaces

Objectives

This lesson shows you how to create a toolpath that follows any line or polyline element across multiple surfaces.

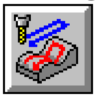
Overview

Use Projected to create a toolpath that follows any line or polyline element across multiple surfaces. The resulting toolpath is gouge free for all supported tools, regardless of the topology. The number of surfaces that the toolpath can cross is limited only by available memory.

Use Projected when you do not want the toolpath pattern produced by straight-line machining or when the topography of surfaces is not suitable for straight-line cutting.

Projected relies on data from the Insert property bar, the active step, and the active group of lines and polylines.

Using Projected



Model File: FFMPROJM.PM4

1. Open the model file **`FFMPROJM.PM4`**.

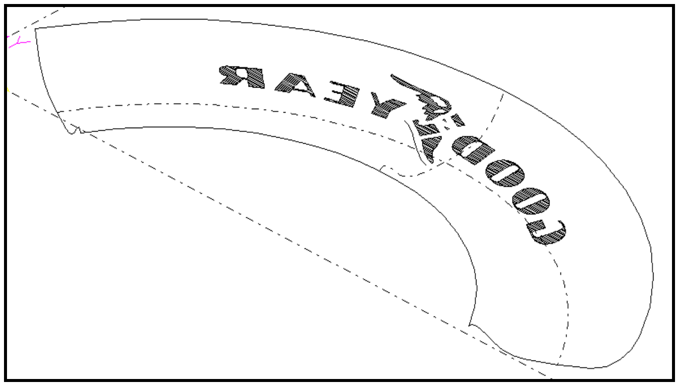
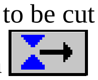
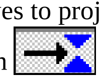
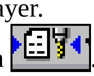


Figure 15-5
Open FFMPROJM.

2. Group the surfaces to be cut and the curves to project.
3. Set the **Before** icon  or **After** icon  on the Insert property bar to indicate if the insert point is to be before or after the selected element, process step, tool, or layer.
4. Set the **With Step** icon .
5. Select **Process—Surface Machine—Projected**. The **Projected** control panel is displayed.

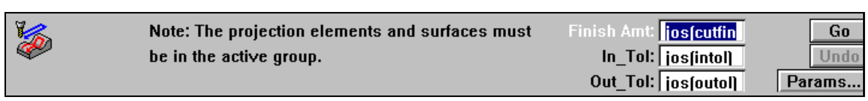


Figure 15-6
Set the values on the Projected control panel.

6. Set the following parametes on the control panel:

   - Set the **Finish Amt** input field to the thickness of the material to be left on the part after the machining operation. For certain applications (for example, die making), you can enter a negative number. The absolute value of the negative finish amount must be less than or equal to the tool corner radius.
   - Set the **In Tol** input field to the maximum allowable distance for any point on the toolpath to deviate toward the part surface. Use any value greater than or equal to 0. The sum of the **In Tol** and **Out Tol** fields must be at least 0.00001.
   - Set the **Out Tol** input field to the maximum allowable distance for any point on the toolpath to deviate away from the part surface. Use any value greater than or equal to 0. The sum of the **In Tol** and **Out Tol** fields must be at least 0.00001.

7. Select the **Params...** button to set additional parameters. The **Projected Parameters** dialog box is displayed.

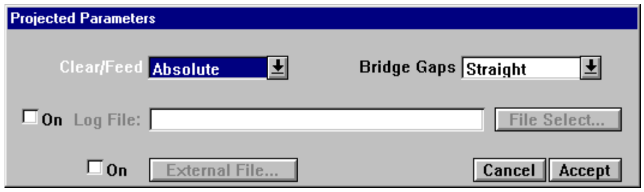


Figure 15-7 *Set additional machining parameters in the Projected Parameters dialog box.*

8. Select the **Accept** button. The parameters you have set in the dialog box will be used by SmartCAM to control the Projected machining process.

9. Select the **Go** button.

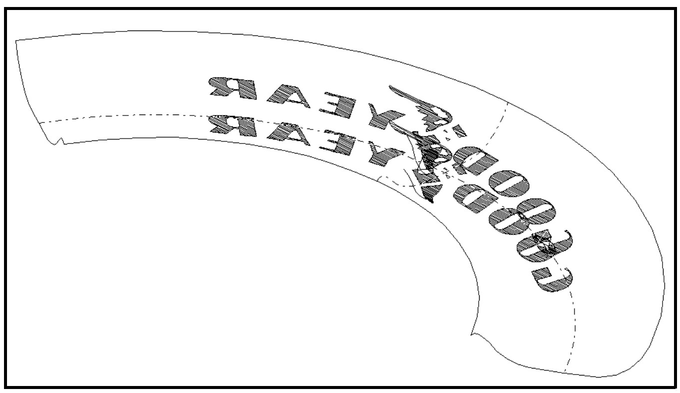


Figure 15-8 *This is the result of projecting the cuts on the surfaces.*

Creating Z-Level Roughing

Objectives

This lesson shows you how to use Contour surface machining to generate roughing levels to use in Cavity roughing.

Overview

Use Contour to create a precise, gouge-free toolpath that is a series of profile cuts at successive Z levels around the selected surfaces. A toolpath is created for all possible profiles at each level. If there are multiple profiles at the same level, the tool retracts to the clearance plane to move between them, allowing the machining of cavities and islands.

Use Cavity to rough an opening in preparation for finishing cuts. Make one roughing pass for each planar cut level. Before using Cavity, complete the following steps:

Using Contour

Model File: FFMCAV.PM4

1. Open the model file **FFMCAV.PM4**.

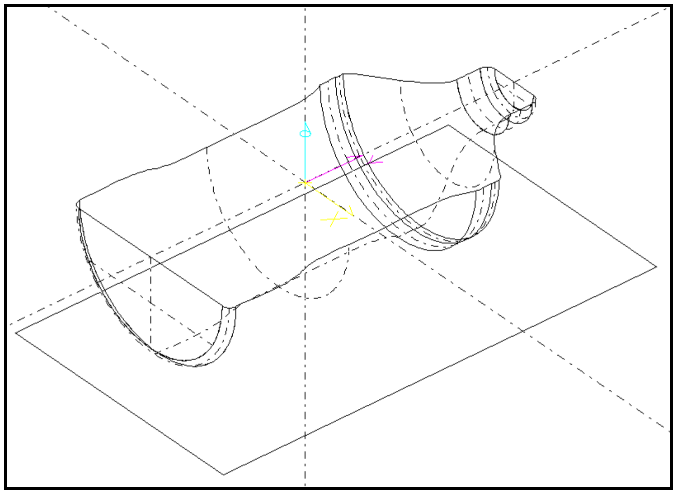
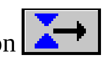
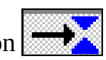
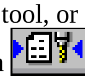


Figure 15-9
Open FFMCAV.

2. Group the surfaces to be roughed.

3. Set the insert location:

   - Set the **Before** icon  or **After** icon  on the Insert property bar to indicate if the insert point is to be before or after the selected element, process step, tool, or layer.
   - Set the **With Step** icon , and select step **18**.
   - Set the **C** (Clear) input field to **.25[6.35]**.

4. Select **Process—Surface Machine—Contour**. The **Contour** control panel is displayed.

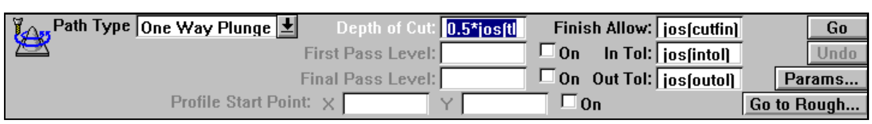


Figure 15-10
Set the values on the Contour control panel.

5. Set additional parameters as necessary on the **Contour Parameters** dialog box.

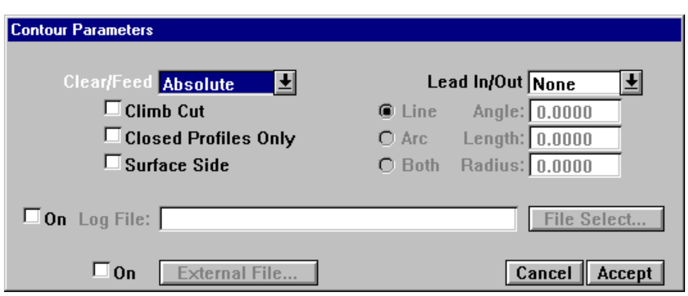


Figure 15-11
Set the values on the Contour Parameters dialog box.

6. Select the **Accept** button. The parameters you set will be used by SmartCAM for the Contour machining process.

7. Use **Contour** to create planar profiles at each Z level of the mesh profile, and put them in the active group.

8. Select **Go To Rough** to link directly to the **Cavity** control panel. Contour sends information to the Cavity procedure. The **Cavity** control panel is displayed.

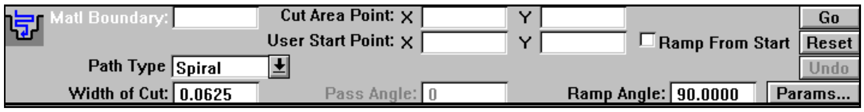


Figure 15-12 *Set the values on the Cavity control panel.*

9. Select the **Material Boundary** input field, and select an element from the material boundary. Each planar cut is extended to meet the material boundary and form closed areas to machine. The material boundary must be a closed profile.

10. Set the **Path Type** selector switch to one of these options:

    - **Spiral** creates a spiral roughing operation.
    - **Zig Zag** creates a zigzag roughing operation.
    - **Linear** creates a linear roughing operation.

11. Set the following fields on the control panel as necessary:

    - Set the **Width of Cut** input field to the maximum stepover distance for each pass. The default is half the active tool diameter.
    - Set the **Cut Area Point**. This should be a point within the area to be cut.
    - Set the **User Start Point** input field to the coordinates for the beginning of the cut. (This is best selected from a top view.) If you enter a start point in the **User Start Point** input field, SmartCAM will create a move directly from the user start point location to the automatic start point of the toolpath.
    - Turn on the **Ramp From Start** on/off switch to create ramp moves directly from the user start point location to the automatic start point of the toolpath. Turn off this switch to create a vertical plunge at the user start point.
    - Select the **Ramp Angle** input field. Enter the angle in degrees for tool entry when the **Ramp From Start** switch is off and the **User Start Point** field is not defined. A 90-degree angle results in a vertical plunge.
    - Set the **Pass Angle** input field to the angular orientation of roughing passes for linear and zigzag cutting. This location indicates the inside of the cavity. This is best selected from a top view. You must enter a cut area point for a cavity.

12. Select the **Params...** button. The Cavity Pameters dialog box is displayed.

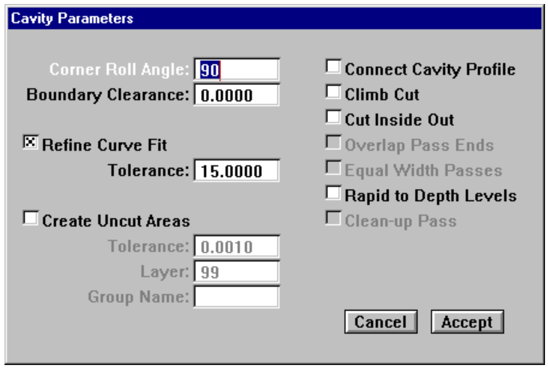


Figure 15-13
Set the values on the Cavity Parameters dialog box.

13. Set the following parameters on the dialog box:

    - Turn on the **Connect Cavity Profile** on/off switch if the roughing tool is the same as the one used for the planar cuts. This resequences each roughing pass to occur before the planar cut at the same level and connects them into a single, continuous profile.
    - Turn on the **Create Uncut Areas** on/off switch to create geometry for areas that the assigned roughing tool cannot cut. This geometry can then be used to create a toolpath to finish the roughing with a different tool.
    - Turn on the **Climb Cut** on/off switch to create climb cuts. Turn off this switch to create conventional cuts.
    - Set the **Boundary Clearance** input field to the distance to cut beyond the material boundary. If you enter **0**, the center of the tool will stop at the material boundary profile.
    - Set additional parameters as necessary.

14. Select **Accept**.

15. Select the **Go** button. Rough toolpath is added to the part, and Contour toolpath is added after every roughing pass.

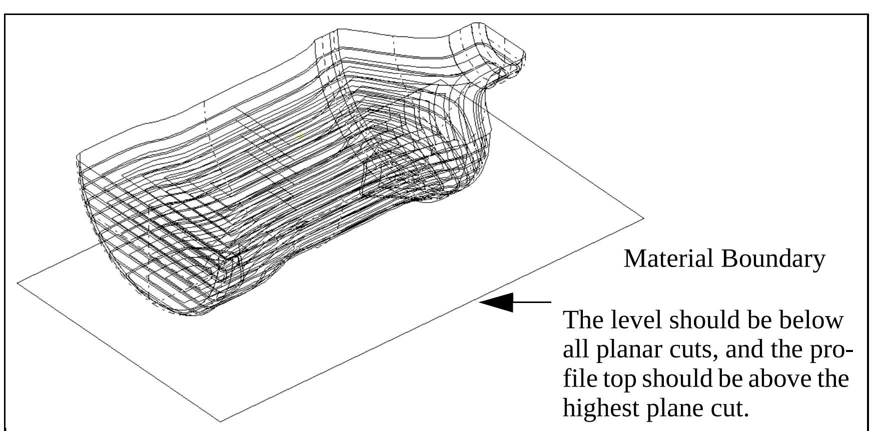


Figure 15-14
This is a result of Rough and Contour machining.

Producing Roughing Cuts on Multiple Surfaces

Objectives

This lesson shows you how to make a series of roughing cuts with limited depth on multiple surfaces.

Overview

Use Roughing to make a series of roughing cuts limited in Z depth. You can select either surfaces or toolpath elements (lines or polylines) as the input for the roughing process.

Using Roughing

Model File: FFMGATOR.PM4

Roughing relies on data from the Insert property bar and the active step.

1. Open the model file **FFMGATOR.PM4**.

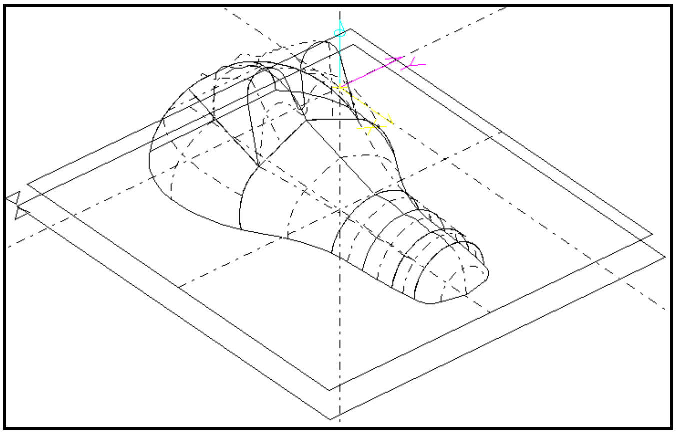
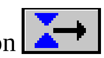
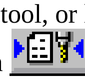


Figure 15-15
Open FFMGATOR.

2. Group the surfaces to cut.

3. Set the insert location:

   - Set the **Before** icon  or **After** icon  on the Insert property bar to indicate if the insert location is to be before or after the selected element, process step, tool, or layer.

   - Set the **With Step** icon .

4. Select **Process—Surface Machine—Roughing**. The **Roughing** control panel is displayed.

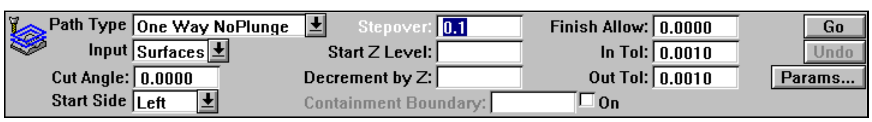


Figure 15-16
Set the values on the Roughing control panel.

5. Set the Path Type field to one of the following options:

   - **ZigZag** connects the start point of a move to the end point of the previous move.

   - **One Way NoPlunge** connects start points with a linear motion. The tool goes back to the previous cut start point and then moves to the new start point without plunging.

   - **One Way Plunge** connects start points with a retract, with the tool plunging at the new start point.

6. Set these parameters on the control panel as necessary:

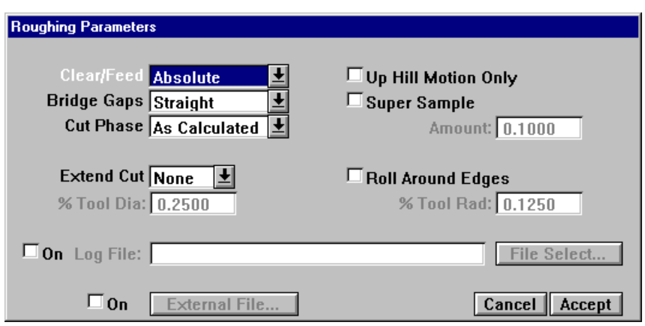
   - Set **Input** to **Surfaces** to use surfaces in the active group.

   - Set the **Finish Allow** input field to the thickness of the material to be left on the part after the machining operation. For certain applications (for example, die making), you can enter a negative number. The absolute value of the negative finish amount must be less than or equal to the tool corner radius.

- Set the **In Tol** input field to the maximum allowable distance for any point on the toolpath to deviate toward the part surface. Use any value greater or equal to 0. The sum of the **In Tol** and **Out Tol** fields must be at least 0.00001.
- Set the **Out Tol** input field to the maximum allowable distance for any point on the toolpath to deviate away from the part surface. Use any value greater than or equal to 0. The sum of the **In Tol** and **Out Tol** fields must be at least 0.00001.
- Set the **Cut Angle** input field to the cutting angle of the straight line cuts measured about the Z axis, starting from the positive X-axis direction (counterclockwise). Use any value from -360 to 360 degrees.
- Set the **Start Side** selector switch to the side of the cut angle to start the cuts, either **Left** or **Right**. The side is determined from the origin of the angle indicator, looking in the direction of the cut angle.
- Set the **Stepover** input field to the distance between consecutive toolpath passes. The minimum value you can use is 0.0001 inch. If you enter a smaller number, FreeForm Machining will use the minimum value.
- Set the **Start Z Level** input field to the Z value of a theoretical plane on which the first roughing pass is to be generated. If the start Z level is below the bottom of the part, only one pass, which will be an exact copy of the input when it is toolpath, is created. If the start Z level is a surface, only one finish pass is created.
- Set the **Decrement by Z** input field to the Z distance between levels on which multiple roughing passes are to be generated. Levels continue to be generated downward from the start Z level until there are no more surfaces cut by the cutting plane.

7. Select the **Params...** button. The **Roughing Parameters** dialog box is displayed.

Figure 15-17
Set the fields on the Roughing Parameters dialog box.



8. Set the parameters as needed, and select **Accept** on the dialog box.
9. Select the **Go** button.

Creating Flow Line Cuts Across a Surface

Objectives

This lesson shows you how to generate an isoparametric toolpath in the U or the V direction.

Overview

Use Flow Line to generate an isoparametric toolpath in either the U direction or the V direction for a single surface element. Flow Line supports ball, bull, and flat tools with no side angle.

Flow Line is a good tool to use on single surfaces having a cylindrical or spherical shape. Flow Line is helpful because it creates uniform toolpath across the surface.

There is no gouge correction for Flow Line. Surfaces with severe discontinuities or vertical walls are poor candidates for this function.

Using Flow Line

Model File: FFMFLOW.PM4

1. Open the model file **`FFMFLOW.PM4`**.

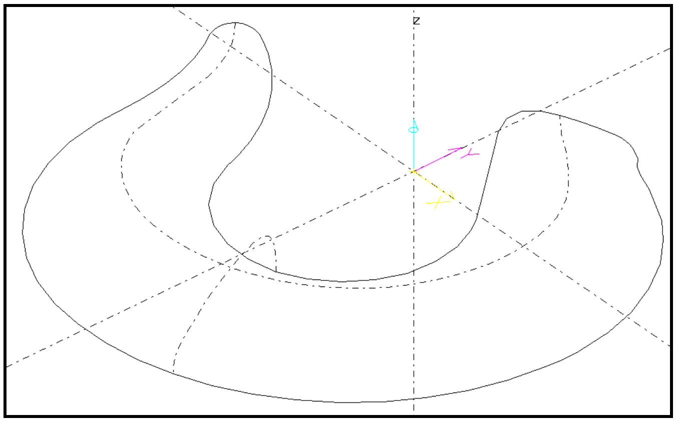


Figure 15-18
Open FFMFLOW.

2. Select **Process—Surface Machine—Flow Line**. The **Flow Line** control panel is displayed.

Figure 15-19
Set the values on the Flow Line control panel.

3. Set the **Surface Element #** input field to the element number of the surface to be machined, or select the surface element from the graphic view or the list view.

4. Set the **Path Dir** selector switch to the direction for the toolpath, as follows:

   - **U-direction** creates the toolpath on the surface of the U direction. The graphic view display of the surface shows the insert markers in the U direction.

   - **V-direction** creates the toolpath on the surface in the V direction.

5. Set the **Spacing** selector switch to the method for creating polylines for the initial mesh, as follows:

   - **Distance** creates polylines at a specified maximum distance apart.

   - **Count** creates a specified number of polylines an equal distance apart.

   - **Mid Tol** creates polylines with a specified maximum chord height tolerance.

6. Set the **Spc Val** input field to the value for the spacing of polylines, as follows:

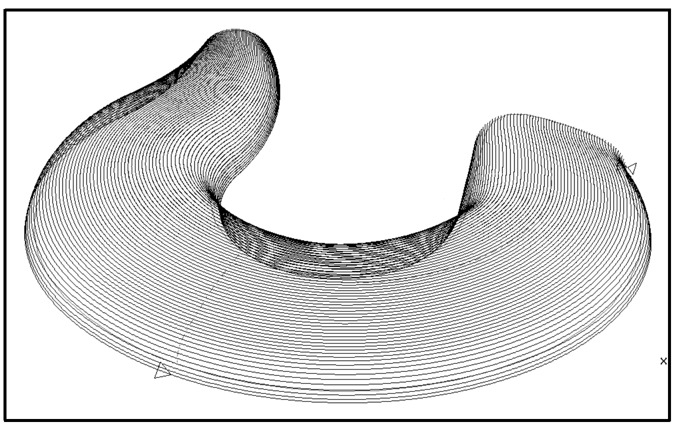
   - **Distance** sets the maximum allowable distance between polylines.

   - **Count** sets the number of polylines to be created in equal spacing across the surface.

   - **Mid Tol** sets the maximum chord height tolerance between mesh curves.

7. Set the **Divisions** selector switch to the method to use for determining the number of segments in the polylines, as follows:
   - **Distance** creates polylines using the specified maximum segment length.
   - **Count** creates a specified number of segments in each polyline at an equal distance apart.
   - **Mid Tol** creates segments using the specified maximum chord height tolerance.
8. Set the **Div Val** input field to the value for creating the polyline segments.
9. Set the **Offset** selector switch to the tool offset direction. When you select the offset direction, arrows are displayed in the graphic view, indicating the offset direction.
   - **Pos** offsets the tool to the positive surface normal side at the center of the surface.
   - **Neg** offsets the tool to the negative surface normal side at the center of the surface.
10. Select the **Go** button.

Figure 15-20
This is a result of Flow Line surface machining.



Creating Contour Cuts Across Multiple Surfaces

Objectives

This lesson shows you how to create a toolpath with a series of profile cuts around multiple surfaces.

Overview

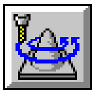
Use Contour to create a precise, gouge-free toolpath that is a series of profile cuts at successive Z levels around the selected surfaces. A toolpath is created for all possible profiles at each level. If there are multiple profiles at the same level, the tool retracts to the clearance plane to move between them, allowing the machining of cavities and islands.

Contour is useful for finish machining curved vertical walls and near vertical walls, with or without islands, in mold cavity and core models. Contour is also useful for roughing hard materials with flat end mills or large roughing mills with inserts, where minimizing tool tip plunging is critical. By using a User Start Point in Rough—Cavity with a drilled hole, there is no plunging of the tool tip into the material.

Contour supports the use of ball, bull, flat, and tapered cutter types.

Contour requires that all surface normals point in the correct direction. If such normals are not oriented properly when you use Surface Side, you may gouge the part.

Using Contour for Multiple Surfaces



Model File: FFMCAV.PM4

1. Open the model file **FFMCAV.PM4**.

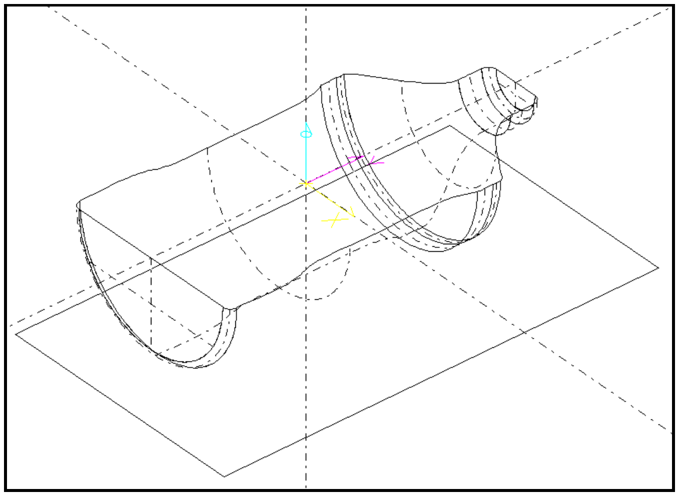
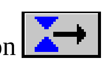
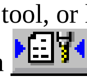


Figure 15-21
Open FFMCAV.

2. Group the surfaces to be roughed.

3. Set the insert location:

   - Set the **Before** icon  or **After** icon  on the Insert property bar to indicate if the insert point is to be before or after the selected element, process step, tool, or layer.

   - Set the **With Step** icon .

4. Select **Process—Surface Machine—Contour**. The **Contour** control panel is displayed.

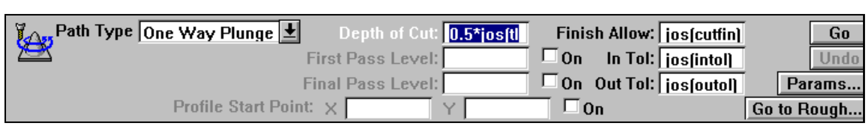


Figure 15-22
Set the values on the Contour control panel.

5. Set the **Depth of Cut** input field to the constant Z distance for each contour pass.

6. Set the **Finish Allow** input field to the thickness of the material to leave on the surface for the finish pass. You can enter a negative number, but the absolute value of the negative finish amount must be less than or equal to the tool corner radius. The active step sets the default value for this input field.

7. Set the **In Tol** input field to the maximum allowable distance for any point on the toolpath to deviate toward the part surface. Use any value greater than or equal to 0. The sum of the **In Tol** and **Out Tol** fields must be at least 0.00001.

8. Set the **Out Tol** input field to the maximum allowable distance for any point on the toolpath to deviate away from the part surface. Use any value greater than or equal to 0. The sum of the **In Tol** and **Out Tol** fields must be at least 0.00001.

9. Set additional parameters as necessary on the **Contour Parameters** dialog box.

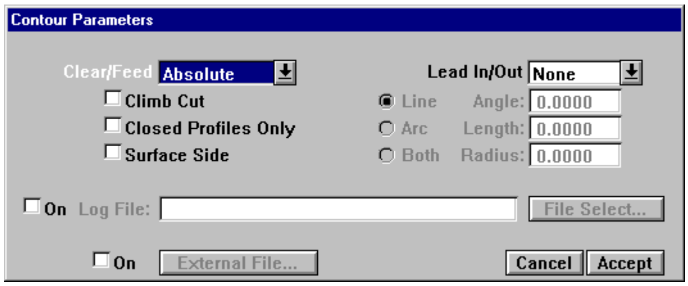


Figure 15-23
Set the values on the Contour Parameters dialog box.

10. Turn on the **Surface Side** on/off switch to use the direction of the surface-normal vector at the center of the surface (from the ACIS model) to determine which side of each surface to machine. Turn off this switch to enable Contour to determine which side of each surface to machine, based on a view of the model from the top.

11. Set the other parameters on the dialog box as necessary.

12. Select the **Accept** button. The parameters you set will be used by SmartCAM for the Contour machining process.

13. Select the **Go** button.

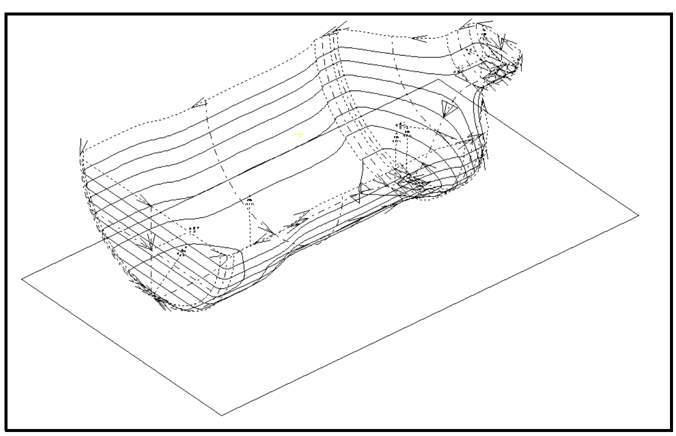


Figure 15-24
This is a result of Contour surface machining.

Points to Remember

- ❑ The number of surfaces that the toolpath can cross is limited only by available memory.
- ❑ Use Projected when you do not want the toolpath pattern produced by straight-line machining or when the topography of surfaces is not suitable for straight-line cutting.
- ❑ Use Contour to create a precise, gouge-free toolpath that is a series of profile cuts at successive Z levels around the selected surfaces.
- ❑ Use Cavity to rough an opening in preparation for mesh profile finishing cuts.
- ❑ You can select either surfaces or toolpath elements (lines or polylines) as the input for the roughing process.
- ❑ There is no gouge correction for Flow Line. Surfaces with severe discontinuities or vertical walls are poor candidates for this function.
- ❑ Contour requires that all surface normals point in the correct direction.
- ❑ Set the Cut Area Point input field to the specific location, *within* the boundary containing the deepest Z level, for cutting to begin.

Self-Test

Directions

Test your understanding of the concepts and procedures in this section by answering the following questions. The answers for each self-test are in *Appendix A* of this manual.

_______________ **1. What is the benefit of using higher U and V values?**

a) redraws faster than lower values
b) displays more lines than lower values
c) consumes less space than lower values
d) there is no benefit

_______________ **2. The generator profile elements need to connect to use the Spun Surface feature.**

a) true
b) false

_______________ **3. What feature should you use to create a surface that is defined by straight lines drawn between two profiles?**

a) Lofted Surface
b) Draft Surface
c) Ruled Surface
d) From Mesh

_______________ **4. What feature should you use to create a surface from an existing polyline mesh?**

a) Lofted Surface
b) Draft Surface
c) Ruled Surface
d) From Mesh

_______________ **5. Render Image shades hidden elements.**

a) true
b) false

_______________ **6. Surface grid settings affect the internal surface structure of a surface.**

a) true
b) false

_______________ **7. Property Change modeling tools require an active group.**

a) true
b) false

_______________ **8. Surfaces can be split to an intersecting curve or a tangent surface.**

a) true
b) false

_______________ **9. How close should the meshes used to generate a blend be?**

a) The meshes should both touch the blend.
b) One mesh should touch the blend.
c) The meshes should be within .001 of the blend.
d) The meshes should completely overlap each other and the blend.

______________ **10. You can punch isolated surfaces.**

a) true
b) false

_______________**11. You should avoid generating a mesh from a surface.**

a) true
b) false

______________ **12. What feature do you use to create profiles on the intersection of a plane with an ACIS solid body?**

a) Lofted Surface
b) Profile Intersection
c) Ruled Surface
d) Slice

______________ **13. When should you use surface editing tools to extract the surfaces?**

a) after creating profiles
b) before creating profiles

______________ **14. If a hole element is created with a drill as the active tool, how is the depth determined?**

a) The full length of the drill becomes the hole depth.
b) The cylinder depth becomes the hole depth.
c) The stock box depth becomes the hole depth.
d) The tolerance value is added to the stock box depth.

______________ **15. What is the maximum number of surfaces that the toolpath can cross?**

a) 0
b) 1
c) not limited
d) limited by available memory

______________ **16. What can you use as an input for a roughing process?**

a) surfaces
b) toolpath
c) both a and b
d) neither a nor b

______________ **17. How do you use gouge protection with Flow Line machining?**

a) Turn on the Gouge Protection option switch.
b) Gouge protection is used by default.
c) Set the Tolerance value.
d) Gouge protection cannot be used with Flow Line.

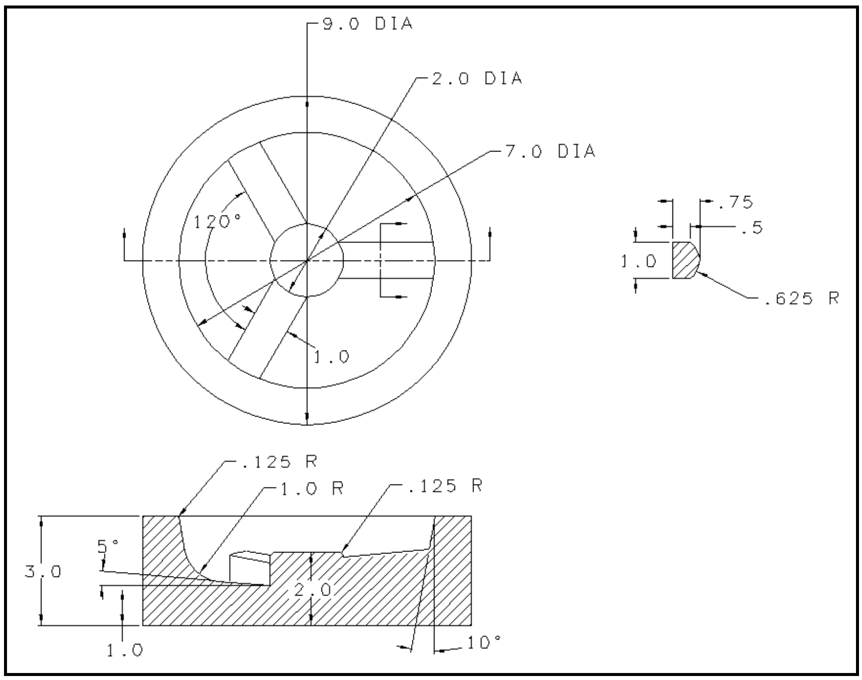
Challenge Project

Directions

Use the part print to build a surface, rough the part, and finish machining the surface.

Part Print

Figure 15-25
Use the part print to build, rough, and finish a surface.



Appendix A: Self-Test Answers

Overview

Use the answers to check your self-tests. Review lessons that correspond to questions that you completed incorrectly.

Production Milling Answers

Becoming Acquainted with SmartCAM

____B____ **1. It is possible to select dimmed menu items.**
Lesson *Using Workplace Areas,* page 1-4

____C____ **2. What does an asterisk (*) in the control panel indicate?**
Lesson *Using Workplace Areas,* page 1-4

____C____ **3. Which of these workplace items does each control panel correspond to?**
Lesson *Using Workplace Areas,* page 1-4

____A____ **4. What do process model files graphically represent?**
Lesson *Learning SmartCAM File Types,* page 1-17

____B____ **5. What do job operation files contain?**
Lesson *Learning SmartCAM File Types,* page 1-17

____C____ **6. Which View features magnify specific areas of the view?**
Lesson *Using Window,* page 1-22, *Using Zoom,* page 1-23

____D____ **7. What View features manipulate the orientation of the view?**
Lesson *Using Pan,* page 1-24, *Using Full,* page 1-24, *Using Last View,* page 1-24, *Using Get View,* page 1-24, *Using Name View,* page 1-26, *Using Dynamic View,* page 1-27

____B____ **8. What feature do you use to control how the geometry is displayed in the graphic view?**
Lesson *Using Display Modes,* page 1-27

Working with SmartCAM

B 1. **Layer geometry generates code.**
Lesson Overview, page 2-3

A 2. **Step geometry generates code.**
Lesson Overview, page 2-3

B 3. **Which input field defines the height that the tool retracts to after making a cut?**
Lesson Setting Levels, Clearance, and Profile Top, page 2-10

C 4. **Which input field defines the height of the top of your part?**
Lesson Setting Levels, Clearance, and Profile Top, page 2-10

C 5. **What type of plane determines the orientation of geometry?**
Lesson Overview, page 2-13

B 6. **What type of plane determines the plane the cutter axis is perpendicular to?**
Lesson Points to Remember, page 2-18

B 7. **Which SmartCAM feature enables you to use points and other element properties from existing geometry to enter coordinate values in input fields when you create or edit geometry?**
Lesson Operating Snap, page 2-20

B 8. **Using Status interrupts the current task.**
Lesson Checking Levels, Clearance, and Profile Top, page 2-11

A 9. **How do you know if tools require groups if there are no active groups?**
Lesson Using the Group Arrow Icon, page 2-24

Using Job Operations

C 1. **What pieces of job information are required?**
Lesson Overview, page 3-3

D 2. **What values must be set to complete a process step?**
Lesson Overview, page 3-3

B 3. **How many steps or tools can you move at a time when you use the Move feature?**
Lesson Moving Steps and Tools, page 3-22

A 4. **You cannot remove active steps from a job.**
Lesson Removing Steps and Tools, page 3-21

B 5. **Where do you define material information that you want to use on a regular basis?**
Lesson Setting Up a File with the Material Librarian, page 3-29

A 6. **Once material information is set up, how do you access it from Production Milling?**
Lesson *Opening a Material Librarian File,* page 3-38

A 7. **Which report should you print if you want to know about the job information for the open process model?**
Lesson *Printing a Job Information Report,* page 3-42

Working with Elements

C 1. **What do you need to do before creating new geometry?**
Lesson *Overview,* page 4-4

A 2. **Associating geometry with a step will result in code.**
Lesson *Overview,* page 4-4

B 3. **Associating geometry with a layer will result in code.**
Lesson *Overview,* page 4-4

B 4. **You can select and code masked elements**
Lesson *Using Show/Mask,* page 4-48

D 5. **How many elements in a model can you view modeling data for?**
Lesson *Using Element Data,* page 4-29

B 6. **Viewing element data affects the geometry database.**
Lesson *Overview,* page 4-29

A 7. **Changing the color of elements:**
Lesson *Using Color Change,* page 4-50

B 8. **Elements to be trimmed or extended should be on different work planes.**
Lesson *Trimming and Extending Geometry,* page 4-32

B 9. **How many groups does trimming by group impact?**
Lesson *Trimming a Group of Elements,* page 4-34

B 10. **How many deleted groups can be recovered with the Undo button?**
Lesson *Deleting Elements,* page 4-43

D 11. **Which feature verifies the sequence and toolpath before you cut parts?**
Lesson *Showing the Toolpath,* page 4-54

A 12. **When you use Optimize, should you save the file before or after selecting modeling tools?**
Lesson *Optimizing the Order of Hole Operations,* page 4-62

Generating and Verifying Roughing Toolpath

A 1. **More that one island can exist inside closed profiles that you use for pocketing.**
Lesson Pocketing, page 5-4

B 2. **Which feature do you use to create roughing profiles for removing material when you want the tool to overlap the outside of a profile boundary?**
Lesson Facing a Part, page 5-10

A 3. **How many islands can you have for Open Profile machining?**
Lesson Cutting an Open Profile, page 5-13

D 4. **What feature do you use to verify roughing toolpath?**
Lesson Overview, page 5-23

B 5. **How do you determine the size of the stock box to create?**
Lesson Overview, page 5-23

Generating Code

B 1. **.tmp files contain this information:**
Lesson Learning File Types for Code Generation, page 6-3

A 2. **.smf files contain this information:**
Lesson Learning File Types for Code Generation, page 6-3

B 3. **Code is generated from the process model in this form:**
Lesson Learning File Types for Code Generation, page 6-3

B 4. **SmartCAM generates code for all step-property elements in the database.**
Lesson Learning File Types for Code Generation, page 6-3

D 5. **SmartCAM ships this text editor:**
Lesson Learning File Types for Code Generation, page 6-3

Advanced Milling Answers

B 1. **Selecting the first and last element of the profile curve selects only those elements and the most central element.**
Lesson Overview, page 7-1

B 2. **When polylines are used as profile curves, the boundary tangent angle is used.**
Lesson Using Tangent Profiles, page 7-2

D 3. **Can non-tangent profiles be edited to become tangent?**
Lesson Using Non-Tangent Profiles, page 7-3

A 4. **How many gaps can be present in a profile curve?**
Lesson Checking for Gaps in Profile Curves, page 7-4

D 5. **How many of the profile's elements should connect to make a surface mesh profile?**
Lesson Creating a Spun Mesh, page 8-2

A 6. **For translated meshes, is the director curve parallel to the generator curve?**
Lesson Creating a Translated Mesh, page 8-6

A 7. **When you generate a coons patch mesh, how many sharp corners can you have?**
Lesson Creating a Coons Patch Mesh, page 8-17

C 8. **When you extend existing mesh profiles, do the properties and sequence for the selected range change?**
Lesson Extending Existing Mesh Profiles, page 9-11

C 9. **When you insert a solid element with a step, how does SmartCAM determine the offset?**
Lesson Overview, page 10-9

D — **10. What does creating slice profiles change?**
Lesson Overview, page 10-13

B — **11. Use surface editing tools to extract the surfaces:**
Lesson Overview, page 10-13

A — **12. If a hole element is created with a drill as the active tool, how is the depth determined?**
Lesson Overview, page 10-15

D — **13. How close should the meshes that you blend be?**
Lesson Overview, page 10-19

A — **14. Surface grid settings affect the visual representation.**
Lesson Overview, page 11-7

C — **15. Is it easier to generate and clean up the planar cuts if they are generated from untrimmed meshes?**
Lesson Creating Planar Cuts across Meshes, page 12-3

B — **16. How many roughing passes are made for each planar cut level?**
Lesson Roughing a Cavity, page 12-5

FreeForm Machining Answers

B 1. **What is the benefit of using higher U and V values?**
Lesson *Controlling Surface Construction*, page 13-7

A 2. **The generator profile elements need to connect to use the Spun Surface feature?**
Lesson *Creating a Spun Surface*, page 13-9

C 3. **What feature should you use to create a surface that is defined by straight lines drawn between two profiles?**
Lesson *Creating a Ruled Surface*, page 13-13

D 4. **What feature should you use to create a surface from an existing polyline mesh?**
Lesson *Creating a Surface From a Mesh*, page 13-17

B 5. **Render Image shades hidden elements.**
Lesson *Overview*, page 14-3

B 6. **Surface grid settings affect the internal surface structure of a surface.**
Lesson *Overview*, page 14-7

A 7. **Property Change modeling tools require an active group.**
Lesson *Overview*, page 14-7

B 8. **Surfaces can be split to an intersecting curve or a tangent surface.**
Lesson *Overview*, page 14-3

A 9. **How close should the meshes used to generate a blend be?**
Lesson *Overview*, page 14-5

A 10. **You can punch isolated surfaces.**
Lesson *Overview*, page 14-13

___**B**___ **11. You should avoid generating a mesh from a surface.**
Lesson Generating a Mesh from a Surface, page 14-19

___**D**___ **12. What feature do you use to create profiles on the intersection of a plane with an ACIS solid body?**
Lesson Overview, page 16-13

___**A**___ **13. When should you use surface editing tools to extract the surfaces?**
Lesson Overview, page 16-13

___**A**___ **14. If a hole element is created with a drill as the active tool, how is the depth determined?**
Lesson Overview, page 16-15

___**D**___ **15. What is the maximum number of surfaces that the toolpath can cross?**
Lesson Overview, page 15-3

___**C**___ **16. What can you use as an input for a roughing process?**
Lesson Overview, page 15-15

___**D**___ **17. How do you use gouge protection with Flow Line machining?**
Lesson Overview, page 15-19

Glossary

Accept

Select this action button to accept entries into the active control panel.

ACIS, see *Surface*

An industry standard geometry kernel that is used to store solids and Non-Uniform Rational B-Spline (NURBS) surfaces. ACIS technology is unique for these reasons:

- It stores the surface and solid geometry in a single database.
- It represents objects by storing their topological features, such as vertices, edges and faces, and the corresponding geometric features such as curves, points, and surfaces.
- It maintains the spatial relationship between object features.
- It can represent both manifold (closed) solid objects and nonmanifold (open) objects such as surfaces.
- It can easily exchange solid models between the CAD and CAM systems that use it.

ACIS Face

Surface elements are stored in ACIS solid objects as faces. The ACIS face contains topological information about vertices and edges, and geometric features such as points, curves, and surfaces. The spatial relationship between solid object features is maintained.

Action Button

Control panel token that is labeled according to what action it carries out.

Active Group

The elements selected for an operation.

After

Setting on the Insert property bar. Used with the Before switch. One of the switches must be selected when you create new geometry to determine where a new element is to be placed in relation to an existing element, profile, step, tool, or layer.

Air-Cut Time

The time between when the tool starts to feed and when it starts cutting stock.

Area of Uncut Material (AUM)

The difference between the actual and nominal net shapes represents the area of uncut material. (For example, when the selected tool is too large to cut all areas of an operation.) The user defines the nominal net shape by indicating a set of profiles while creating a machining process.

ATS (Automated Transfer Specification)

A binary file format used in CAMAX's Camand applications.

Axial Length

Distance input. Enter the length of the thread as measured parallel to the Z axis.

Ball Mill

Conventional end mill with a full radius bottom (corner radius equals tool radius). It may be straight (cylindrical) or tapered (conical).

Base Envelope

A viewing window which is defined when you start a new model. It enables you to redisplay the intended part size quickly after using the various viewing options.

Base Surface Edges

The curves that define the outer edges of the underlying surface for a trimmed surface.

Before

Setting on the Insert property bar. Used with the After switch. One of the switches must be selected when you create new geometry to determine where a new element is to be placed in relation to an existing element, profile, step, tool, or layer.

Boring

The Boring operation type provides the primary operation parameters for single-point boring operations. Use it with single-point boring tools.

Boring Tool

A single-point tool used to finish predrilled holes to a precise size tolerance.

Boundary Profile

The profile enclosing the material to be removed.

Boundary Vector

The indicator that shows the direction and length of one side of a surface extension in the Edit—Surfaces—Extend function. When you select a surface edge to extend, the boundary vectors are tangent to the surface curvature at the end points of the selected edge.

When you select contiguous continuity, the boundary vectors determine a direction along which to make an offset copy of the selected edge curve. The resulting surface is ruled between the edge curve and the offset copy.

When you select tangent continuity, each boundary vector is tangent to the surface curvature and normal to the edge curve at the end point. The resulting surface remains tangent to the surface along the edge curve. The direction is determined by interpolation between the start and end vectors.

Bridge Gaps, see *Surface*

A surface machining option that allows you to specify how to handle gaps and overlaps between NURBS surface elements. There are two options: Straight and Clear/Feed. The Straight option creates machining moves over any gap to the nearest point on the surface on the other side. The Clear/Feed option performs a retract to an absolute or incremental Z level to jump over the gap. Overlaps are recognized, so that the topmost surface is machined and the underneath surface is not. By selecting bridge gaps, you avoid gouging your part surface. It is a FreeForm Machining term.

Bull Mill

Conventional end mill with a flat bottom and rounded corners. May be either straight (cylindrical) or tapered (conical).

C0 Continuity

Signifies that two adjacent surfaces or curves are contiguous (their end points coincide).

C1 Continuity

Signifies that two adjacent surfaces or curves are tangent.

C2 Continuity

Signifies that two adjacent surfaces or curves have a constant rate of curvature.

Cancel

Select this button to close the dialog box or panel without saving any changes.

Canned Cycle

A series of preset directions, or machine cycle, that a machine control uses to perform repetitive tasks. The function is called by a code containing information about where to place the event. For example, hole cycles are typically between the G76 and G89 code numbers.

Cavity

A volume of material bounded by a set of surfaces.

CCW (Counterclockwise)

Sets the rotation direction of a tool, or the toolpath direction along an arc element.

Center Drill

A stepped drill with an outer body diameter and a smaller drill diameter that are connected by a 60-degree, included point angle. In some cases the transition between the 60-degree point angle and the outer body diameter is broken by a 120-degree point angle beginning at the bell diameter. This tool type is typically used to create a precision center location for subsequent turning or grinding operations.

Center Drilling

The Center Drilling operation type provides the primary operation parameters for center drilling.

Center UV Curves

The curve in the U direction and the curve in the V direction that are at the center of a surface element.

Chordal Deviation

The distance between the midpoint of an arc and the midpoint of a line drawn between the ends of that same arc. This distance is the width of the tolerance band that SmartCAM should use when interpolating arcs or curves into lines. The band restricts the size of the polyline segments that SmartCAM creates when approximating curves. The larger the size of the band, the greater the chordal deviation and resulting polyline segments.

Clean-up Pass

Cutting tool performs a clean-up pass on the profile to remove any inconsistencies in the surface after roughing.

Clear

The Z-height necessary for the tool to make positioning moves safely. The point from which the tool begins to feed.

Climb Cut

Roughing cut option in which the tool's rotation pulls the tool in the same direction as the cutting motion. This is the opposite of conventional cutting in which the tool rotation pushes the tool in the opposite direction from the cutting motion.

Close

Use to close an active panel. If changes are not saved, a prompt will be displayed, enabling you to save or discard the changes.

Construction Geometry

The geometric elements and the construction method used to create a surface using the Create—Surface modeling tools. SmartCAM stores construction geometry inside the surface when it is created.

Containment Boundary

A 2-D profile which is projected onto 3-D part surfaces to define a limited area within which to machine.

Contour Machining

A method of machining that creates precise, gouge-free toolpath that is a series of profile cuts at successive Z levels around the selected NURBS surfaces. toolpath is created for all possible profiles at each level. If there are multiple profiles at the same level, the tool retracts to the clearance plane to move between them, allowing the machining of cavities and islands. Contour machining retracts the tool when gaps are encountered and recognizes the machine overlaps. Contour machining is useful for finish machining curved vertical walls or protrusions (for example, islands and bosses) in mold cavity and core models, as well as for roughing hard materials with flat end mills or large roughing mills with inserts.

Control Panel

Each modeling tool has its own control panel, with input switches and buttons to operate the tool. A control panel appears at the bottom of the screen display when a tool is selected and remains open as long as the tool is active.

Conventional Cut

Roughing cut option in which the tool's rotation pushes it in the opposite direction from the cutting motion. This is the opposite of climb cutting in which the tool's rotation pulls it in the same direction as the cutting motion.

Coons Surface

A composite surface defined by a starting profile, an ending profile, and any number of interior profiles that define interior patch boundaries.

Copy Milling

The Copy Milling operation type provides the primary operation parameters for creating toolpath on multiple surfaces. Use Copy Milling for creating FreeForm Machining multi-surface machining operations.

Counterbore

A piloted flat bottom cutting tool used to create counterbores in existing holes. This tool is often used to create recesses for cap screw fasteners.

Counterboring

The Counterboring operation type provides the primary operation parameters for counterboring an existing hole. Use Counterboring with a piloted counterboring tool.

Countersink

A short tool used to create chamfers at the top of existing holes. This tool typically contains an 82-degree, included point angle, and it is primarily used to create recesses for flat head screw fasteners.

Countersinking

The Countersinking operation type provides the primary operation parameters for countersinking an existing hole. Use with 82-degree counterbore tools.

Crest Start

Distance input. Enter the outside radius if it is an external thread or the inside radius if it is an internal thread. When a face-grooving tool is active, Crest Start applies to a Z-axis value.

Cursor

The location indicator on the screen, moved by manipulating the mouse; the cursor is displayed in one of five forms, depending on its location.

CW (Clockwise)

Sets the rotation direction of a tool, or the toolpath direction along an arc element.

Decrement by Z Distance

The distance between Z levels in a Surface Machine—Roughing toolpath. To create surface roughing toolpath, first define a starting Z level (typically the top of the part) and a Decrement by Z Distance value. During creation of the roughing toolpath, the Decrement by Z Distance value is subtracted from the current Z level for each subsequent pass until all surfaces of the part are cut (that is, the roughing toolpath has reached the bottom of the part). Roughing projects a linear toolpath (similar to Straight) down onto the surfaces, and then projects the toolpath elements up to each of the defined Z levels. The Z levels are ordered from top to bottom, so that the material is cut away one Z level at a time.

Delta Major

The radial distance from the Nominal diameter to Major diameter.

Delta Minor

The radial distance from Nominal diameter to Minor diameter.

Design Model File

Enter the name of the design model file created by your CAD program. If you need to search for the file, use the File Select action button at the top of the form.

Design Ref

Represents the functional design perspective of the part.

Dialog Boxes

Dialog boxes open temporarily to allow selection or control of a variety of events. A dialog box may appear anyplace on the screen display and closes when its action is completed or canceled. If a dialog box is open, you must complete the required action or cancel it before doing anything else to the model.

Dimmed Text

When the text for a menu item is dimmed, you cannot select the menu item until some other action is performed. For example, most of the options on the top menu bar remain dim until you load a model file or until a current group is active.

Director Curve, see *Generator Curve*

The curve used to control the direction or path along which to sweep a generator curve to define the shape of a surface or mesh.

Display Curve Tol

The setting that controls the maximum amount that the display grid curves can deviate from the exact surface. FreeForm Machining calculates the tolerance from the chord height of each grid curve polyline segment.

Display Grid

The set of isoparametric curves in the U and V directions of the surface that SmartCAM uses to display the surface. You can change the settings for the number and tolerance of curves in the Utility—Surface Modes or Edit—Property Change—Surfaces dialog boxes.

Double D Punch

The tool type used to enter the tool parameters for a Double D punching tool. The width is the distance between the flats.

Draft Surface

A surface defined by sweeping a 2-D generator profile along a director curve. The generator rotates as it sweeps so that its projection remains perpendicular to the 2-D projection of the director curve into the active tool plane.

Drilling

The Drilling operation type provides the primary operation parameters for basic hole drilling. Use Drilling to create holes that can be completed with a single feed motion.

Edge Milling

The Edge Milling operation type provides the primary machining parameters for edge machining. Use Edge Milling for machining chamfers or round corners in customized machining procedures (such as macros).

Element

The toolpath or boundary definition that SmartCAM places in the database and displays in the list view and Element Data list; element types include lines, arcs, polylines, splines, user events, sub calls, holes, points, ellipses, and helixes.

Element Data

Information about the element coordinates and properties in a model are stored in the database.

End Mill

Conventional end mill with a flat bottom and sharp corners. Cutting edges may be either straight (cylindrical) or tapered (conical).

Evaluator Expression

SmartCAM's evaluator function uses standard mathematical functions and the assigned expressions for each field to perform calculations. The expressions are system terms representing known measurements. For example, tl(tl_len) represents tool length, and it can be multiplied to equal another input such as 3*tl(tl_len).

External Groove Tool

A tool used to machine grooves on the outside diameter of a turned part.

External Thread Tool

A tool used to create threads on the outside diameter of a turned part.

External Turn Tool

A tool used to machine the outside diameter of a turned part.

Face Groove Tool

A tool used to machine grooves on the face of a turned part.

Face Grooving

The Face Grooving operation type provides the primary machining parameters for adding a groove to the face of a turned part.

Face Mill

Arbor-mounted mill intended for machining large flat surfaces. Typically contains replaceable carbide inserts with rounded corners. Cutting edges may be either straight (cylindrical) or tapered (conical).

Face Milling

The Face Milling operation type provides the primary machining parameters for face milling. Use it to create toolpath with the Process—Rough—Face option in SmartCAM.

Facing Tool

A tool used to machine the end of a turned part.

File Select

Select this to open the File Select dialog box to search for a file. This button can only be selected if the active field requires entry of a file name.

Finish Allowance

Used for Rough, Face, and Surface Milling operations. Enter the thickness of the material to be left on the part after the machining operation.

Finish Amount

The amount of material to be left on the sides of the boundary profile after roughing. (This material can later be removed with a finish pass.)

Finish Milling

The Finish Milling operation type provides the primary machining parameters for finish profile machining. Use Finish Milling for machining any SmartCAM finish profile.

Finish Pass

The cutting moves made next to a desired profile to complete the part's shape. A finish pass occurs next to the pocket's outer profile and next to each island profile.

Finish Turning

The Finish Turning operation type provides the primary machining parameters for finishing internal or external turning, face, or profiling operations.

Form Hole

Special tool containing a custom edge profile. This tool type typically contains multiple diameters and shoulders, and it is used to finish existing predrilled holes to a specific shape.

Form Hole Making

A hole making operation that uses a tool with multiple cutting diameters.

Form Mill

Special end mill with a custom cutting edge profile. This tool type typically has a non-cutting body diameter as well as a cutting tool diameter. Use the Form Mill tool type to define tools such as Chamfer Mills, Radius Mills, and similar specialty tools.

From Mesh Surface

A FreeForm Machining surface defined by fitting a surface through the set of points defined by the segments of an existing polyline mesh.

Generator Curve

The curve that is swept through space, usually along the path of a director curve, defining the shape of a surface or mesh.

Go

Activates the new sequence according to the input values.

Graphic View

The graphic view provides a visual representation of the process model. Additions or changes to the graphic view are reflected immediately. The displayed model may be viewed from any angle, rotated, moved, enlarged, or panned. You can view the toolpath and make changes quickly and easily.

Grooving

The Grooving operation type provides the primary machining parameters for adding grooves to the internal diameter or external diameter of a turned part.

Group

Use Group to create an active group of elements. You can use the Group Arrow icon or the Group tool palette found on the readout line below the graphic view.

Help

Online Help provides information about each menu item, toolbox, modeling tool, control panel, and dialog box.

Hot Keys

Shortcut keystroke combinations that provide a quick way of performing a task or setting a mode of operation. The hot key assignments shipped with SmartCAM and information about customizing hot keys are available from the Help menu Keyboard option.

Icon Bar

The icon bar provides shortcuts for accessing SmartCAM functions.

IGES

A universal graphics file format that is used to convert CAD/CAM files from proprietary software systems to a format that can be read by other systems.

Input Fields

Control panels and dialog boxes contain input fields for specific types of information, such as a file name, a line angle, or coordinates. An input field can contain various types of information.

Insert

The Insert function enables you to add geometry. You can add geometry with a step or on a layer.

In Tol

The inner machining tolerance, which is the maximum allowable deviation of the toolpath on the part side of an element or surface. The In Tol and Out Tol values work together to establish a tolerance band or overall machining tolerance for the toolpath. Part of the tool always remains within the tolerance band as it cuts. No part of the tool ever crosses the In Tol boundary. The two values must be zero or positive, with at least one of them being positive.

Internal Groove Tool

A tool used to machine grooves in the inside diameter of a turned part.

Internal Thread Tool

A tool used to create threads on the inner diameter of a turned part.

Internal Turn Tool

A tool used to machine the inside diameter of a turned part.

Intersecting Surface

A single surface element or a group of surfaces that are used to cut through a surface or surfaces in a splitting, trimming, or other surface editing operation. For example, in a splitting operation, the surface elements are cut by an intersecting surface, much like a knife can be used to cut through a slice of cheese.

Intersection Curve(s)

A single polyline curve or a series of curves that make up a profile that defines the intersection of two NURBS surface elements. (These can be single surfaces or groups of surfaces.) An intersection curve theoretically lies on both surfaces and is an approximation arrived at by using an internal tolerance. Thus, if the group of surfaces to edit is split by an intersecting group of surfaces, the intersection curve lies on the intersection of the two sets of surfaces. FreeForm Machining term.

Isoparametric Curve

A curve that follows the flow of a surface at a constant U or V parameter value.

Job Operations File (`.jof`)

The file that contains the job operations setup information. It is saved when you save the associated process model file. The file has the same file name as the model file and is assigned a `.jof` extension. To use a different job operations setup file, use File—Load Job File to find and load the desired .jof file.

Job Operations Setup

The information about tools, operations, and machines to use for machining a part. It is stored in the job operations file (`.jof`). To enter job operations setup information, select File—Planner to open the Job Operation Planner.

Job Plan

In earlier versions of the software, the job plan file contained the tooling information used by the process model. It had a `.jof` extension. If you open a model file that uses a job plan file, SmartCAM will use the information in the **`.jsf`** file to create a job operations setup (`.jof`) file.
SmartCAM CAM Connections still require the use of a .jsf file when converting CAD files.

Layers

Layer information is CAD geometry. Each layer in a model has one of sixteen colors, and the maximum number of layers is 99. Items such as clamps, fixtures, and material boundaries are examples of geometry to draw on layers and do not represent toolpath. In addition, no tool or operation information is associated with layer geometry.

Level

A specific position on the Z axis of the active work plane's local coordinate system.

LH Tool

Left-hand tool. A tool is left-handed if its flutes twist away from the observer in a counterclockwise direction when viewed from either end of the cutter. Cutting occurs on the left-hand side of the tool for Climb cutting and on the right-hand side for Conventional cutting.

List View

A list of the elements, tools, steps, work planes, or layers that comprise the process model is displayed in the list view. To change the type of list, select the appropriate tool on the workbench. Additions or changes can be made to items on any of the lists. Changes are displayed in the graphic view.

Live Tooling

Some turning centers include motorized turrets with the capability to perform milling/drilling operations. Live tooling describes the action of the turret's rotary cutting tool motion as opposed to the normal stationary cutting position. The tool is "live" because it rotates.

Lofted Surface

A surface defined by blending through a series of curves.

Log File

A file that FreeForm Machining creates to record the messages and errors generated while importing files, exporting files, or processing surface machining functions.

Lookup

This action button calculates values for Crest Start, Root Start, Root End, and 1st Pass Depth. The information comes from the thread table file, using the Nominal Diameter and Pitch values that you enter.

Machine File

Machine files have an `.smf` extension and consist of a list of questions and options that tell SmartCAM how to format code for your machine. For information about machine files see the *SmartCAM Code Generation Guide*.

Match View

Match generated view. A Show Cut option which returns the generated view to match the existing graphic view orientation.

Material Elmt.

Text/Selection input. Select an element in the material profile. This defines the material boundary profile of which the element is a part.

Menu Bar

The application menu is displayed as a bar across the top of the SmartCAM screen display. Select items from the menu bar to display pull-down menus with selections to open toolboxes, dialog boxes, or submenus.

Mesh

A wireframe approximation used to represent a surface. If you create a mesh using a process step, the mesh represents toolpath.

Mill-Turn

A turning center (machine) that can do both turning and milling operations.

Model Space

The three-dimensional coordinate system used to build the model. The model remains fixed with respect to this coordinate system. Rotating a view is achieved by rotating the geometric model and the model space axes.

NC File

Enter the name of the numeric control code file to be created.

Nested Pocket

A pocket that lies entirely within an island profile.

Nominal Diameter

This is a distance input that specifies the diameter (nominal size), which is the general identification of a thread.

Nub

A special area of uncut material created during spiral roughing at the corner of adjacent roughing passes when the width of cut is greater than the tool radius.

Number of Passes, see *Material Elmt.*

Integer input. Enter the number of cutting passes you want to make. This field is dim if you select a material boundary.

NURBS (Non-Uniform Rational B-Spline Curves)

A method of representing complex sculptured shapes. NURBS geometry can exactly represent points, lines, arcs, conics, Bezier geometry, and conventional B-Spline geometry. A NURB can represent an entire arc or conic without approximation.

Offset Side

The side of the primary profile to which the wire is to be offset as viewed from the direction of tool travel on the primary profile. The choice are Left, Right, and None; None does not offset the tool.

On/Off Switches

On/Off switches are used to set a mode or turn a function or input field on or off. An on/off switch is on when an x appears in the box and off when the box is empty. You can have more than one on/off switch turned on at a time. Click the mouse on a switch to turn it on or off, or TAB to the box and press Enter.

Operation

The process parameters used with a cutting tool to perform a process step.

Option Switches

Option switches enable you to select one of a pair or group of options. Option indicators are round and become highlighted when selected. To select an option from the keyboard, press the Tab key until the desired input field is highlighted.

Out Tol

The outer machining tolerance, which is the maximum allowable deviation of the toolpath on the tool side of an element or surface. The Out Tol and In Tol values work together to establish a tolerance band or overall machining tolerance for the toolpath. Part of the tool always remains within the tolerance band as it cuts. The two values must be zero or positive, with at least one of them being positive.

Outer Profile

A profile that is either an outermost profile or the first profile that is entirely within an island profile.

Outermost Profile

A profile that lies entirely outside any other profile.

Overlap Passes

This is an on/off switch that creates an overlap along the profile after each roughing pass. This eliminates the “stair stepping” that can occur on the profile during roughing passes.

Peck Drilling

The Peck Drilling operation type provides the primary operation and increment parameters for deep hole drilling. Use Peck Drilling for creating holes that require multiple in-feed moves to complete.

Pinch-Turning

This is an industry term to describe two tools simultaneously cutting on opposite sides of a part. The equalized tool pressure stabilizes the material, minimizing deflection, and cuts the part twice as fast.

Pitch (Milling)

The pitch of the tap in inches or a metric unit.

Pitch (Turning)

The distance along the Z axis between adjacent thread crests. The pitch of a thread is the lead divided by the number of starts.

Pocket

A false cavity represented by a set of profiles which contain an outer profile and any number of island profiles.

Point Set Fitting

The tolerance to use for fitting a NURBS surface through a set of points. The tolerance is calculated by measuring the distance from one of the points to the exact point on the surface along a normal vector. This setting is used for creating surfaces with the Draft, Coons, and From Mesh surface creation tools.

Polyline Point

A connecting point between two polyline segments in the wire’s movement or toolpath.

Pressure Angle

The contact angle between the involute spline curve of the gear tooth. The available pressure angles are 14.5, 20, 25, 30, 37.5, and 45 degrees. The default is 14.5.

Primary Range

Values used to identify the first and last elements of the primary profile. The primary range must not overlap with the secondary range.

Process Model

The interactive graphic image you build which represents true tool motion. It is used to create and modify the actual process for the cutting of parts.

Process Model File

The part model file produced by SmartCAM.

Process Plan Step List

The list of steps which make up one process.

Process Step

The combination of a tool and an operation used to perform one step in a process. Also called a *step*.

Production Turning

Product name for the turning package without any advanced capabilities.

Prof Top

Identifies the top of the profile (thickness) for layers and milling operations.

Profile

A series of related elements linked to create a single geometry feature. Elements in a profile must:

- be linked (The end point of one element is the start point of the following element.)
- be in the same direction
- be assigned to the same tool or layer
- be on the same work plane

An open profile has a separate start point and end point. It may be used to define the outline of a part or a feature. In a closed profile the start point and the end point are at the same coordinate location, thus creating a closed feature such as a pocket or an island.

Profile Start Point

The point on a profile on which the toolpath for that profile begins.

Project

Tool that allows you to project geometry to a plane. The projection takes place along the Z axis of the active work plane.

Properties

Assigns machining parameters such as tool selection, depths, tool offset direction, and machine/control behavior to the toolpath.

Pull-down Menu

Selection of a menu item followed by four dots (::) opens a toolbox. Selection of an item followed by three dots (...) opens a dialog box to perform a specific task. Selection of an item followed by a triangle opens a submenu offering further selections.

Rapid Traverse Speed

The rapid speed for the machine.

Read-out Line

The read-out line is visible below the bottom left corner of the graphic view and occasionally in the control panels. The information displayed depends on the task in progress.

Reamer

A non-center cutting multifluted tool used to finish predrilled holes to a precise size tolerance.

Reaming

The Reaming operation type provides the primary operation parameters required to generate precise diameters for existing holes. Use Reaming for reaming holes.

Regen

Regenerate. Erases the results of the previous simulation and returns the display of the material and fixture layers to the window.

Region

An area of material which can be removed by a continuous toolpath without tool retractions.

Remove synch

Modeling tool that enables you to select and delete a pair of matching Wait commands, or all Start, End, and Wait synch commands associated with the active group. Use this tool when you want to modify existing synchronized operations.

Reset

Returns the input fields to their previous settings or default entries at the time the panel was opened. Use Reset or Revert to clear entries before you enter new information.

Revert

Returns the graphic view display to what it was when you opened the dialog box.

RH Tool

Right-hand tool. A tool is right-handed if its flutes twist away from the observer in a clockwise direction when viewed from either end of the cutter. All tools are assumed to be right-handed. Cutting occurs on the right side of the tool for Climb cutting, and on the left side for Conventional cutting.

Rough Milling

The Rough Milling operation type provides the primary machining parameters for roughing procedures. Use Rough Milling to rough out pockets and create other roughing toolpath.

Rough Turning

The Rough Turning operation type provides the primary machining parameters for roughing procedures during external and internal turning, facing, or groove making operations.

RPM

Select to specify the speed mode.

Ruled Surface

A surface defined by straight lines drawn between corresponding segments of two profiles.

SAB file

An ACIS binary file which uses an **`.sab`** file extension.

SAT file

An ACIS text file which uses an **`.sat`** file extension.

Secondary Range

Values used to identify the first and last elements of the secondary profile. The secondary range must not overlap the primary range.

Section

The set of profiles in a cavity or pocket created by intersecting the cavity or pocket with a plane at a specified level.

Selector Switches

Selector switches provide short lists of options in a control panel or dialog box. Move the pointer over the selector switch, and click the mouse to display the list of options.

Sequence

Specifies the order of machining operations and toolpath, such as roughing/finishing, drilling/tapping, or feed/speed changes.

Slitting

The process of cutting along single or multiple lines with a single-edge or a gang of circular blades.

Slotting

The process of cutting or punching an elongated hole or rectangular slug.

SMF File

A file used by SmartCAM to create machine code. The **`.smf`** file contains information about the machine that will manufacture the part. For more information about machine files, refer to the *SmartCAM Code Generation Guide*.

Snap

The Snap on/off switch appears in the read-out line below the graphic view. An x appears in the box when Snap is on. The box is blank when Snap is off. When Snap is on, the cursor "snaps" like a magnet to the nearest element and displays the element's X, Y, and Z coordinate values. Use Increment to set the pick limit and tolerance for Snap. (See *Increment* in the reference manual for your application.)

Solution Indicators

SmartCAM uses asterisk (*) symbols in the input fields to indicate that entering a value in the field will trigger a solution.

Spindle

The rotational part of a turning machine that holds the part.

Spot Drill

A short, stout drill used to create accurate start hole locations for subsequent drilling operations. This tool typically contains a 90-degree included point angle, and it is often used to spot the hole location to a depth large enough to leave a chamfer on the top of the finished hole.

Spot Drilling

The Spot Drilling operation type provides the primary operation parameters for creating accurate start locations for subsequent drilling procedures. Use Spot Drilling with 90-degree spot drilling tools.

Spot Facing

The Spot Facing operation type provides the primary machining parameters for creating a flat surface in preparation for subsequent machining operations. Use Spot Facing with flat bottom end cutting tools.

Spun Surface

A surface defined by rotating a 2-D generator curve around an axis vector line.

Station

A specific tool station on a machine.

Station Number

The physical location specified for tool setup on machines with automatic tool changers or turrets. This is related to but separate and distinct from tool number. Although these are usually the same, in cases such as manual tool change and coded tooling, they are different.

Step, see *Process Step*

A unique number for the new step. This field defaults to the number automatically assigned when the Insert action button on the control bar is selected.

OR...

A combination of the tool and the operation used to perform one step in a process.

Stepover Value

The distance between consecutive toolpath passes. The minimum value you can use is 0.0001 inch [0.00254 mm]. If you enter a smaller number, SmartCAM will use the minimum value.

SUPM (Surface Units Per Minute)

Used on the Step dialog box to indicate that the output will be in surface units per minute.

Surface

A SmartCAM element that describes a free-form shape representing the finished piece part. It may curve in both directions. These elements are represented either as an ACIS NURBS surface or an ACIS analytic surface.

Create surfaces from FreeForm Machining construction geometry, an IGES surface (trimmed or untrimmed), or an ACIS face (trimmed or untrimmed).

Surface Envelope

The defined cube of space that encompasses a surface. The length of each side of the envelope is determined by the maximum length of the surface in each direction.

Surface Milling

The Surface Milling operation type provides the primary machining parameters for machining single surfaces. Use Surface Milling to create toolpath from single surface offset mesh.

Surface Normal

A direction vector that is perpendicular to the surface at a given point on the surface. The FreeForm Machining surface normals display provides an arrow to indicate the surface normal at the center of the surface. You can use the surface normal indicator to determine the side of the surface to use (for example, which side of the surface to machine).

To reverse the surface normal direction for a group of surfaces, use Edit—Property Change—Surfaces. To change the display of surface normals, select the desired display options from the Utility—Display Modes dialog box.

Surfaces to Edit

A single surface element or a group of surfaces that you want to edit by splitting, trimming, or other surface editing operation. For example, in a splitting operation, the surface elements are cut by an intersecting surface, much like a knife can be used to cut through a slice of cheese.

SWEL Commands

The SmartCAM Workplace Environment Language (SWEL) is a set of system commands and symbols used for special configuration of the hot keys or icon buttons. A complete list of commands for customizing hot keys is available by selecting Keyboard—Customizing Hot Keys from the Help pull-down menu.

Tap

A tool with helical threads at a defined pitch used to create internal threads in predrilled holes.

Taper Change

A user command used to change the taper angle in the profile.

Tapping

The Tapping operation type provides the primary machining parameters for the tapping of holes. Use with solid body taps for creating interior threads in an existing hole.

Template File

Template files have a `.tmp` extension and determine the format SmartCAM uses to generate your code. SmartCAM combines information from the model's database, Job Operations (`.jof`) and machine (`.smf`) files, and outputs it through a template file. For more information about template files, refer to the *SmartCAM Code Generation Guide*.

Thread Depth

Depth of a thread based on its nominal diameter and pitch.

Thread Lead In

This is a distance input in which you enter the incremental distance to start threading before the root start. This enables you to start threading off the end of the part. The default is two times the pitch.

Thread Pitch

The distance from a point on a thread to a corresponding point on the next thread.

Thread Table

This is a text input field in which you enter the name of the thread table you want to use to look up data for the thread diameter. Selecting Lookup enables you to automatically input values to the control panel from values in the thread table.

Threading

The Threading operation type provides the primary machining parameters for adding threads to the internal diameter or external diameter of a turned part.

Title Bar

The title bar is displayed across the top of the SmartCAM screen display. It contains the application name and the current path and file name.

TMP File

A template file (`.tmp`) SmartCAM uses to format information in a process model into the code for a machine. For more information about template files, refer to the *SmartCAM Code Generation Guide*.

To Size

On/Off switch. Turn on this switch if you want the final groove pass to produce the actual groove size. When off, the final pass is separated from the groove size by the amount you specify with finish allowance.

Tolerance

The range of variation permitted in the toolpath creation for an element, or the graphic display of an element.

Tool Graphics

Standard program and user-customized files that display basic machine tool profiles during Show Path operations.

Toolpath

The graphic representation of the tool movement and travel as it machines the part. You can assign toolpath to geometry as you create it, or use Edit—Property Chg—Toolpath to assign toolpath to existing geometry.

Toolpath for a surface usually consists of a polyline mesh that is offset from the surface element based on the remaining stock amount and the cut settings (that is, contact point, tool tip, or center point). SmartCAM uses the surface element, the job operations step, and surface machining settings to create the toolpath.

Toolbox

Each toolbox contains a set of modeling tools used to create and change the model. For example, the Geometry toolbox contains modeling tools that enable you to create geometric elements, such as lines, arcs, holes, and points.

Select toolboxes from pull-down menus or from the workbench. The three most recently used toolboxes are always available in the workbench. The tools in the selected toolbox are displayed in a list below the workbench.

Tooling

Physical “hard” tooling such as cutting tools, forming tools, work-holding tools, and fixtures as well as “soft” meta-tools such as process features.

Tool plane

A tool plane defines the tool's normal orientation to a work plane. You can assign tool planes to an existing work plane and position them at an angle to a work plane.

Topology

Logical relationships between the vertices, edges, and faces of a surface. The geometric representations of them (points, curves, surfaces).

Translated Surface

A surface defined by sweeping a 2-D generator profile along a director curve. The generator profile maintains a perpendicular orientation to the tangent vector for each segment as it moves along the director curve.

Traverse Cut

The roughing movement of a tool as it removes excess material.

Triggers

SmartCAM provides a triggering mechanism that gives you more flexibility for defining geometry without completing all of the input fields on a control panel. SmartCAM indicates it is ready to “trigger” a solution for an element by placing an asterisk next to the last required input field.

Trimmed Surface

A surface, imported from an IGES or ACIS file, with curves that limit the displayed and machined part of the surface to a smaller area than that defined by base surface edges.

Trimming Curves

The curves that define the trimming boundary of a trimmed surface that was imported from an IGES or ACIS file. These curves limit the displayed and machined part of the surface to a smaller area than that defined by the base surface.

Turn Rough

Toolbox with modeling tools which enable you to create roughing toolpath for an existing profile.

Twist Drill

A conventional fluted drill used to create blind or through holes. This tool typically contains a 118-degree, included point angle.

U and V Direction

The isoparametric flow along the curve of a surface in two directions. A surface is represented by parameter curves and model space curves. The parameter space curves are 2-D and have U and V coordinates. The model space curves are the shapes displayed in the graphic view and have X, Y, and Z coordinates.

Undo

Undo removes the most recently completed operation from the graphic display and the database.

Units

The measurement units, inches or metric, used in the process model. Set the units for a job in the Job Information dialog box in the Job Operation Planner. Do not change the units for an existing job because the Planner will not convert values in existing steps from one system to another.

Untrimmed Surface

A complete surface, unlimited by trimming curves, with edges defined at the outer boundaries of the base surface edges.

Up Cutting

The process of clearing chips away from the part during milling operations.

UPM

Specifies the feed mode in units per minute for output.

V and U Direction

The isoparametric flow along the curve of a surface in two directions. A surface is represented by parameter curves and model space curves. The parameter curves are 2-D. They have U and V coordinates. The model space curves are the shapes displayed in the graphic view. They have X, Y, and Z coordinates.

View Space

The three-dimensional coordinate system that represents your viewpoint of the model. The origin of the view space coordinate system is located at the center of your computer screen. The XY plane is parallel to the computer screen and the Z-axis points out of the screen toward you. For example, the XYZ coordinate (1,1,0) is located at the upper right corner of your screen. Objects located in view space, such as vector light sources, do not move with manipulations of the model space. Rotating a view, for example, rotates the geometric model and the model space axes while the view space axes remained fixed.

Virtual Mesh Tolerance

Used with certain modeling tools that require a uniform polyline mesh to operate. When profiles consist of basic primitive elements or a range of polyline profiles with an uneven count, the application converts them into uniform polylines using this setting. Virtual mesh tolerance helps determine the length of polyline segments by establishing a tolerance band width that the segments must fit in. The default setting is 0.001.

Which Solution

When the Which on/off switch is on, you can step through the possible solutions when multiple solutions exist. When it is off, SmartCAM uses the most likely solution.

Width of Cut

The distance (step over) between roughing passes.

Wire Electrode

The tool type used to enter the tool parameters for Wire EDM wire electrodes.

Work Plane

A flat or level surface on which to work or assign geometry. SmartCAM has three reserved system work planes positioned according to the X, Y, and Z axes to build a process model. You can also define your own work planes.

Workbench

This portion of the screen provides quick access to frequently used toolboxes. The three most recently used toolboxes occupy the three spaces on the workbench. To add a new toolbox to the workbench, select it from a menu.

ZCHK (check distance)

The distance above the Prof Top where the tool starts to feed. #ZCHCK is assigned in the SMF file.

Index

D

E

F

G

J

K

L

M

Q

R

S

T