SmartCAMcnc

# Exploring

# SmartCAM® Turning V11.5

## Production Turning™

## Advanced Turning™

Doc SC006-MRS

Originally Published: November 1997
Update Published: December 2004

SmartCAMcnc

# Contents

## 2 Working with SmartCAM

## 3 Using Job Operations

# 4 Working with Elements

## 8 Learning Milling Basics for Advanced Turning



## 9 Using Rough Machining Processes



## 10 Creating Work Planes

## Appendix A: Self-Test Answers



## Glossary

## Index

# Welcome

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

## Overview

There are two sections to this course: Production Turning and Advanced Turning. You may need to use only one or both sections of the exploring guide, depending on your needs or the course you are attending. However, you need to know Production Turning before learning the Advanced Turning 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 Turning Exploring Program

SmartCAM Turning exploring sessions are intended to give you a broad introduction to the SmartCAM Turning 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 Turning Objectives

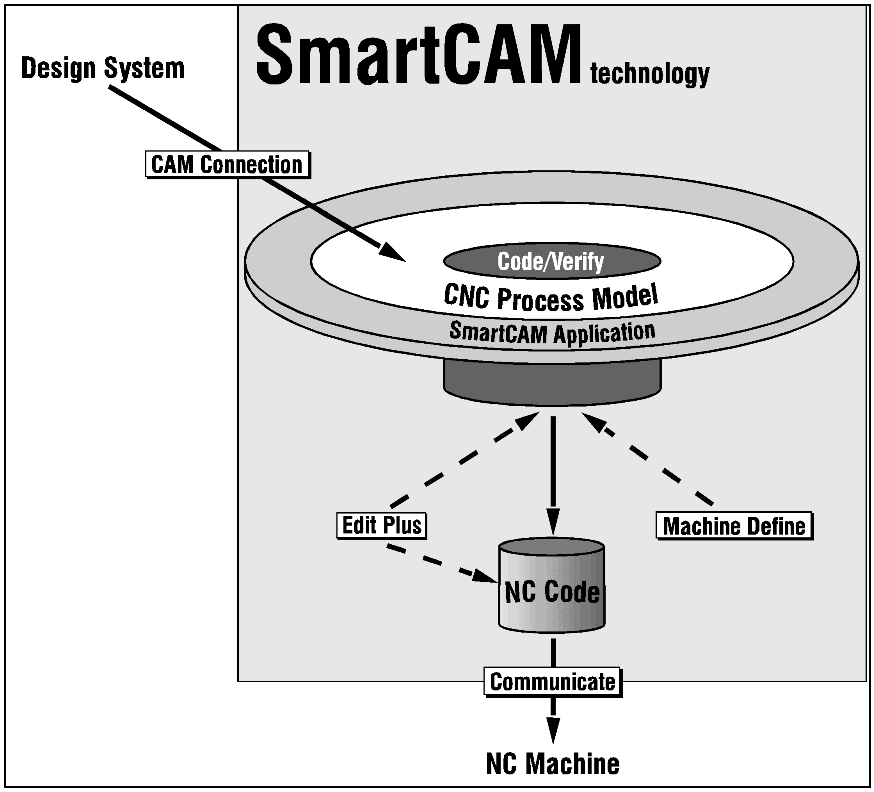
- Prepare to use SmartCAM.
- Work with SmartCAM.
- Use Job Operations.
- Work with elements.
- Use processes.
- Use utilities.
- Generate code.

## Exploring Advanced Turning Objectives

- Use 4-axis synchronization.
- Learn milling basics for Advanced Turning.
- Use Rough machining processes.
- Create work planes.

# 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

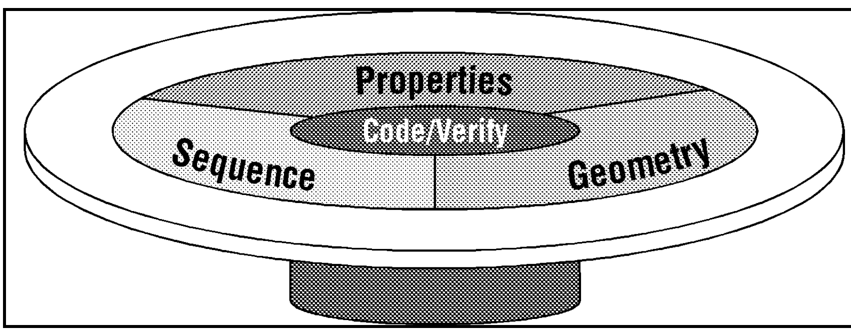
A SmartCAM CNC Process Model is a dynamic, sequential toolpath database. Part geometry is immediately converted into toolpath, sequenced in the way the machine will cut the part. Any changes you make to the model immediately update the database.

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

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 Turning units and at the end of the Advanced Turning exploration. 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 Turning

# Exploring SmartCAM Production Turning

## Welcome

SmartCAM Production Turning creates efficient toolpath and CNC code for 2-axis lathes. As the first tier in the SmartCAM family of turning 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 Toolpath
- Generating Code

1

# BecomingAcquainted with SmartCAM

## Overview

It is important for you to become acquainted with the SmartCAM environment before you performTurning 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 Turning** 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).

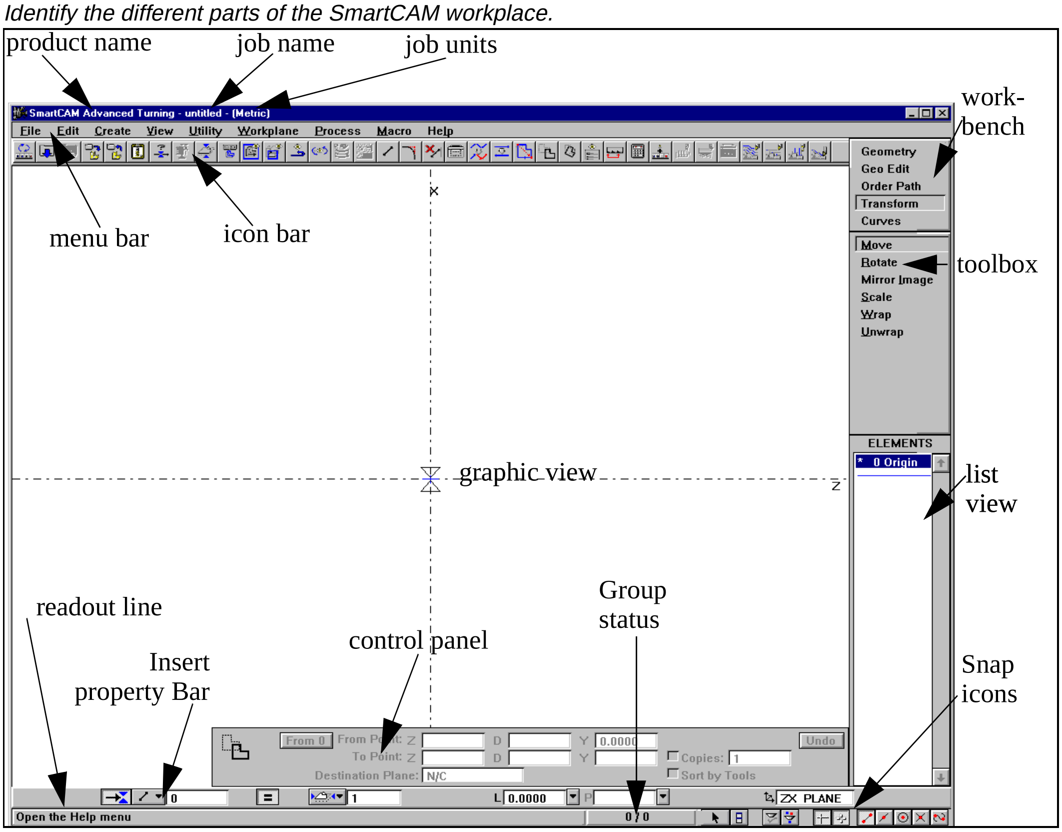
The SmartCAM application window is displayed on the screen.

# 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.*

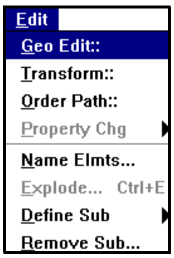



## Menu Bar


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

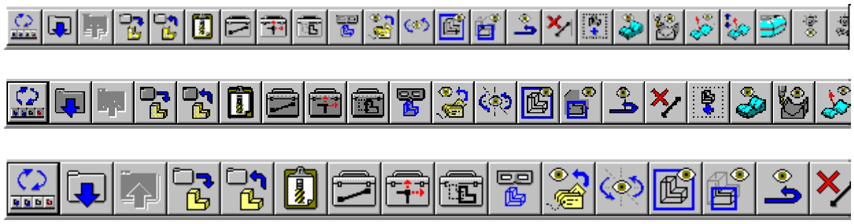
Pull-down menus are accessed through the menu bar. Perform these tasks to select from the Edit pull-down menu:

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.

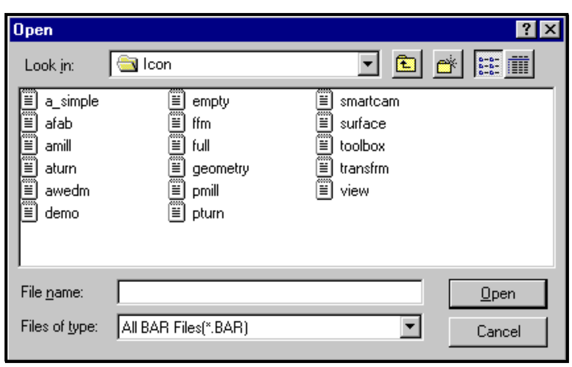



***Figure 1-3***
*Set the values on the Open dialog box.*

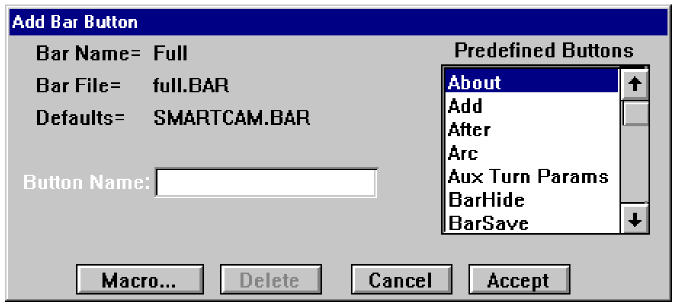
3. Select an icon bar file.
4. Select the **Open** button. The **Choose Icon Bar** dialog box becomes active.
5. Select the **Accept** button.

### 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

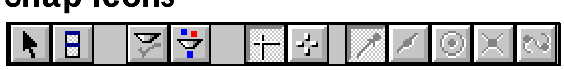
You can change any of the icons that are displayed on the icon bar. One way is to open your `.bar` file in a text editor and edit it in that way. However, you can perform these tasks to edit your icon bar from within SmartCAM:

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.

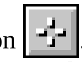
## Snap Icons



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.

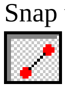
  **Note** If no snap points are found within the element picking distance, Snap returns the coordinates of the last point that you selected.

- **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.

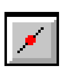
- **Automatic mode** requires that you activate the Snap Mode icon and the Free Coordinates Mode icon. The system switches between the modes depending on the distance of the pointer from the nearest available point. When the pointer is within the pick limit from an existing point, the Snap mode is in 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:

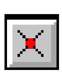


**Endpoint** uses the snap points at the start and end points of all elements in the process model.

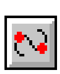


**Midpoint** uses the snap points at the midpoint of all elements in the process model, except for polylines and splines.

**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

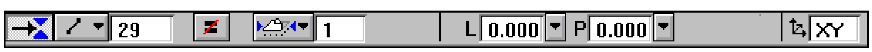

The readout line is always visible below the graphic view. The information displayed to the left depends on the task in progress. As you move the cursor over any menu, icon, toolbox, or tool, the readout line displays a short statement identifying its use. The readout line displays information about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

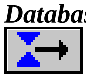



Chapter 2 presents detailed information about the Insert property bar.

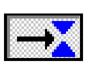
### Insert Positions

These icons control where the new geometry resides:

***Database Location***



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

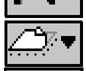


The **After** icon inserts geometry into the database after the current position.

***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.

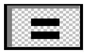


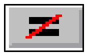
The **Layer** icon specifies that the insert position is relative to a layer.

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

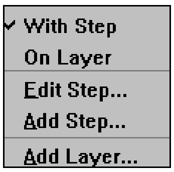
The **Tool** icon specifies that the insert position is relative to a tool.

***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.

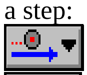
The **Add Step...** item opens the Add Process Step dialog box, which enables you to create new process steps.

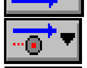
The **Add Layer...** item opens the Add Layer dialog box, which enables you to create a new layer with full control over layer properties.

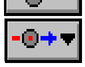
## Insert Properties

### *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*

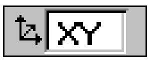
The **L** selector switch accepts a value for the level of the geometry on a specific workplane. You can enter a value in the input field, snap to a level in the graphic view, use the default level, or select one of the five different most recently used levels.


### *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.

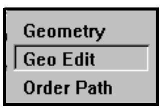

### *Clear*

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.



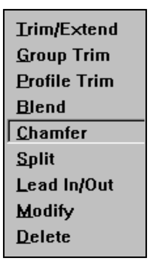

### *Work Plane*

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.



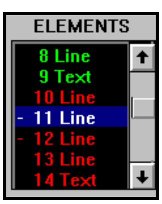

## Workbench

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**.



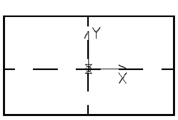

## Toolbox

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

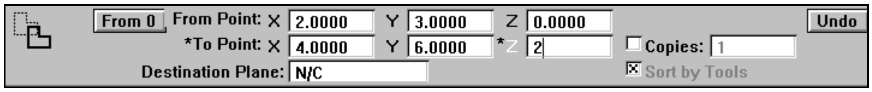
The list view displays information for the tool that you select from the open toolbox. For example, Element Seq is the selected tool, so element information is displayed sequentially in the list view.




## 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.

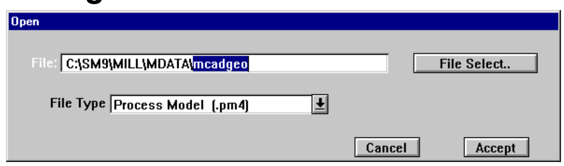
## Control Panel




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.

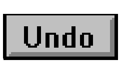
## Dialog Box




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.

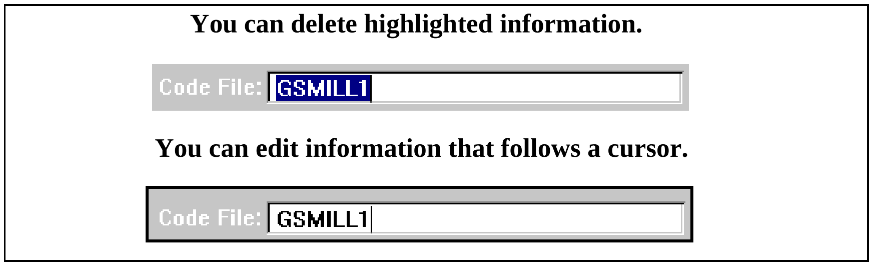
Select a button by placing the cursor over it and pressing the left mouse button or by pressing the Tab key until the button is highlighted and then pressing the Enter key.


## 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.

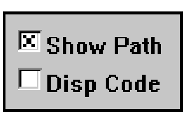



Selecting in the input field places the cursor to the right of the last character in the input field. If only a flashing bar (cursor) is visible, the current value can be appended to or edited by using the arrow and Backspace keys.

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:

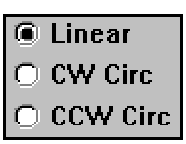
1. Select the Z input field label to make the input field active.
2. Select the end of the existing element (with Snap on) to enter the Z coordinate of the existing element end point.
3. Enter **+1** in the input field after the element input coordinate.

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



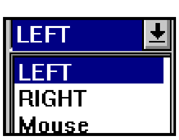

## On/Off Switches

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 Turning.

## Learning SmartCAM File Types

SmartCAM creates and uses process model and job operations files.

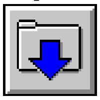
### Process Model

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: PTTURN.PM4**

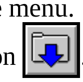
When you first open your application, most of the options on the menu bar and the icon bar are dimmed, and you cannot select them. This is because you have not yet opened a model file.

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\turn_e`, and metric data files are in `d:\train\turn_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 `PTTURN.PM4`, the metric file name would be `PTTURNM.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 **`PTTURN.PM4`**.
5. Select the **Accept** button.

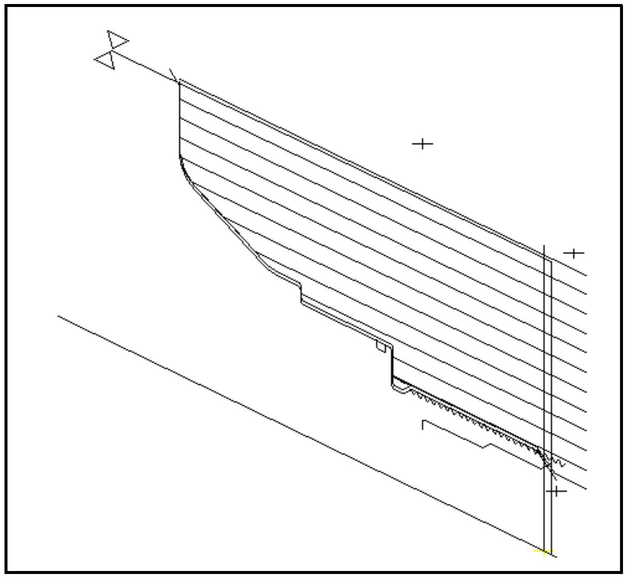


***Figure 1-7***
*Open PTTURN.*

## Saving a File

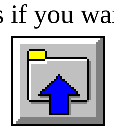
To avoid losing your work, save your model frequently while you are working on it. Avoid saving model files that are shipped with the software because they are often used for multiple lessons. However, if you do save changes to one of these files on your local drive, you can return to the `\train` directory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

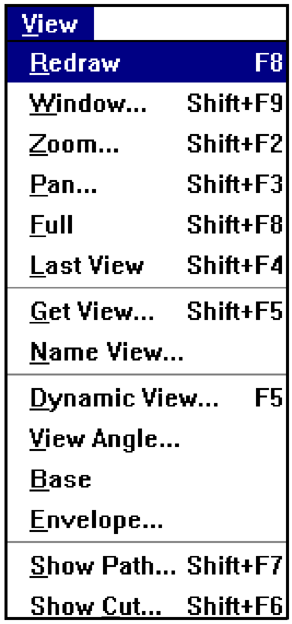



***Figure 1-8***
*Specify options from the View menu.*

## Using Window

**Model File: PTTURN.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 **`PTTURN.PM4`**.

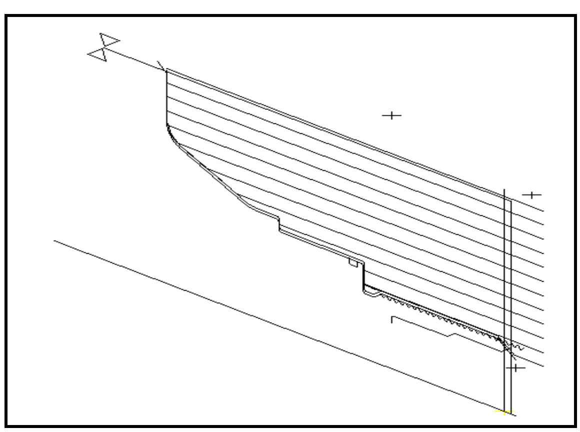
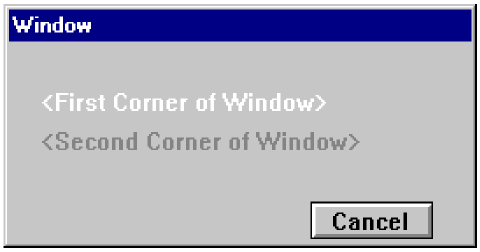


***Figure 1-9***
*Open PTTURN.*

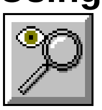
2. Select **View—Window**. The **Window** dialog box is displayed.

***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

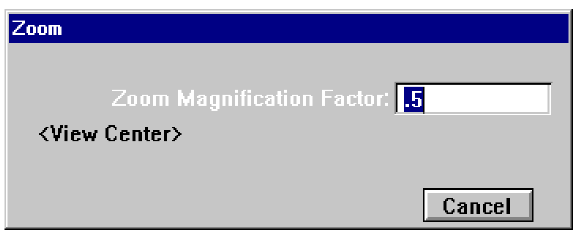


**Model File: PTTURN.PM4**

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

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

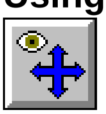
***Figure 1-11***
*Specify a zoom magnification factor in the Zoom dialog box.*




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

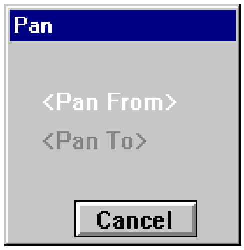


**Model File: PTTURN.PM4**

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

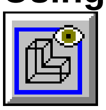
1. Select **View—Pan**. The **Pan** dialog box is displayed.

*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.

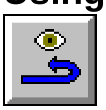
## Using Full



**Model File: PTTURN.PM4**

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

## Using Last View



**Model File: PTTURN.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: PTVIEW.PM4**

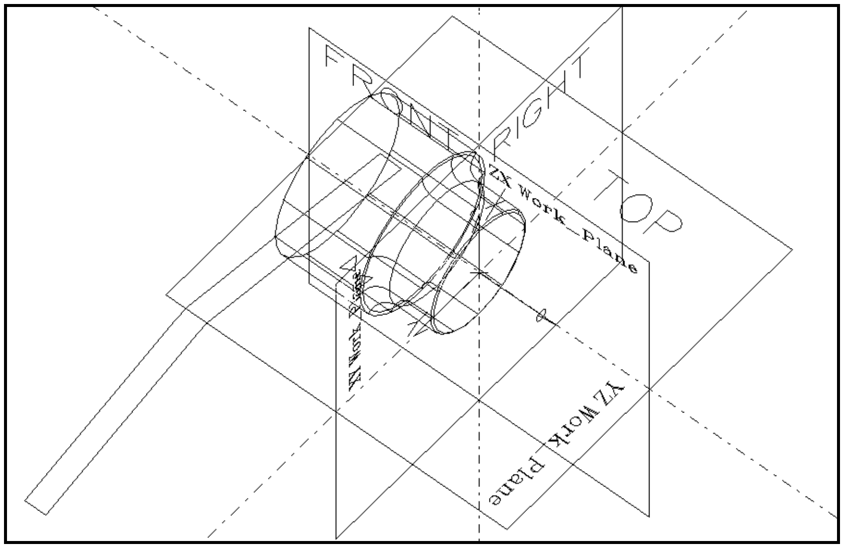
Use the Get View feature to view different orientations that you saved. By default, four different views are available (Top, Right, Front, Isometric). You can use the Get View dialog box or these hot keys to view your model from different orientaions:

- 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:

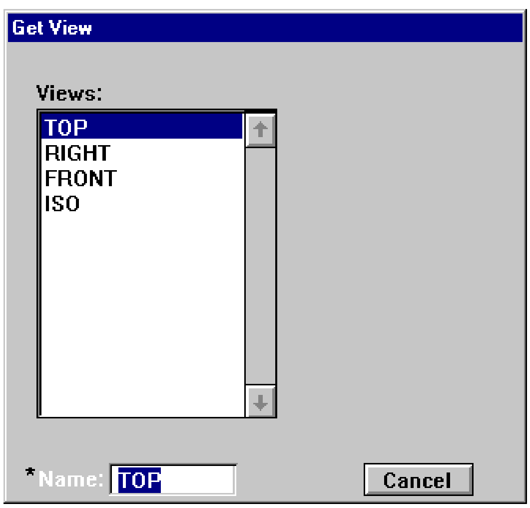
1. Open the model file **`PTVIEW.PM4`**.

*Figure 1-13*
*Open PTVIEW.*

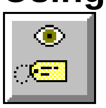



2. Select **View—Get View**. The **Get View** dialog box is displayed.
3. Select the view you want.

*Figure 1-14*
*Specify the view that you want.*



## Using Name View



**Model File: PTVIEW.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.

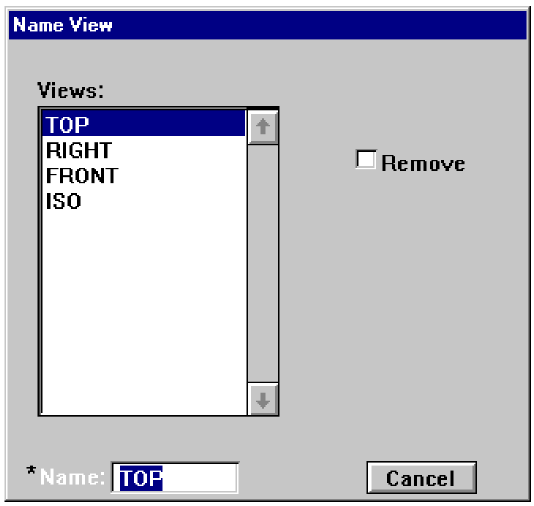
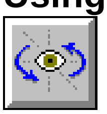



***Figure 1-15***
*Set the Name input field to save the new view.*

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

## Using Dynamic View



**Model File: PTVIEW.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.

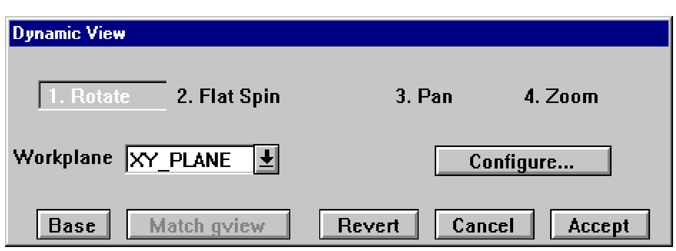
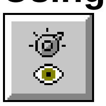



***Figure 1-16***
*Specify a way to move the view in the Dynamic View dialog box.*

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



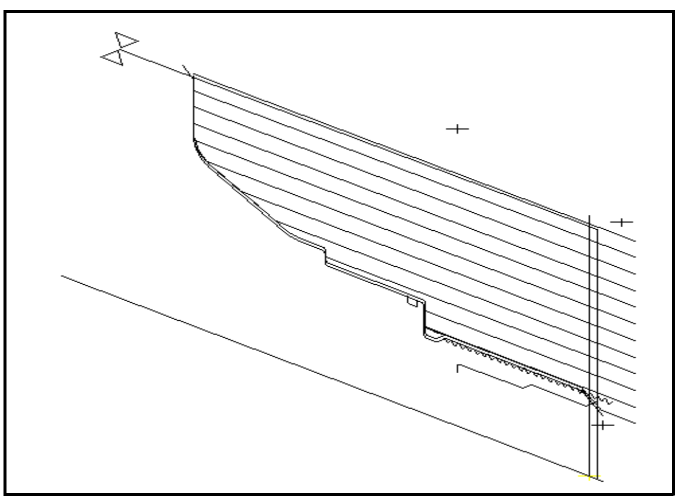
**Model File: PTTURN.PM4**

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:

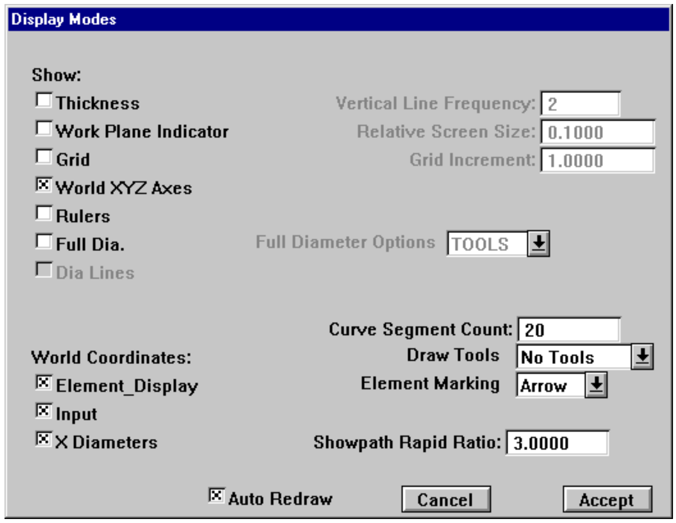
1. Open the model file **PTTURN.PM4**.

***Figure 1-17***
*Open PTTURN.*



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

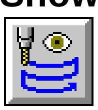
*Figure 1-18*
*Set the values on the Display Modes dialog box.*




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**.

## Showing the Toolpath



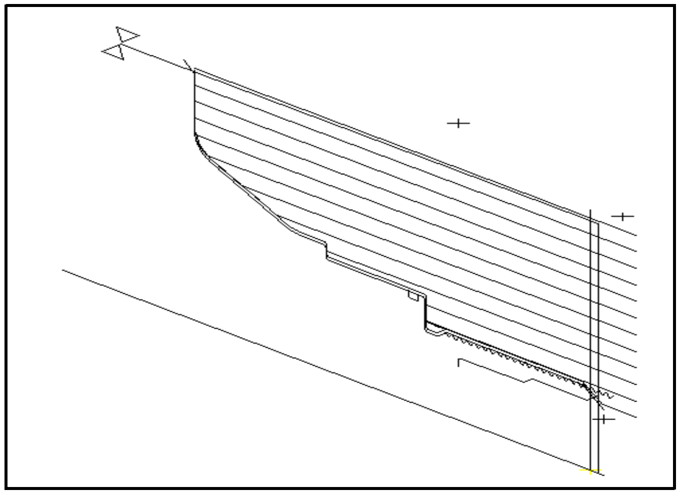
Show Path provides a representation of the toolpath before any parts are cut. Use Show Path to verify the sequence and toolpath of the model based on your machine's capabilities.

**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 the tooling section.

**Model File: PTTURN.PM4**

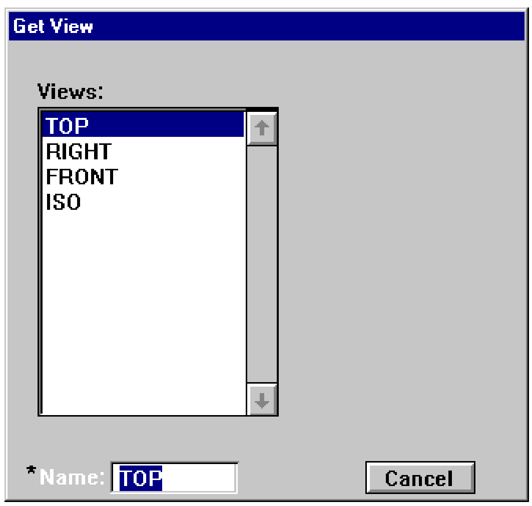
1. Open the model file **`PTTURN.PM4`**.

*Figure 1-19*
*Open PTTURN.*



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

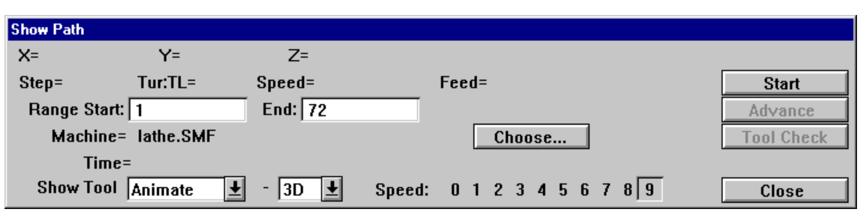
*Figure 1-20*
*Specify the view that you want.*




3. Select **Front** from the dialog box.

4. Select **View—Show Path**. The **Show Path** dialog box is displayed.

*Figure 1-21*
*Set the values in the Show Path dialog box.*




5. Set **Show Tool** to **Animate - 3D**.

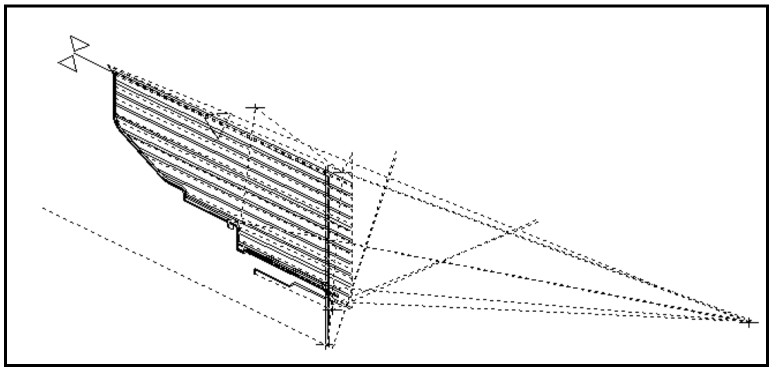
6. Set **Speed** to **5**.

7. Select the **Start** button. SmartCAM simulates the toolpath. You can see whether the geometry has the necessary sequence and manufacturing properties.

   **Note** Use the number keys on your keyboard to alter Show Path's speed at any time during the simulation.

8. Press **ESC** to stop **Show Path**.

***Figure 1-22***
*This is the result of Show Path.*



## 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.
- ❑ 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 the toolpath.

# 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 Clearance
- 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.

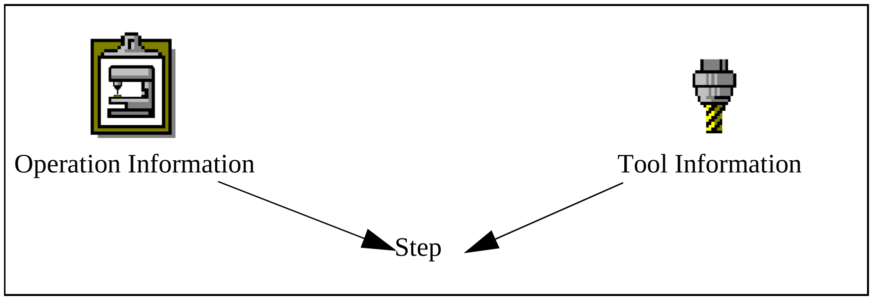
## Overview

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 (turn, groove, or thread)
- Type (diamond, round, or square cutter)
- Shape (round)
- Optional custom tool graphics
- Operation information, which includes the following:
  - Feed rate in IPM or IPR
  - Spindle speed in SFM or RPM

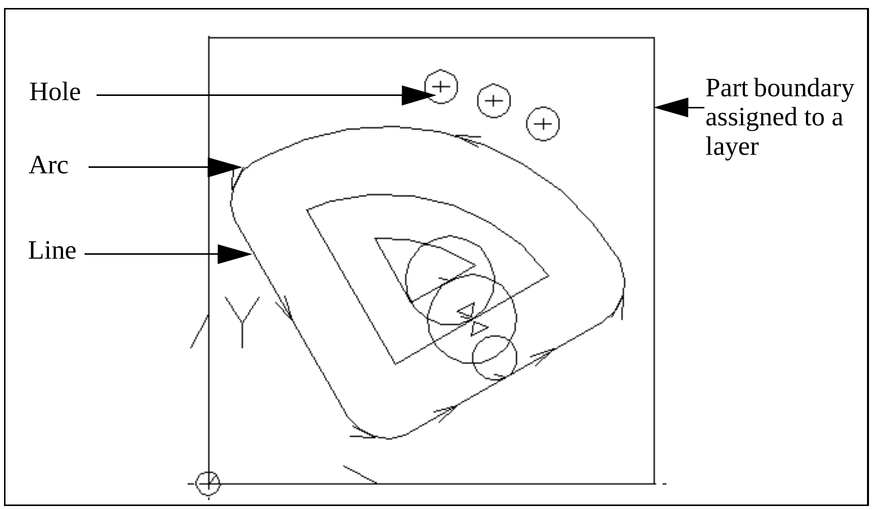
*Figure 2-1*
*A step is made of operation and tool information.*




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.

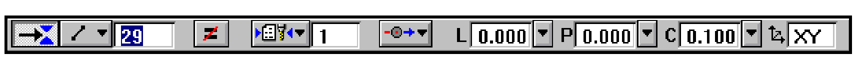
Elements are displayed in the following figure.

*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, 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.

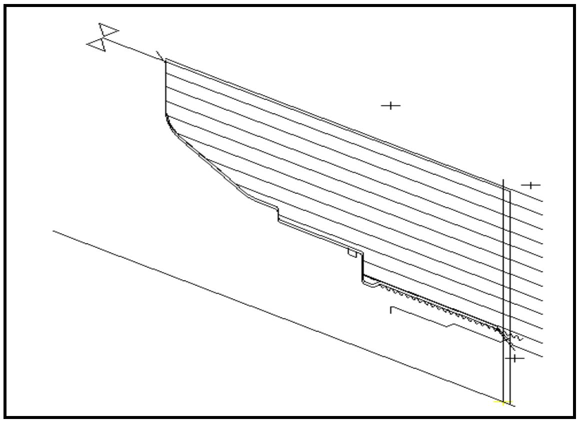
## Inserting Geometry

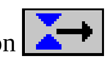
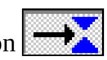
**Model File: PTTURN.PM4**

Perform these tasks to insert geometry:

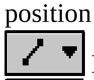
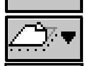
1. Open the model file **`PTTURN.PM4`**.

*Figure 2-4*
*Open PTTURN.*



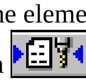
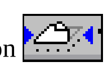
2. Set the **Before** icon  or **After** icon  on the **Insert** property bar to indicate if the insert point is to be before or after the selected element, process step, tool, layer, or profile.

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

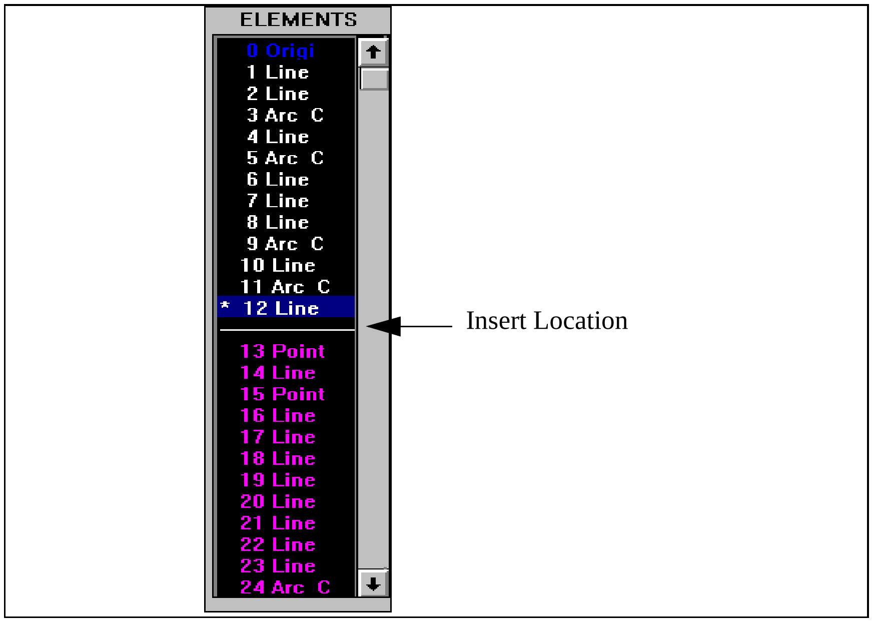
   Notice that the list view is updated to correspond to the icon you select.

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.

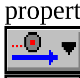
6. Select the step or layer to use from the list view or graphic view.

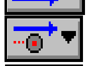
*Figure 2-5*
*Identify the insert location in the list view.*

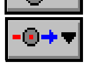



7. Set the **C (Clear)** input field on the **Insert** property bar. This is the primary retract plane for the tool. You can set this value only when you insert with a step. Setting a clearance value for a step ensures that subsequent uses of inserting on a layer use this data for layer geometry.

8. 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.

9. 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 Clearance

## Objectives

This lesson shows you how to perform these tasks:

- Define the clearance height.
- Set the Insert property bar.

## Overview

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.

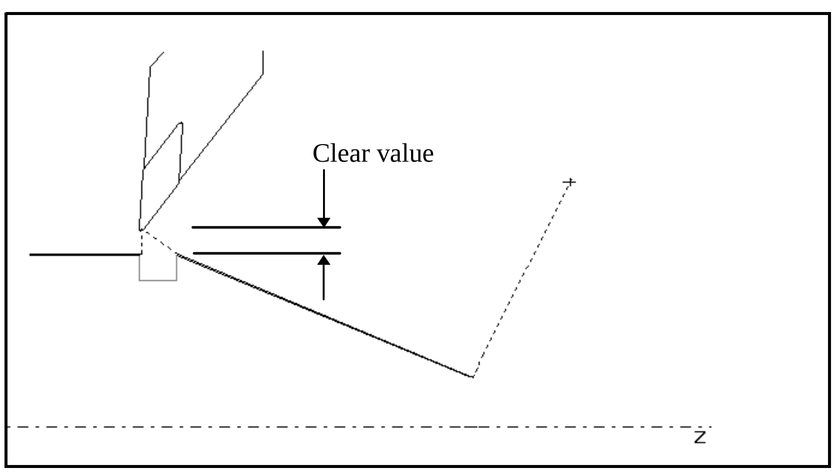


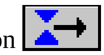
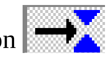
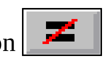

***Figure 2-6***
*Set the Clear field to the distance to retract the tool tip from the part.*

# Setting the Insert Property Bar

When you set clearance on the Insert property bar, you must turn on the on/off option for the Clear field to be active.

# Setting Clearance

Perform these tasks to set clearance:

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 turning tool from the list view.
6. 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.

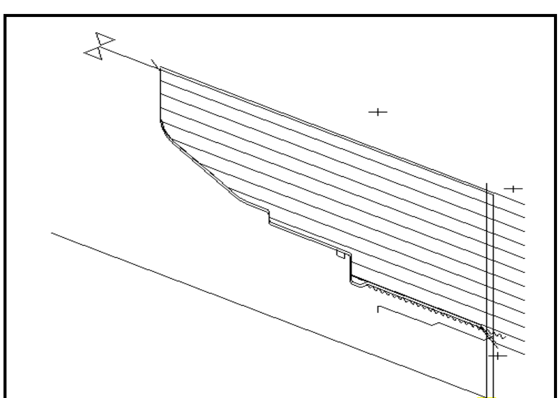
# Checking Clearance

**Model File: PTTURN.PM4**

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

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

***Figure 2-7***
*Open PTTURN.*



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

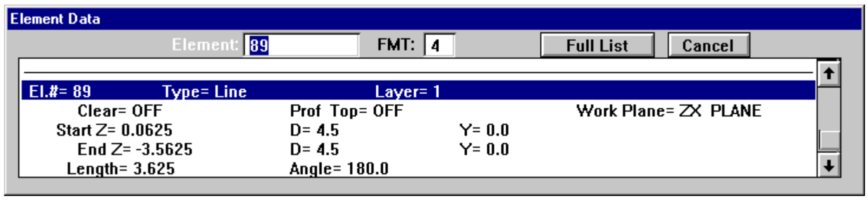



***Figure 2-8*** *Verify your data with the Element Data dialog box.*

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.

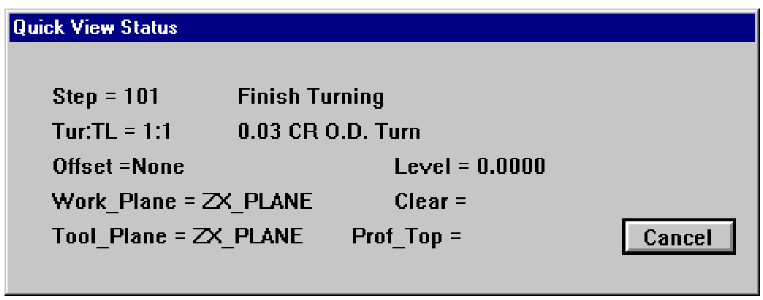



***Figure 2-9*** *View the status of the current operation with the Quick View Status dialog box.*

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.
- ❏ 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.

## Overview

Work planes determine the plane orientation of planar elements such as lines, arcs, and ellipses. SmartCAM has four pre-defined work planes that are reserved: XY, YZ, ZX,  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.

Use the Workplane menu to create or manipulate work planes.

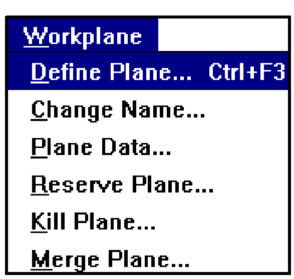
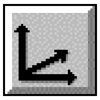



***Figure 2-10***
*Use the Workplane menu.*

## Creating Work Planes



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.

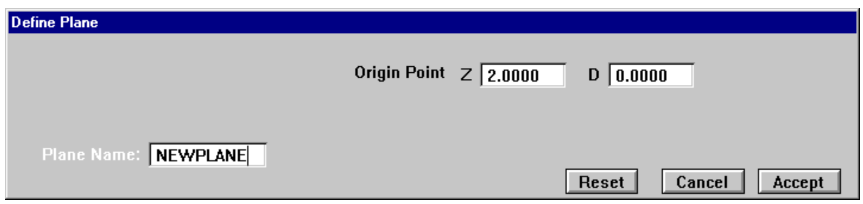
Use Define Plane to create a new work plane for the model. When you define and accept a work plane, it becomes the active work plane. To make an existing work plane active, use the Insert property bar.

### Defining a Work Plane from Three Points

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

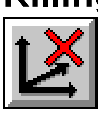
1. Select **Workplane—Define Plane**. The **Define Plane** dialog box is displayed.

***Figure 2-11***
*Set the values on the Define Plane dialog box.*




2. Set the **Origin Point** input fields.
3. Enter a name for the plane in the **Plane Name** input field.
4. 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.

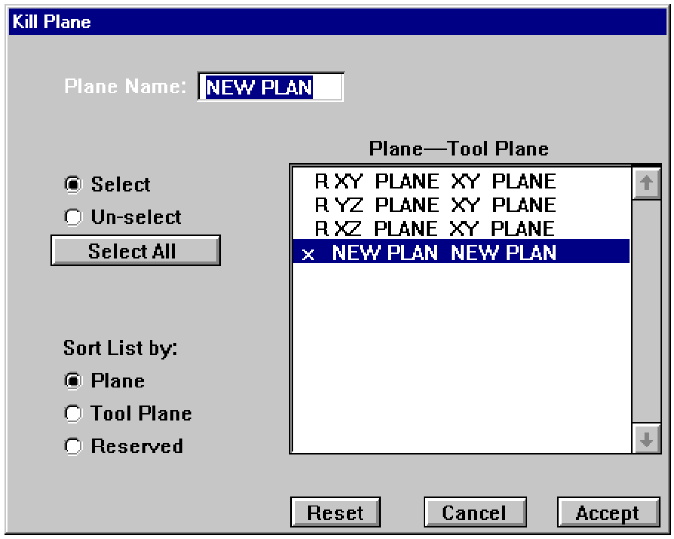



***Figure 2-12*** *Select the plane to delete on the Kill Plane dialog box.*

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.

## Points to Remember

- ❑ Work planes determine the orientation of geometry.
- ❑ Delete unused work planes using Kill Plane.

# Using Snap

## Objectives

This lesson shows you how to perform these tasks:

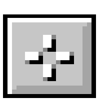
- Learn Snap selection modes.
- Use Snap.

## 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.

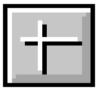
### Snap Selection Modes

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:

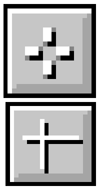


- **Snap mode** requires that you activate the Snap Mode icon. When Snap mode is on, the pointer is displayed as a short cross hairs. Use any of the snap point settings icons (End Point, Midpoint, Center Point, and so forth), which are on the readout line, to set the active point settings and then snap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

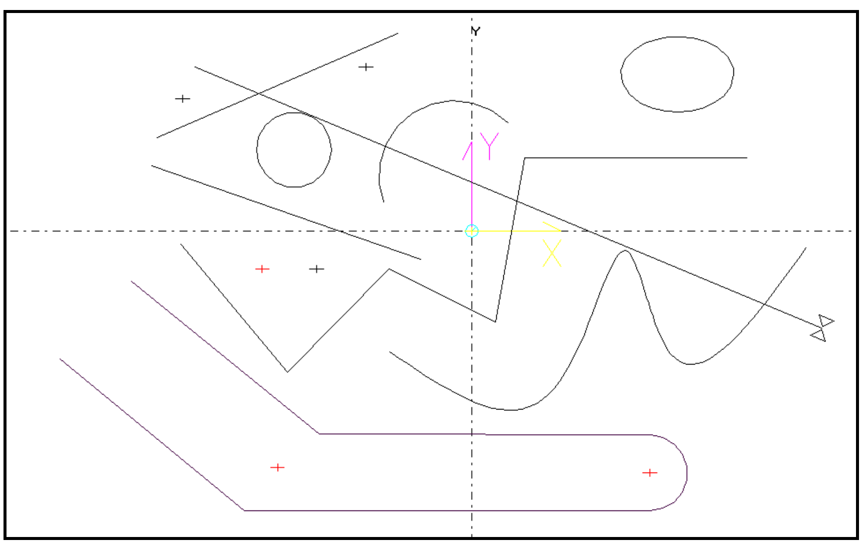
## Operating Snap

**Model File: PTSNPGRP.PM4**

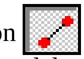
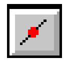
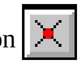
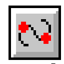
Snap point settings control which element points are used for snap points. You can use any combination of the snap point settings, but you must always have at least one setting turned on. Practice using Snap mode and the Snap point settings by performing these tasks:

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

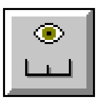
*Figure 2-13*
*Open PTSNPGRP.*




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



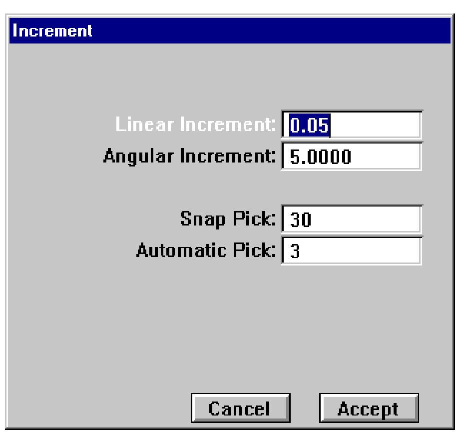
**Model File: PTSNPGRP.PM4**

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 **`PTSNPGRP.PM4`**.
2. Select **Utility—Increment**. The **Increment** dialog box is displayed.

***Figure 2-14***
*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.

## 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.
- The acceptability of the Snap Pick and Automatic Pick values is affected by your monitor resolution.

# Grouping Elements

## Objectives

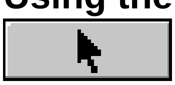
This lesson shows you how to practice these tasks:

- 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



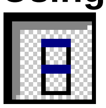
Use the Group Arrow icon to add, edit, or delete elements from the active group or to create a new active group if you operate without the Group tool palette showing.

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 4-50. 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.

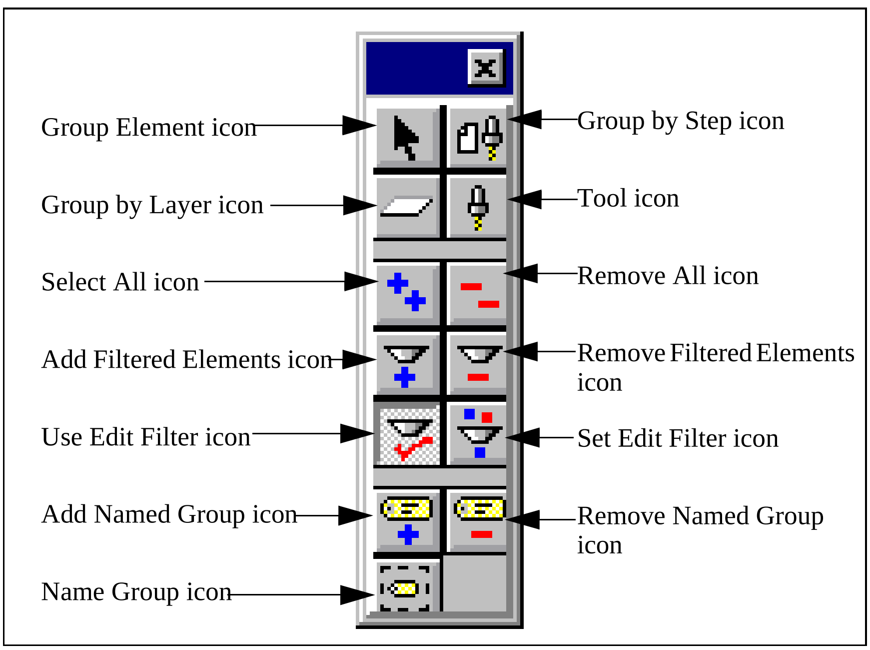
## Using the Group Tool Palette



**Model File: PTTURN.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 2-15***
*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 **`PTTURN.PM4`**.

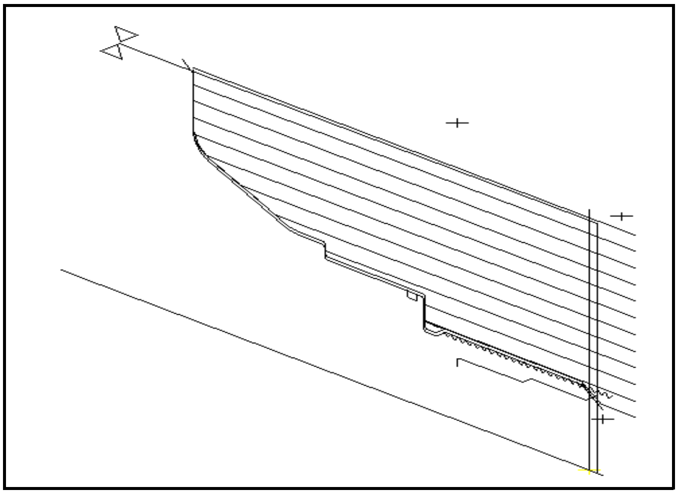


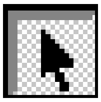
***Figure 2-16***
*Open PTTURN.*

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.



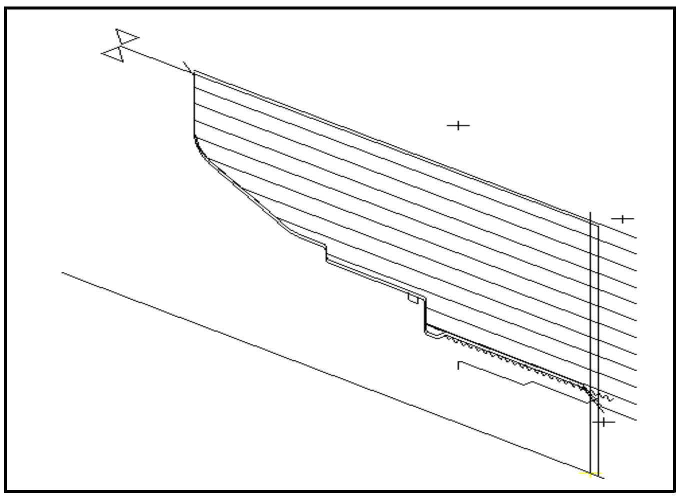
## Grouping Geometry by Element

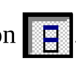
**Model File: PTTURN.PM4**

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 **`PTTURN.PM4`**.

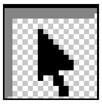
***Figure 2-17***
*Open PTTURN.*



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.

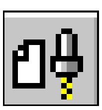


## Grouping Geometry by Box

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:

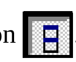
1. Select the **Group Tool Palette** icon . The **Group** tool palette is displayed.
2. Select the **Group Element** icon .
3. Click and drag the cursor in the graphics view to create a selection box to select geometry. There are two variations on box selection:
   - **Box Complete** is invoked by pressing 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.

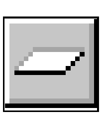
**Note** Hold the Control (ctrl) key while performing either a Box Complete or a Box Partial selection to ungroup items.



## Grouping Geometry by Step

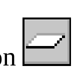
Grouping by step enables you to group or ungroup all elements that are defined with a selected step number. Perform these tasks to group geometry by step:

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

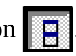
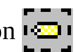
Grouping by layer enables you to group or ungroup all elements that are on the selected layer. Perform these tasks to group geometry by layer:

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.

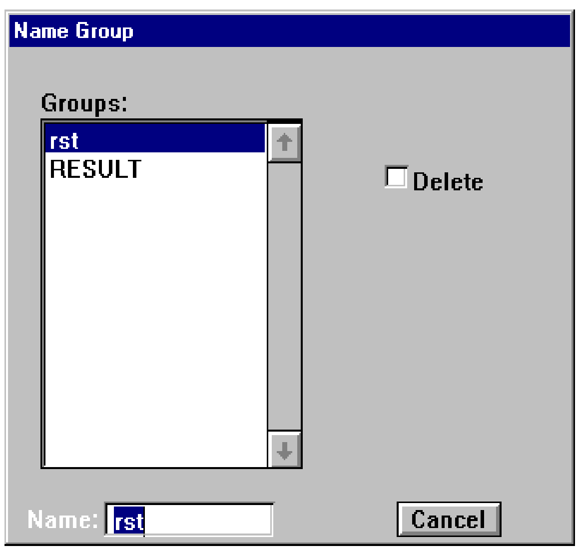



***Figure 2-18***
*Set the values on the Name Group dialog box.*

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.

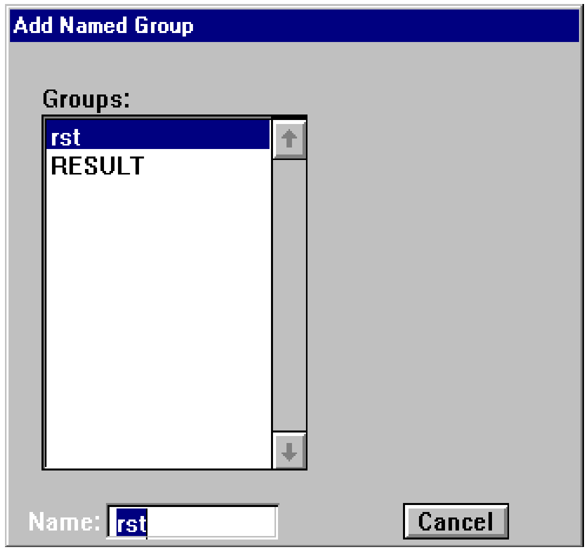
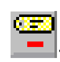



***Figure 2-19***
*Set the values on the Add Named Group dialog box.*

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.

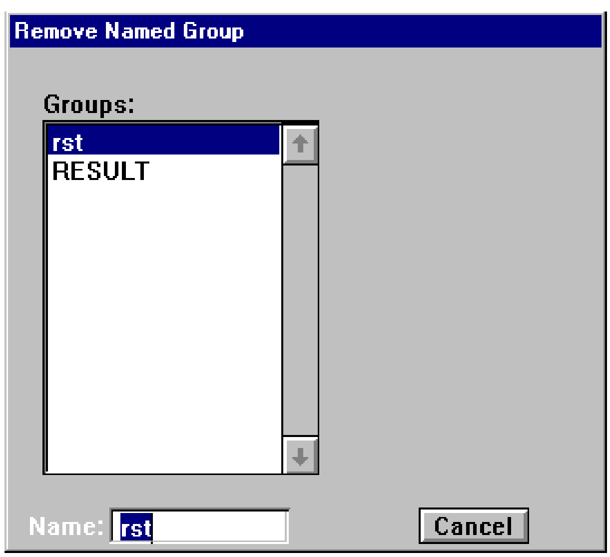



***Figure 2-20***
*Set the values on the Remove Named Group dialog box.*

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

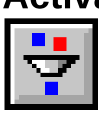
This lesson shows you how to perform these tasks:

- 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

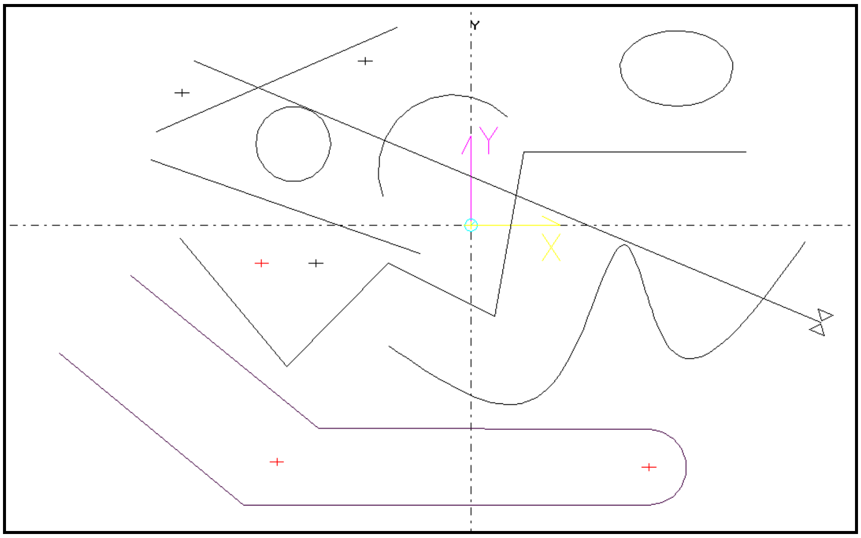


**Model File: PTSNPGRP.PM4**

Perform these tasks to activate edit filters:

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

*Figure 2-21*
*Open PTSNPGRP.*



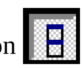
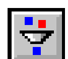
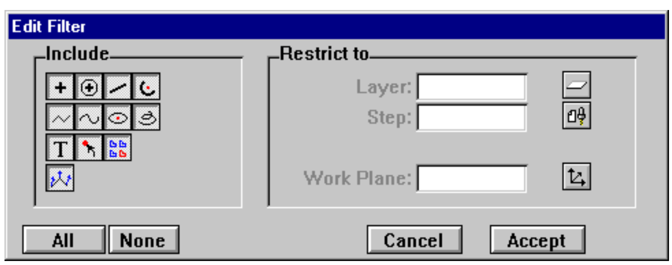

2. Select the **Group Tool Palette** icon  to open the **Group** tool palette.
3. Select the **Edit Filter** icon  from the **Group** tool palette. The **Edit Filter** dialog box is displayed.

*Figure 2-22*
*Enter Include and Restrict values on the Edit Filter dialog box.*

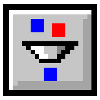



4. Under the **Include** heading, specify the element types for the system to recognize for the group functions. You can choose all element types by selecting the **All** button or exclude all element types by selecting the **None** button.

   **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: PTSNPGRP.PM4**

Perform these tasks to activate edit filters:

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

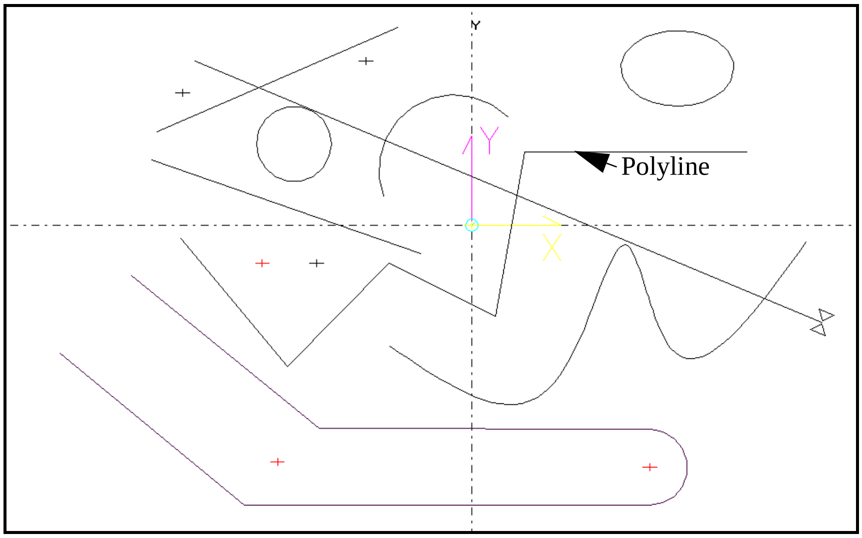
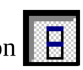
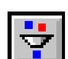



***Figure 2-23***
*Open PTSNPGRP.*

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.

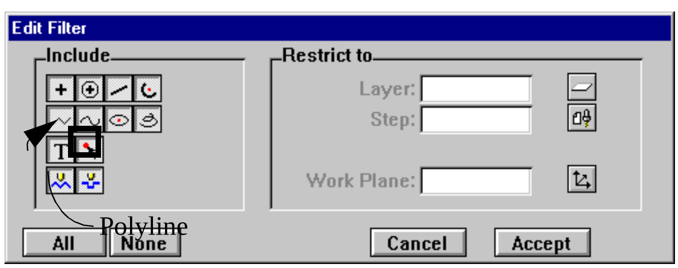
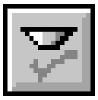



***Figure 2-24***
*Enter Include and Restrict to values on the Edit Filter dialog box.*

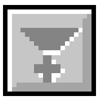
4. Include only polyline elements.
5. Select the **Accept** button.
6. Set the Snap mode to **Automatic**, and turn on all of the snap point settings that are on the readout line.
7. Select **Create—Geometry—Line**.
8. Move the cursor in the graphic view. Notice that the automatic snapping recognizes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

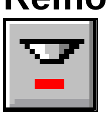
## Adding Filtered Elements



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
- Layers
- Work Planes
- Steps
- Thread
- Groove

## 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. What type of plane determines the orientation of geometry?**

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

_______________ **5. 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

_______________ **6. Using Status interrupts the current task.**

a) true
b) false

_______________ **7. 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

# 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

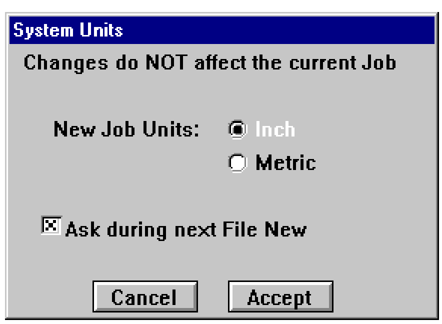
## Creating a .jof File

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.

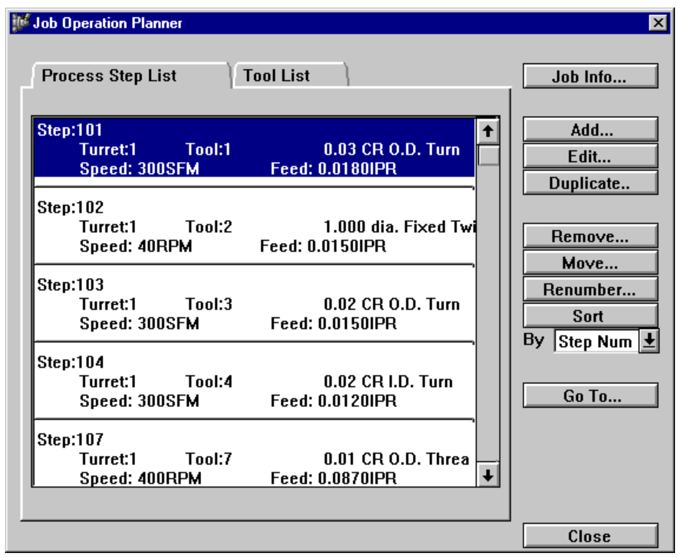
***Figure 3-2***
*File—New invokes the New Job Units dialog box.*


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

c. Select the **Accept** button.

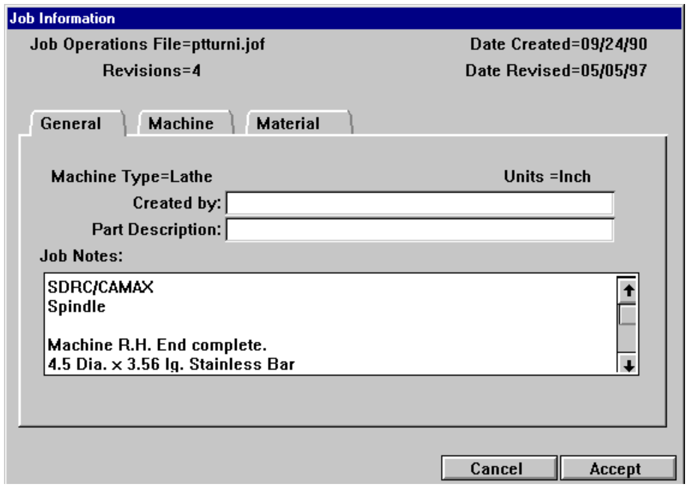
3. Select **File—Planner**. The **Job Operation Planner** dialog box is displayed.

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



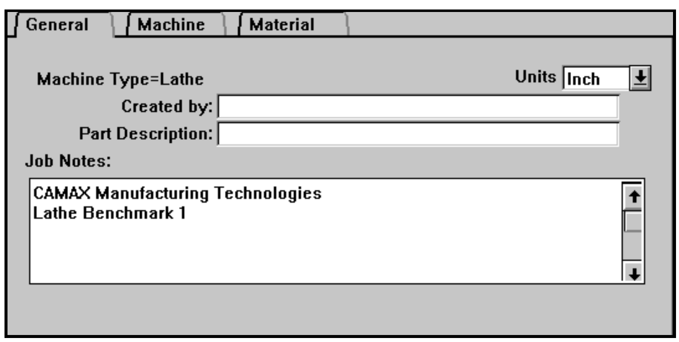

4. Select the **Job Info** button in the upper right corner of the **Job Operation Planner** dialog box. The **Job Information** dialog box is displayed.

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




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

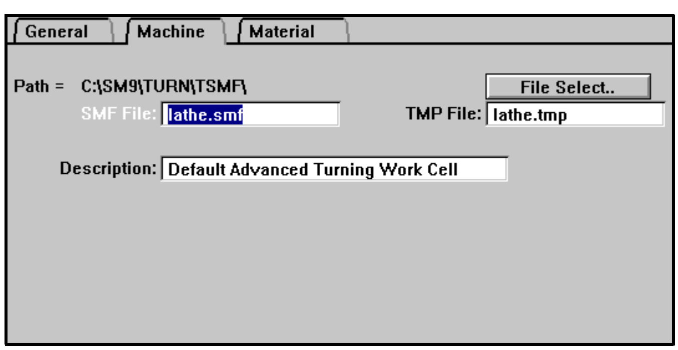
*Figure 3-5*
*The General tab invokes the General page.*




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.

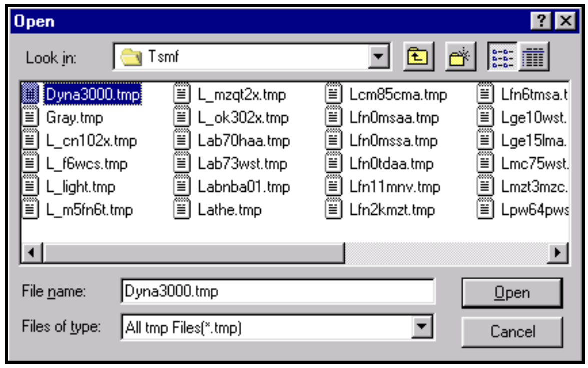



***Figure 3-7***
*Select an SMF file to use from the TSMF directory.*

9. Select an SMF file to use from the TSMF directory (for example: `CONROUGH.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.

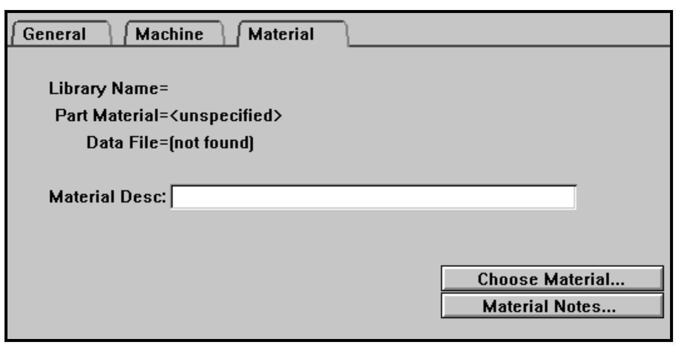



***Figure 3-8***
*Use the Material page to set the material description.*

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

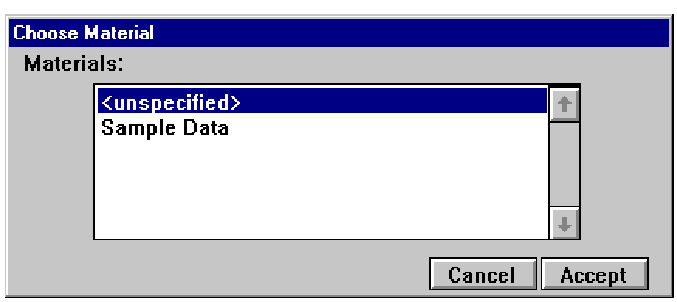



***Figure 3-9***
*Use the Choose Material dialog box to specify the kind of material to use.*

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 3-27.

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.

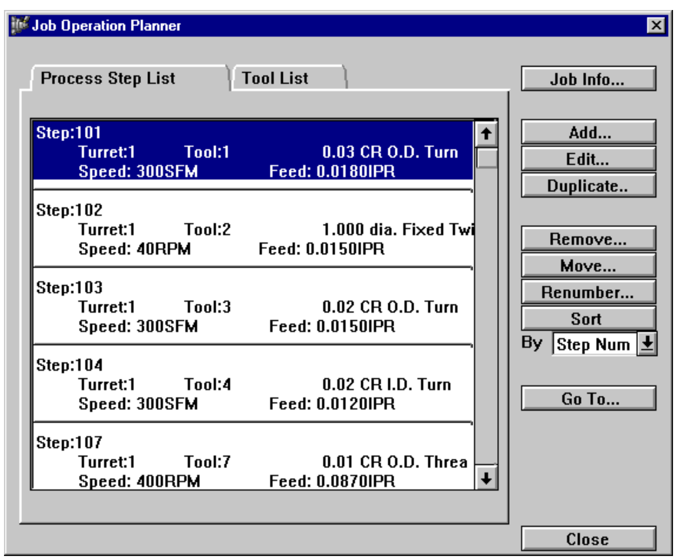
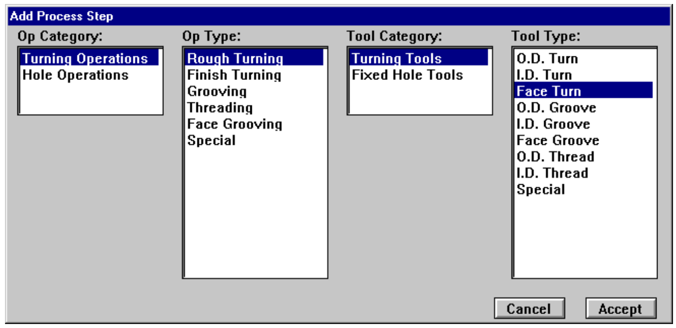



***Figure 3-10*** *File—Planner invokes the Job Operation Planner dialog box.*

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.*

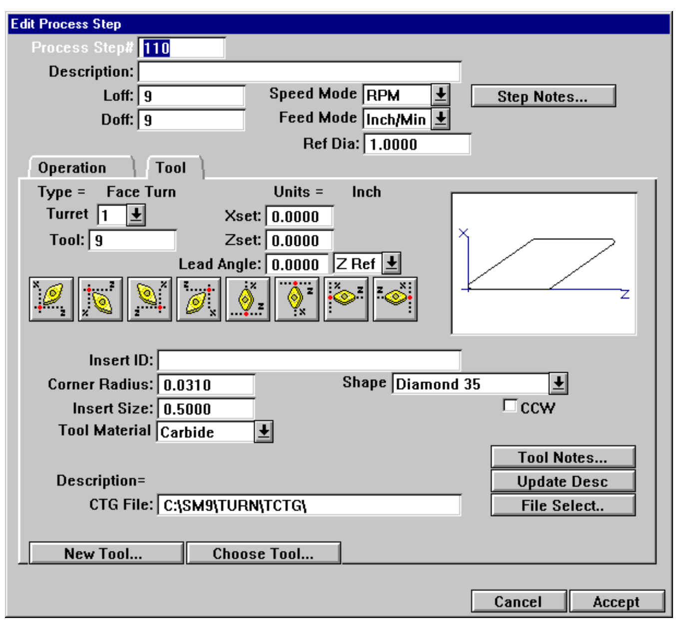



3. Complete the **Add Process Step** dialog box by selecting one operation or tool from each of the four fields:

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

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.*



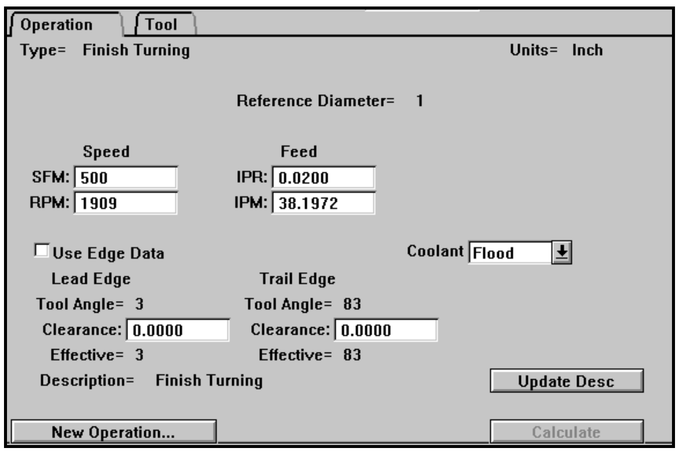

5. Set the **Process Step #** input field if needed; otherwise, a default value is placed in the input field.

**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. You can set a reference diameter on the **Operation** tab to help calculate feeds and speeds.
7. Select the **Tool** tab, and complete the **Tool** page.
8. Select the **Operation** tab, and complete the **Operation** page.

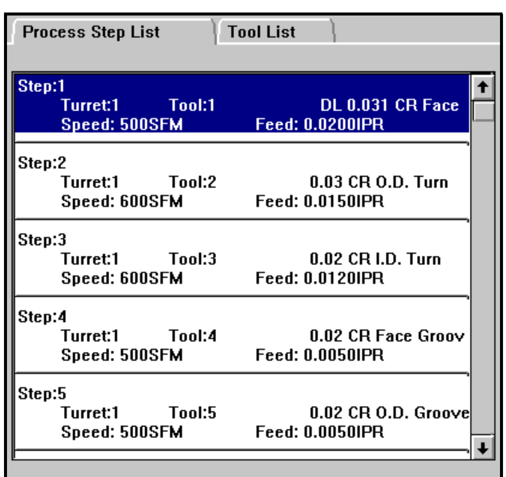
**Note** The **Feed** and **Speed** input fieldsare for Showpath (displaying the toolpath) purposes only. Code output depends on your machine, .smf file, and .tmp file.

*__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.*



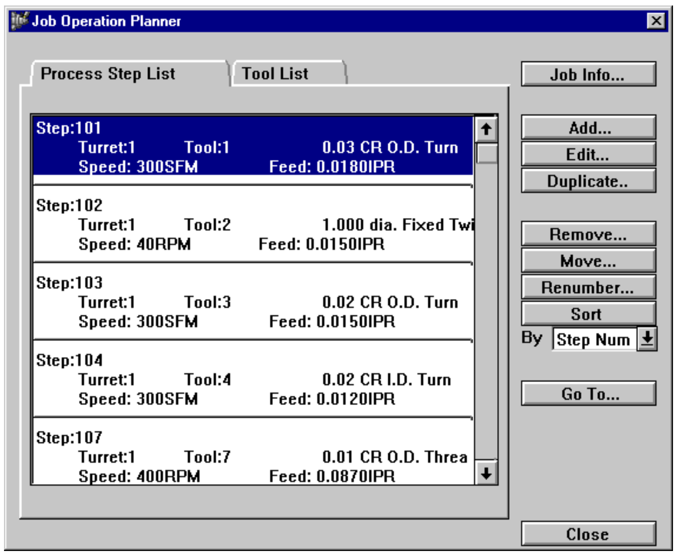
## Creating a Step Using Duplicate

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.

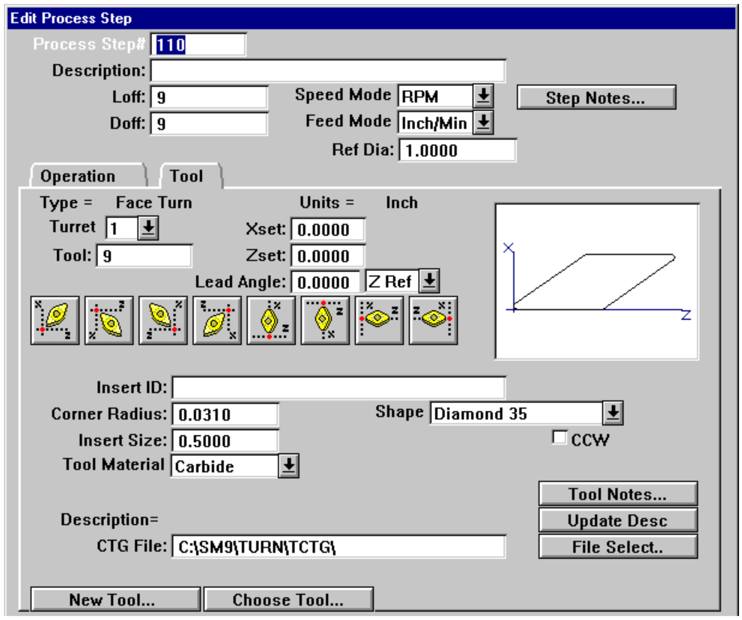



***Figure 3-16*** *Select the Duplicate button to open the Edit Process Step dialog box.*

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 3-14.

- 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.

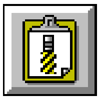
  c. Select a new tool.

  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.

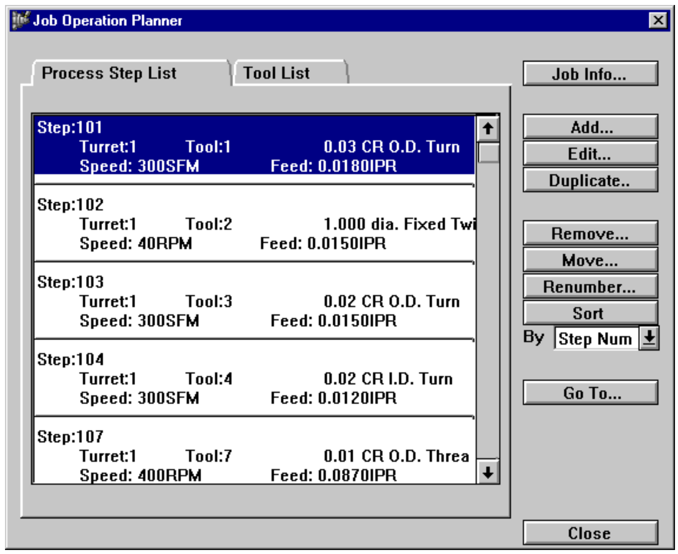



***Figure 3-17***
*File—Planner invokes the Job Operation Planner dialog box.*

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.

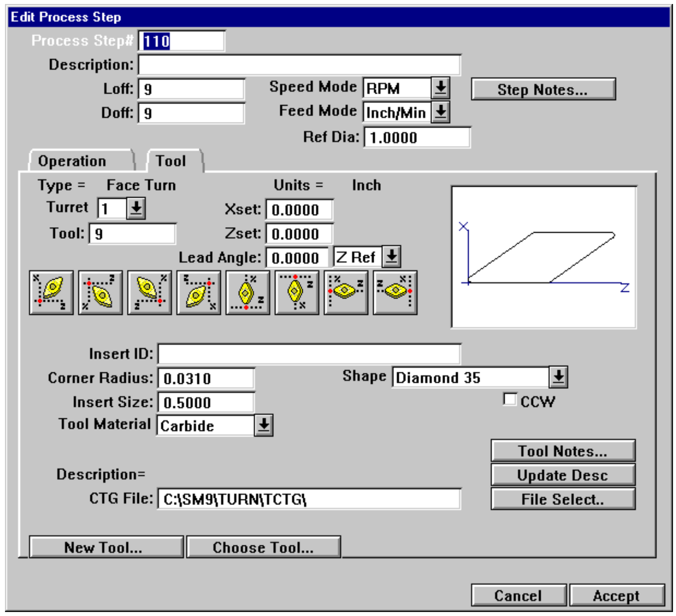



***Figure 3-18***
*Select the Edit button to open the Edit Process Step dialog box.*

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 turning tool to a facing tool. Perform these tasks to select a new tool to use:

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

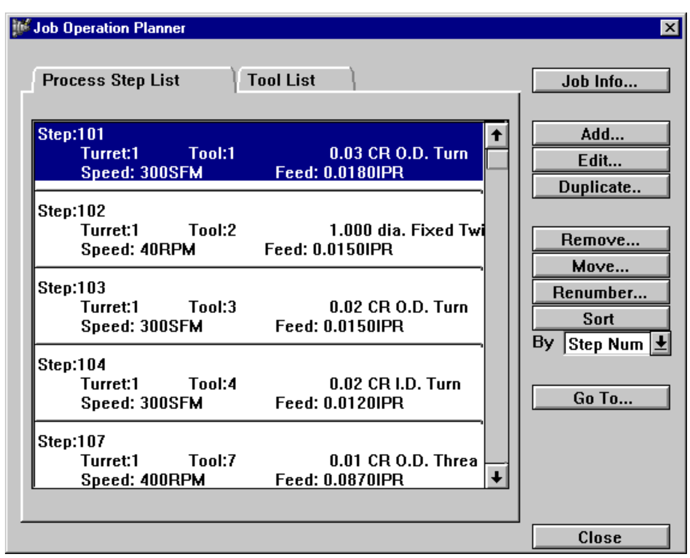
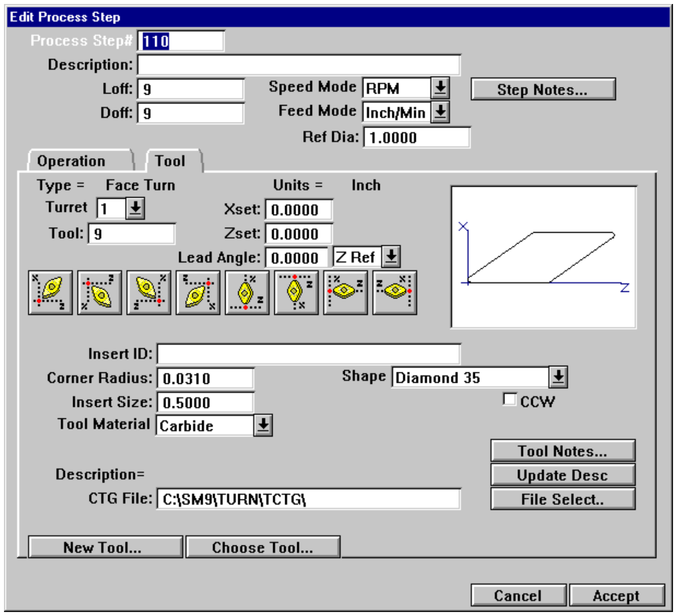



***Figure 3-19***
*File—Planner invokes the Job Operation Planner dialog box.*

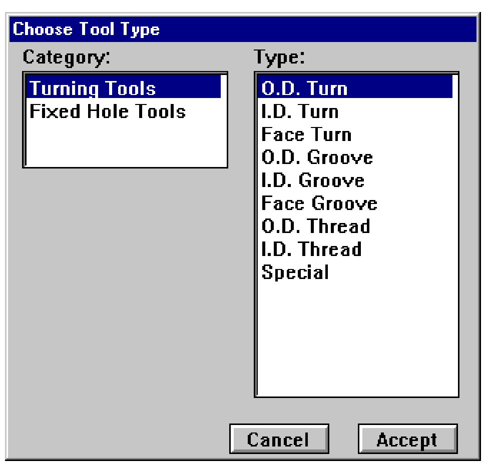
2. Select a step to edit. The step is highlighted.
3. Select the **Edit** button on the right side of the **Job Operation Planner** dialog box. The **Edit Process Step** dialog box is displayed.

***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.

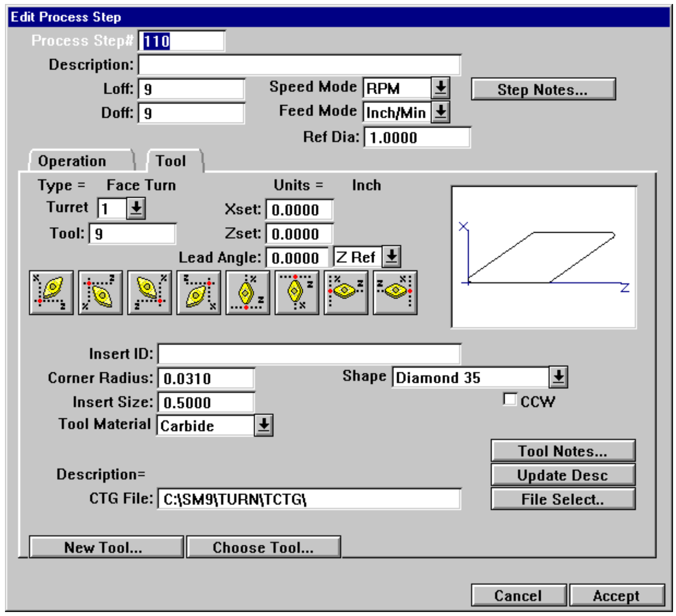



***Figure 3-22*** *Select the Accept button to open the Edit Process Step dialog box.*

9. Complete the input fields on the **Tool** page to describe the new tool.

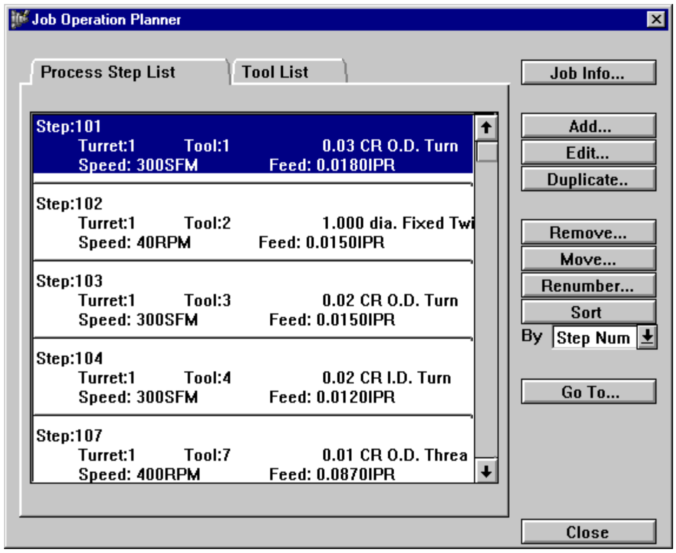
10. Select the **Accept** button.

## 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:

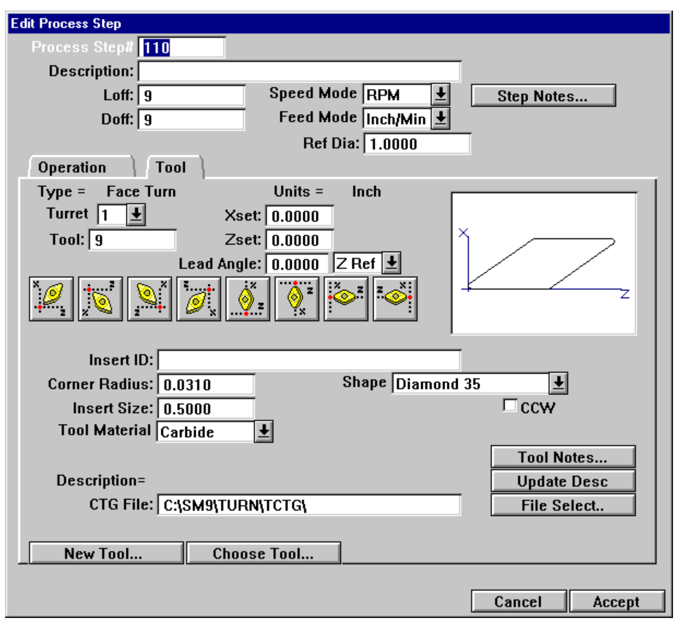
1. Select **File—Planner**. The **Job Operation Planner** dialog box is displayed.

*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.

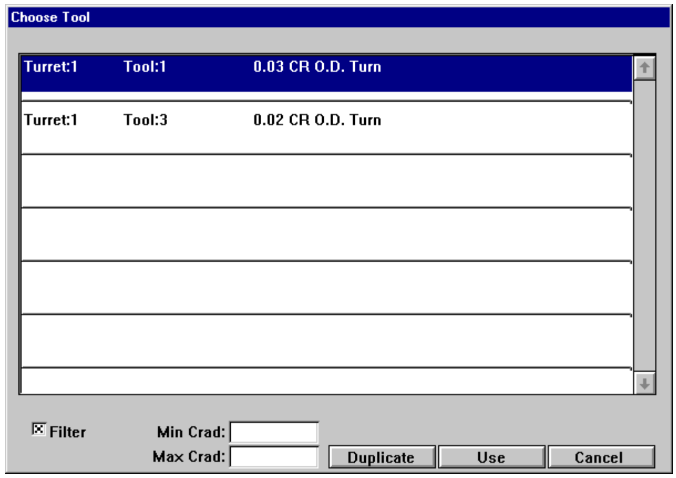



***Figure 3-25***
*Use the Choose Tool dialog box to select an existing tool.*

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 groove tool, only existing groove tools 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.

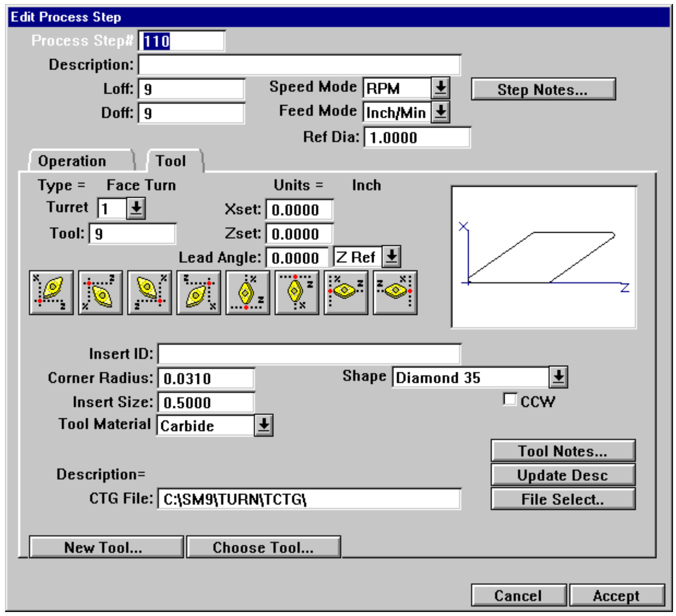



***Figure 3-26***
*Select the Duplicate or Use button to open the Edit Process Step dialog box.*

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:

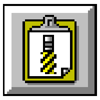
  a. Press **SHIFT +F1** to display a  cursor. Place the cursor anywhere on the Tool or Operation tab and click. A Help topic about Edit Process Step is displayed with an overview of information. If you select the **Fields** button for this topic, a new Help topic is displayed that describes the input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

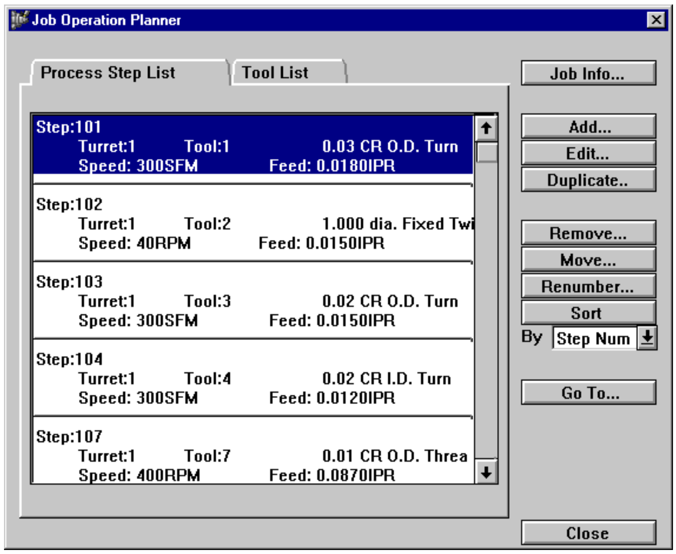



***Figure 3-27***
*File—Planner invokes the Job Operation Planner dialog box.*

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

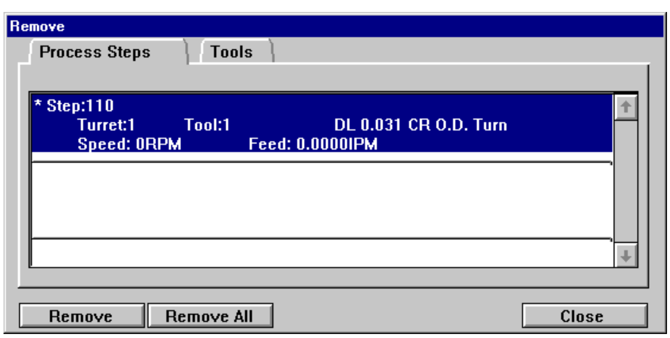
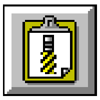



***Figure 3-28***
*Use the Remove dialog box to specify steps or tools to remove from the Job Operation Planner.*

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.

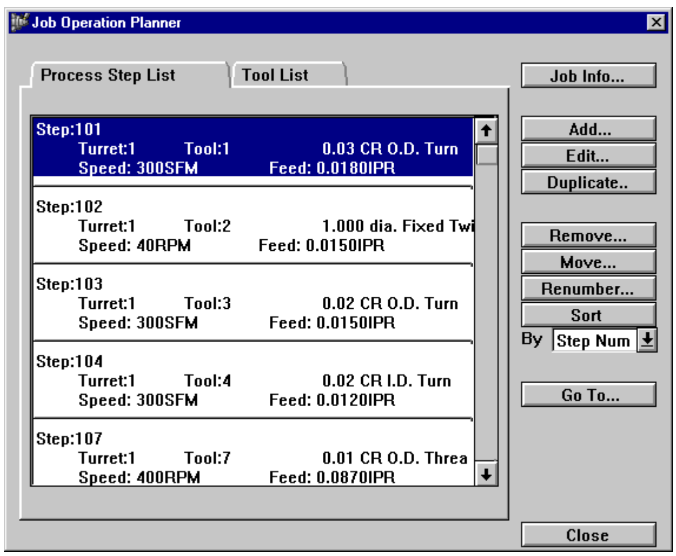



***Figure 3-29***
*File—Planner invokes the Job Operation Planner dialog box.*

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.

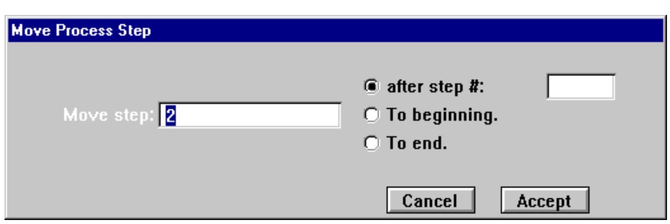



***Figure 3-30***
*Use the Move Process Step dialog box to specify where to move the step.*

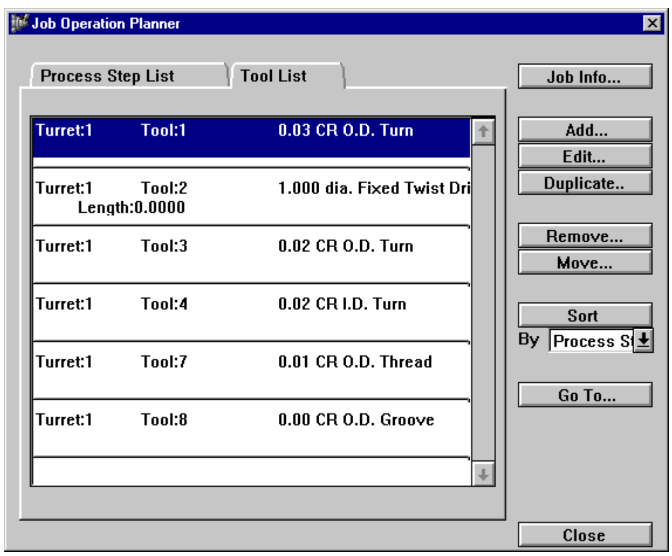
5. Specify where to move the step. You can move the step to these locations:
   - After a specified step number
   - To the beginning of the list of steps
   - To the end of the list of steps
6. Select the **Accept** button. The step is moved to its new location.

## 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:

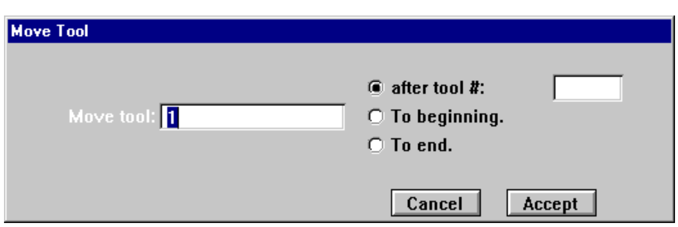
1. Select **File—Planner**. The **Job Operation Planner** dialog box is displayed.

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



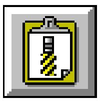

2. Select the **Tool List** tab to move a tool. The **Tool** page is displayed.
3. Select the tool to move.
4. Select the **Move** button on the right side of the dialog box. The **Move Tool** dialog box is displayed.

***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—When you specify the tool, both the turret and tool number are used and are separated by a colon (for example, turret:tool).
   - 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.

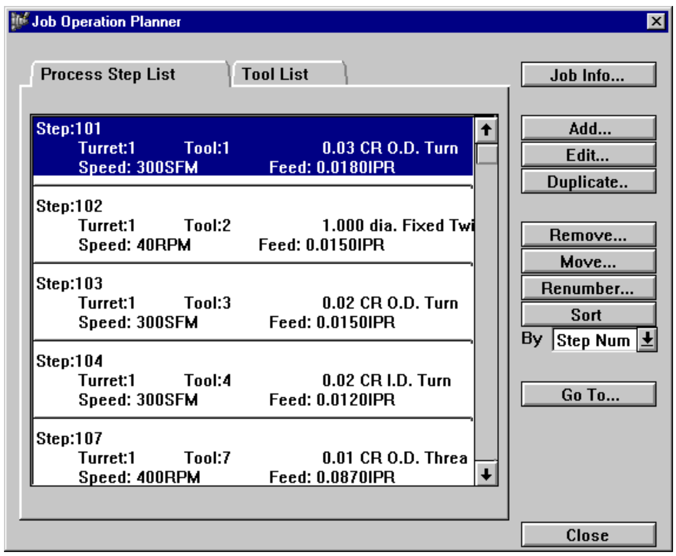



***Figure 3-33*** *File—Planner invokes the Job Operation Planner dialog box.*

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.

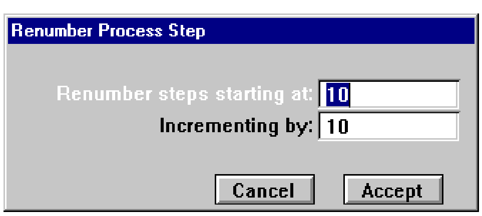
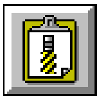



***Figure 3-34*** *Set the start and increment numbers by which to renumber steps.*

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 turret: tool number.

### Sorting Steps

Perform these tasks to sort steps:

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

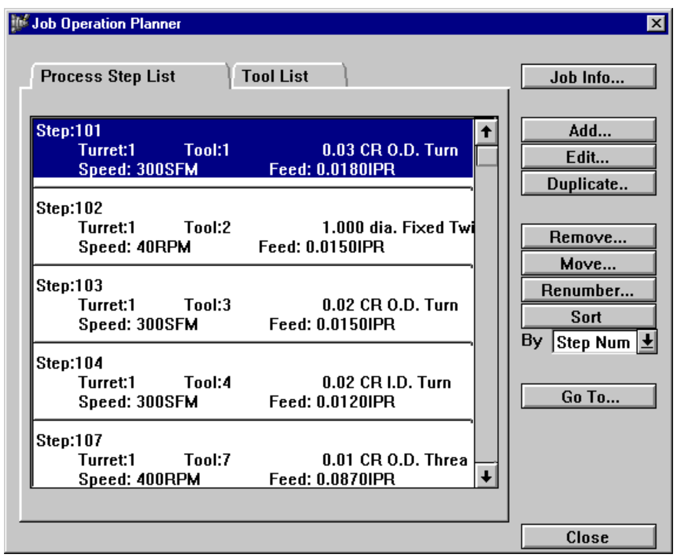



***Figure 3-35***
*File—Planner invokes the Job Operation Planner dialog box.*

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.

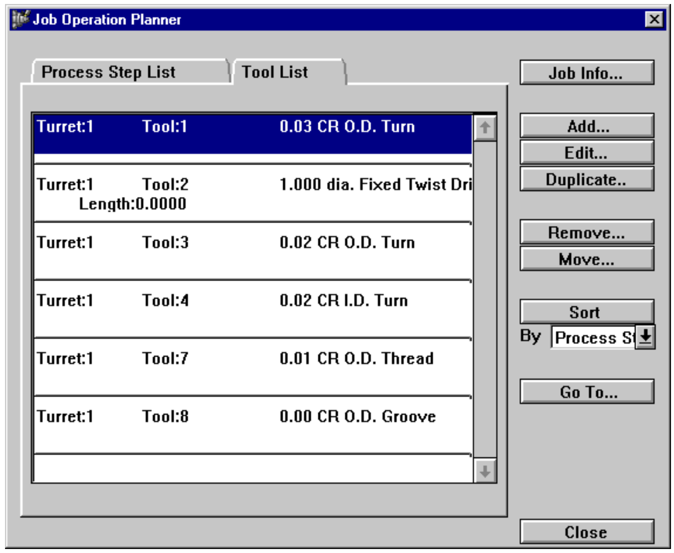



***Figure 3-36*** *File—Planner invokes the Job Operation Planner dialog box.*

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
   - **Turret/Tl** to sort by the turret: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 turret: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

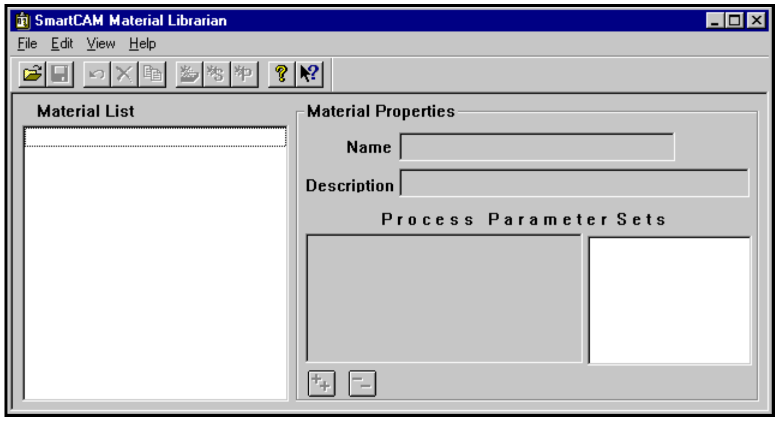
Perform these tasks to set up a file with the Material Librarian:

- 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

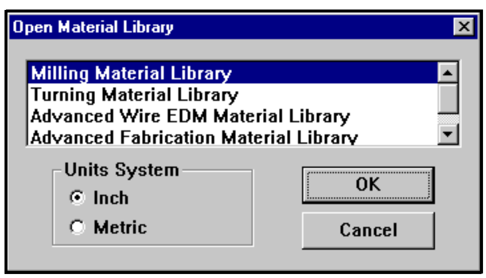
1. Select the **Material Librarian** icon from the SmartCAM program group. The **SmartCAM Material Librarian** is displayed.

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



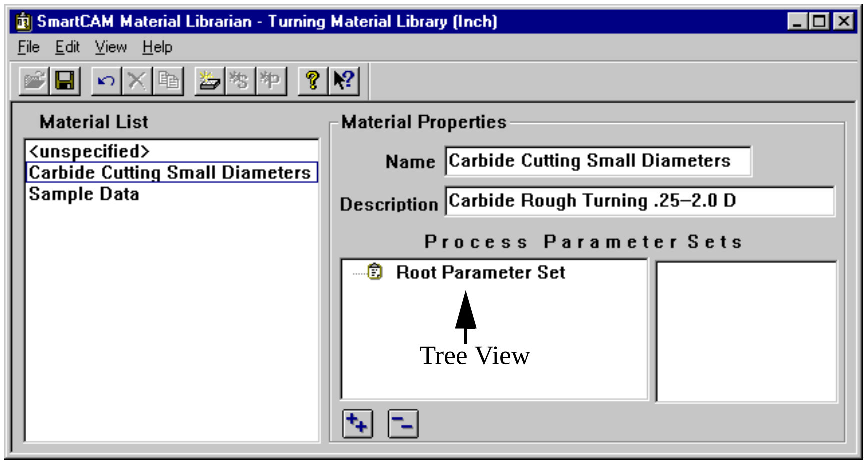

2. Select **File—Open Library**. The **Open Material Library** dialog box is displayed.

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




3. Select **Turning 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.*

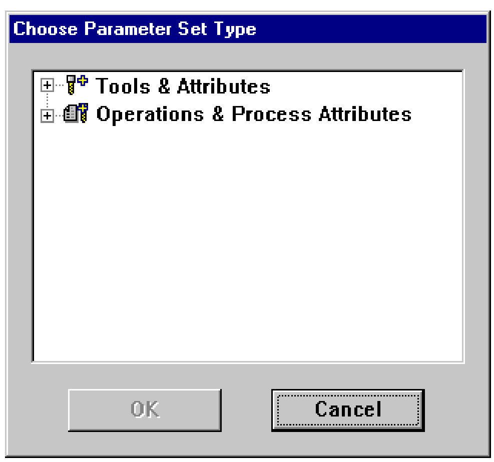



9. Keep the Material Librarian open and continue with the next procedure.

## Inserting a Parameter Set

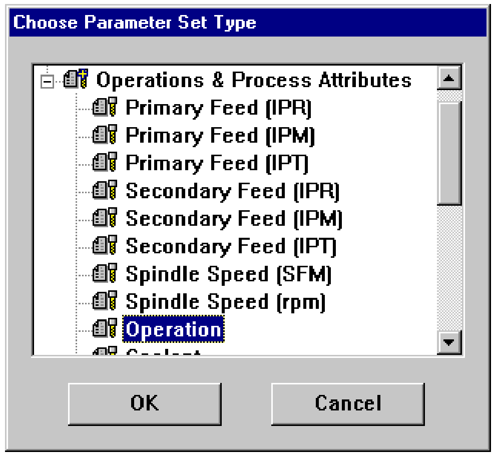
1. Highlight **Root Parameter Set**, which is displayed in the tree view, see Figure 3-39.
2. Select **Edit—Insert Parameter Set**. The **Choose Parameter Set Type** dialog box is displayed.

*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.

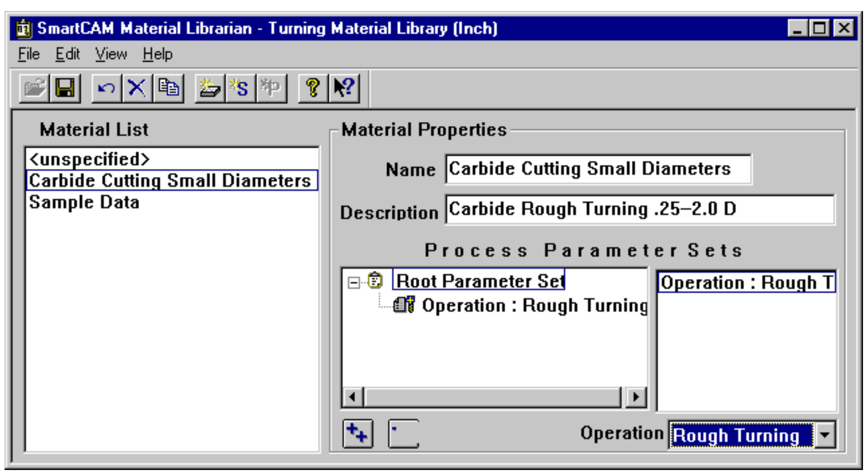
*Figure 3-41*
*Select Operation from the list.*




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 Turning**. 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

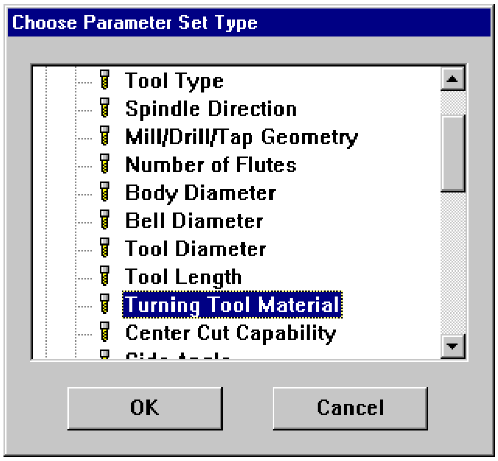
1. Select **Operation: Rough Turning** from the root parameter set in the tree view.

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.

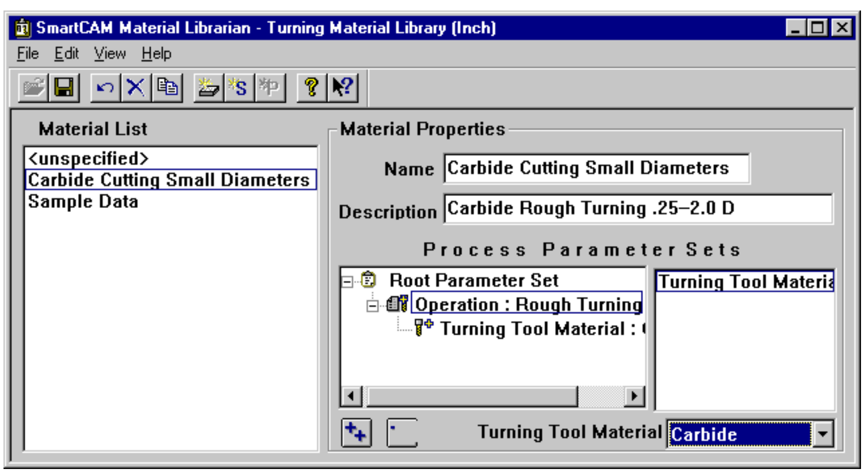
4. Select **Turning Tool Material** from the list.

*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 **Turning Tool Material** selector switch to **Carbide**. 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.

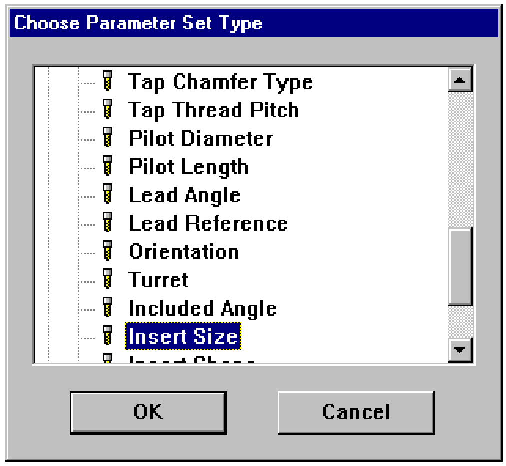
## Adding a Tool Diameter Parameter Set

1. Select **Turning Tool Material: Carbide** 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 Insert Size from the list.

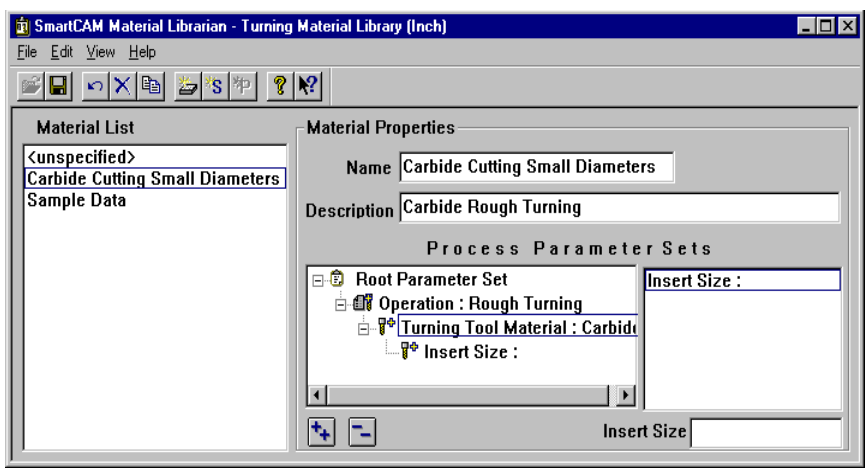
*Figure 3-45*
*Select Insert Size.*




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

6. Set the **Insert Size** input field to **.25**.

*Figure 3-46*
*Set the Insert Size value.*




7. Press Enter.

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

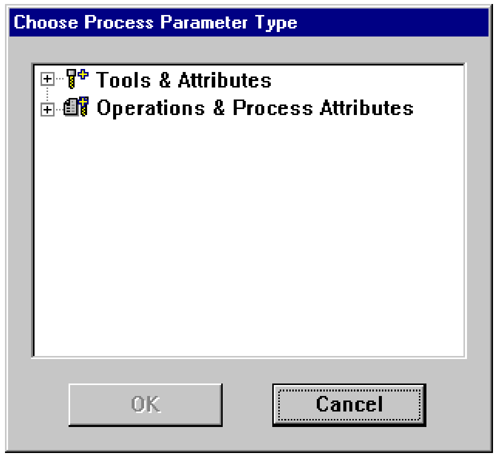
## Defining Process Parameters

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 **Insert Size: .25** from the tree view.

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

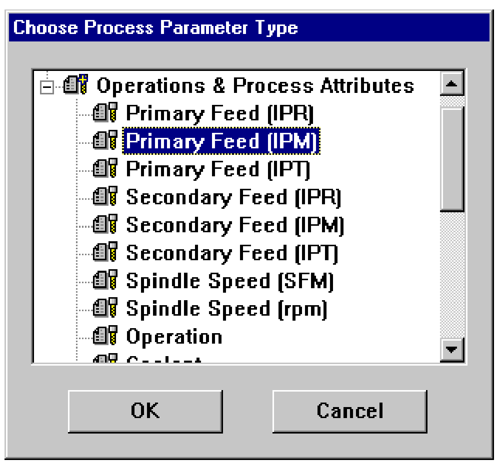
***Figure 3-47***
*Open the Choose Process Parameter Type dialog box.*




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

4. Select **Primary Feed.**

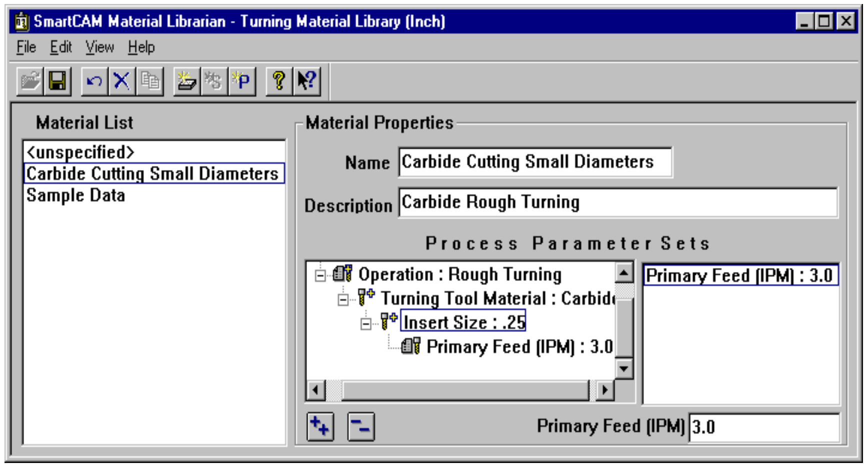
***Figure 3-48***
*Select Primary Feed.*




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 advanced turning 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.

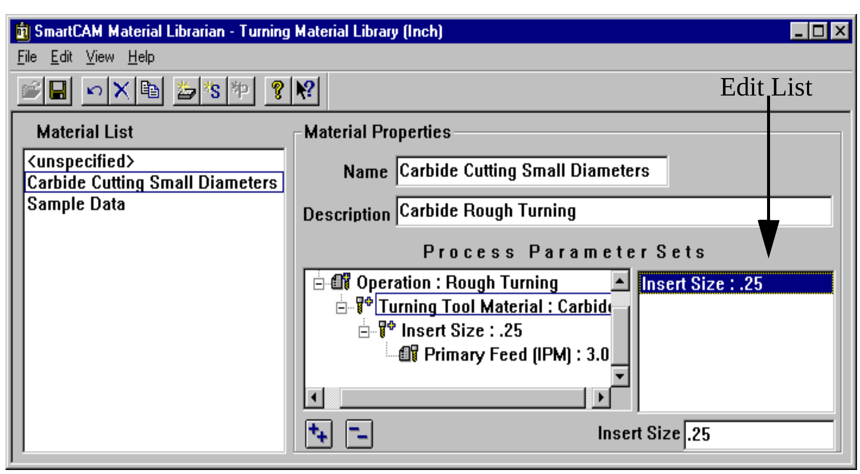
## Duplicating a Parameter Set

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 **Turning Tool Material: Carbide** from the tree view.
2. Select Insert Size: .25 from the edit list.

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

***Figure 3-50***
*Select Insert Size: .25 from the edit list.*



3. Select **Edit—Duplicate** to copy the **Insert Size: .25** parameter set and all of its parameters.
4. Select the new **Insert Size: .25**. (It is displayed below the original.)
5. Set the **Insert Size** input field to **2** to change the tool diameter.
6. Press Enter.
7. Select **Insert Size 2** from the tree view. The **Primary Feed** for **Insert Size 2** is 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. 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.

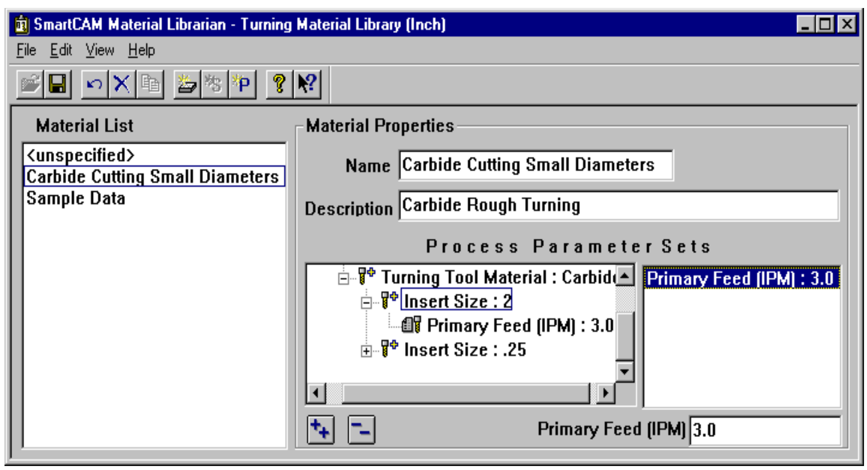



***Figure 3-51***
*The feeds and speeds for the operation are set.*

## Viewing Results

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

## Saving and Exiting

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

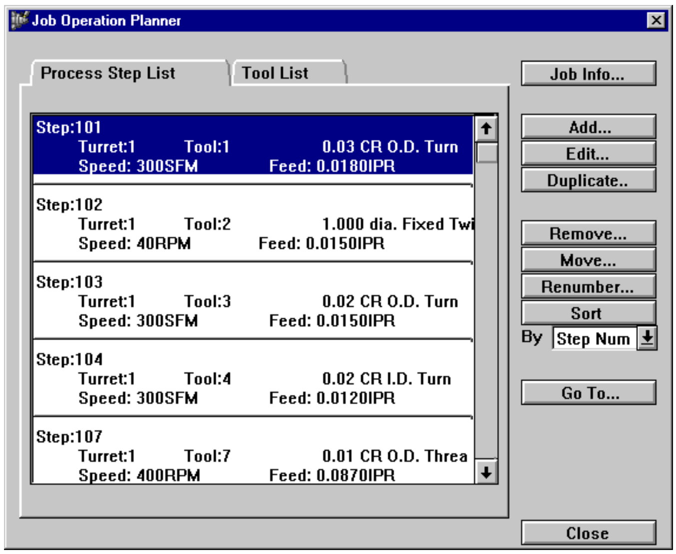
1. Select **File—Save Changes** to save the file.
2. Select **File—Exit** to close the Material Librarian.

# 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:

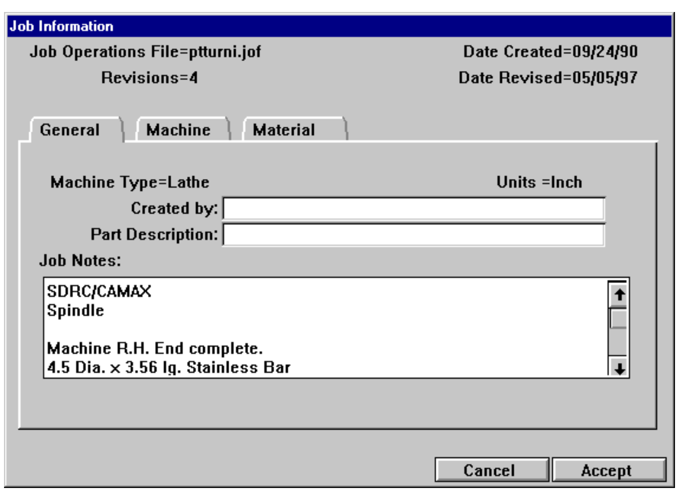
1. Select **File—Planner** from the Production Turning application. The **Job Operation Planner** dialog box is displayed.

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



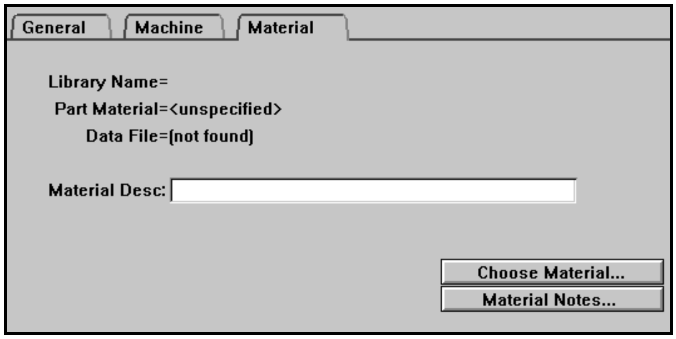

2. Select the **Job Info** button. The **Job Information** dialog box is displayed.

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



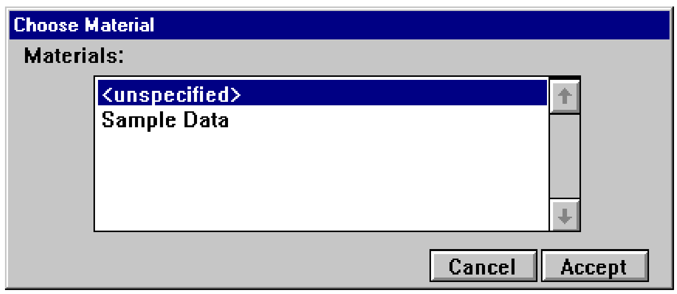

3. Select the **Material** tab. The **Material** page is displayed.

***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 Turning 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.

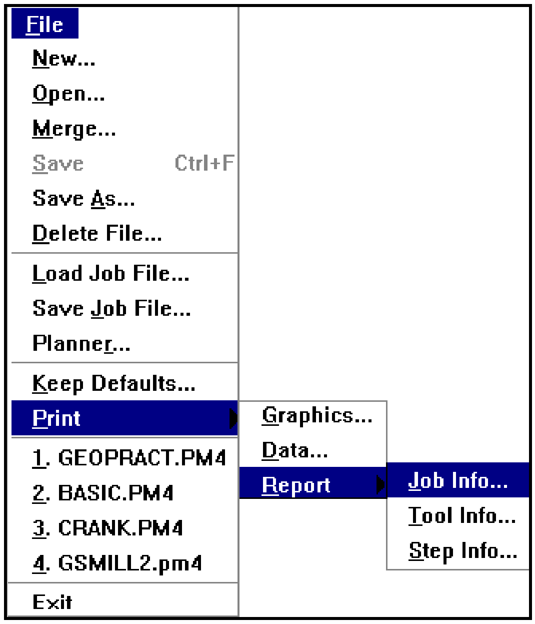



***Figure 3-56***
*Select File—Print—Report, and specify the type of report to work with.*

# 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.

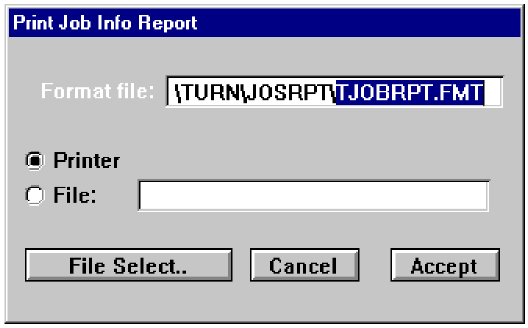



***Figure 3-57***
*Use the Print Job Info Report dialog box to create operator instructions.*

### 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.

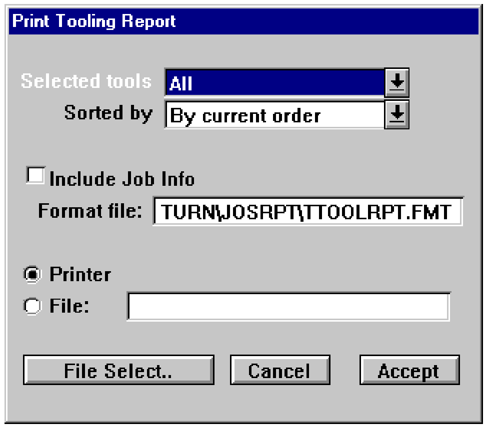



***Figure 3-58***
*Use the Print Tooling Report dialog box to limit the report.*

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.

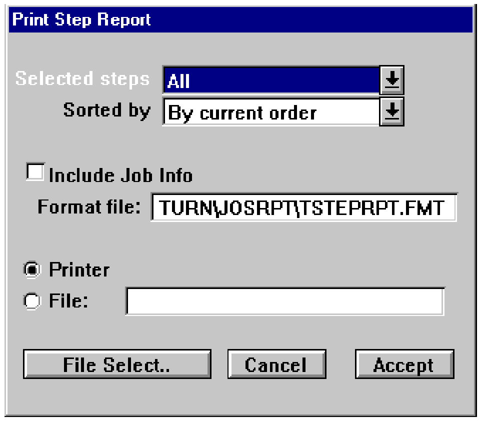



***Figure 3-59***
*Use the Print Step Report dialog box to limit the report to specific steps.*

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 Turning?**

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

# 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

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 Order Path
- Transforming Geometry
- Importing a CAD File
- Exporting a SmartCAM File

# Creating Elements

## Objectives

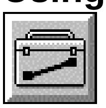
This lesson shows you how to perform these tasks:

- 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

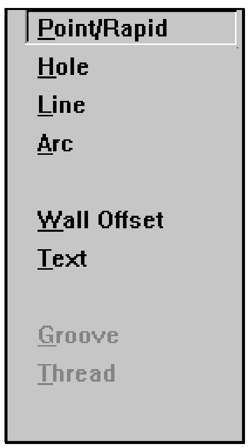
With SmartCAM, you can create a variety of geometric elements from various points. This geometry is affected by the way in which the X Parameters switch is set. If you have diameter information and not radius information, select Utility—Display Modes, and turn on the X Parameters on/off switch. Otherwise, turn this switch off. 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

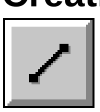


Use the Geometry toolbox modeling tools to create point rapid, hole, line, and arc elements.

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




## Creating Lines



**Model File: PTLINES.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 **`PTLINES.PM4`**.

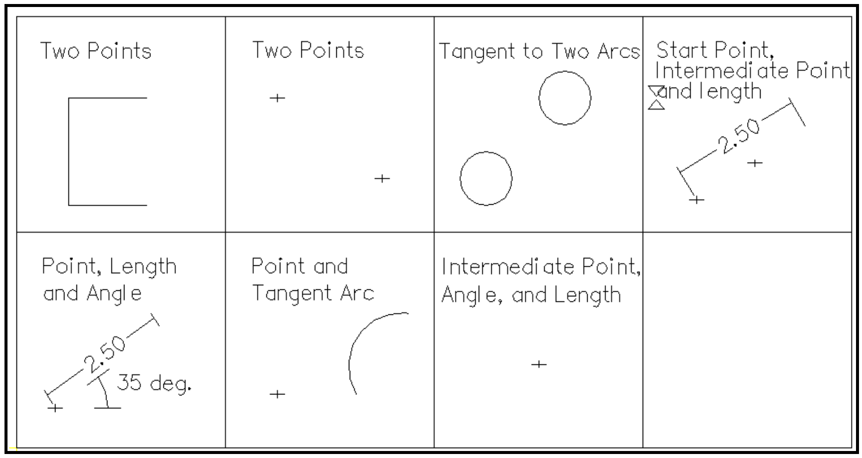



***Figure 4-2***
*Open PTLINES.*

2. Set the insert location.

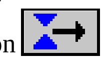
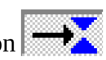
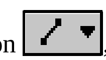
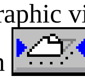
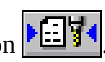
   - Set the **Before** icon  or **After** icon  to indicate if the insert point is to be before or after the selected element, process step, tool, or layer.
   - Select the **Element** icon , and select the element to be before or after in the list view or graphic view.
   - Select the **On Layer** icon  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 **Clear** and **Offset** 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.

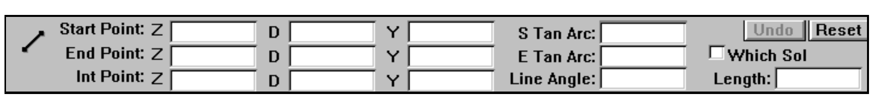



***Figure 4-4***
*Set the values on the Line control panel.*

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.

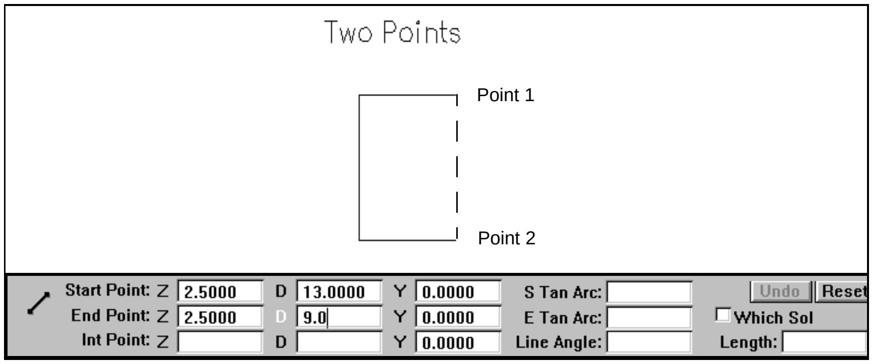



***Figure 4-5***
*This line has two known points.*

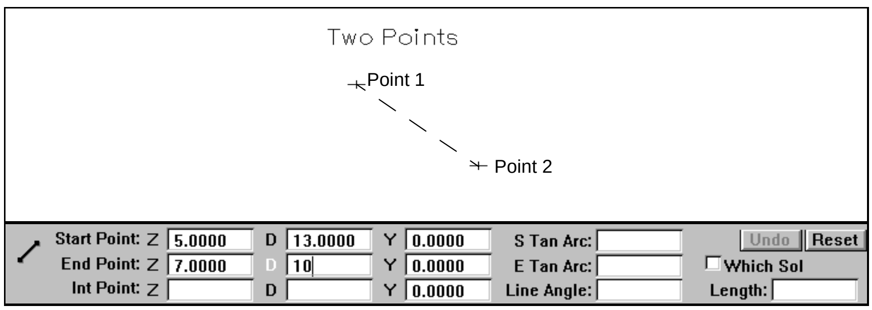



***Figure 4-6***
*This line has two known points.*

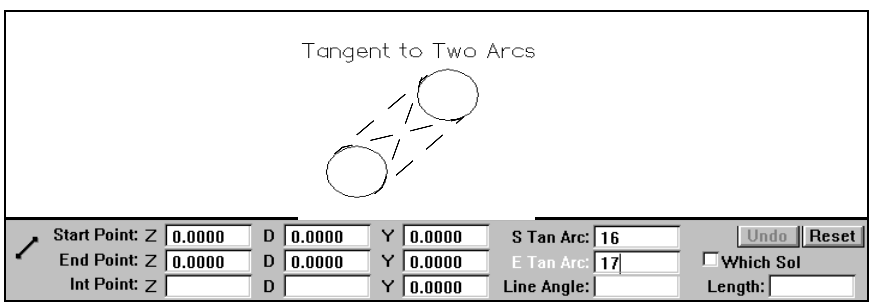



***Figure 4-7***
*This line is tangent to two arcs.*

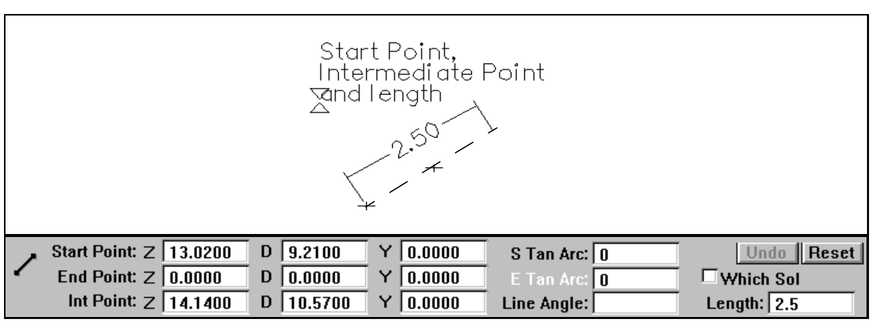



***Figure 4-8***
*This line has a known start point, intermediate point, and length.*

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

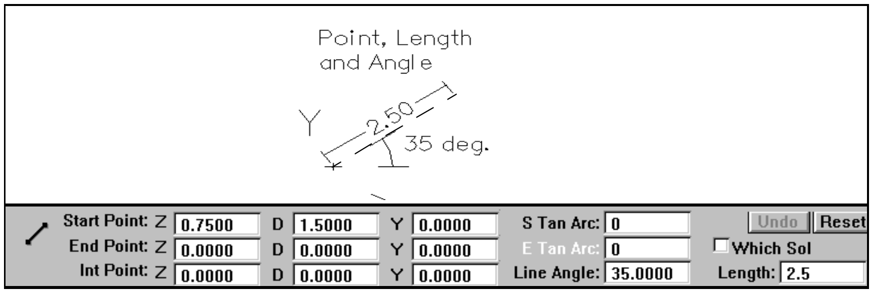



**Figure 4-10**
*This line has a known end or intermediate point, and a tangent arc.*

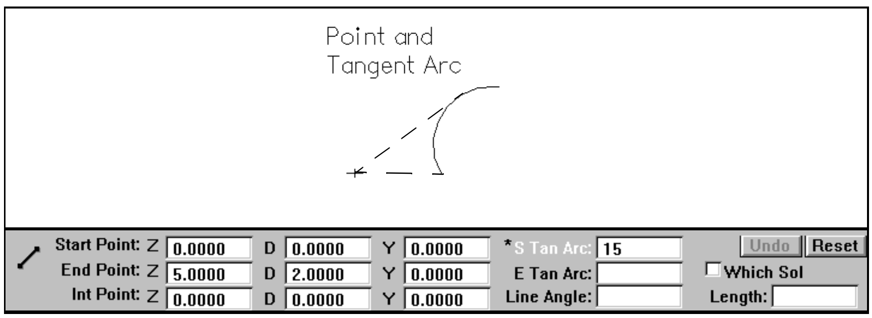
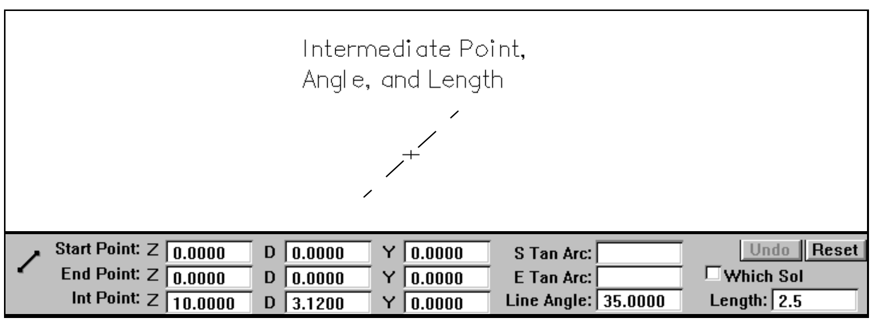
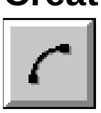



**Figure 4-11**
*This line has a known intermediate point. Specify any angle and length.*




## Creating Arcs



**Model File: PTARCS.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 **`PTARCS.PM4`**.

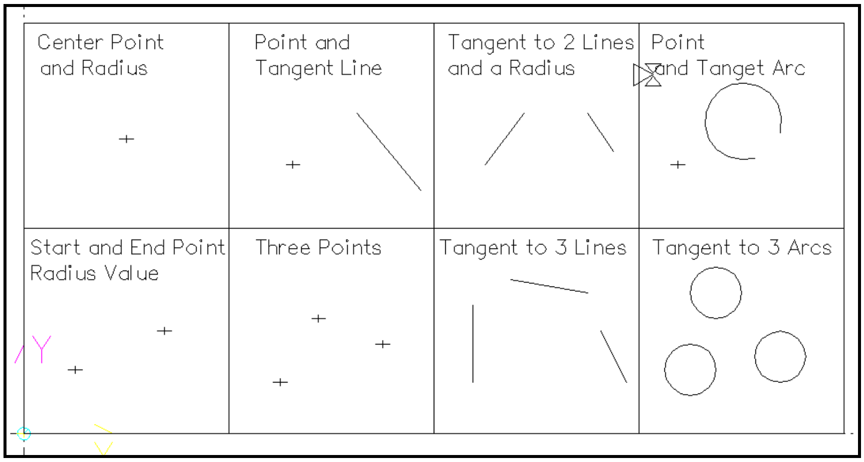



***Figure 4-12***
*Open PTARCS.*

2. Set the insert location.

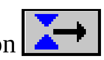
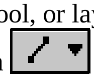
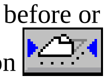
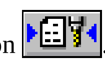
   - Set the **Before** icon  or **After** icon  on the Insert property bar to indicate if the insert point is to be before or after the selected element, process step, tool, or layer.
   - Select the **Element** icon  on the Insert property bar.
   - Select the element to be before or after in the list 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 **Clear** and **Offset** input fields on the Insert property bar. If you are inserting with a layer, set the **Work Plane** and **Layer** 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.

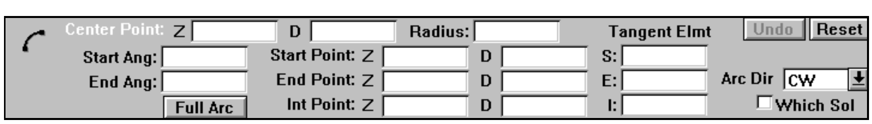



***Figure 4-14***
*Set the values on the Arc control panel.*

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**.

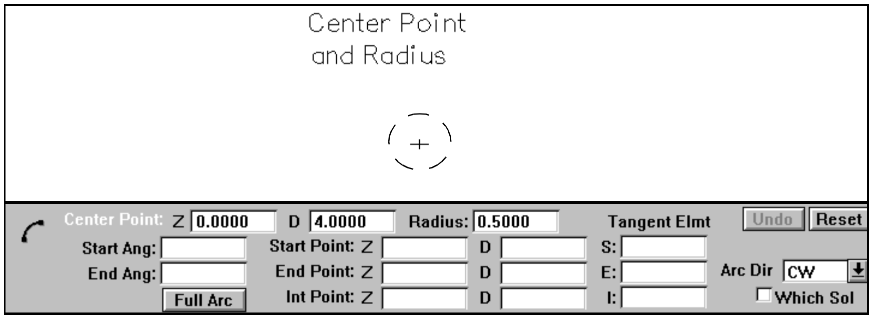



***Figure 4-15***
*Create an arc that has a known center point and any radius that you specify.*

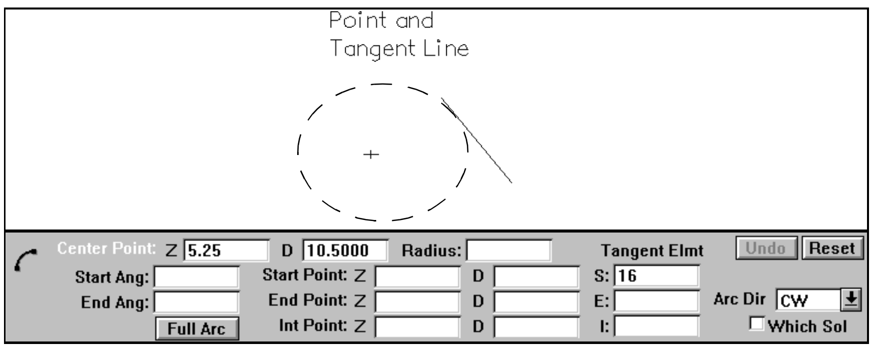



***Figure 4-16***
*Create an arc that has a center point and a tangent line.*

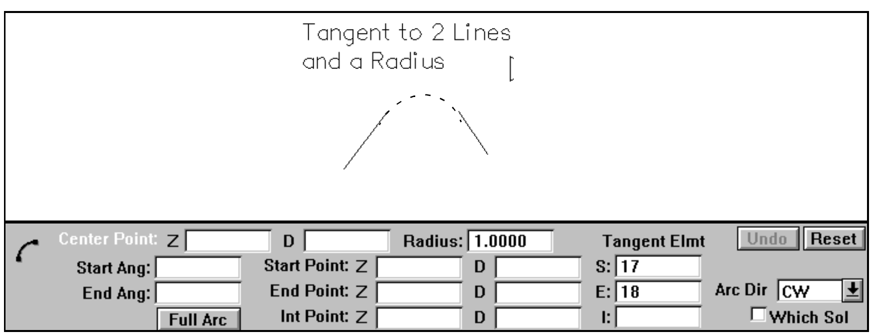



***Figure 4-17***
*Create an arc that is tangent to two lines and has any radius that you specify.*

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

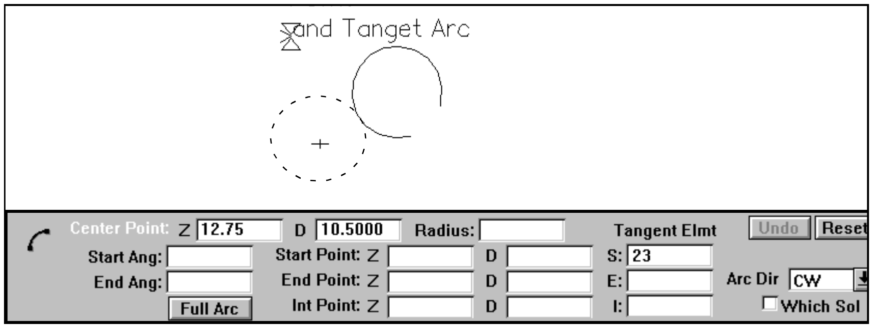



***Figure 4-19***
*Create an arc with a start point, end point, and radius.*

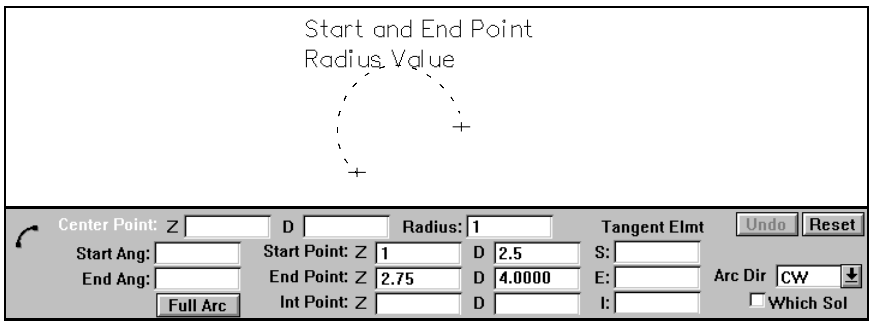



***Figure 4-20***
*Create an arc with a known intermediate point, start point, and end point.*

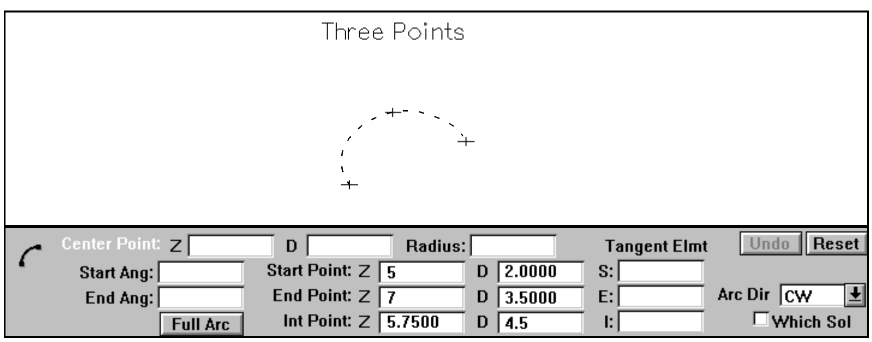



***Figure 4-21***
*Create an arc that is tangent to three lines.*

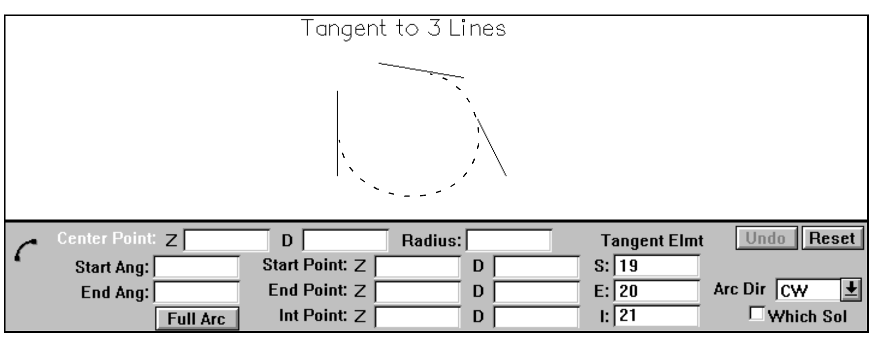
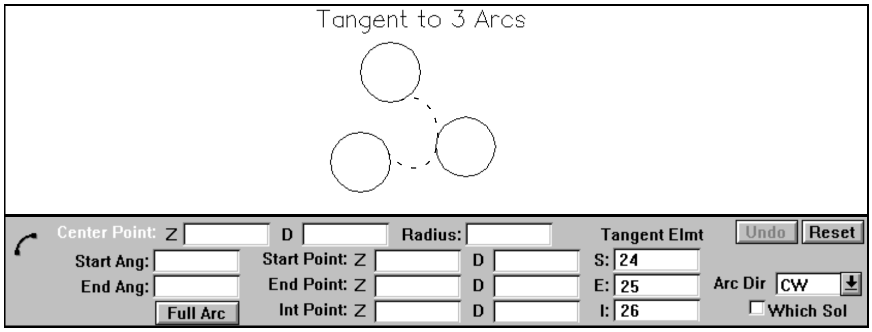
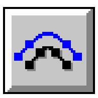


*Figure 4-22*
*Create an arc that is tangent to three arcs.*




## Creating Wall Offsets

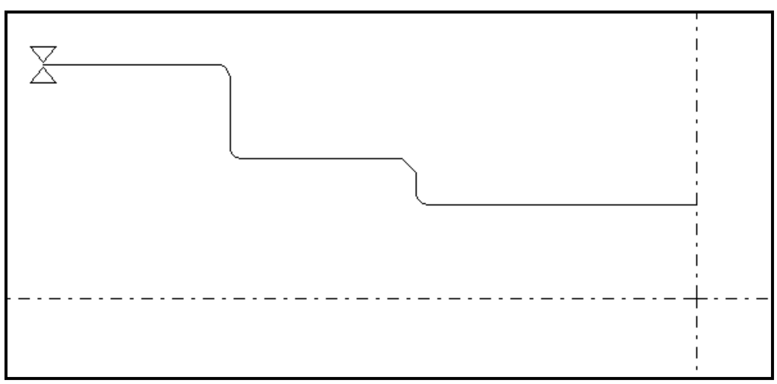


**Model File: PTOFFSET.PM4**

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 **PTOFFSET.PM4.**

*Figure 4-23*
*Open PTOFFSET.*



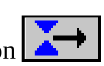
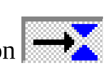
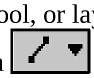
2. Set the insert location:
   - Set the **Before** icon  or **After** icon  on the Insert property bar to indicate if the insert point is to be before or after the selected element, process step, tool, or layer.
   - Select the **Element** icon  on the Insert property bar.
   - Select the element to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

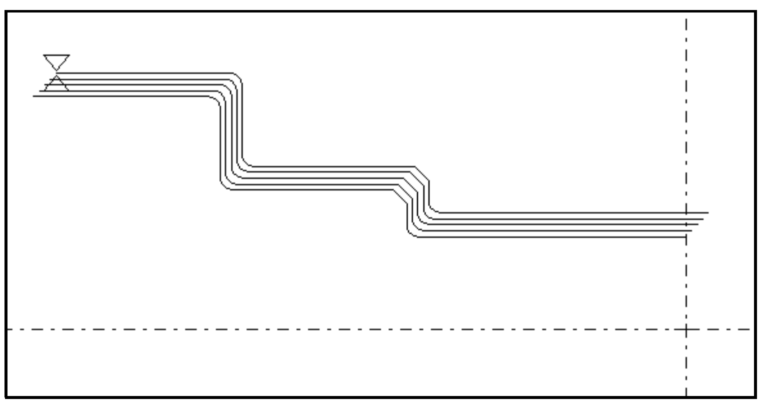
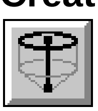


***Figure 4-26***
*This is a result of creating wall offsets.*

## Creating Holes

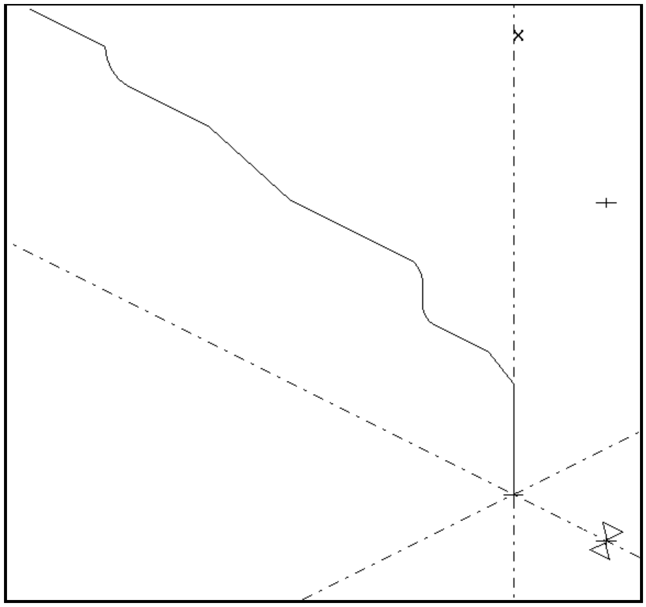


**Model File: PTHOLES.PM4**

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 **`PTHOLES.PM4`**.

***Figure 4-27***
*Open PTHOLE.*



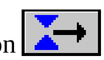
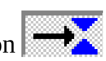
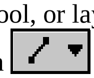
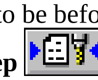
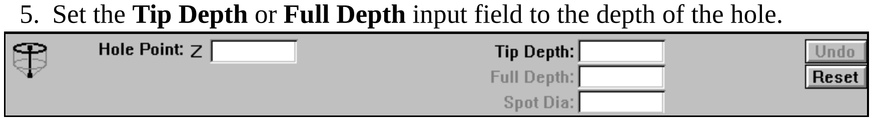
2. Set the insert location.
   - Set the **Before** icon  or **After** icon  on the Insert property bar to indicate if the insert point is to be before or after the selected element, process step, tool, or layer.
   - Select the **Element** icon  on the Insert property bar.
   - Select the element to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

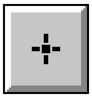



6. Locate the position of the hole by setting the **Hole Point** input field. Multiple hole operations can be performed at a single location.

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

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.

## Creating Rapid Points

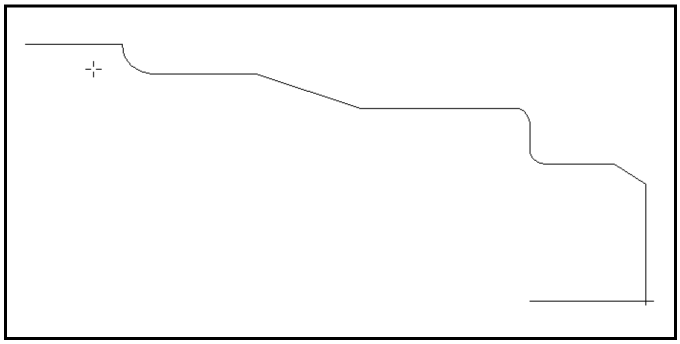


**Model File: PTRAPID.PM4**

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 **`PTRAPID.PM4`**.

***Figure 4-30***
*Open PTRAPID.*



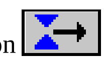
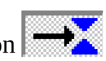
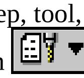
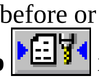
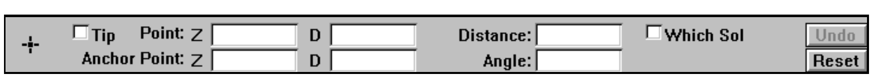
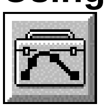
2. Set the insert location:
   - Set the **Before** icon  or **After** icon  on the Insert property bar to indicate if the insert point is to be before or after the selected element, process step, tool, or layer.
   - Select the **Step** icon  on the Insert property bar.
   - Select the step to be before 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 4-31***
*Set the values on the Point/ Rapid control panel.*




5. 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.

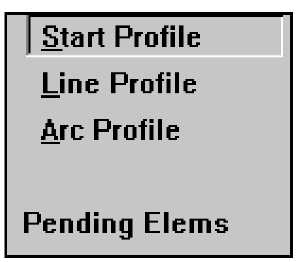



***Figure 4-32***
*Use the Profiles toolbox to create profiles.*

## 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:

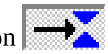
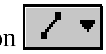
- Set the **After** icon  on the Insert property bar to indicate if the insert point is to be before or after the selected element, process step, tool, or layer.
- Select the **Element** icon  on the Insert property bar.
- Select the element to be before or after in the list view.
- Select the **With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


***Figure 4-33***
*Set the Insert property bar.*

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

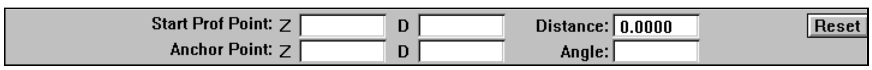



***Figure 4-34***
*Set the values on the Profile Start control panel*

5. Locate the start position by setting the **Start Prof Point** input fields, or by selecting the input field and using the snap icons.

# 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.

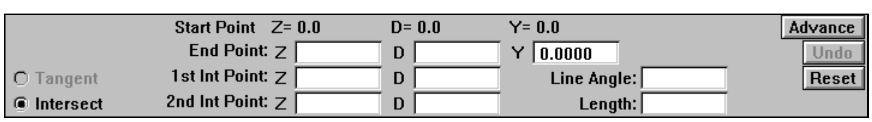



***Figure 4-35***
*Set the values on the Line Profile control panel.*

3. Locate the end position by setting the **End Point** input fields, 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.

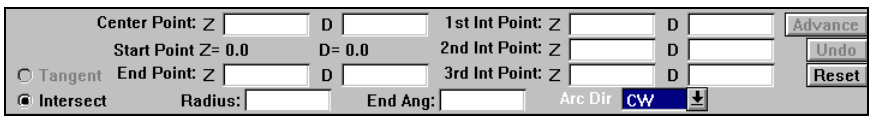



***Figure 4-36***
*Set the values on the Arc Profile control panel.*

3. Set these values on the control panel:
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 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 consists of a continuous toolpath.

As you practice with profiles, you will create a start profile, a line profile, and an arc profile. Then you will complete this part drawing.

## Beginning the Profile Drawing

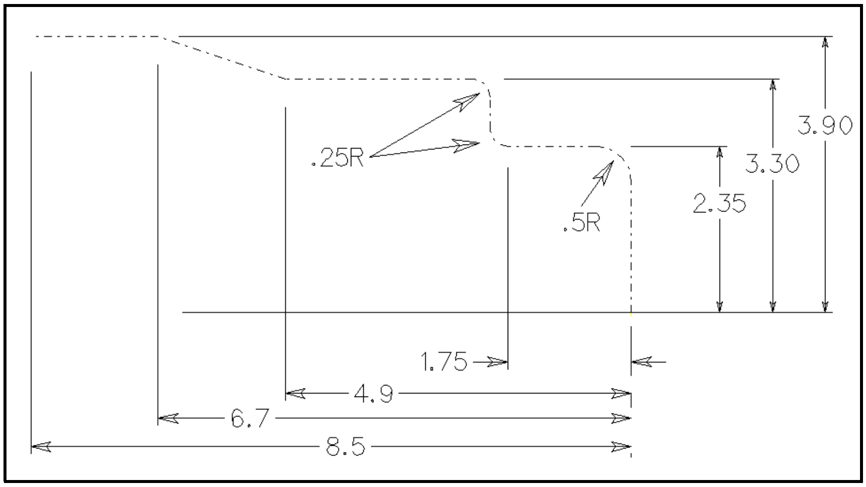



***Figure 4-37*** *As you practice with profiles, you will complete this part drawing.*

1. Set the Insert location.
2. Select **Create—Profile—Start Profile**. The **Start Profile** control panel is displayed.

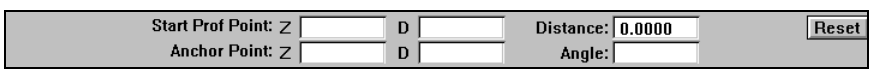



***Figure 4-38*** *Set the values on the Start Profile control panel.*

3. Locate the start position by setting **Start Prof Point Z** to **0** and **X** (or **D**) to **0**.
4. Select **Create—Profile—Line Profile**. The **Line Profile** control panel is displayed.

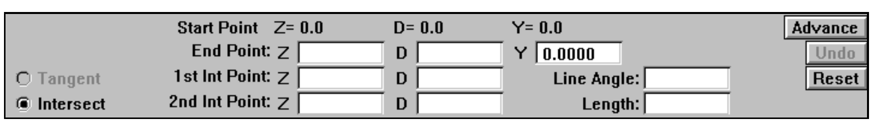



***Figure 4-39*** *Set the values on the Line Profile control panel.*

5. Set the **Line Angle** input field to **90**.
6. Select the **Advance** button. The line profile is considered a pending element.
7. Select **Create—Profile—Arc Profile**. The **Arc Profile** control panel is displayed.

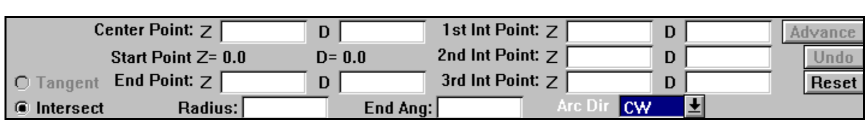



***Figure 4-40*** *Set the values on the Arc Profile control panel.*

8. Turn on the **Tangent** option switch.
9. Set the **Arc Direction** selector switch to **CCW**.
10. Set the **Radius** input field to **.5** [**12.7**].
11. Set **End Point** along the X axis to **2.35** [**59.69**].

12. Set the **End Angle** input field to **90**.

## Completing the Profile Drawing

1. Select **Line Profile** from the toolbox. The **Line Profile** control panel is displayed.
2. Make these settings on the **Line Profile** control panel:
   - Set the **End Point** input fields to **Z -1.75** [**44.45**] and **X 2.35** [**59.69**].
   - Select **Arc Profile** from the toolbox. The **Arc Profile** control panel is displayed.
3. Make these settings on the **Arc Profile** control panel:
   - Relation **Tangent**
   - Arc Dir **CW**
   - Radius **.25 [6.35]**
   - Angle **180**
4. Select **Line Profile** from the toolbox. The **Line Profile** control panel is displayed.
5. Make sure that the **Relation** option switch is set to **Tangent**.
6. Select the **Advance** button.
7. Select **Arc Profile** from the toolbox. The **Arc Profile** control panel is displayed.
8. Make these settings on the **Arc Profile** control panel:
   - Relation **Tangent**
   - Arc Dir **CCW**
   - Radius **.25 [6.35]**
   - End Point **X 3.3 [83.82]**
   - Angle **90**
9. Select **Line Profile** from the toolbox. The **Line Profile** control panel is displayed.
10. Make these settings on the **Line Profile** control panel:
    - End Point **Z -4.9 [-124.46], X 3.3 [83.82]**
    - Line Angle **180**
11. Select **Line Profile** from the toolbox. The **Line Profile** control panel is displayed.

12. Set **End Point** on the **Line Profile** control panel to Z **-6.7 [-170.18]**, X **3.9 [99.06]**.
13. Select **Line Profile** from the toolbox. The **Line Profile** control panel is displayed.
14. Set **End Point** to Z **-8.5 [-215.9]**, X **3.9 [99.06]**. Your profile should match the part in the drawing, see Figure 4-37.

## Using the Curves Toolbox

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

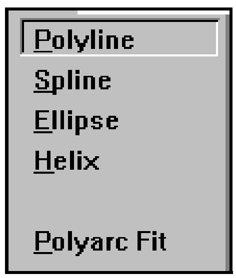
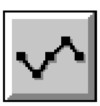



***Figure 4-41***
*Use the Curves toolbox to create polylines, splines, ellipses, and helixes.*

## 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:

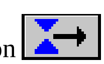
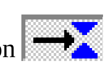
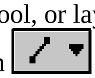
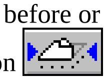
1. Set the insert location:
   - Set the **Before** icon  or **After** icon  on the Insert property bar to indicate if the insert point is to be before or after the selected element, process step, tool, or layer.
   - Select the **Element** icon  on the Insert property bar.
   - Select the element to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


***Figure 4-42***
*Set the Insert property bar.*

2. Select **Create—Curves**.

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

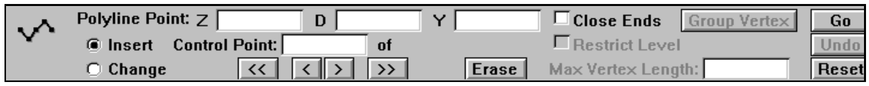
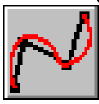



***Figure 4-43***
*Set the values on the Polyline control panel.*

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.

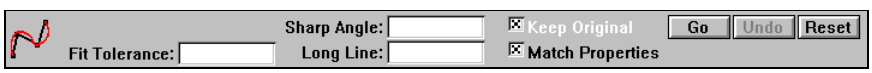
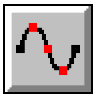



***Figure 4-44***
*Set the values on the Polyarc Fit control panel.*

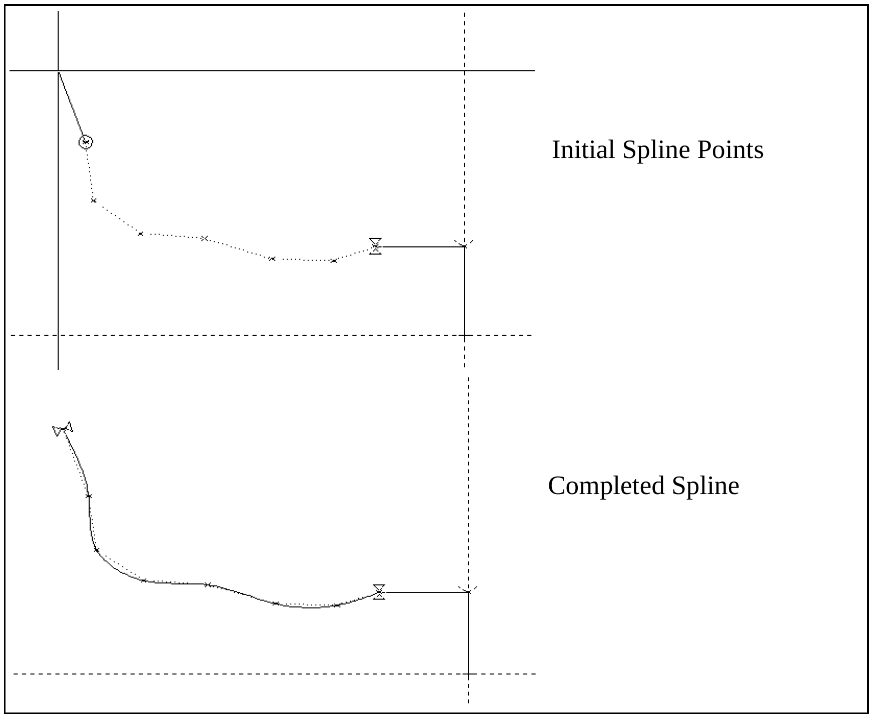
4. Group the polylines that you want to fit.
5. Set the **Fit Tolerance** input field.
6. Set the **Sharp Angle** input field.
7. Set the **Long Line** input field to the length beyond which chords are not fit.
8. Turn off the **Keep Original** on/off switch to erase the original polylines after the 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 4-45__*
*Compare initial spline points to a completed spline.*




Perform these tasks to create a spline:

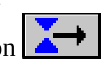
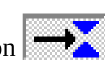
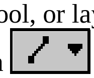
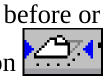
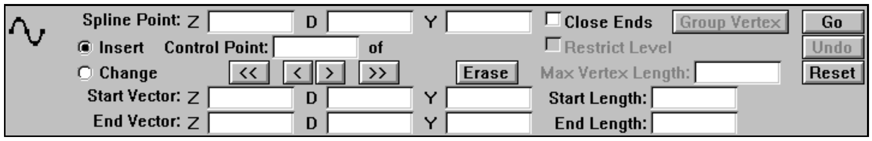
1. Set the insert location:
   - Set the Before icon  or After icon  on the Insert property bar to indicate if the insert point is to be before or after the selected element, process step, tool, or layer.
   - Select the **Element** icon  on the Insert property bar.
   - Select the element to be before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

*__Figure 4-46__*
*Set the Insert property bar.*

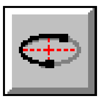

2. Select **Create—Curves**.
3. Select **Spline** from the toolbox. The **Spline** control panel is displayed.

***Figure 4-47***
*Set values on the Spline control panel.*



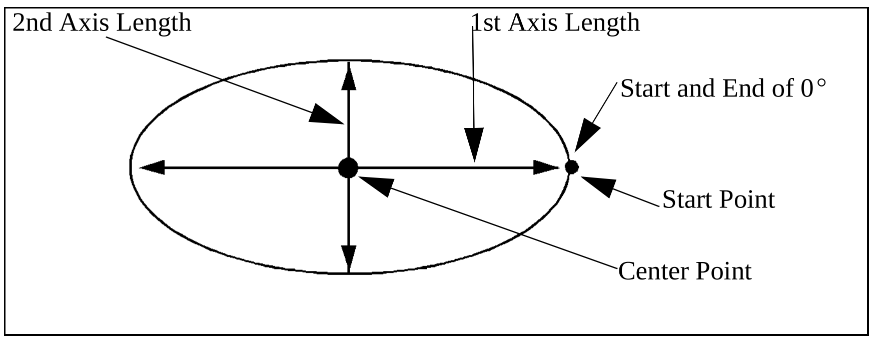

4. Set the **Spline Point** input fields to the value for each of the points.
5. Select the **Go** button.

## 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.

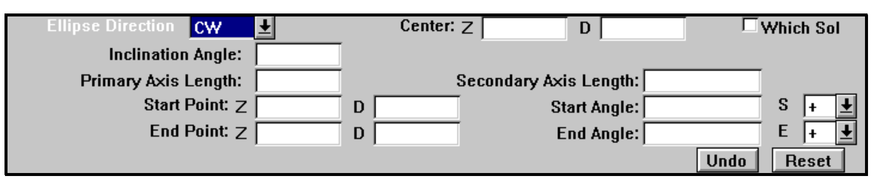
***Figure 4-48***
*Construct an ellipse from different locations.*




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.

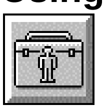
***Figure 4-49***
*Set the ellipse values.*




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 Z 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.

## Using the User Elements Toolbox



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

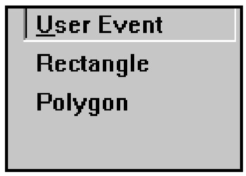
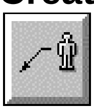



***Figure 4-50***
*Use the User Elements toolbox.*

## 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.

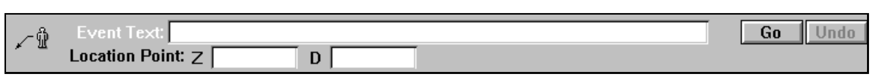
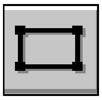



***Figure 4-51***
*Enter a command in the Event Text input field.*

4. Enter the command that you want in the **Event Text** input field.
5. 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.

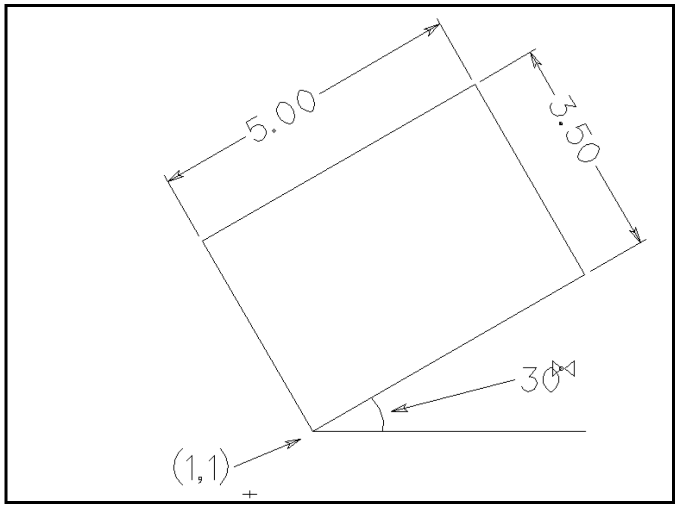



***Figure 4-52*** *Draw this rectangle.*

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.

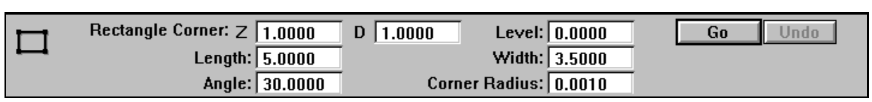
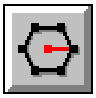



***Figure 4-53*** *Enter rectangle information.*

4. Enter values in the **Rectangle Corner**, **Length** (along Z axis), **Angle** (from Z axis, positive is counterclockwise), **Width** (along X axis), and **Corner Radius** input fields.
5. Select the **Go** button.

## Creating a Polygon



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

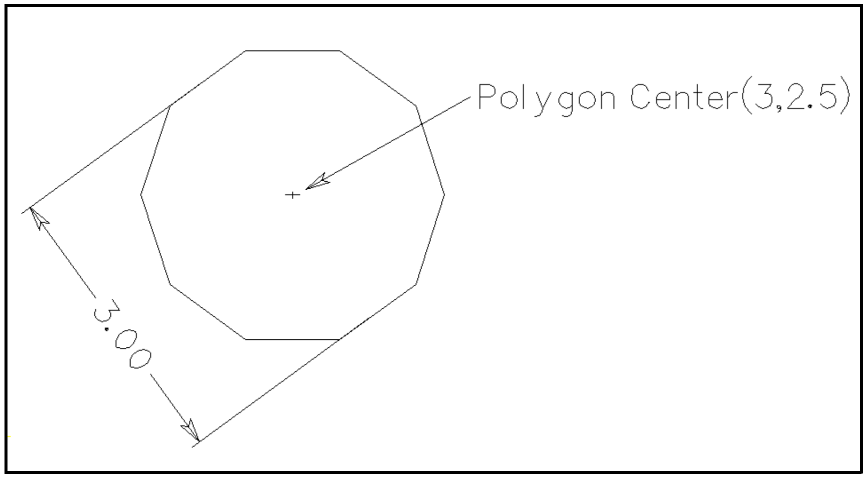



***Figure 4-54***
*Draw this polygon.*

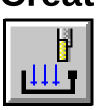
Perform these tasks to create a polygon:

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


***Figure 4-55***
*Enter polygon information.*

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

## Creating Grooves



**Model File: PTGROOV.PM4**

Grooves can be created to clean out and finish small areas in a part that a normal facing or turning operation cannot fit into. You must specify a grooving step in the Job Operation Planner, and you must select it as the active step to access the Groove control panel.

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

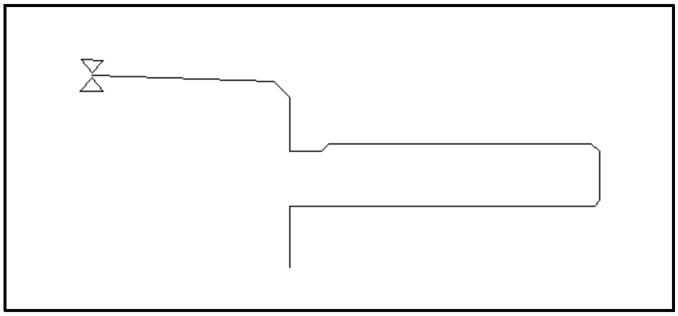
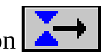
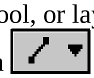


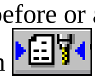
***Figure 4-56***
*Open PTGROOV.*

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  on the Insert property bar.

   ■ Select a groove step from the list view.

   ■ Set the **C (Clear)** field.

3. Select **Create—Geometry—Groove**. The **Groove** control panel is displayed.

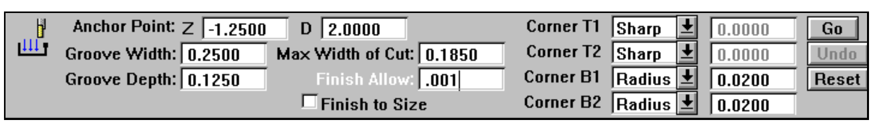



***Figure 4-57***
*Set the values on the Groove control panel.*

4. Set the Z and X positions of the anchor point. The anchor point is always in the upper left corner of the groove if **Groove Width** is positive. The anchor point is always in the upper right corner of the groove if **Groove Width** is negative.

5. Set the **Groove Width** input field to **.25[6.35]**. This is the width of the groove, measured from the anchor point. This field accepts positive or negative values, but for this lesson, use a positive value.

6. Set the **Groove Depth** input field to **.125[3.175]**. This is the depth of the groove parallel to the X axis.

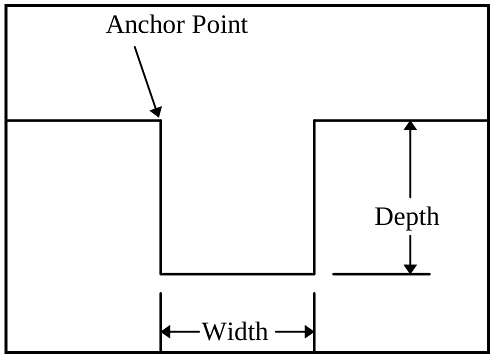



***Figure 4-58***
*Insert anchor points.*

7. Set the **Max Width of Cut** field if you want a value other than the one calculated by SmartCAM.
8. Set the **Finish Allow** field to the amount of material you want left behind. With this field active, you can also turn on the **Finish to Size** on/off switch to have a cleanup pass remove the finish allowance.
9. Set the **Corner T1**, **Corner T2**, **Corner B1**, and **Corner B2** selector switches to one of these options:
   - **Sharp** creates a sharp corner.
   - **Radius** creates a rounded corner.
   - **Chamfer** creates a 45-degree angle at the corner.

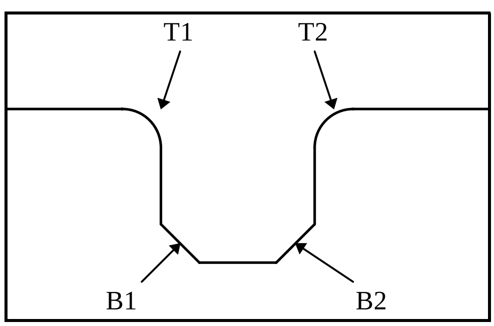
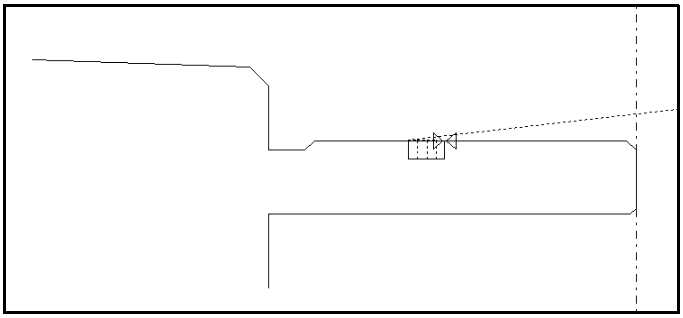


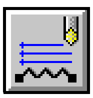

***Figure 4-59***
*Specify chamfers and radii.*

10. Select the **Go** button.
11. Select **View—Show Path**. The **Show Path** control panel is displayed.
12. Select the **Start** button. The toolpath is displayed.

*Figure 4-60*
*This is the toolpath for creating a groove.*



## Creating Threads

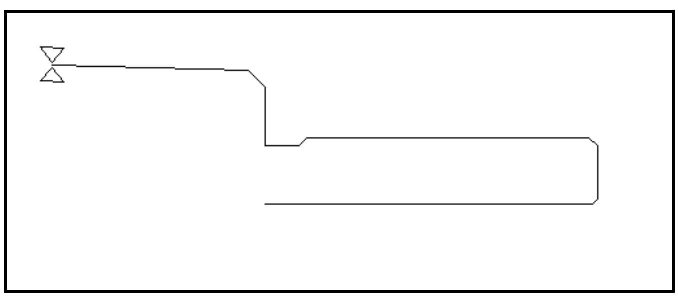


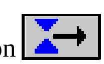
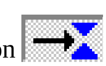
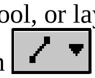
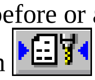
**Model File: PTHREAD.PM4**

The Thread menu is used to create ID and OD threads. A threading tool must be specified in the active step to access this menu.

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

*Figure 4-61*
*Open PTHREAD.*



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  on the Insert property bar.
   - Select a thread step from the list view.
   - Set the **Clear** field.
3. Select **Create—Geometry**.
4. Select **Thread** from the toolbox. The **Thread** control panel is displayed.

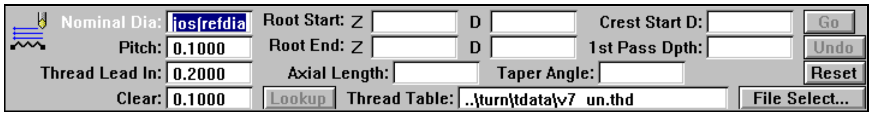



***Figure 4-62***
*Set the values on the Thread control panel.*

5. Set the following fields on the control panel:
   - Set the **Nominal Dia**. input field to the nominal diameter for the thread.
   - Set the **Pitch** input field to the pitch value of the thread (1 / thread per inch). SmartCAM calculates the **Clear** and **Thread Lead In** fields.
6. Select the **Lookup** button. SmartCAM searches a thread file to find the correct **Root Start D**, **Root End D**, **Crest Start D**, and **1st Pass Dpth** values for your thread.

   **Note** The thread file is a text file that you can customize. The Thread Table field lists the thread table file that SmartCAM uses.
7. Set the Z values for the **Root Start** and **Root End** input fields. SmartCAM calculates the **Axial Length** and **Taper Angle** fields.
8. Select the **Go** button.
9. Select **View—Show Path**. The **Show Path** control panel is displayed.
10. Select the **Start** button. The toolpath is displayed.

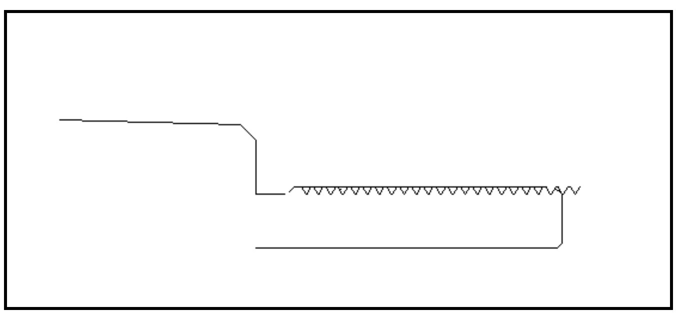
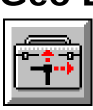


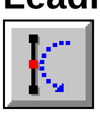
***Figure 4-63***
*This is the toolpath for creating a thread.*

## 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:PTLEAD.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

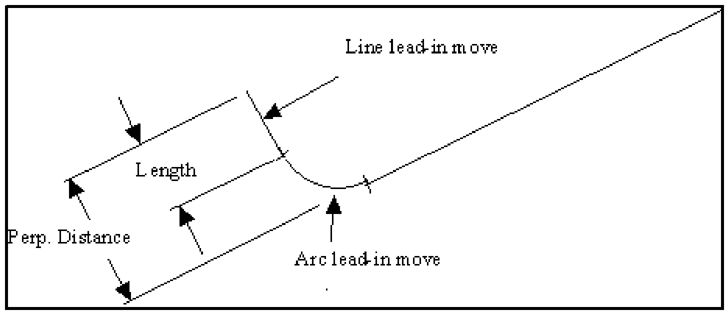
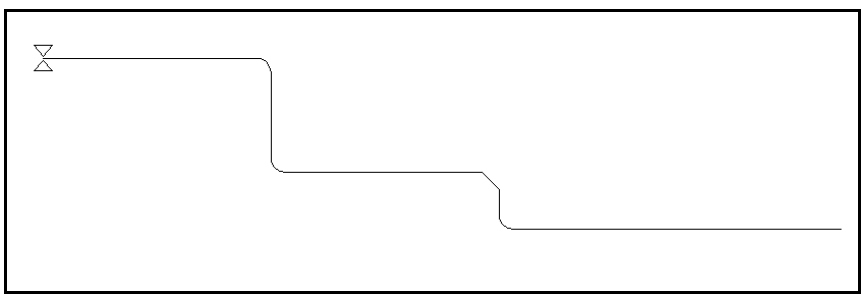



***Figure 4-64***
*Compare the line and arc lead-in moves.*

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

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



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

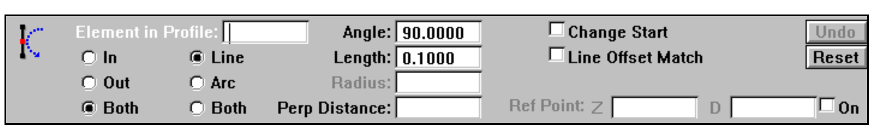



***Figure 4-65***
*Enter the Lead In/Out information.*

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. 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.

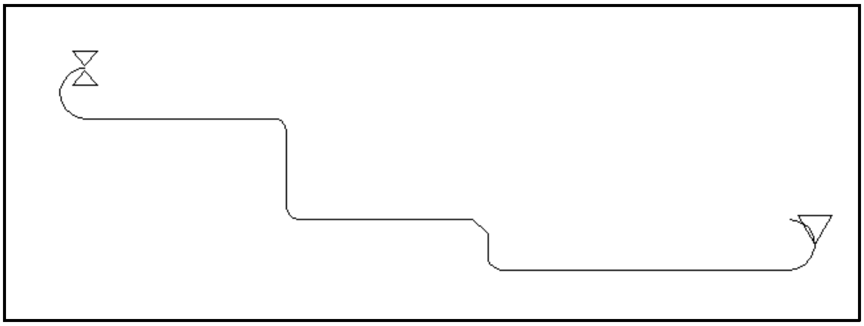


***Figure 4-66*** *This is the toolpath for creating lead in/ out move.*

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.

## 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: PTTURN.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 **`PTTURN.PM4`**.

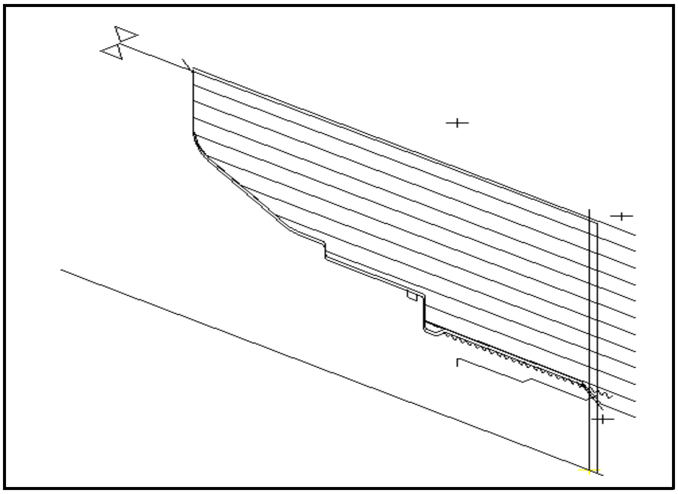


***Figure 4-67***
*Open PTTURN.*

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

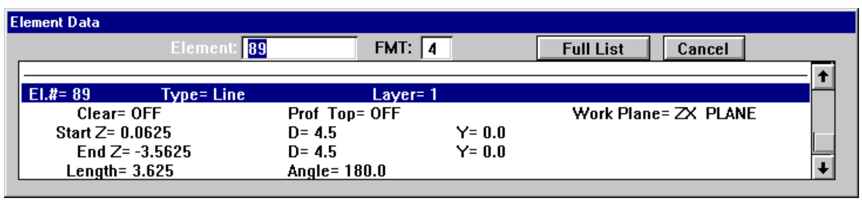



***Figure 4-68***
*Set the values on the Element Data dialog box.*

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

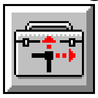
This lesson shows you how to perform these tasks:

- 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.

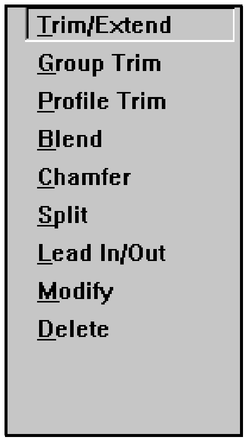
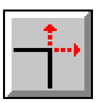



***Figure 4-69***
*Use the options in the Geo Edit toolbox.*

## Trimming and Extending Geometry



**Model File: PTTRIMEX.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 **`PTTRIMEX.PM4`**.

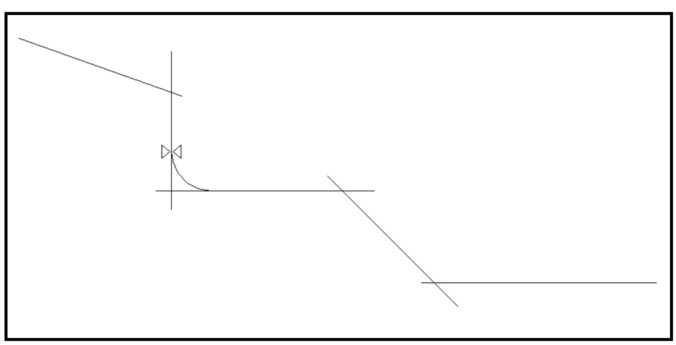


***Figure 4-70***
*Open PTTRIMEX.*

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

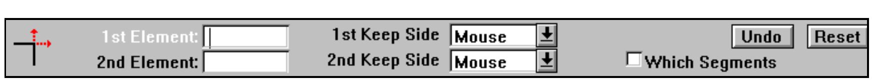



***Figure 4-71***
*Set the values on the Trim/ Extend control panel.*

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 4-72***
*Trim the lines.*

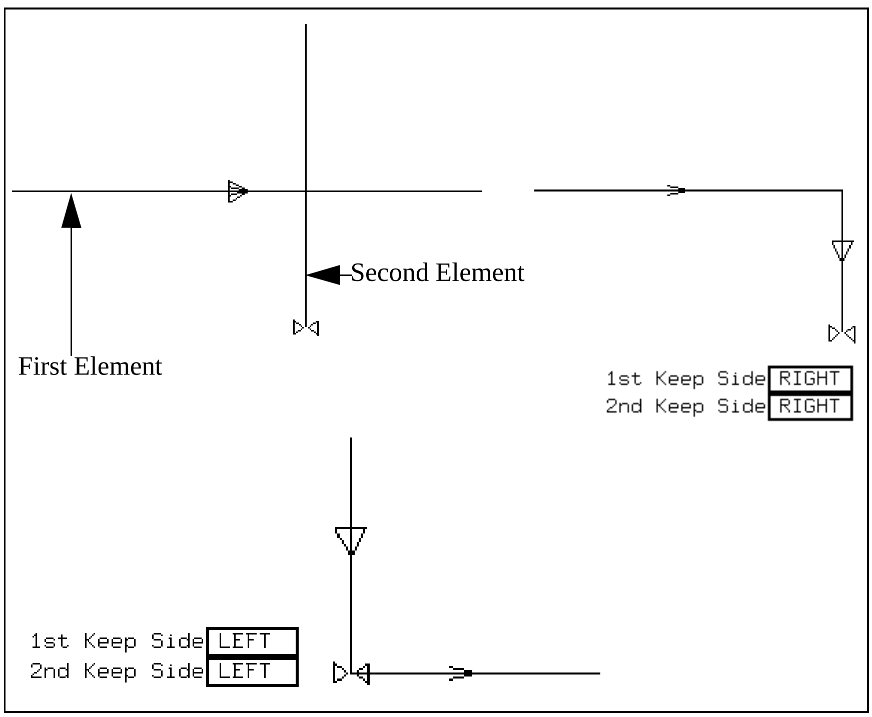
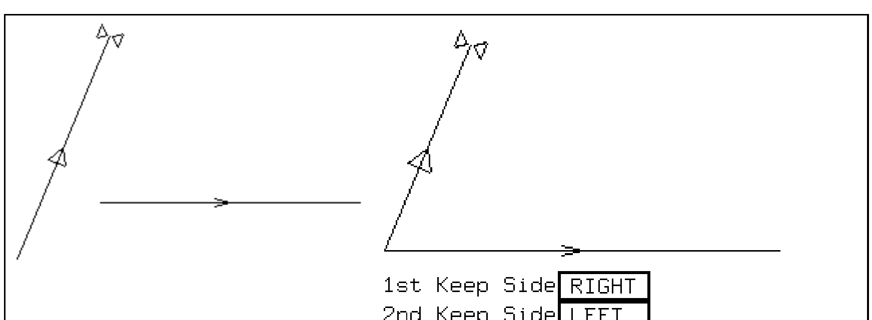



***Figure 4-73***
*Extend the lines.*




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

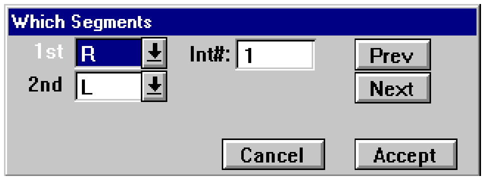
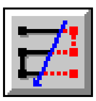



***Figure 4-74***
*Specify the segments that you want to use.*

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: PTGRPTRM.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 **`PTGRPTRM.PM4`**.

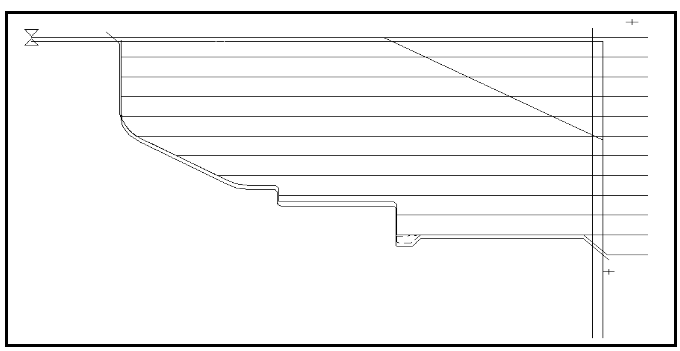


***Figure 4-75***
*Open PTGRPTRM.*

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

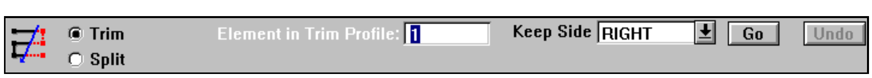



***Figure 4-76***
*Set the values on the Group Trim control panel.*

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.

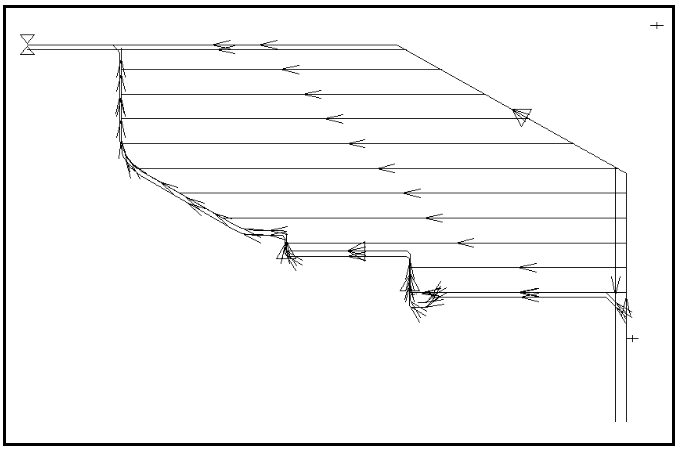
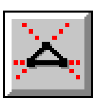


***Figure 4-77*** *This is the result of trimming the group.*

## Trimming a Profile



**Model File: PTPRFTRM.PM4**

Use Profile Trim to intersect, trim or extend, and sequentially order groups that are on the same 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 **`PTPRFTRM.PM4`**.

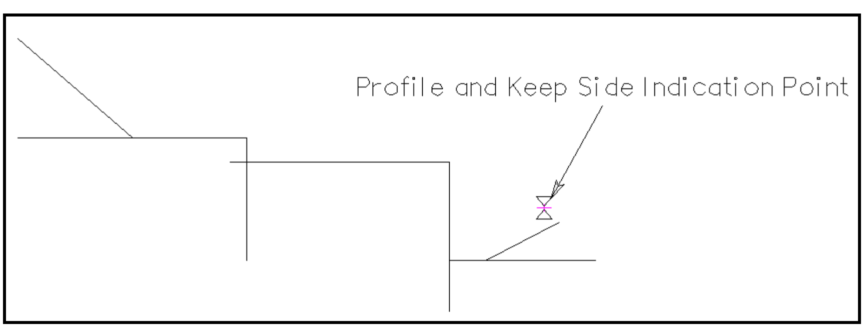



***Figure 4-78*** *Open PTPRFTRM.*

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.

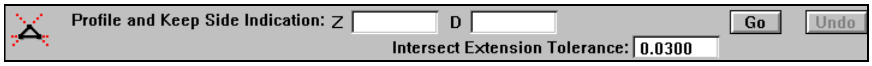



***Figure 4-79***
*Set the Profile and Keep Side Indication input fields on the Profile Trim control panel.*

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 Z and D fields or selecting a position in the graphics view.

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

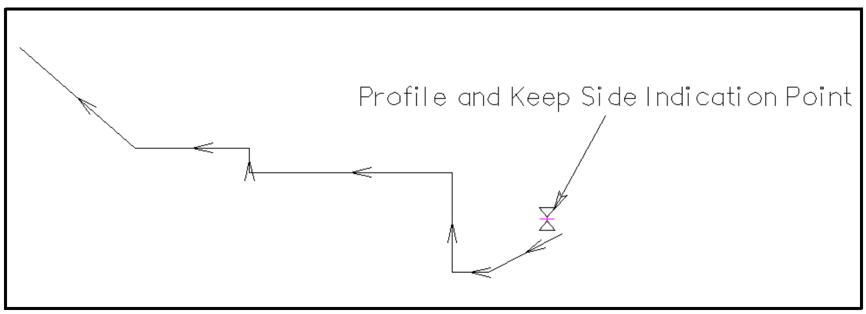
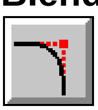



***Figure 4-80***
*This is a result of trimming the profile.*

## Blending Lines and Arcs



**Model File: PTBLEND.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 **`PTBLEND.PM4`**.

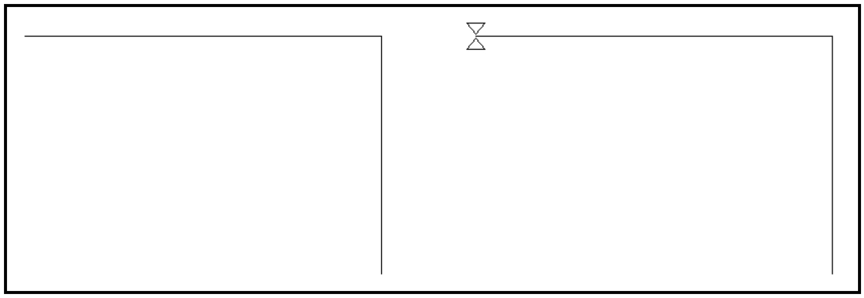


***Figure 4-81***
*Open PTBLEND.*

2. Select **Edit—Geo Edit**.

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

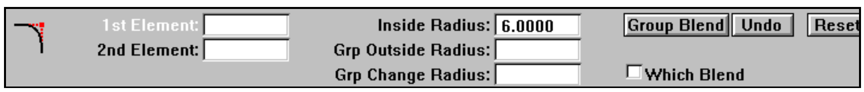



***Figure 4-82***
*Set the values on the Blend control panel.*

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.

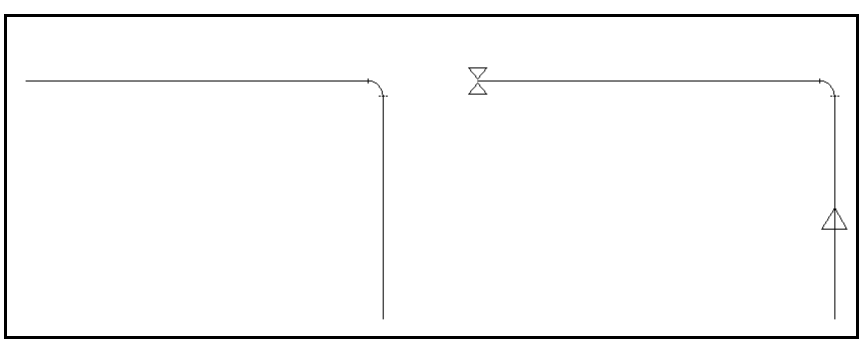
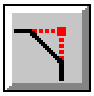


***Figure 4-83***
*This is a result of blending the curves.*

## Creating a Chamfer



**Model File: PTCHAMF.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 **`PTCHAMF.PM4`**. You will chamfer the two lines.

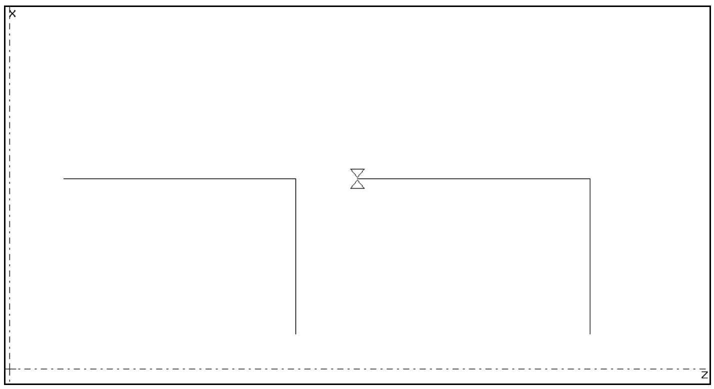


***Figure 4-84***
*Open PTCHAMF.*

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

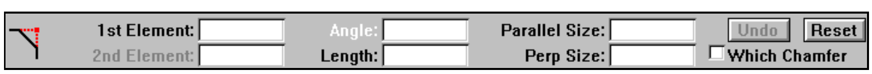



***Figure 4-85***
*Set the values on the Chamfer control panel.*

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.

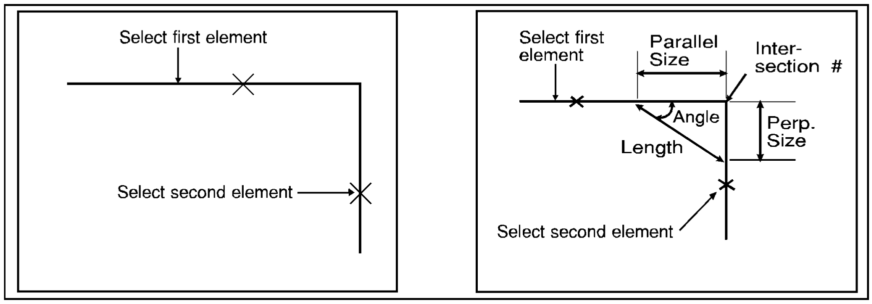
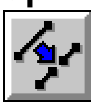



***Figure 4-86***
*Specify the solution that you want to use.*

## Splitting a Curve



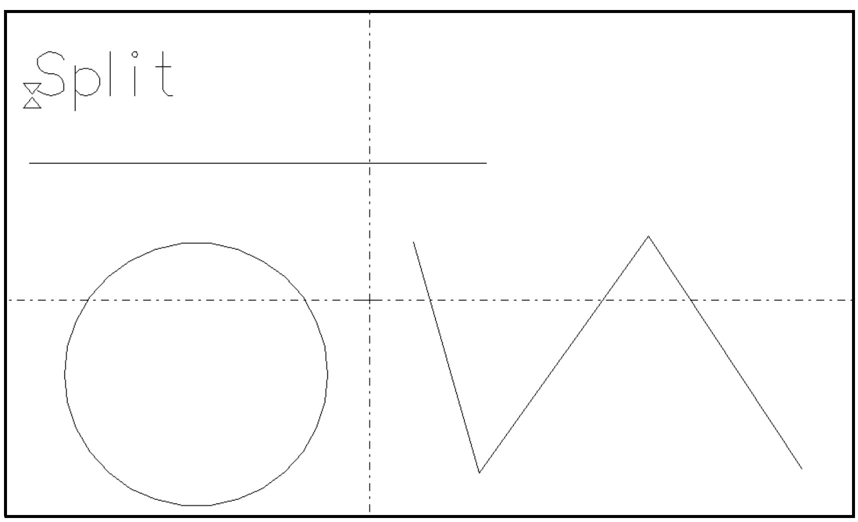
**Model File: PTSPLIT.PM4**

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 **`PTSPLIT.PM4`**.

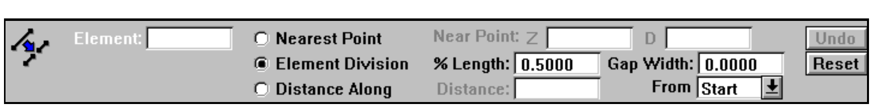
***Figure 4-87***
*Open PTSPLIT.*




2. Select **Edit—Geo Edit**.

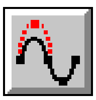
3. Select **Split** from the toolbox. The **Split** control panel is displayed.

***Figure 4-88***
*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: PTMODIFY.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 **`PTMODIFY.PM4`**.

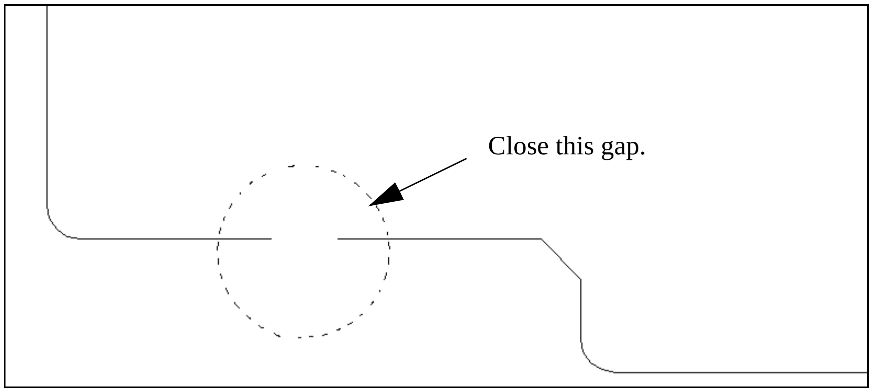



***Figure 4-89***
*Open PTMODIFY.*

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


***Figure 4-90***
*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 Line dialog box, is displayed.

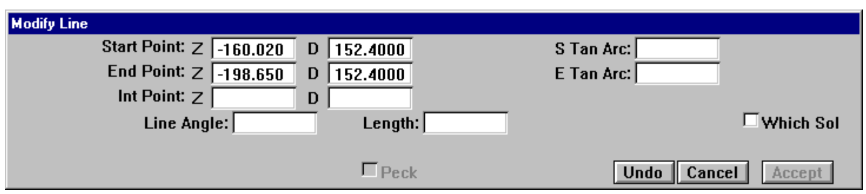
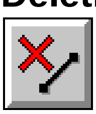



***Figure 4-91***
*The Modify Line dialog box is displayed.*

6. Make the necessary changes, and select the **Go** button.
7. Select the **Accept** button.

## Deleting Elements

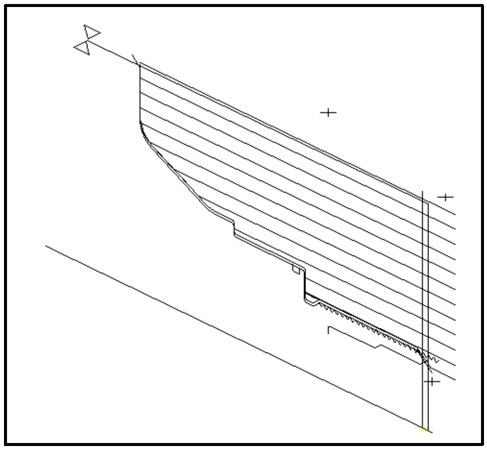


**Model File: PTTURN.PM4**

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 **`PTTURN.PM4`**

***Figure 4-92***
*Open PTTURN.*



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

***Figure 4-93***
*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:

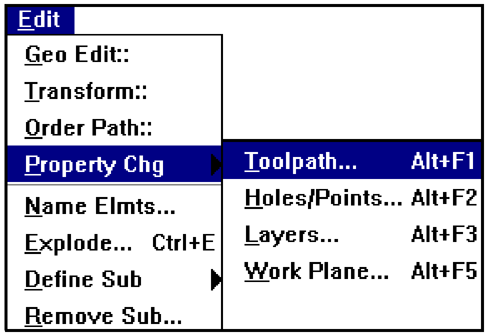
- The clearance heights
- The step associated with the geometry
- The offset direction
- Whether a piece of geometry is drawn on a layer or with a step

## Using the Property Chg Submenu

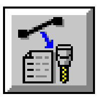
Use the Property Chg submenu to update modeling properties associated with existing elements.

**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 4-94* Learn the components of the Property Change submenu.




## Changing a Toolpath Property

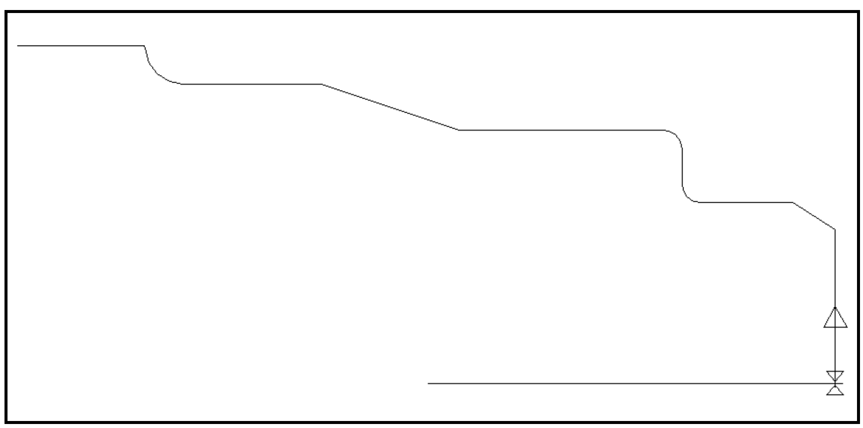


**Model File: PTPROPCH.PM4**

Use Toolpath to change one or more toolpath properties for an active group of elements. Properties that you can change include step, tool offset, and clear. 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:

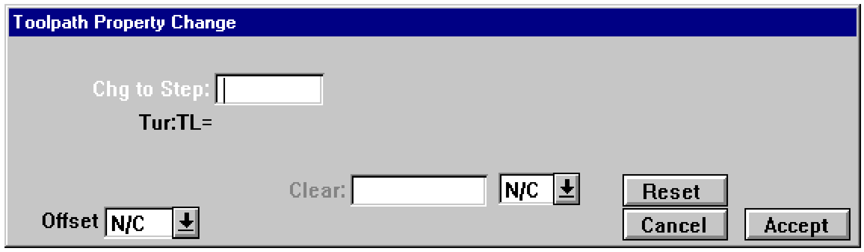
1. Open the model file **PTPROPCH.PM4**.

*Figure 4-95* Open PTPROPCH.



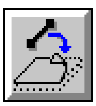
2. Group the geometry to change.
3. Select **Edit—Property Chg—Toolpath**. The **Toolpath Property Change** dialog box is displayed.

*Figure 4-96*
*Set the values on the Toolpath Property Change dialog box.*




4. Enter the changes on the dialog box. To change the , **Clear** value, you must change the **Offset** selector switch from **N/C** to another value.
5. Select the **Accept** button.

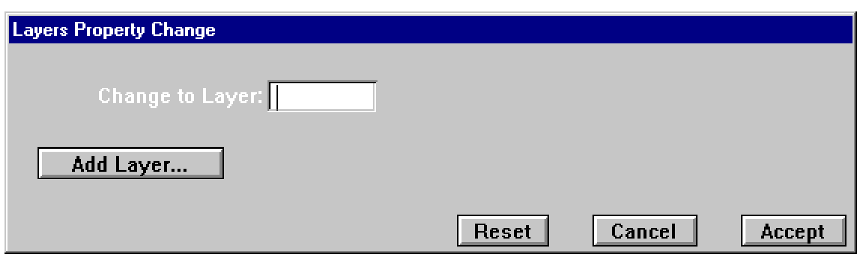
## Changing a Layer Property



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:

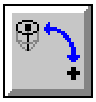
1. Group the geometry that you want to change.
2. Select **Edit—Property Chg—Layers**. The **Layers Property Change** dialog box is displayed.

*Figure 4-97*
*Set the values on the Layers Property Change dialog box.*




3. Enter the changes on the dialog box.
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, 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.

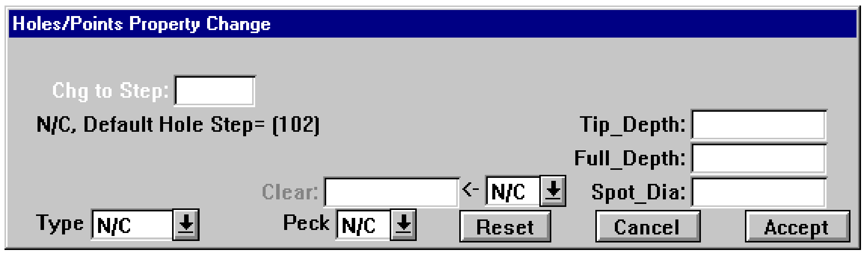
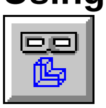



***Figure 4-98*** *Set the values on the Holes/ Points Property Change dialog box.*

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: PTTURN.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 **`PTTURN.PM4`**.

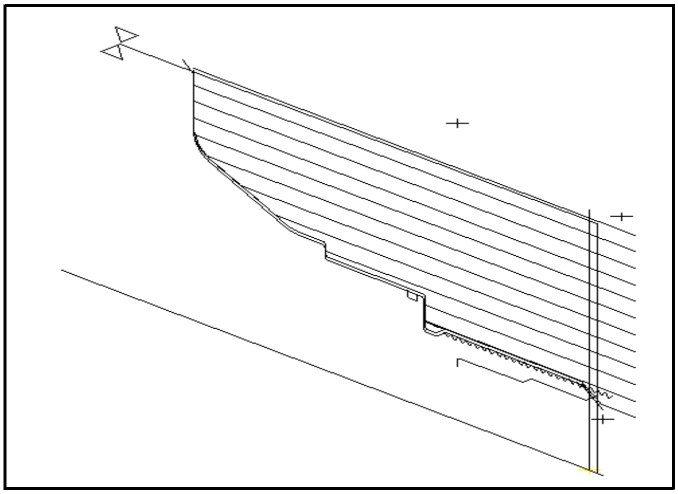


***Figure 4-99***
*Open PTTURN.*

2. Select **Utility—Show/Mask**. The **Show/Mask** dialog box is displayed.

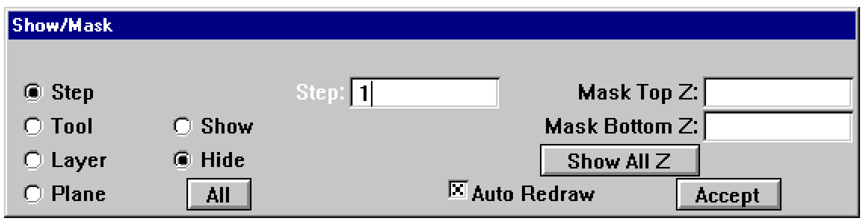


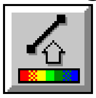

***Figure 4-100***
*Set the values on the Show/ Mask dialog box.*

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: PTTURN.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 **PTTURN.PM4**.

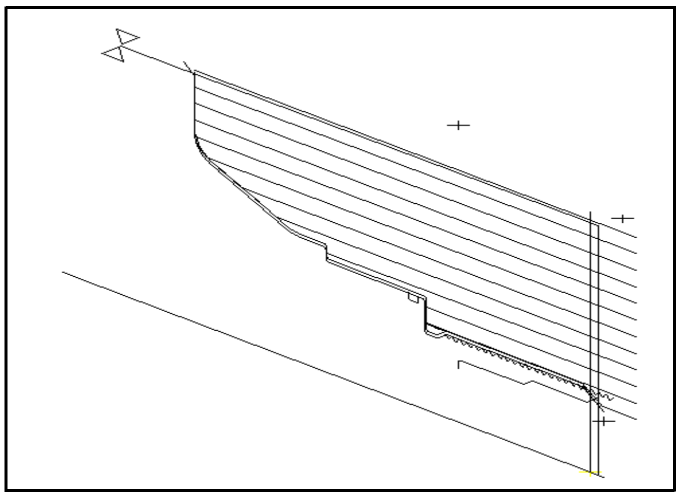


***Figure 4-101***
*Open PTTURN.*

2. Select **Utility—Color Change**. The **Color Change** dialog box is displayed.

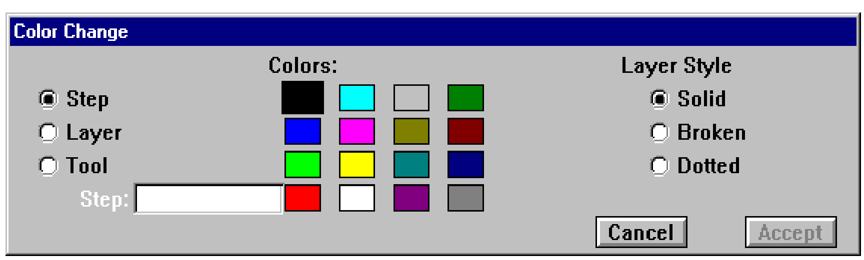



***Figure 4-102***
*Set the values on the Color Change dialog box.*

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 Order Path

## Objectives

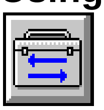
This lesson shows you how to perform these tasks:

- 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.

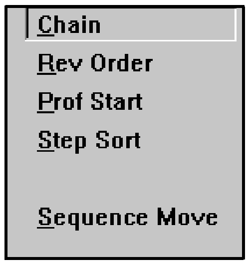
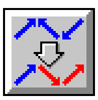



***Figure 4-103***
*Open the Order Path toolbox.*

## Chaining Geometry



**Model File: PTORDER1.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 **`PTORDER1.PM4`**.

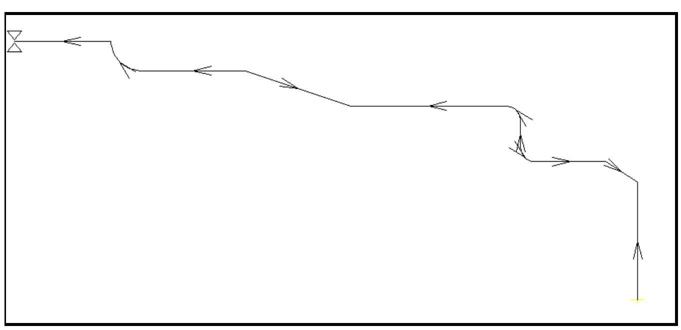


***Figure 4-104***
*Open PTORDER1.*

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.

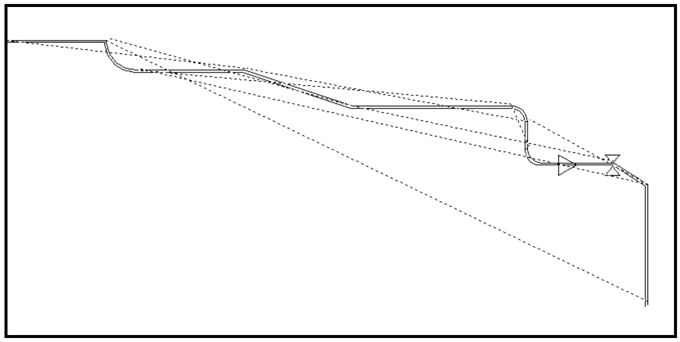


***Figure 4-105***
*Notice the initial toolpath is out of order.*

5. Select **Edit—Order Path**.
6. Select **Chain** from the toolbox. The **Chain** control panel is displayed.


***Figure 4-106***
*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.

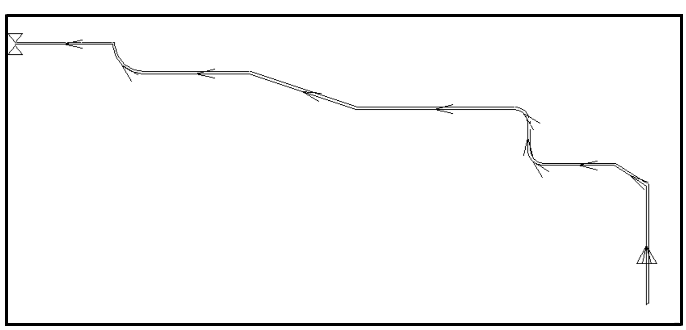
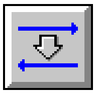


***Figure 4-107***
*Notice that the final toolpath is in order and does not gouge the part.*

## Reversing the Order of Geometry



**Model File: PTORDER1.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 **`PTORDER1.PM4`**.

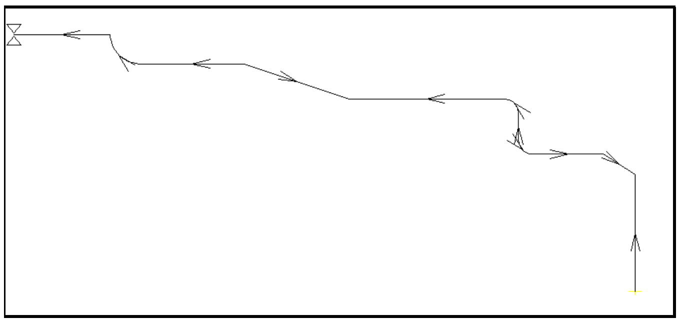


***Figure 4-108***
*Open PTORDER1.*

2. Select **Edit—Order Path**.
3. Select **Rev Order** from the toolbox. The **Reverse Order** control panel is displayed.

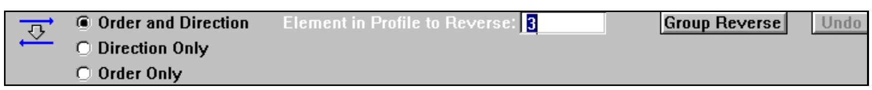
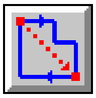



***Figure 4-109***
*Specify values for the Reverse Order control panel.*

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: PTORDER1.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 **`PTORDER1.PM4`**.

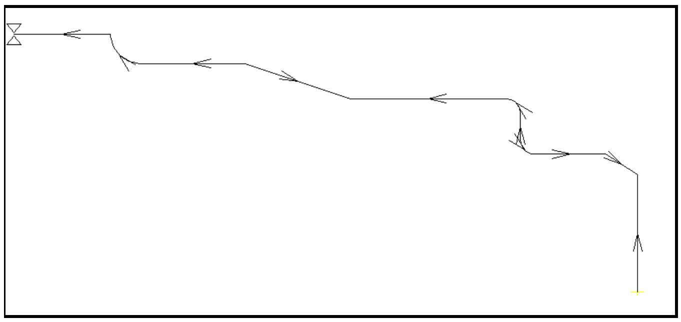


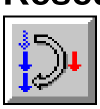
***Figure 4-110*** *Open PTORDER1.*

2. Select **Edit—Order Path**.
3. Select **Prof Start** from the toolbox. The **Prof Start** control panel is displayed.


***Figure 4-111*** *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: PTORDER1.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 **`PTORDER1.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.

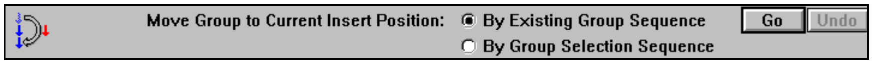



***Figure 4-112*** *Specify values for the Sequence Move control panel.*

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.

## 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.

# *Transforming Geometry*

## Objectives

This lesson shows you how to perform these tasks:

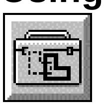
- Move geometry.
- Rotate geometry.
- Mirror geometry.
- Scale geometry.
- Show and mask geometry.

## 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.

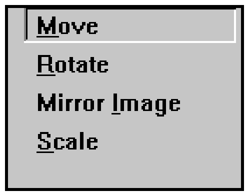
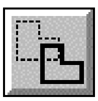



***Figure 4-113*** *Use the Transform toolbox.*

## Moving Geometry



**Model File: PTMOVE.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 **`PTMOVE.PM4`**.

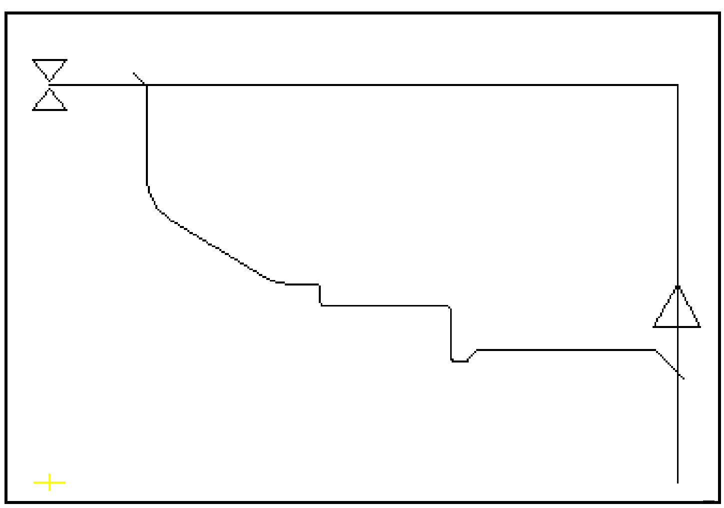


***Figure 4-114*** *Open PTMOVE.*

2. Group the elements to move.
3. Select **Edit—Transform**.
4. Select **Move** from the toolbox. The **Move** control panel is displayed.

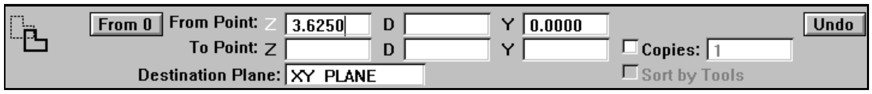
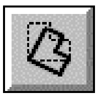



*Figure 4-115*
*Set the values on the Move control panel.*

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: PTROTATE.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 **`PTROTATE.PM4`**.

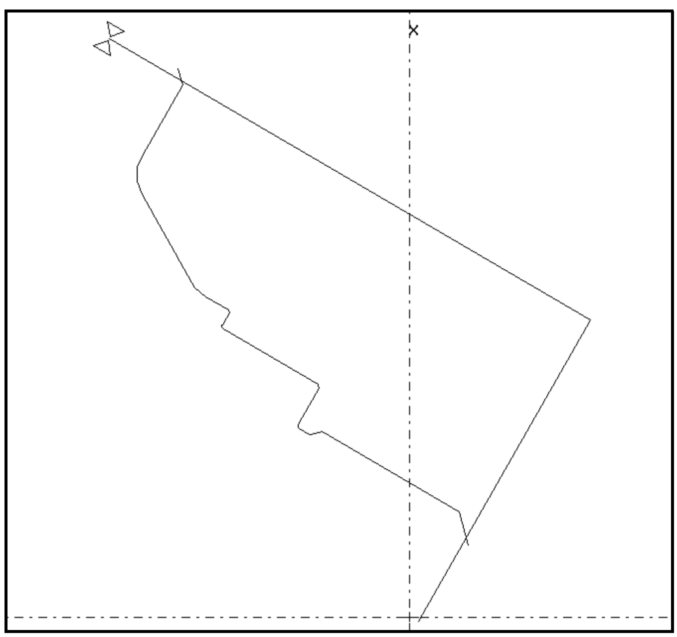


*Figure 4-116*
*Open PTROTATE.*

2. Group the elements to rotate.
3. Select **Edit—Transform**.
4. Select **Rotate** from the toolbox. The **Rotate** control panel is displayed.

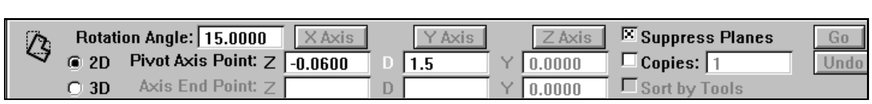
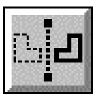



*Figure 4-117*
*Set the values on the Rotate control panel.*

5. Set the **Pivot Axis Point** input fields to the location of the pivot point(-.06[-1.524], 1.5[38.1].)
6. 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.
7. Set the **Rotation Angle** input field.
8. Select the **Go** button.

## Mirroring Geometry



**Model File: PTMIRROR.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 **`PTMIRROR.PM4`**.

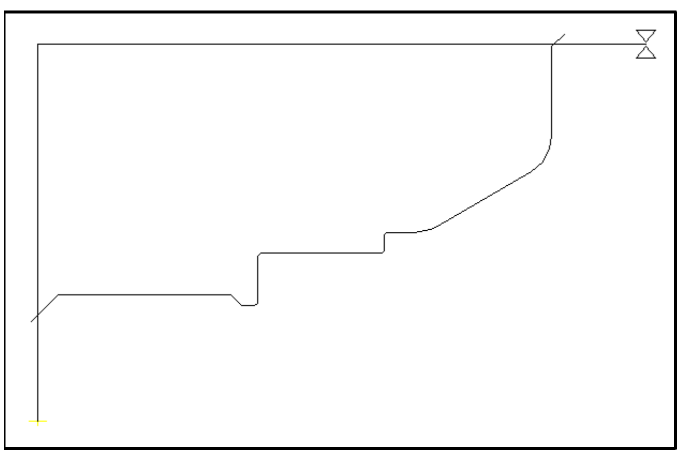


*Figure 4-118* *Open PTMIRROR.*

2. Group the elements to mirror.
3. Select **Edit—Transform**.
4. Select **Mirror Image** from the toolbox. The **Mirror Image** control panel is displayed.

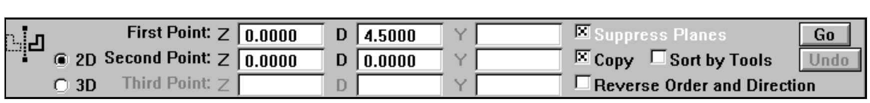
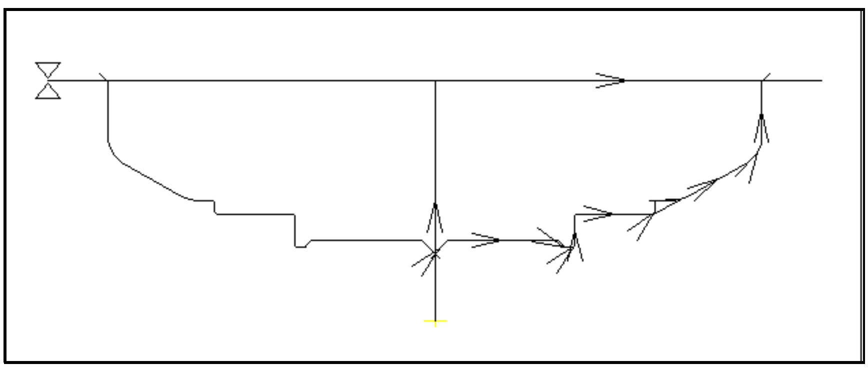


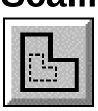

*Figure 4-119* *Set the values on the Mirror Image control panel.*

5. Set the **First Point** input fields to the starting point of the line along which to mirror the image.
6. Set the **Second Point** input fields to the ending point of the line along which to mirror the image.
7. Turn on the **Copy** on/off switch on to create a copy of the original group of elements on the mirror side.
8. 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.
9. 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.
10. Select the **Go** button.

***Figure 4-120***
*This is the result of mirroring the geometry.*



## Scaling Geometry



**Model File: PTSCALE.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 **PTSCALE.PM4**.

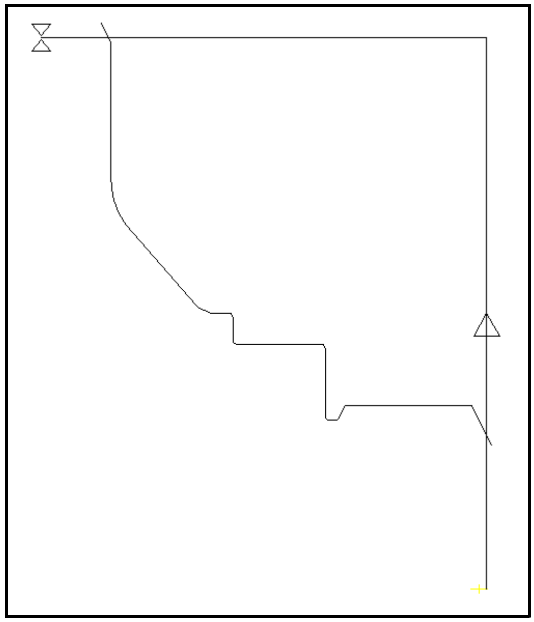


*Figure 4-121* Open PTSCALE.

2. Group the elements to scale.
3. Select **Edit—Transform**.
4. Select **Scale** from the toolbox. The **Scale** control panel is displayed.

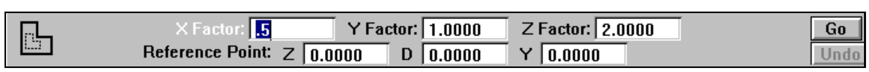



*Figure 4-122* Set the values on the Scale control panel.

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.

## 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:

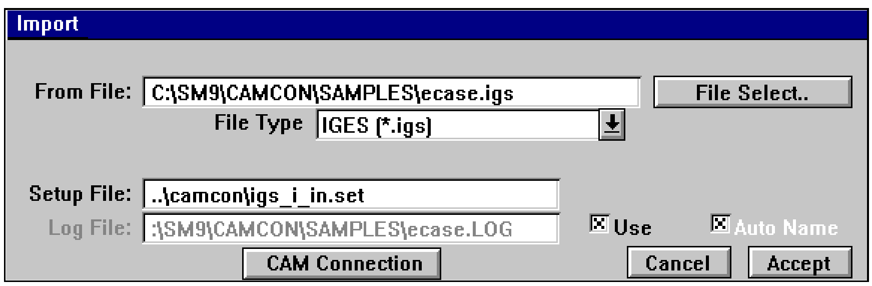
- Import into a model file that has the same units.
- Import into a model file that has different units, and adopt the units of the existing file.
- Import into a model file that has different units, and adopt the units of the incoming file.

## 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 4-123*
*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 4-70.
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 4-70.
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.

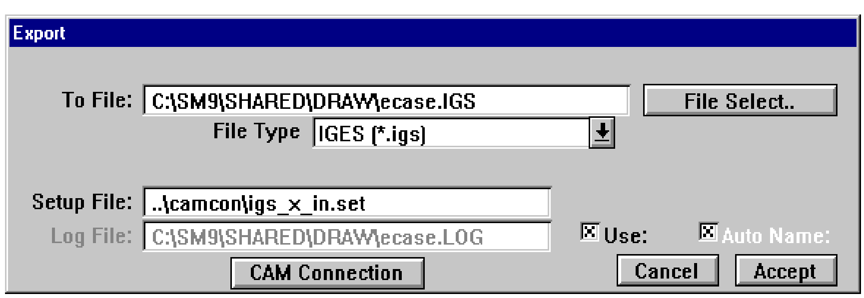



***Figure 4-124***
*Open the Export dialog box.*

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

# 5

# Generating and Verifying Toolpath

## Overview

These lessons show you how to perform Rough machining to generate toolpath. You will then use Show Path to verify the toolpath.

## Lessons for This Unit

- Generating Roughing Toolpath
- Verifying Roughing Toolpath

# Generating Roughing Toolpath

## Objectives

This lesson shows you how to perform these tasks:

- Use linear roughing to produce toolpath.
- Use contour roughing to produce toolpath.
- Use groove roughing to produce toolpath.
- Use profile roughing to produce toolpath.

## Overview

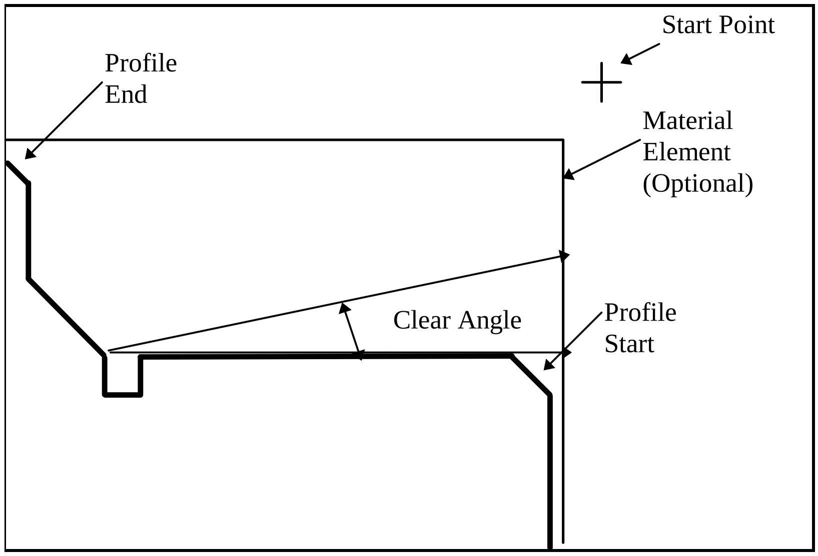
Use the tools in the Rough toolbox to create roughing toolpath for existing profiles. You can also define a material boundary curve to help minimize unnecessary cuts. Both linear roughing and contour roughing produce a toolpath with a turning or facing tool. Groove roughing requires a grooving tool to produce a toolpath.

## Using the Rough Toolbox

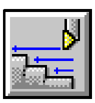
Use the Rough toolbox to create roughing toolpath to remove a large volume of material using multiple cutting moves. The material that you remove is defined by the XY position of the finish profile, combined with the Level and Prof Top (Z) position.

These are the important portions of your part to be aware of when you perform roughing.

*Figure 5-1*
*These are important options to specify for roughing.*




## Performing Linear Roughing

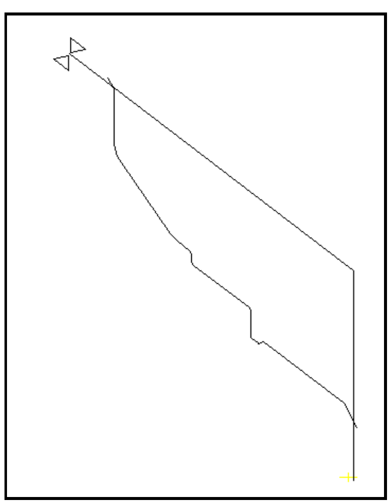


**Model File: PTRUFF1.PM4**

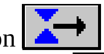
Linear roughing enhances and accelerates the creation of roughing passes. The tool passes are straight line cuts created with a turning or facing tool parallel to the Z or X axis. Cuts are automatically generated with an offset. Notice the important options to specify as you perform linear roughing.

1. Open the model file **PTRUFF1.PM4**.

*Figure 5-2*
*Open PTTRUFF1.*



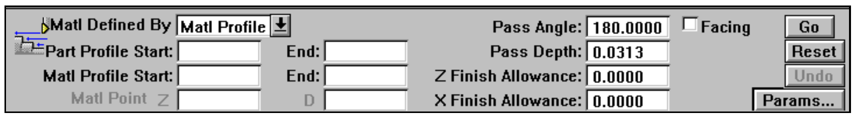
2. Set the insert location:

- Set the **Before** icon  or **After** icon .
- Select the **Element** icon  or **Step** icon , and select the element or step to be before or after.
- Set the **With Step** icon , and select a lathe step from the list view.

3. Select **Process—Turn Rough**.

4. Select **Linear** from the **Turn Rough** toolbox. The **Linear** control panel is displayed.

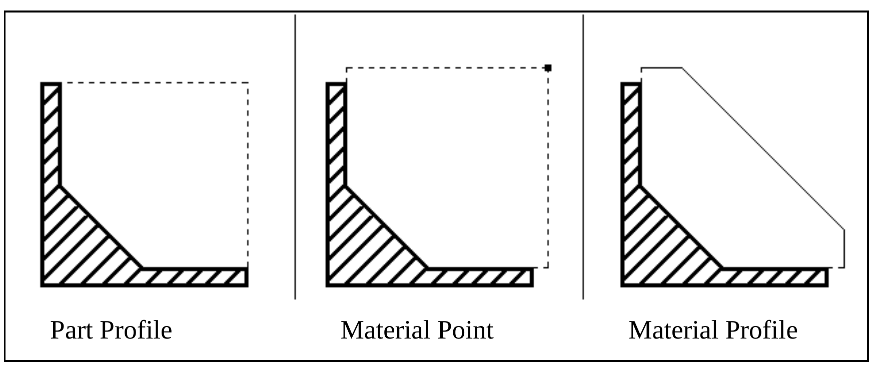
*Figure 5-3 Set the values on the Linear control panel.*




5. Define the material area to be removed using the **Matl Defined By** option switch. Depending on the option you select, you must provide additional information to complete the definition as described below:

   - **Matl Profile**—Select a profile that represents the material boundary. The results of your selection appear in the **Matl Profile Start** and **End** fields.
   - **Matl Point**—Supply a point to represent the outermost corner of the material boundary by making a selection in the graphics view or typing a point value in the **Matl Point Z** and **D** fields.
   - **Part Profile**—Use the extremes of the bounding box of the supplied part profile as the material boundary.

*Figure 5-4 Define the material boundary.*




6. Set the **Part Profile Start** and **End** input fields. These elements must be continuous in the element list.

7. Set the remaining fields on the control panel.

   - Set the **Pass Angle** input field. An angle of zero is parallel to the Z axis. If a turning type is set, the pass angle should be near 0. For a facing type, use a pass angle value near 90.
   - Set the **Pass Depth** input field to the depth to cut.

- Set the **Z Finish Allowance** input field to the amount of stock to leave on the part profile. A negative value is permissible.
- Set the **X Finish Allowance** input field to the amount of stock to leave on the part profile. A negative value is permissible.
- Turn on the **Facing** on/off switch to make facing linear cuts; otherwise, turning linear cuts are used.

8. Select the **Params...** button. The **Linear Parameters** dialog box is displayed.

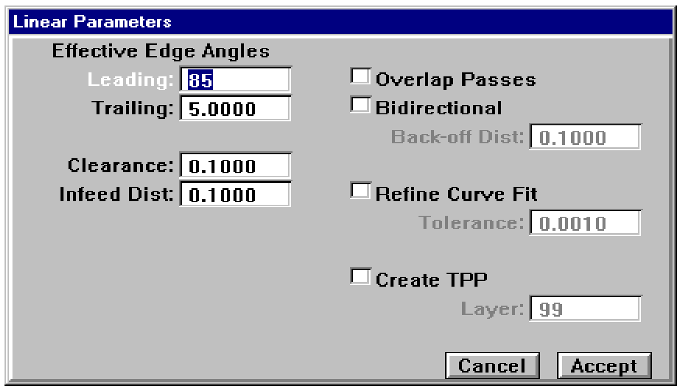
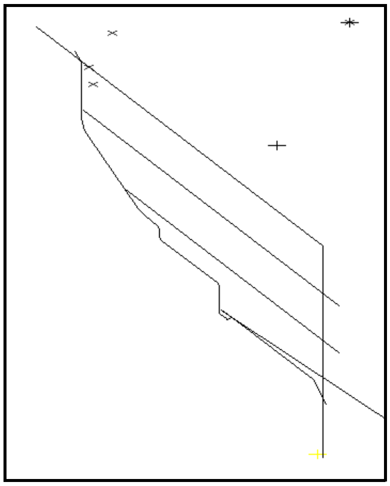



***Figure 5-5***
*Set the values on the Linear Parameters dialog box.*

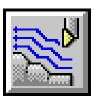
9. Set these values on the **Linear Parameters** dialog box:
   - Set the **Effective Edge Angles** input fields (**Leading** and **Trailing**). The default values are the same as those set on the **Operation** tab of the **Edit Process Step** dialog box.
   - Set the **Clearance** input field, which is the distance for the tool to clear the part during approach and retract moves.
   - Set the **Infeed Dist** input field, which extends toolpath elements that intersect the material boundary.
   - Turn on the **Overlap Passes** 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 **Bidirectional** on/off switch if you want to use the tool to cut in two directions to maximize the time the tool is in the cut and to minimize the time the tool is in the air. Otherwise, turn it off.
   - 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 TPP** on/off switch if you want to approximate a material boundary from a theoretical part profile. You must also specify a layer if this is on. Otherwise, turn it off.
10. Select the **Accept** button to close the Linear Parameters dialog box.

11. Select the **Go** button on the control panel.

*Figure 5-6* *Notice the results of linear roughing.*



## Performing Contour Roughing

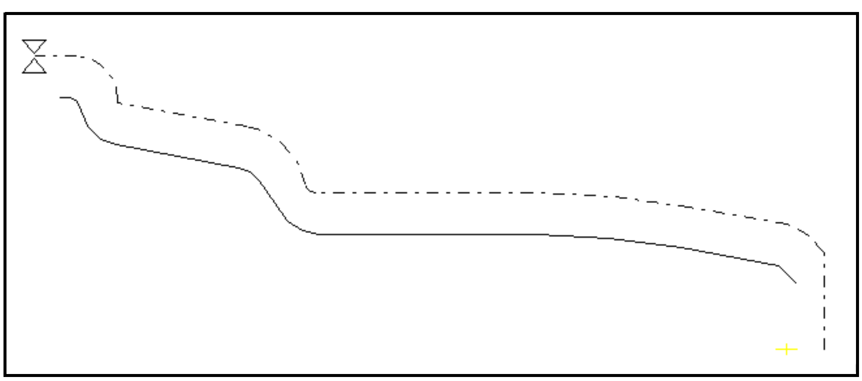


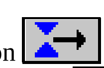
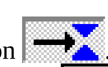
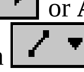
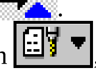
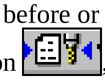
**Model File: PTCONRF.PM4**

Contour roughing creates roughing passes that are parallel to the contour of the existing part profile. Multiple passes can be used to rough an area between the material boundary and part profile.

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

*Figure 5-7* *Open PTCONRF.*



2. Set the insert location:
   - Set the **Before** icon  or **After** icon .
   - Select the **Element** icon  or **Step** icon , and select the element or step to be before or after.
   - Set the **With Step** icon , and select a lathe step from the list view.
3. Select **Process—Turn Rough**.

4. Select **Contour** from the **Turn Rough** toolbox. The **Contour** control panel is displayed.

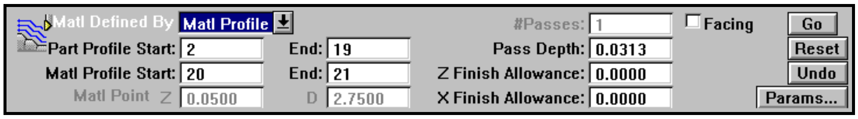



***Figure 5-8***
*Set the values on the Contour control panel.*

5. Define the material area to be removed using the **Matl Defined By** option switch. Depending on the option you select, you must provide additional information to complete the definition as described below:

   - **Matl Profile—**Select a profile that represents the material boundary. The results of your selection appear in the **Matl Profile Start** and **End** fields.
   - **Matl Point—**Supply a point to represent the outermost corner of the material boundary by making a selection in the graphics view or typing a point value in the **Matl Point Z** and **D** fields.
   - **Part Profile—**Uses the extremes of the bounding box of the supplied part profile as the material boundary.
   - **#Passes**—SmartCAM calculates the material to remove using the values in the **#Passes** and **Pass Depth** fields.

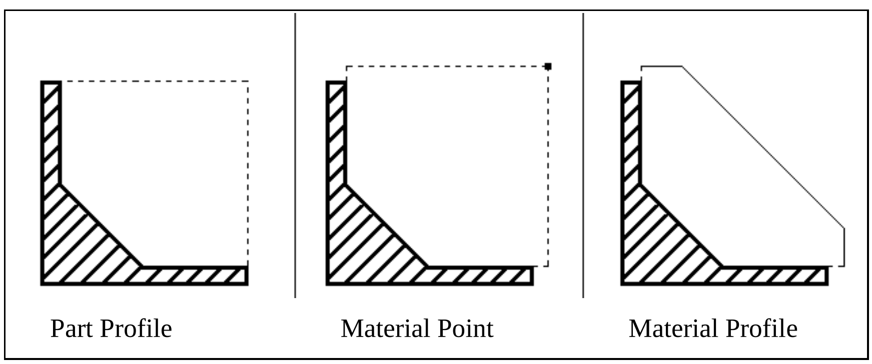


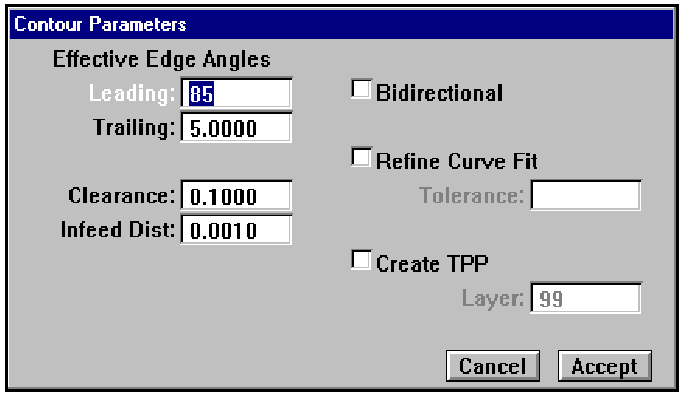

***Figure 5-9***
*Define the material boundary.*

6. Set the remaining fields on the control panel.

   - Set the **Pass Depth** input field to the depth to cut.
   - Set the **Z Finish Allowance** input field to the amount of stock to leave on the part profile. A negative value is permissible.
   - Set the **X Finish Allowance** input field to the amount of stock to leave on the part profile. A negative value is permissible.
   - Turn on the **Facing** on/off switch to make facing linear cuts; otherwise, turning linear cuts are used.

7. Select the **Params...** button. The **Contour Parameters** dialog box is displayed.

***Figure 5-10***
*Set the values on the Contour Parameters dialog box.*



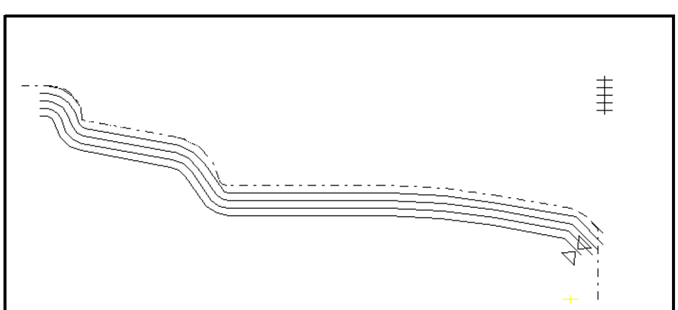

8. Set these values on the **Contour Parameters** dialog box:

   - Set the **Effective Edge Angles** input fields (**Leading** and **Trailing**). The default values are the same as those set on the **Operation** tab of the **Edit Process Step** dialog box.
   - Set the **Clearance** input field, which is the distance for the tool to clear the part during approach and retract moves.
   - Set the **Infeed Dist** input field, which extends toolpath elements that intersect the material boundary.
   - Turn on the **Bidirectional** on/off switch if you want to use the tool to cut in two directions to maximize the time the tool is in the cut and to minimize the time the tool is in the air. Otherwise, turn it off.
   - 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 TPP** on/off switch if you want to approximate a material boundary from a theoretical part profile. You must also specify a layer if this is on. Otherwise, turn it off.

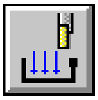
9. Select the **Accept** button to close the **Contour Parameters** dialog box.

10. Select the **Go** button on the **Contour** control panel.

***Figure 5-11***
*Notice the results of contour roughing.*



## Performing Groove Roughing



**Model File: PTRUFF2.PM4**

Groove roughing enhances and accelerates the creation of roughing passes that are perpendicular to the Z axis when you use an outside diameter grooving tool. The cuts may be parallel to the Z axis if a face grooving tool is used. The active step must contain a grooving tool.

Cleanup passes for Groove are made in a down motion or across a flat area. The cleanup cuts cannot be a continuous path.

1. Open the model file **`PTRUFF2.PM4`**.

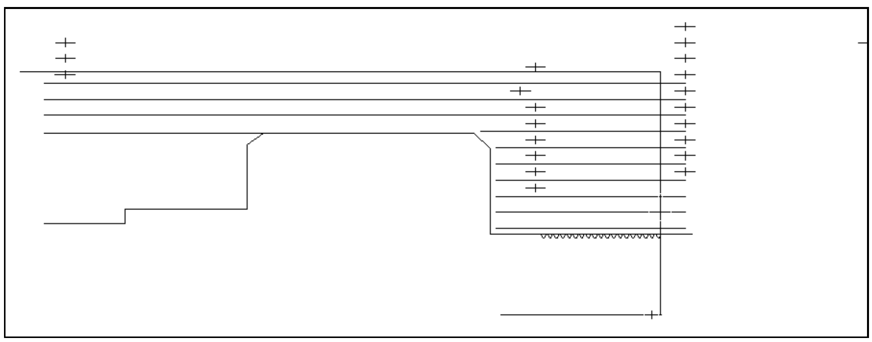
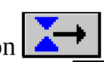
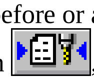


*Figure 5-12* *Open PTTRUFF2.*

2. Set the insert location:
   - Set the **Before** icon  or **After** icon .
   - Select the **Element** icon  or **Step** icon , and select the element or step to be before or after.
   - Set the **With Step** icon , and select a groove step from the list view.
3. Select **Process—Turn Rough**.
4. Select **Groove** from the **Turn Rough** toolbox. The **Groove** control panel is displayed.

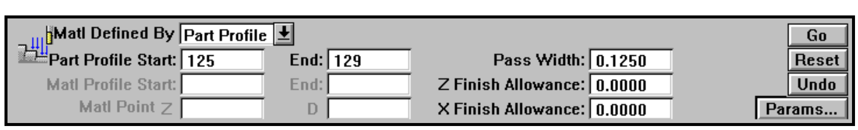



*Figure 5-13* *Set the values on the Groove control panel.*

5. Define the material area to be removed using the **Matl Defined By** option switch. Depending on the option you select, you must provide additional information to complete the definition as described below:
   - **Matl Profile—**Select a profile that represents the material boundary. The results of your selection appear in the **Matl Profile Start** and **End** fields.

- **Matl Point—**Supply a point to represent the outermost corner of the material boundary by making a selection in the graphics view or typing a point value in the **Matl Point Z** and **D** fields.
- **Part Profile—**Uses the extremes of the bounding box of the supplied part profile as the material boundary.

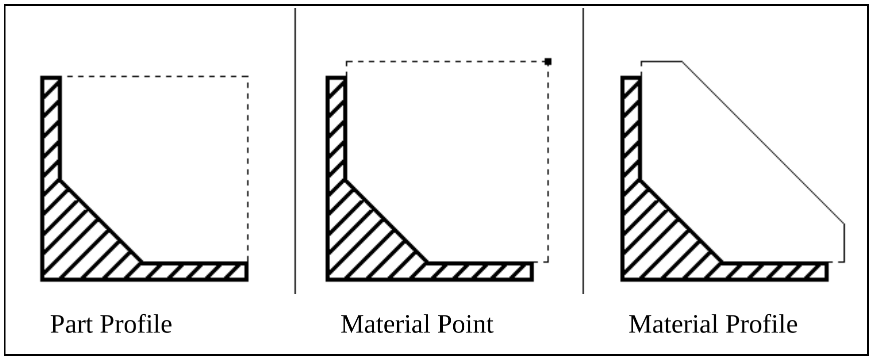



***Figure 5-14*** *Define the material boundary.*

6. Set the remaining fields on the control panel.
   - Set the **Pass Width** input field to the width to cut.
   - Set the **Z  Finish Allowance** input field to the amount of stock to leave on the part profile. A negative value is permissible.
   - Set the **X Finish Allowance** input field to the amount of stock to leave on the part profile. A negative value is permissible.
7. Select the **Params...** button. The **Groove Parameters** dialog box is displayed.

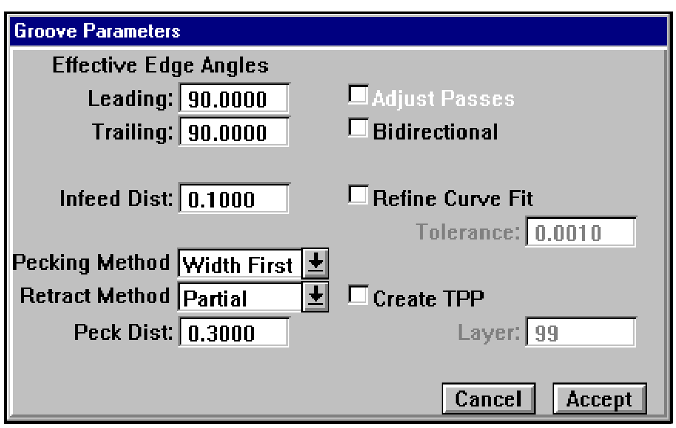


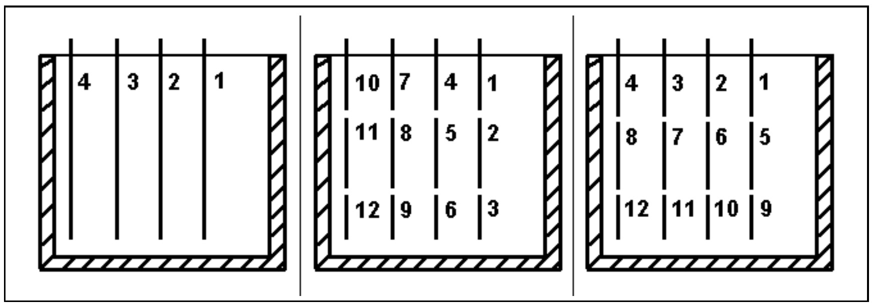

***Figure 5-15*** *Set the values on the Groove Parameters dialog box.*

8. Set these values on the **Groove Parameters** dialog box:
   - Set the **Effective Edge Angles** input fields (**Leading** and **Trailing**). The default values are the same as those set on the **Operation** tab of the **Edit Process Step** dialog box.

- Set the **Infeed Dist** input field, which extends toolpath elements that intersect the material boundary.
- Set the **Pecking Method** to define the order of the plunging movements you intend to make with your toolpath. Plunging options are **None**, **Depth First**, and **Width First**.

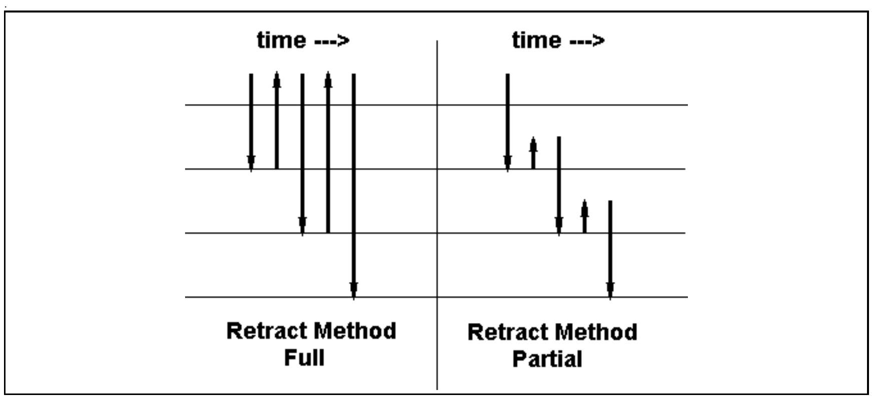
  The **Bidirectional** parameter provides additional control over the order of **Width First** toolpath. In this case, alternate levels of toolpath elements would traverse the recess in opposite directions.

***Figure 5-16*** *Groove toolpath examples of Depth First, Width First and Width First not modified by the bidirectional option.*




- Set the Retract Method to either Full or Partial. This controls the distance the tool will move during retract moves that are made between consecutive plunge moves. Partial retracts the tool by the distance specified by the Infeed Dist parameter. Full retracts the tool to the edge of the material area. This field is not available unless the Pecking Method is set to either Depth First or Width First.

***Figure 5-17*** *Retract Method examples.*




- Turn on the **Adjust Passes** on/off switch to justify the sides of the grooving insert against any vertical walls in the part profile, and evenly distribute passes (without exceeding the stepover distance set in Pass Width) across any region bounded by vertical walls.

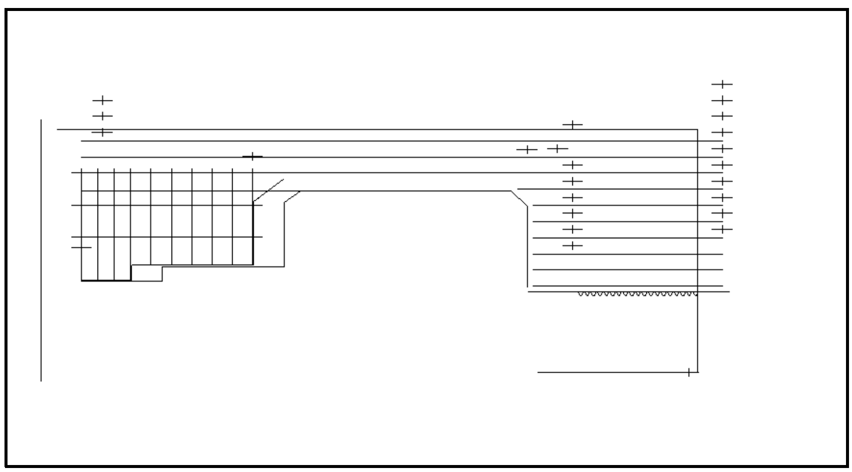
  Set this switch to off for all passes to be the distance apart specified by the Pass Width parameter.

**Note** Although setting the switch to off maximizes stepover distance, it can create a condition in which the last pass represents a “sliver cut” where the width of cut is not optimal.

- Turn on the **Bidirectional** on/off switch if you want to use the tool to cut in two directions to maximize the time the tool is in the cut and to minimize the time the tool is in the air. Otherwise, turn it off. See figure 5-16 for an illustration of how this setting can affect Width First toolpath.
- 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 TPP** on/off switch if you want to approximate a material boundary from a theoretical part profile. You must also specify a Layer if this is on. Otherwise, turn it off.

9. Select the **Accept** button. The **Groove** control panel is displayed.
10. Select the **Go** button on the **Groove** control panel.

*Figure 5-18*
*This is the result of performing groove roughing.*



## Performing Profile Roughing

**Model File: PTPRFRUF.PM4**

Profile roughing enables you to create a semi-finishing or finishing pass for parts with critical final tolerances. You can leave excess stock on the part after the initial roughing, change tools, and bring the final cut to an exact tolerance level with adjustments for cutter compensation.

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

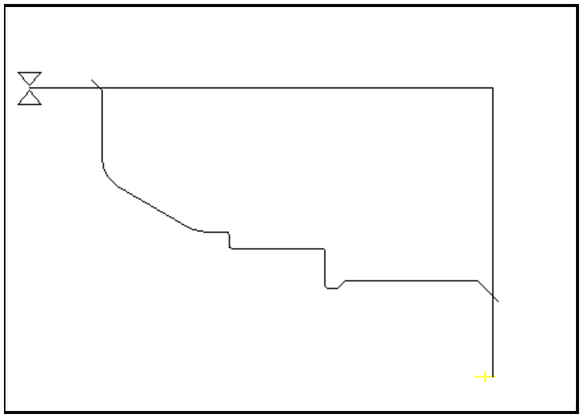
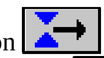
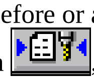


***Figure 5-19***
*Open PTPRFRUF.*

2. Set the insert location:
   - Set the **Before** icon  or **After** icon .
   - Select the **Element** icon  or **Step** icon , and select the element or step to be before or after.
   - Set the **With Step** icon , and select a groove step from the list view.
3. Select **Process—Turn Rough**.
4. Select **Profile** from the **Turn Rough** tool list. The **Profile** control panel is displayed.

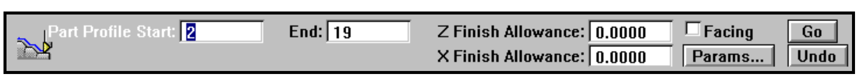



***Figure 5-20***
*Set the parameters on the Profile control panel.*

5. Set the **Part 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.
6. Set the remaining fields on the control panel.
   - Set the **Z Finish Allowance** input field to the amount of stock to leave on the part profile. A negative value is permissible.
   - Set the **X Finish Allowance** input field to the amount of stock to leave on the part profile. A negative value is permissible.
   - Turn on the **Facing** on/off switch to reverse the roles of the effective leading and trailing edges of the tool, controlling offset and toolpath direction. The initial state of this check box is determined by the tool definition.
7. Select the **Params...** button. The **Profile Parameters** dialog box is displayed. Use it for lead in/out, plunge, and retract movements.

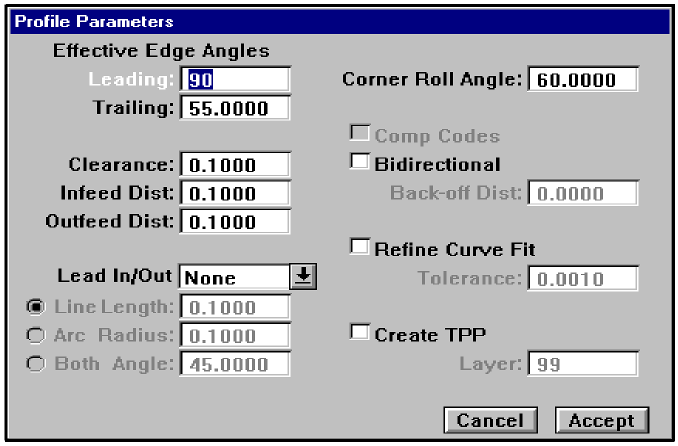



***Figure 5-21***
*Set the values on the Profile Parameters dialog box.*

8. Set these values on the **Profile Parameters** dialog box:

- Set the **Effective Edge Angles** input fields. The default values are the same as those set on the **Operation** tab of the **Edit Process Step** dialog box.
- Set the **Clearance** input field, which is the distance for the tool to clear the part during approach and retract moves.
- Set the **Infeed Dist** input field, which extends toolpath elements that intersect the material boundary.
- 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 switch 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**, or **Both**.

  For the **Line** option switch, set the **Length** input field to the length of any lead in/out line elements. The length must be greater than zero.

  When Line is selected, the angle of the approach and recession of the tool as it engages and disengages the profile toolpath are relative to the start and end tangent of the profiling toolpath.

  For the **Arc** option switch, set the **Radius** input field to the radius of any lead in/out arc elements. The radius must be greater than zero.

When Arc is selected, the angle of the approach and recession of the tool as it engages and disengages the profile toolpath determine the arc span.

For the **Both** option switch, set the **Angle** input field to the approach and recession directions of the tool as it engages and disengages the profiling toolpath.

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.

- Enter a value for **Corner Roll Angle**. Any interior angle that is less than this value is blended with an arc.
- Turn on the **Comp Codes** on/off switch to add cutter compensation commands to the toolpath.
- Turn on the **Bidirectional** on/off switch to use the tool to cut in two directions to maximize the time the tool is in the cut and to minimize the time the tool is in the air. You can also enter a value for Back-off Dist to relieve cutting forces on the tool when this is on. Otherwise, turn it off, and the Back-off Dist parameter need not be set.
- 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 TPP** on/off switch to approximate a material boundary from a theoretical part profile. Otherwise, turn it off.

9. Select the **Accept** button on the dialog box.
10. Select the **Go** button.

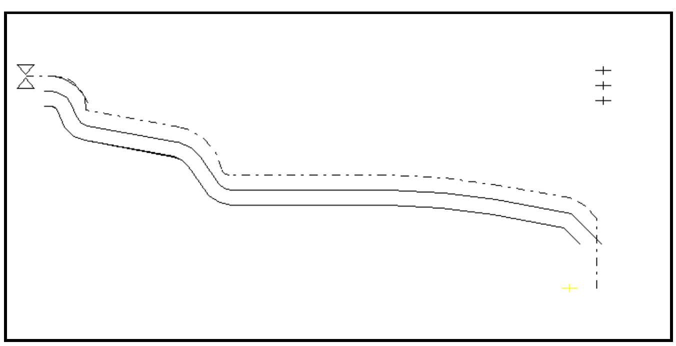


***Figure 5-22***
*This is the result of profile roughing.*

## Points to Remember

- ❑ Use the Rough toolbox to create roughing toolpath to remove a large volume of material using multiple cutting moves.
- ❑ Cuts are automatically generated with an offset when you want the tool to overlap the outside of a profile boundary when you use linear roughing.
- ❑ Contour roughing creates roughing passes that are parallel to the contour of the existing part profile.
- ❑ The active step must contain a grooving tool to use groove roughing.

# 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 feature to further 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: PTTURN.PM4**

Perform these tasks to use Show Cut:

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

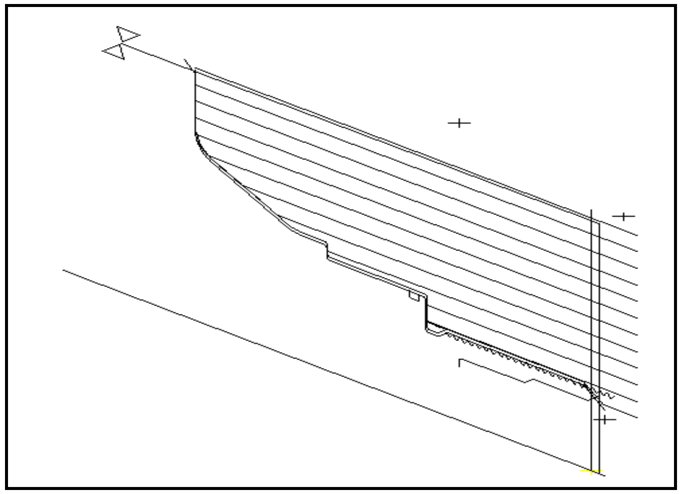


***Figure 5-23***
*Open PTTURN.*

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

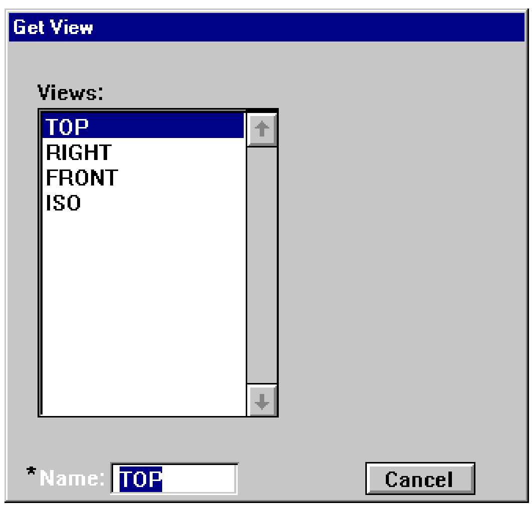



***Figure 5-24***
*Specify the view that you want.*

3. Set the view to **ISO**, or dynamically rotate the part to any desired view.
4. 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.
5. Select **View—Show Cut**. The **Show Cut** dialog box is displayed.

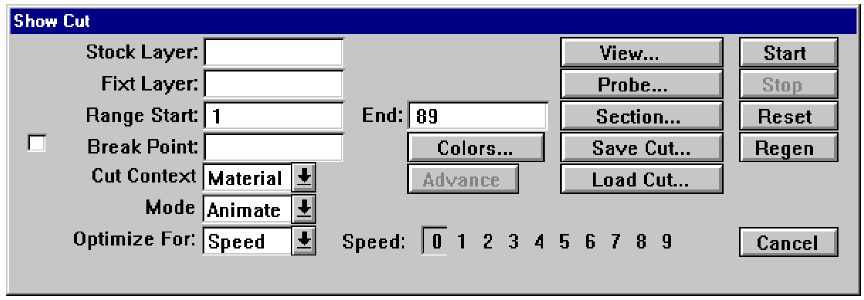



***Figure 5-25***
*Set the values in the Show Cut dialog box.*

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 **Range End** input field to the last element in the range of elements to verify.

9. Set the **Optimize For** selector switch to either **Speed** or **Quality**.

10. Set the **Speed** input field by pressing the **9** button. The speed can be changed during simulation by setting a new number on the keyboard. The mouse will not change the speed.

11. Set the **Cut Context** input field to **Step**. Each cut is then displayed in the color of the step.

12. Select the **Start** button to start the simulation. Select the **Stop** button to stop the step-by-step display of the simulation and return to the Show Cut options. You can also press the ESC key to stop the simulation.

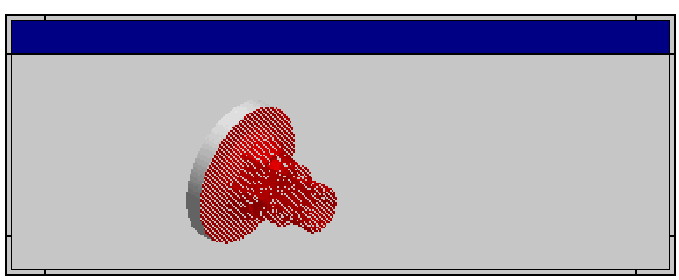


***Figure 5-26***
*This is the result of Show Cut.*

## 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. How do you tell SmartCAM to generate cuts with an offset when you use linear roughing?**

a) You cannot use an offset with linear roughing.
b) You must set an offset in the Offset dialog box after you set the linear roughing parameters.
c) You set the offset in the linear roughing feature.
d) SmartCAM automatically generates cuts with an offset for linear roughing.

______________ **2. Use the Rough toolbox to create roughing toolpath to remove a large volume of material using multiple cutting moves.**

a) true
b) false

______________ **3. How do the roughing passes that contour roughing creates relate to the contour of the existing part profile?**

a) The passes are perpendicular to the contour of the part.
b) The passes are parallel to the contour of the part.
c) The passes are at a 45-degree angle to the contour of the part.
d) The passes are not related to the contour of the part.

______________ **4. What must the active step contain to use groove roughing?**

a) roughing tool
b) active group
c) grooving tool
d) no group

______________ **5. What feature do you use to verify roughing toolpath?**

a) Verify Toolpath
b) Roughing
c) Copy
d) Show Cut

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

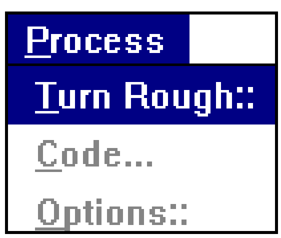
## Objectives

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.

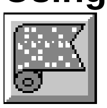
### Machine Definition Files

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 for each turret.

### 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.

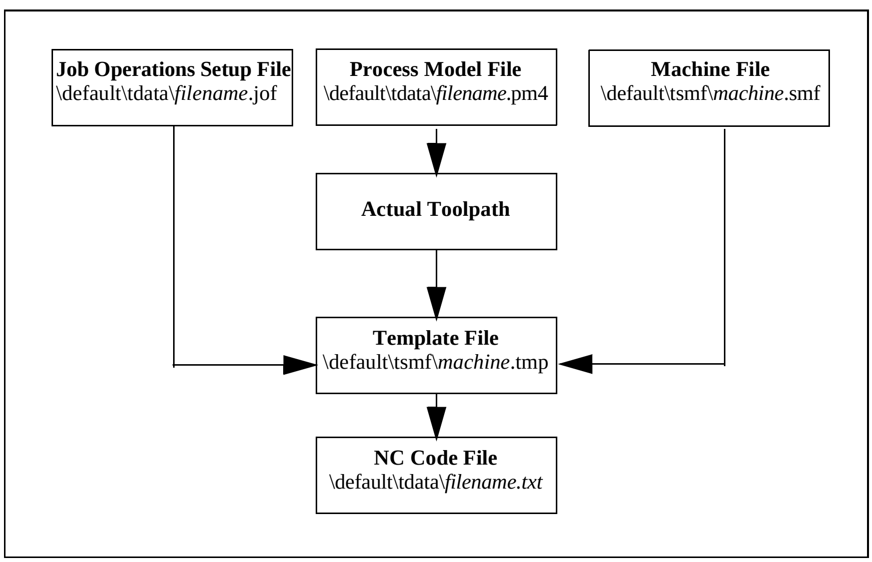
## Using Code



**Model File: PTTURN.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 **`PTTURN.PM4`**.

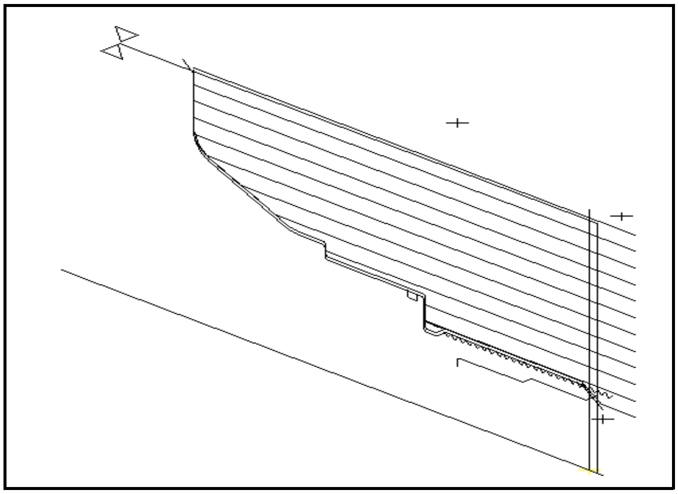


***Figure 6-3***
*Open PTTURN.*

2. Select **Process—Code**. The **Code** dialog box is displayed.

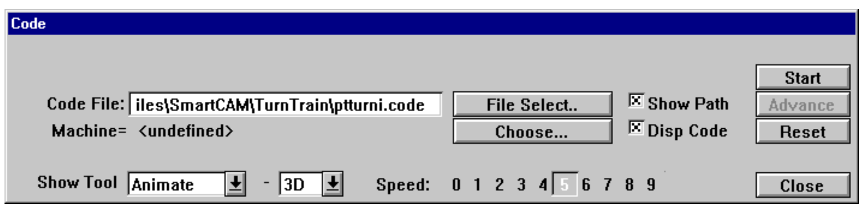



***Figure 6-4***
*Set the values on the Code dialog box.*

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.

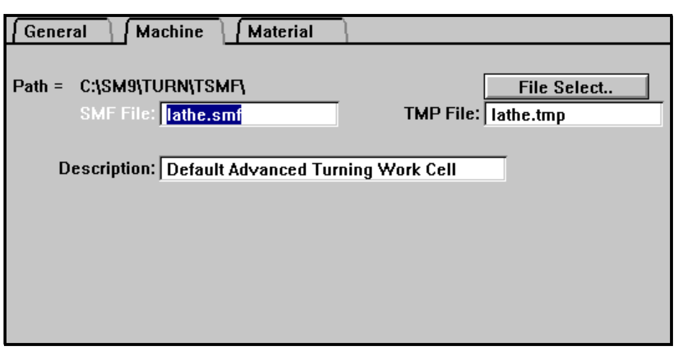



***Figure 6-5***
*Set the .smf and the .tmp files.*

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.
     - **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.

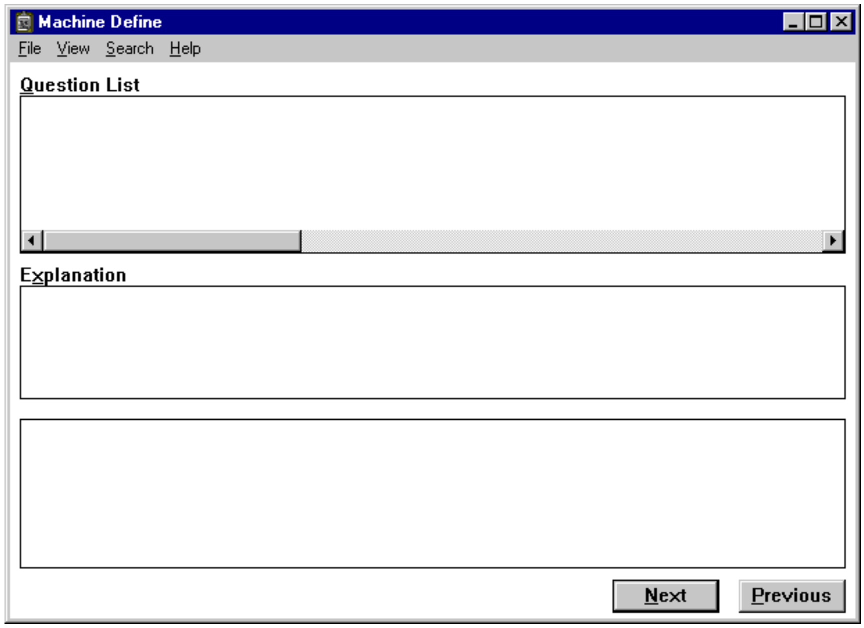



***Figure 6-6***
*Set the values on the Machine Define dialog box.*

2. Select **File—Open SMF**. The **Open SMF** dialog box is displayed.
3. Select the `GRAY.SMF` file to open.

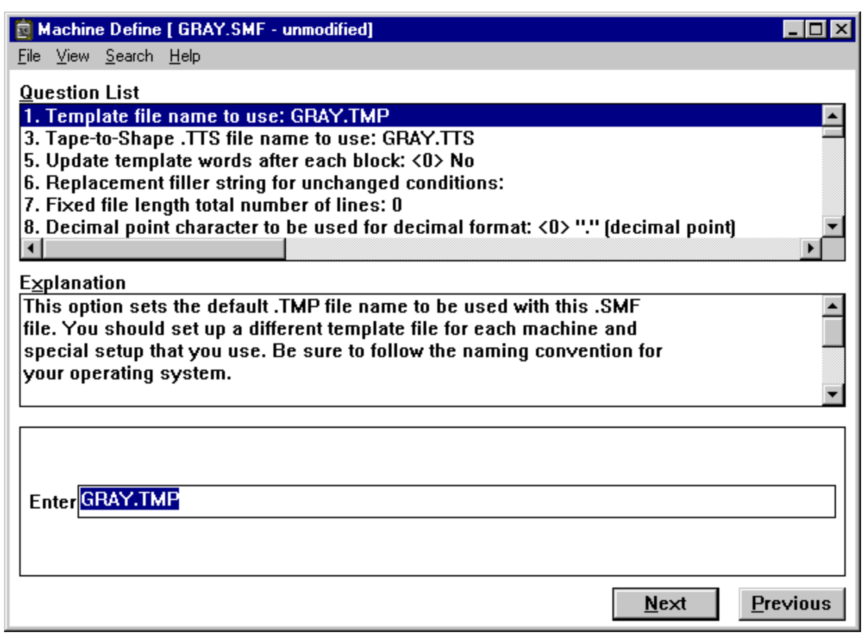



***Figure 6-7***
*Select the question to change and enter the changes.*

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 Turning

# Exploring SmartCAM Advanced Turning

## Welcome

SmartCAM Advanced Turning creates efficient toolpath and CNC code for 2- through 6-axis lathes and mill/turn centers. 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 4-Axis Synchronization
- Learning Milling Basics for Advanced Turning
- Using Rough Machining Processes
- Creating Workplanes

7

# Using 4-Axis Synchronization

## Objectives

This unit shows you how to perform these tasks:

- Add and remove the ability to coordinate two turning operations that occur at the same time.
- Insert and remove Wait commands.
- Create a mirrored toolpath for the operation of a second turret so that both turrets can cut the same toolpath simultaneously.

## Overview

Simultaneous 4-axis turning involves the synchronized motion of two independent turrets. Turrets in 4-axis applications can work independently or simultaneously on the ZX plane.

# Synchronizing Turning Operations

## Objectives

This lesson shows you how to coordinate two turning operations that occur simultaneously.

## Overview

Use synchronization to coordinate two turning operations that occur at the same time. The turning center must have dual turret synchronization capabilities. You can synchronize only turning operations. You cannot nest synchronized operations inside of each other.

Before synchronizing your model, save the current version to disk. Synchronizing changes the sequence of the elements.

You can also add Wait commands to the process. Wait commands cause one of the synchronized tools to wait until the other tool starts cutting.

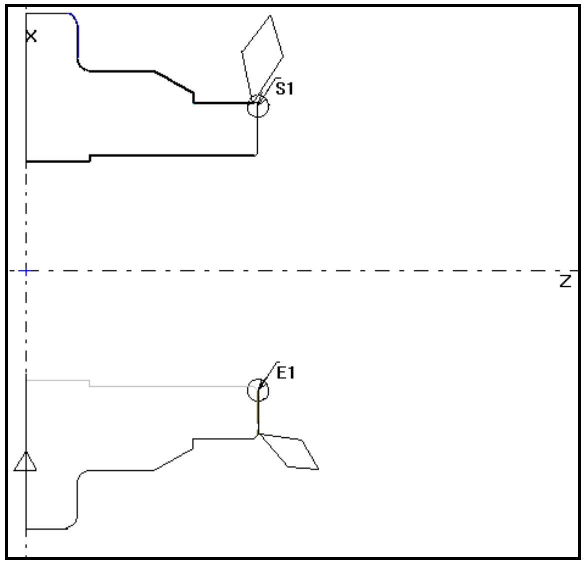
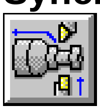



***Figure 7-1***
*These are results of 4-axis turning.*

## Synchronizing a Model



**Model File: ATSYNCH.PM4**

Before synchronizing your model, save the current version with a new name to disk. There is no undo for this feature once you press the Go button. When you use synchronization, the database is reordered, and it cannot be restored by removing the synchronization.

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

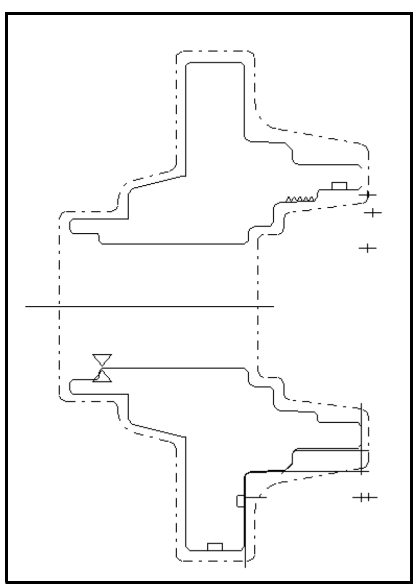


***Figure 7-2***
*Open ATSYNCH.*

2. Select **View—Show Path**. The **Show Path** dialog box is displayed.

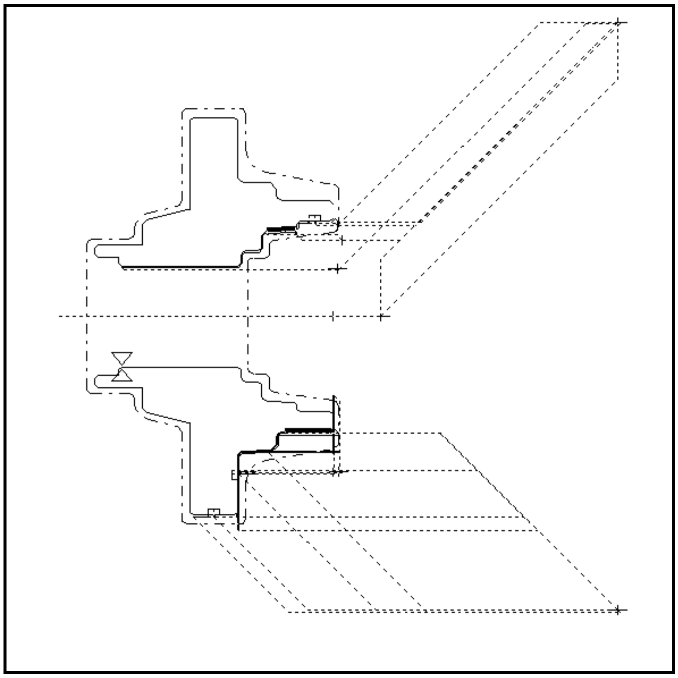


*Figure 7-3* *Notice the toolpath.*

3. Set the **Speed** to **5**, and pay attention to the order of operations by each turret on this part.
4. Group the operations to synchronize, which are steps **102** and **201**.
5. Set the insertion location after element **4** for the synchronized operations. The grouped operations are moved to this location in the database.
6. Select **Process—4-Axis Sync.**

   **Note** The **4-Axis Sync** toolbox is available only when **SMF question 210** is set to **1** (**yes**).

7. Select **Synchronize** from the toolbox. The **Synchronize** control panel is displayed.

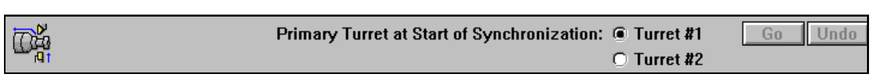



*Figure 7-4* *Set the values on the Synchronize control panel.*

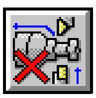
8. Select the turret to use as the primary turret. The primary turret is the turret that controls the spindle speeds during the operations.
9. Select the **Go** button.

   A user command is inserted before and after the operations. All grouped operations are reordererd and placed at the insertion location that you specified in step 5 of this procedure. An S1 and E1 are displayed on the screen, showing the start and end of the first synchronized operations.

10. Select **View—Show Path** again. The **Show Path** dialog box is displayed.

11. Set the **Speed** to **5**, and pay attention to the order of operations by each turret on this part. Notice that the grouped operations occur at the same time, but step **201** remains adjacent to the spindle.
12. Set the insert location after element **36**.
13. Make sure that the **Match Elmt** on/off switch is turned on.
14. Select **Create—Geometry**.
15. Select **Point/Rapid** from the toolbox.
16. Set **Point Z** to **6**, and **D** to **-12.375**.
17. Press Enter. The point/rapid is created. You can view it with the Show Path feature.

## Removing Synchronization



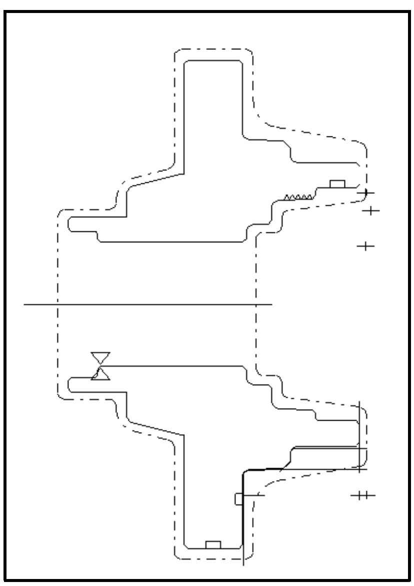
**Model File: ATSYNCH.PM4**

Use Remove synch to select and delete a pair of matching Wait commands, or all Start, End, and Wait synch commands associated with the active group. This enables you to modify existing synchronized operations.

Removing synchronization does not restore the original database order.

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

***Figure 7-5***
*Open ATSYNCH.*



2. Group the synchronized cuts.
3. Select **Process—4-Axis Sync**.

4. Select **Remove synch** from the workbench. The **Remove synch** control panel is displayed.


***Figure 7-6***
*Set the values on the Remove synch control panel.*

5. Select the **Group Remove** button.

## Points to Remember

- ❑ Synchronized operations cannot be nested inside of each other.
- ❑ Before synchronizing your model, save the current version to disk. There is no undo for this feature once you press the Go button.
- ❑ You can add Wait statements to the process; they cause one of the synchronized tools to wait until the other tool starts cutting.
- ❑ Removing synchronization does not restore the original database order.

# Working with Wait Commands

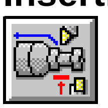
## Objectives

This lesson shows you how to insert and remove Wait commands.

## Overview

Wait commands control the timing of two synchronized turrets. Use Wait commands to cause a turret to stop or pause until the other turret reaches a matching Wait command.

## Inserting Wait Commands



**Model File: ATWAIT.PM4**

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

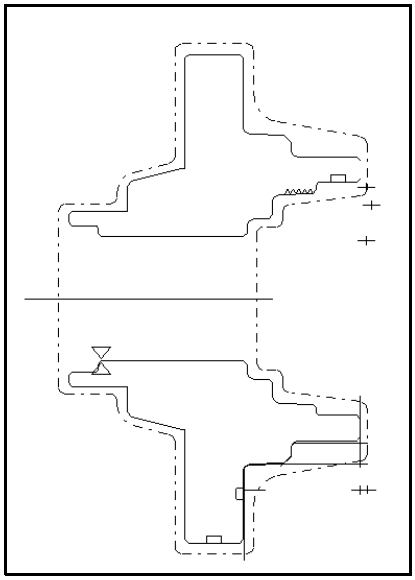


***Figure 7-7***
*Open ATWAIT.*

2. Use Show Path to see the synchronized tools (101 and 201). Notice that the tools come very close to each other. A Wait command will ensure that you avoid a potential collision.
3. Group steps 101 and 201, which are the synchronized steps.
4. Select **Process—4-Axis Sync**.
5. Select **Wait** from the toolbox. The **Wait** control panel is displayed.

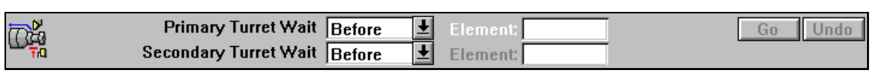



***Figure 7-8***
*Set the values on the Wait control panel.*

6. Set the **Primary Turret Wait** selector switch to **After,** and then set the element input field to **3**.
7. Set the **Secondary Turret Wait** selector switch to **After**, and then set the element input field to **19**.
8. Select the **Go** button.
9. Use Show Path to view the results.

## Points to Remember

- ❑ Wait commands control the timing of two synchronized turrets.
- ❑ You can remove Wait commands after you insert them.

# Auto Balancing Cuts

## Objectives

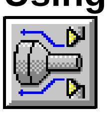
This lesson shows you how to auto balance cuts.

## Overview

Auto Balance creates a mirrored toolpath for the operation of a second turret (pinch turning) so that both turrets can cut the same toolpath simultaneously.

Auto Balance mirrors the active group across the Z axis and assigns the specified tool from the other turret. Synchronization commands are placed at the start and end of both toolpaths.

## Using Auto Balance



**Model File: ATAUOBAL.PM4**

Perform these tasks to use Auto Balance:

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

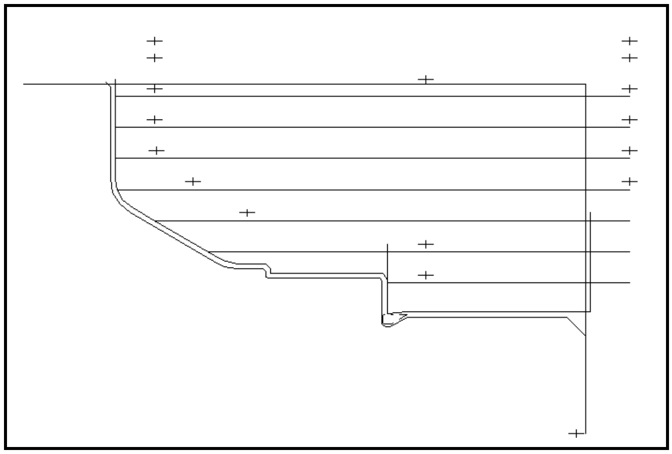


***Figure 7-9***
*Open ATAUOBAL.*

1. Save a copy of the current process model with a different name to disk.
2. Before you begin, be sure you defined steps for both the upper and lower turret and that steps are present for both turrets. You can do this by verifying steps 101 and 201 in the Job Operation Planner and noting the tool orientation and turret for each step.
3. Insert after element **62**.
4. Group the cuts to mirror from one of the turrets, specifically step 101. Only grouped cuts are auto balanced.
5. Select **Process—4-Axis Sync.**
6. Select **Auto Balance** from the toolbox. The **Auto Balance** control panel is displayed.


***Figure 7-10***
*Set the values on the Auto Balance control panel.*

7. Set the **Select Step for Balanced passes** field to step 201, which is from the other turret.
8. Select the **Go** button.
9. Select **View—Full** to view the new autobalanced roughing toolpath.

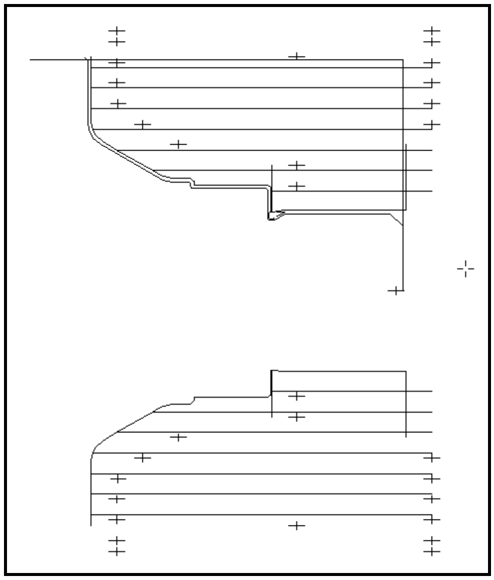


***Figure 7-11***
*This is the toolpath after auto balance is applied to it.*

## Points to Remember

- ❑ Auto Balance creates a mirrored toolpath for the operation of a second turret.
- ❑ Auto Balance mirrors the active group across the Z axis and assigns the specified tool from the other turret. Synchronization commands are placed at the start and end of both toolpaths.
- ❑ Only grouped cuts are auto balanced.

8

# Learning Milling Basics for Advanced Turning

## Objectives

This lesson shows you how to perform these tasks:

- Learn basic milling terminology.
- Use the appropriate work plane to generate milling and drilling operations on the face of the part.
- Wrap geometry around a part.

## Overview

You can use SmartCAM Advanced Turning to perform simple milling operations, as well as simple and complex turning operations.

## Learning Basic Milling Terminology

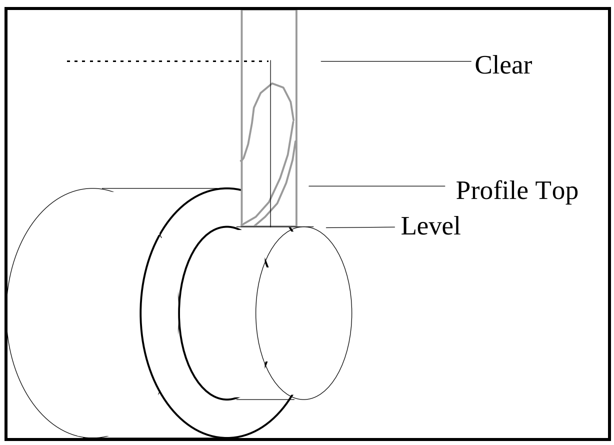
When you select a milling tool as the active tool, you must set Profile Top, Level, and Clear.

Use the P (Profile Top) input field to set the Z-height of the top of your part.

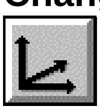
Use the Z (Level) input field to set the Z-height of the geometric elements you draw. When you draw an element with a step, SmartCAM sets the Z-height for the tip of the tool to the level you set in the Level input field.

Use the C (Clear) input field to set the Z-height for how far the tool retracts after making a cut.

***Figure 8-1***
*This part has Clear, Profile Top, and Level values for a simple Milling operation.*




## Changing Work Planes



**Model Files: ATPLANES.PM4**

You must change work planes to perform wrapping, and milling or off-center drilling operations on a part. You can use the existing work planes for most of these operations.

Perform these tasks to change a work plane:

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

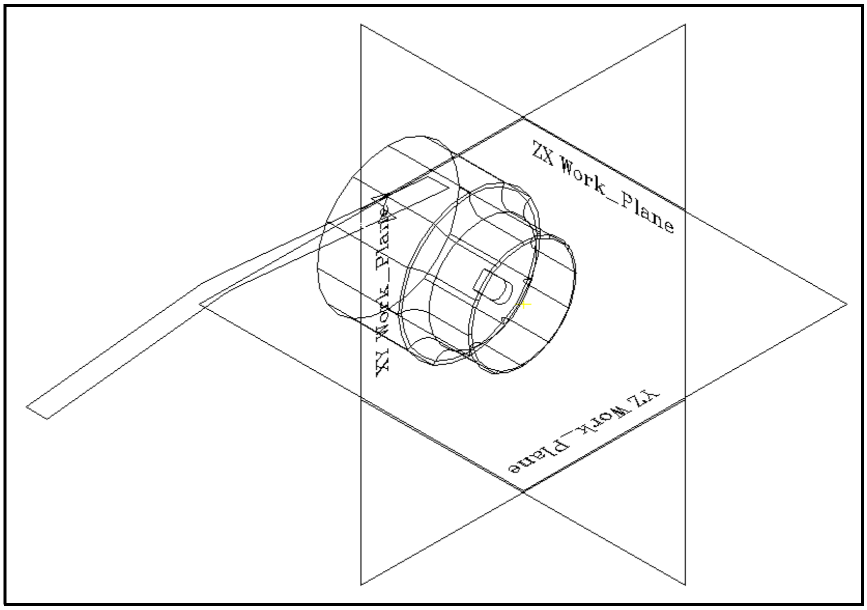
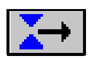
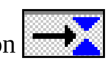
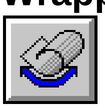



***Figure 8-2***
*Open ATPLANES.*

2. Set the **Before** icon  or **After** icon  on the Insert property bar.
3. Set the **With Step** icon .
4. Set the **Work Plane** input field.
5. Select the correct work plane from the list view.

## Wrapping Geometry



**Model File: ATWRAP.PM4**

Use Wrap to transform the active group of elements so that they wrap around the X and/or Y axis. The property and sequence location of the active group stays the same, but the wrap geometry is assigned to a new work plane. Use this modeling tool to create toolpath for machines with rotary contouring capabilities.

Perform these tasks to wrap geometry:

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

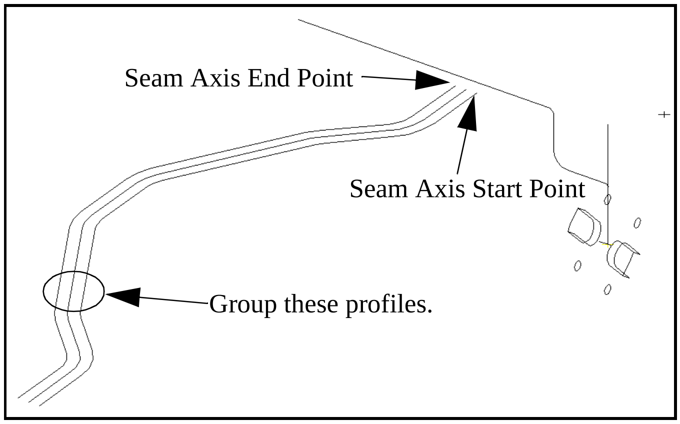



***Figure 8-3***
*Open ATWRAP.*

2. Select the geometry at the proper level and start location on the YZ work plane.
3. Group the three profiles.
4. Select step **105**.
5. Select **Edit—Transform**.
6. Select **Wrap** from the toolbox. The **Wrap** control panel is displayed.

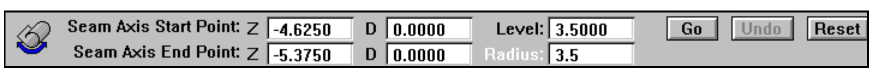
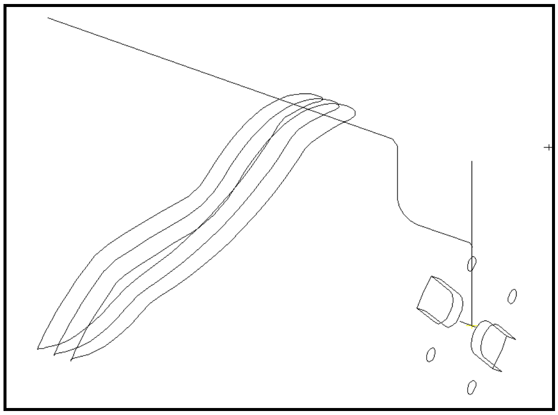


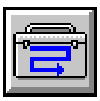

***Figure 8-4***
*Set the values on the Wrap control panel.*

7. Set the following fields on the control panel.
   - Select the **Seam Axis Start Point** input field and select the seam axis start point from the graphic view.
   - Select the **Seam Axis End Point** input field and select the seam axis end point from the graphic view.
   - Set the **Level** input field to the depth of the cut, which is 3.5.
   - Set the **Radius** input field to the radius of the shape, which is 3.5. Typically, this value is the same as the one set in the **Level** input field.
8. Select the **Go** button.

***Figure 8-5***
*This is the result of wrapping geometry.*



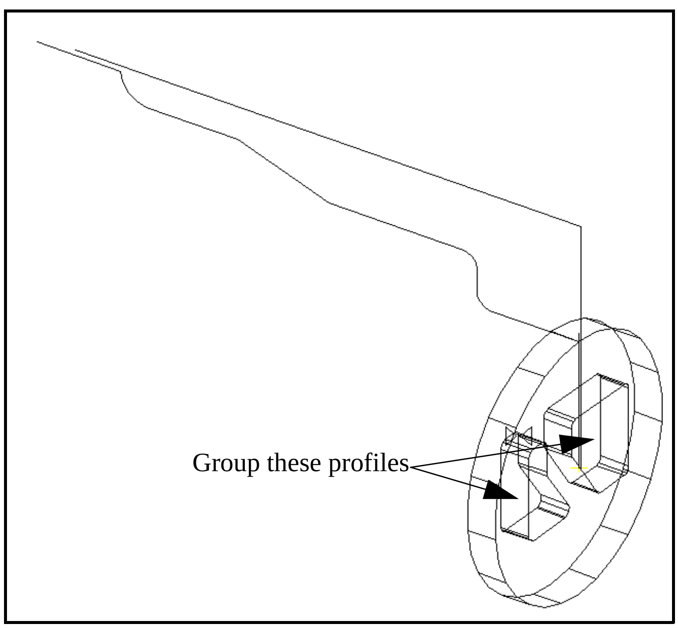
## Milling on a Face

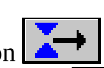
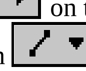


**Model File: ATFACE.PM4**

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

***Figure 8-6***
*Open ATFACE.*




2. Set the insert location:
   - Set the **Before** icon  on the Insert property bar.
   - Select the **Element** icon , and select the element **17** from the list view.

- Set the **With Step** icon , Turn on **With Step**, and select step **17** from the list view
- Set the **C** (Clear) input field to **.25[6.35]**.
- Select the **Work Plane** input field to **XY**.

3. Select **Process—Mill Rough**.
4. Select **Face** from the toolbox. The **Face** control panel is displayed.

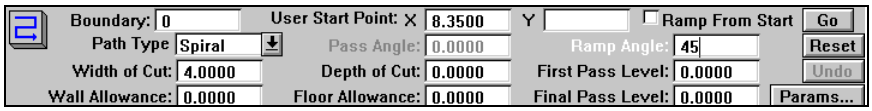



***Figure 8-7***
*Set the values on the Face control panel.*

5. Group the two red, closed profiles.
6. Set these values on the **Face** control panel:
   - Set the **Ramp Angle** input field to **45**.
   - Set the **Boundary** input field to **3.**
   - Set the **Width of Cut** input field to**.25[6.35]**.
   - Set the **First Pass Level** input field to **-.2[-5.08]**.
   - Set the **Depth of Cut** input field to **.25[6.35]**.
   - Set the **Final Pass Level** input field to **-.25[-6.35]**.
7. Select the **Params...** button. The **Face Parameters** dialog box is displayed.

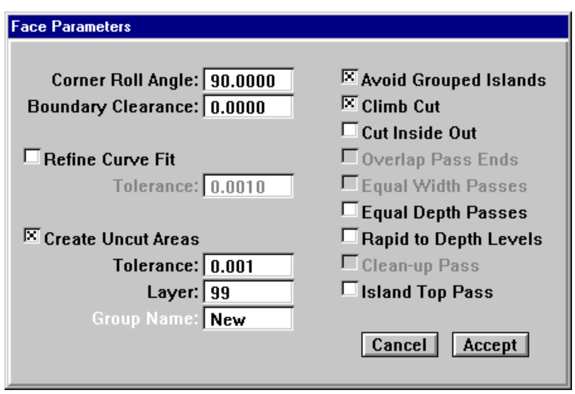
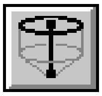



***Figure 8-8***
*Set the values on the Face Parameters dialog box*

8. Set these values on the dialog box.
   - Turn on the **Avoid Grouped Islands** on/off switch.
   - Turn on the **Climb Cut** on/off switch.
9. Select the **Accept** button.
10. Select the **Go** button.

## Performing Off-Center Drilling



**Model File: ATDRILL.PM4**

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

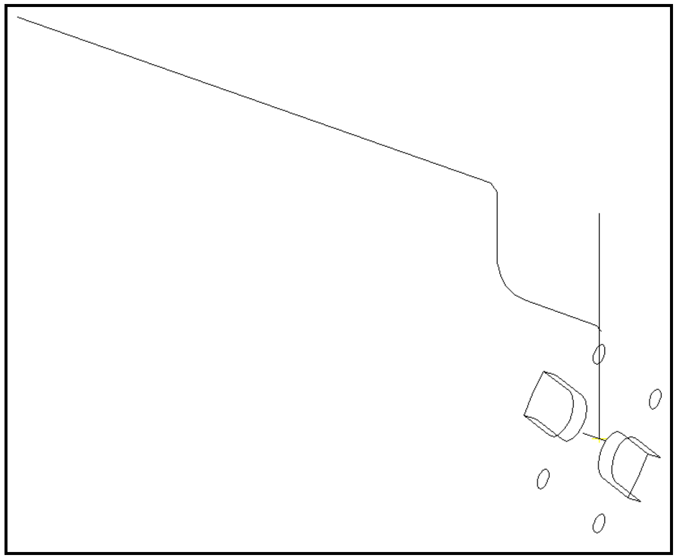


***Figure 8-9***
*Open ATDRILL.*

2. Set the insert location:
   - Set the **After** icon  on the Insert property bar.
   - Select the **Step** icon , and select step **103**.
   - Set the **With Step** icon , and select step **110**.

     **Note** It is important that the drilling step not be a fixed drilling tool.
   - Set the **Work Plane** to **XY**.
3. Select **Create—Geometry**.
4. Select a **Hole** from the toolbox. The **Hole** control panel is displayed.

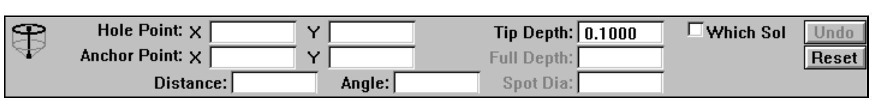



***Figure 8-10***
*Set the values on the Hole control panel.*

5. Set the following values on the control panel.
   - Set the **Full Depth** input field to **1.5[38.1]**.
   - Set **Hole Point** to **1.5[38.1], 0**.
6. Use automatic snap mode and the **Circle Center** snap icon to create holes at the centers of the arcs.

## Points to Remember

- ❑ When you select a milling tool as the active tool, you must set Profile Top, Level, and Clear.
- ❑ You must change work planes to perform wrapping, and milling or off-center drilling operations on a part.

# 9

# Using Rough Machining Processes

## Objectives

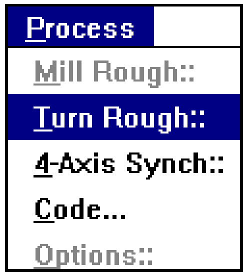
This lesson shows you how to perform these tasks:

- Pocket an area with islands.
- Pocket multiple areas.
- Face a part.
- Cut an open profile.
- Set spiral parameters.
- Set linear parameters.

## 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.

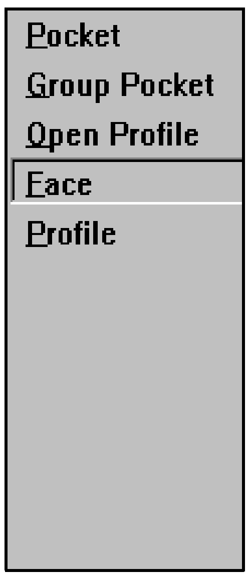
*Figure 9-1*
*Learn the parts of the Process menu.*



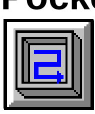

## Using the Mill Rough Toolbox

Use the Mill Rough toolbox to create roughing toolpaths to remove a large volume of material using multiple cutting moves.

*Figure 9-2*
*Learn the parts of the Mill Rough toolbox.*




## Pocketing



**Model File: ATPOCKET.PM4**

Use Pocket to create roughing profiles for the removal of material inside a closed profile. Multiple islands can exist inside the profile.

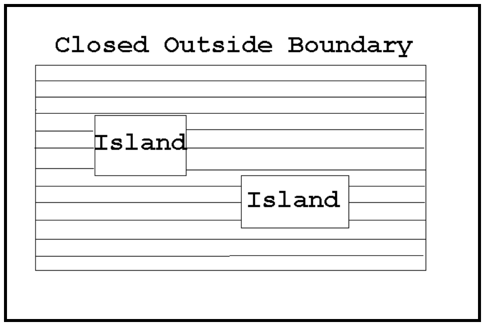



***Figure 9-3***
*Multiple islands are in this profile.*

The outside boundary defines the bottom edge of the pocket. Islands are grouped. Outside boundaries and islands have these characteristics:

- A closed profile
- A profile top value

Follow these steps to create a pocket:

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

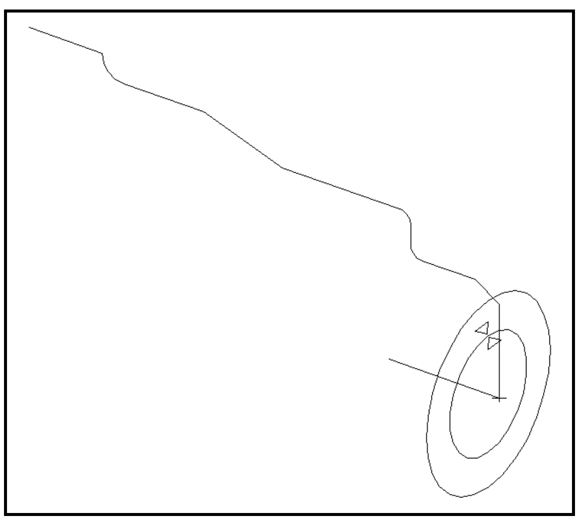
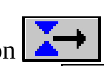
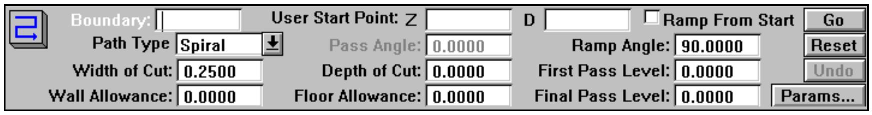


***Figure 9-4***
*Open ATPOCKET.*

2. 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.
3. Set the insert location:
   - Set the **Before** icon , and select element **17**.
   - Set the **With Step** icon , and select step **17**.
   - Set the **C** (Clear) input field.
4. Select **Process—Mill Rough**.

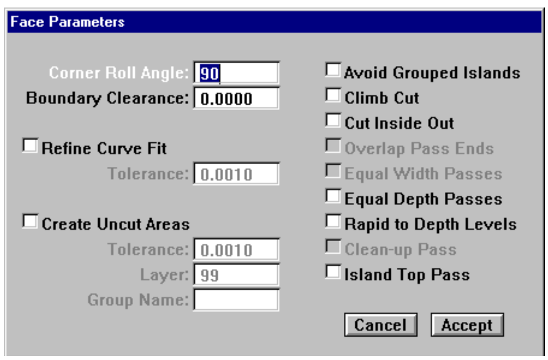
5. Select **Pocket** from the toolbox. The **Pocket** control panel is displayed.

***Figure 9-5***
*Set the values on the Pocket control panel.*



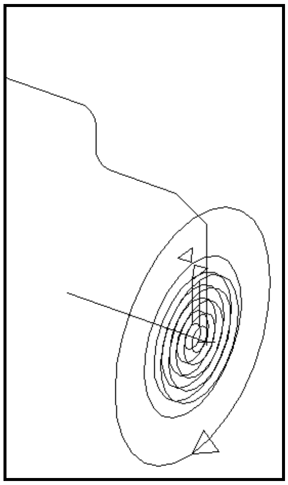

6. Set the **Pocket** selector switch to **Spiral** to creates a spiral roughing operation.
7. Turn on the **Ramp From Start** on/off switch to create ramp moves directly from the user start point location to the automatic start point of the toolpath; otherwise, turn it off to create a vertical plunge at the user start point.
8. Set the other fields as necessary on the control panel.
9. Select the **Params...** button on the toolbox. The **Face Parameters** dialog box is displayed.

***Figure 9-6***
*Set the values on the Face Parameters dialog box.*

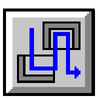



10. Set these values on the dialog box:
    - Turn on the **Cut Inside Out** on/off switch.
    - Set the **Corner Roll Angle** to **90**.
    - Turn on the **Avoid Grouped Islands** on/off switch to avoid an active group of islands or notches in the pocket; otherwise, turn it off.
    - Turn on the **Create Uncut Areas** on/off switch to create geometry for areas that the assigned roughing step cannot cut; otherwise, turn it off.
    - Turn on the **Climb Cut** on/off switch to create climb cuts; otherwise, turn it off to create conventional cuts.
11. Press the **Accept** button.
12. Select the **Go** button to start the operation.

***Figure 9-7***
*This is the result of pocketing.*



## Pocketing Multiple Areas

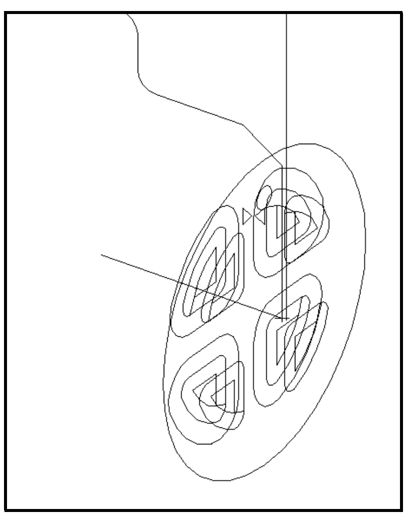


**Model File: ATGRPOCK.PM4**

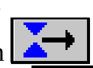
Use Group Pocket to create roughing passes for a group of closed finish profiles. You cannot use island curves.

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

***Figure 9-8***
*Open ATGRPOCK.*



2. Set the insert location:

■ Set the **After** icon .

■ Select the **Step** icon , and select **2**.

■ Set the **With Step** icon , and select **17**.

**Note** The roughing toolpath uses the Z clearance of the outer boundary and not the value in the **Clear** input field.

3. Select **Process—Mill Rough**.
4. Select **Group Pocket** from the toolbox. The **Group Pocket** control panel is displayed.
5. Group all of the cyan profiles on layer 4.

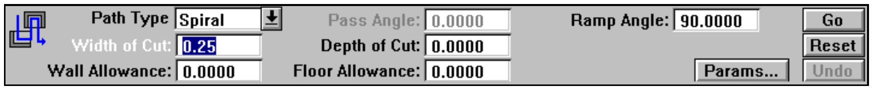



***Figure 9-9*** *Set the values on the Group Pocket control panel.*

6. Set these values on the **Group Pocket** control panel:
   - Set the **Path Type** selector switch to **Spiral** to create a spiral roughing operation.
   - Set the **Width of Cut** input field to **.25[6.35]**.
   - Set the **Wall Allowance** to **.005[.127]**.
   - Set the **Depth of Cut** input field to **.2[5.08]**.
7. Select the **Params...** button. The **Group Pocket Parameters** dialog box is displayed.

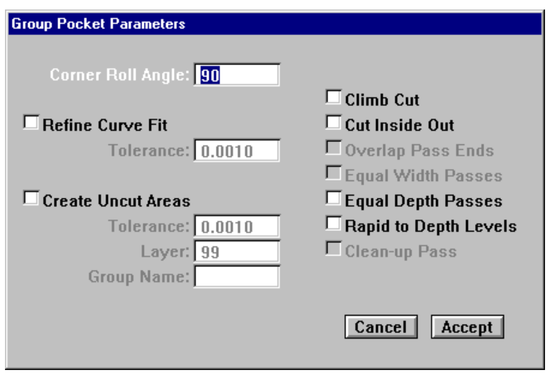



***Figure 9-10*** *Set the values in the Group Pocket Parameters dialog box.*

8. Set the following values on the Group Pocket Parameters dialog box:
   - Turn on the **Climb Cut** on/off switch.
   - Set the other parameters as necessary.
9. Select **Accept**.
10. Select the **Go** button to start the operation.

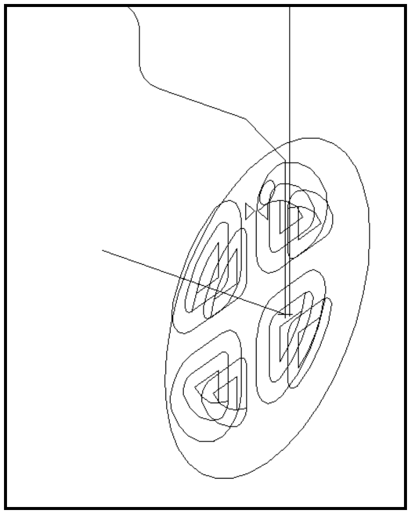
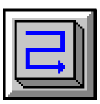


***Figure 9-11***
*This is the result of creating multiple pockets.*

## Pocketing on a Face



**Model File: ATPOCKET.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.

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

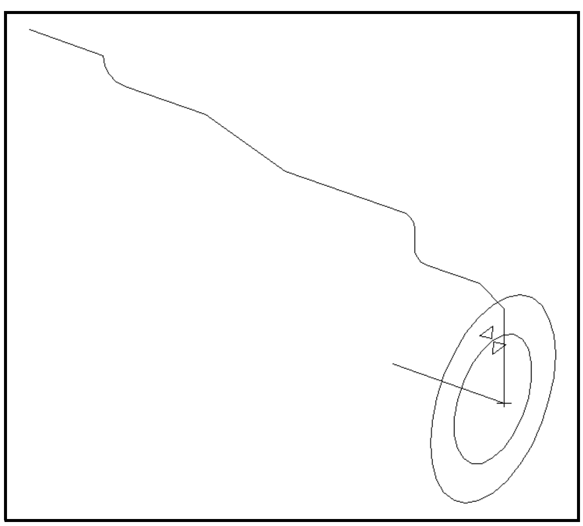
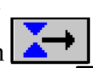
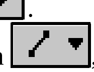


***Figure 9-12***
*Open ATPOCKET.*

2. Set the insert location:
   - Set the **After** icon .
   - Select the **Element** icon , and select element **13**.

- Set the **With Step** icon , and select step **17**.

  **Note** The roughing toolpath uses the Z clearance of the outer boundary and not the value in the **Clear** input field.

3. Select **Process—Mill Rough**.
4. Select **Pocket** from the toolbox. The **Pocket** control panel is displayed.

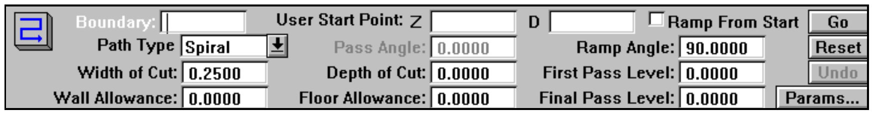



***Figure 9-13*** *Set the values on the Pocket control panel.*

5. Set these values on the **Pocket** control panel:
   - Set the **Path Type** selector switch to **Spiral** to create a spiral roughing operation.
   - Set the **Ramp Angle** input field to **45**.
   - Set the **Boundary** input field to **17**.
   - Set the **Width of Cut** input field to **.25[6.35]**.
   - Set the **Wall Allowance** input field to **.005[.127]**.
   - Set the **First Pass Level** input field to **-.2[-5.08]**.
   - Set the **Depth of Cut** input field to **.125[3.175]**.
   - Set the **Final Pass Level** input field to **-.25[-6.35]**.
   - Set the other fields as needed.
6. Select the **Params...** button on the toolbox. The **Face Parameters** dialog box is displayed.

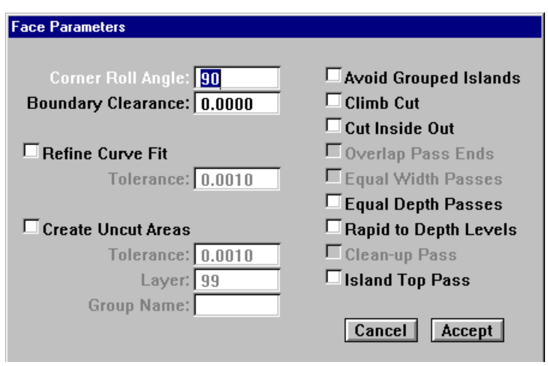


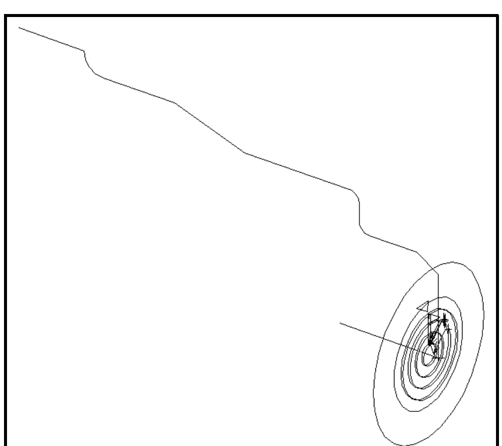

***Figure 9-14*** *Set the values on the Face Parameters dialog box.*

7. Set the values on the dialog box as necessary:
   - Set the **Corner Roll Angle** input field to the determining angle for corner rolling.

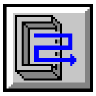
- Set the **Boundary Clearance** input field to the distance the tool is to cut beyond the face boundary. This input field accepts positive or negative numbers.
- Turn on the **Refine Curve Fit** on/off switch to remove colinear points from the part profile. This is useful when the part profile is a 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. Otherwise, turn it off.
- 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 **Overlap Pass Ends** on/off switch to create a small contour move at the end of each pass to clean up the edges of the profile; otherwise, turn it off.
- Turn on the **Equal Width Passes** on/off switch to make the width of all cutting passes the same; otherwise, turn it off.
- Turn on the **Equal Depth Passes** on/off switch to make the depth of all cutting passes the same; otherwise, turn it off.
- Turn on the **Rapid to Depth Levels** on/off switch to rapid the tool to the levels of the previous roughing passes; otherwise, turn it off.
- Turn on the **Clean-up Pass** on/off switch to create a profile pass around the islands and the profile boundary for each roughing pass level; otherwise, turn it off.
- Turn on the **Island Top Pass** on/off switch to create a pass over all island tops at a depth without roughing the entire pocket at that depth; otherwise, turn it off.

8. Select the **Accept** button.
9. Select the **Go** button to start the operation.

***Figure 9-15***
*This is a result of pocketing on a face.*



## Cutting an Open Profile



**Model File: ATOPNPRF.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.

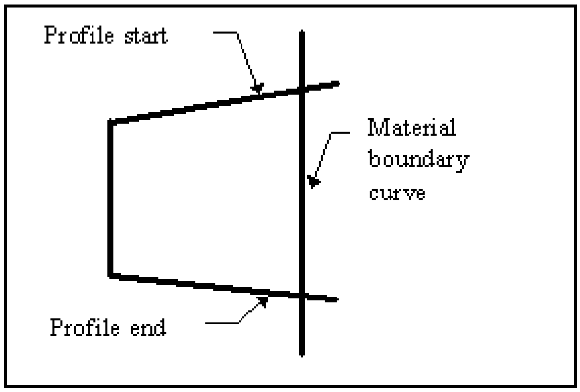



***Figure 9-16***
*Learn the parts of an open profile.*

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

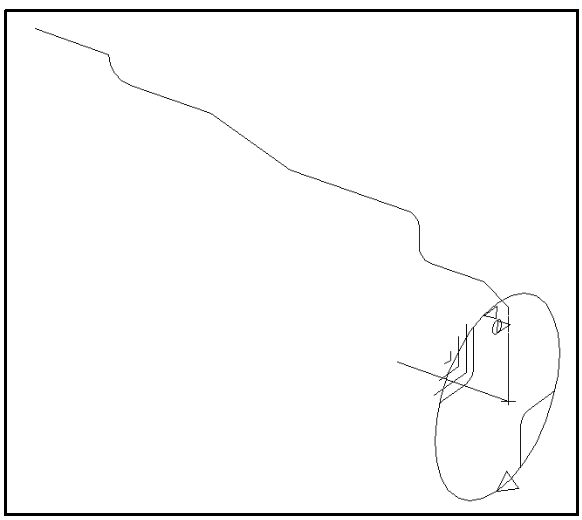
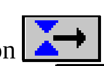
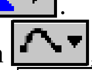


***Figure 9-17***
*Open ATOPNPRF.*

2. Set the insert location:
   - Set the **Before** icon .
   - Select the **Profile** icon , and profile **28**.
   - Set the **With Step** icon , and select **17**.

     **Note** The roughing toolpath uses the Z clearance of the outer boundary and not the value in the **Clear** input field.

3. Select **Process—Mill Rough**.

4. Select **Open Profile** from the tool list. The **Open Profile** control panel is displayed.

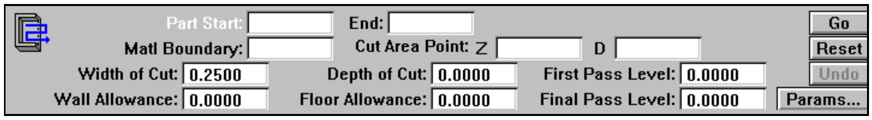



***Figure 9-18*** *Set the values on the Open Profile control panel.*

5. Set the following fields on the control panel:

   - Set the **Part Start** input field to **26**, which is the first curve in the profile.
   - Set the **End** input field to **28**, which is the last curve in the profile.
   - Set the **Matl Boundary** input field to **16**.
   - Select the **Cut Area Point** input field, and select a point anywhere inside the profile.
   - Set the **First Pass Level** input field to **-.2[-5.08]**.
   - Set the **Depth of Cut** input field to **.2[5.08]**.
   - Set the **Final Pass Level** input field to **-.25[-6.35]**.

6. Select the **Params...** button. The **Open Profile Parameters** dialog box is displayed.

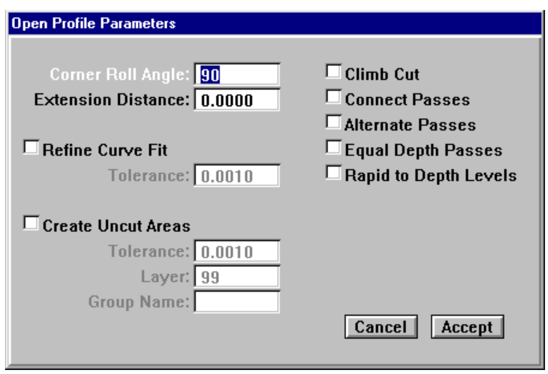



***Figure 9-19*** *Set the values on the Open Profile Parameters dialog box.*

7. Set fields on the dialog box as necessary.

   - Set the **Corner Roll Angle** input field 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. Otherwise, turn it off.
   - 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; otherwise, turn it off.
- Turn on the **Rapid to Depth Levels** on/off switch to rapid the tool to the levels of the previous roughing passes; otherwise, turn it off.

8. Select the **Accept** button.
9. Select the **Go** button.

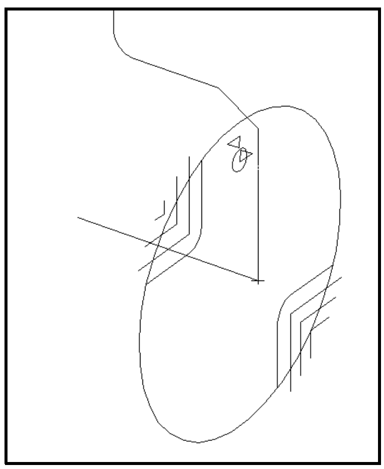


***Figure 9-20***
*This is the result of creating an open profile.*

## Points to Remember

- ❑ Use the Mill Rough toolbox to create roughing toolpaths to remove a large volume of material using multiple cutting moves.
- ❑ Multiple islands can exist inside the profile for Pocketing.
- ❑ You cannot have island curves when you pocket multiple areas.
- ❑ Use Face to create roughing profiles for the removal of material when you want the tool to overlap the outside of a profile boundary.
- ❑ No islands can be used with Open Profile.

# 10

# Creating Work Planes

## Objectives

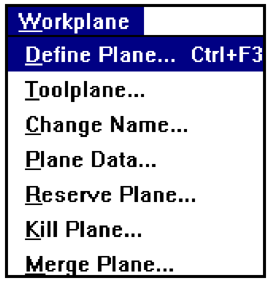
This unit shows you how to perform these tasks:

- Define a work plane.
- Define a tool plane.
- Kill a work plane.

## Overview

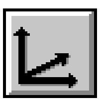
Use the Workplane menu to create or manipulate work planes or tool planes, which are used to define the orientation for planar elements such as arcs and ellipses.

*Figure 10-1*
*Learn the parts of the Workplane menu.*




All basic ID (inside diameter) and OD (outside diameter) turning operations should be assigned to the ZX work plane. Some parts may require drilling operations on the end or face of the part. This geometry is assigned to the XY work plane. When you perform mill-turn operations, you can use the YZ work plane when geometry is wrapped around the OD of the part. When you draw geometry to be wrapped, make sure its level is the same as the radius value when the geometry is wrapped. Often, new work planes are not necessary; use standard work planes wherever possible.

## Defining a Work Plane



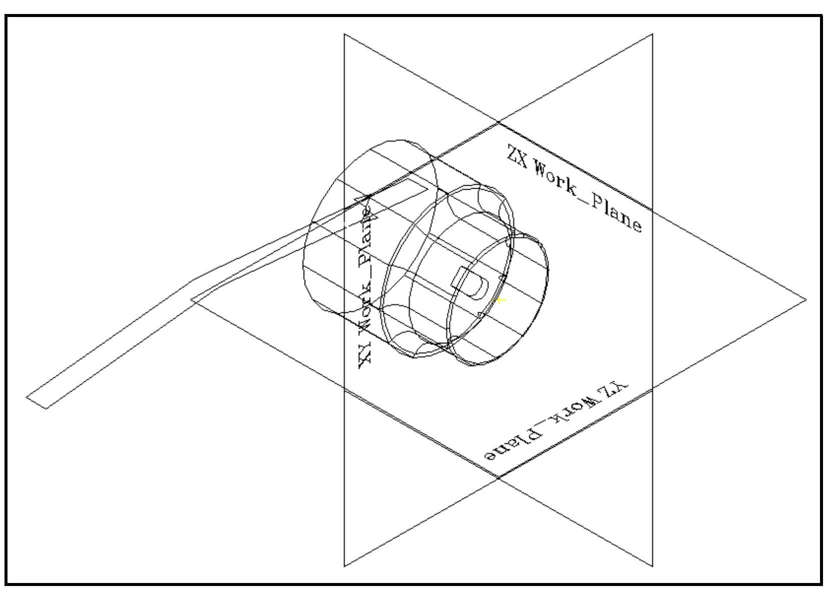
**Model File: ATPLANES.PM4**

Use Define Plane to create a new work plane for the model. You can also use Define Plane to change the parameters for an unreserved work plane, which changes the geometry associated with it. You cannot delete a reserved work plane. 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.

Perform these tasks to define a work plane:

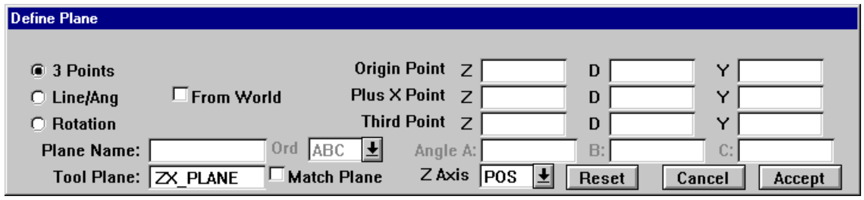
1. Open the model file **`ATPLANES.PM4`**.

*Figure 10-2*
*Open ATPLANES.*



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

***Figure 10-3*** *Set the values on the Define Plane dialog box.*

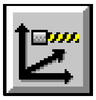



3. Turn on the **3 Points**, **Line/Ang**, or **Rotation** option switch:
   - **3 Points** defines a work plane, using three points.
   - **Line/Ang** defines a work plane, using a defined line and angle.
   - **Rotation** defines a work plane by rotating around a defined point.
4. Set the definition input fields:
   - **3 Points—**Set **Origin Point**, **Plus X Point**, and **Third Point**.
   - **Line/Ang—**Set **Origin Point**, **Plus X Point**, and **Angle**.
   - **Rotation—**Set **Origin Point**, **Ord**, and **Angle**.
5. Turn on the **From World** on/off switch to use the world coordinate system origin point as the reference for setting the new work plane's **Origin Point**, **Plus-X Point**, and **Third Point** input fields.
6. Set the **Plane Name** input field to the name of the new plane or the plane whose parameters you are changing.
7. Set the **Tool Plane** input field to the name of the plane's associated tool plane.
8. Turn on the **Match Plane** on/off switch to make the name of the tool plane the same as the new work plane name.
9. Select the **Accept** button.

## Creating a Work Plane When You Transform Geometry

You can also create a work plane when you transform geometry by turning off the Suppress Plane that is on the Mirror control panel and the Rotate control panel. The work plane you create is based on the current transformation.

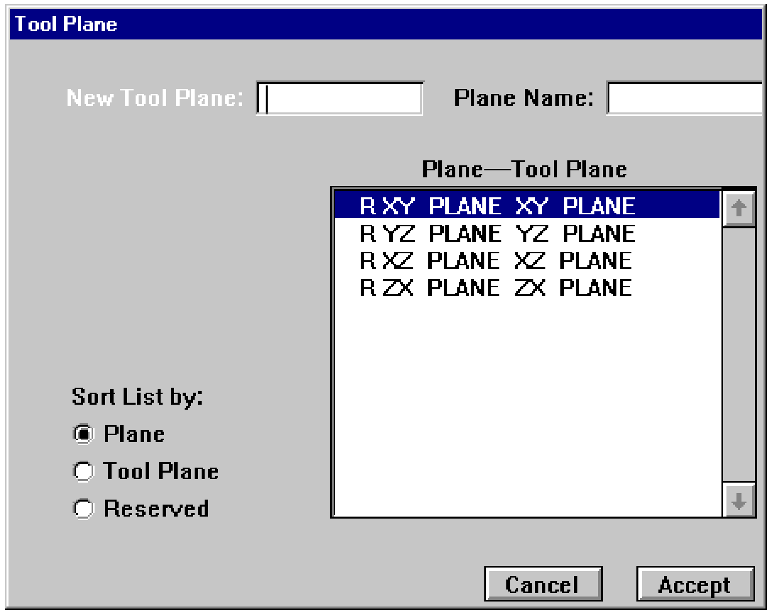
## Defining a Tool Plane



Use Tool Plane to assign a tool plane to an existing work plane:

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

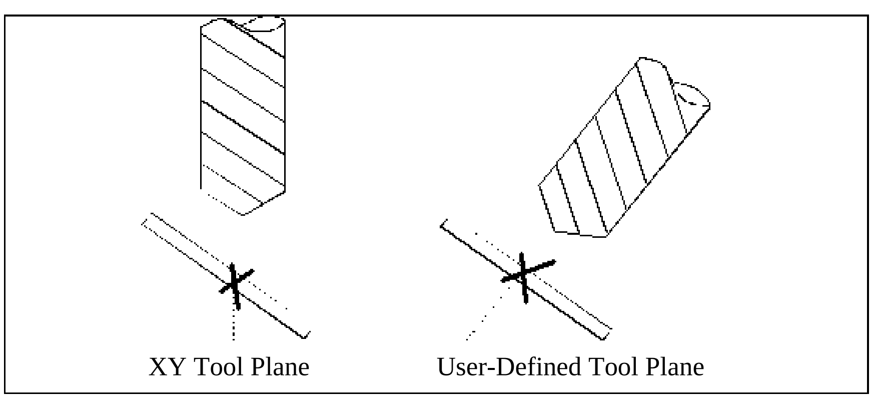
*Figure 10-4* *Set the values on the Tool Plane dialog box.*



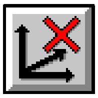

2. Set the **New Tool Plane** input field to the name of the tool plane from the displayed list.
3. Set the **Plane Name** input field to the work plane to assign from the list.
4. Set the **Sort List by** option switch for the plane information associated with the model to one of these:

- **Plane** to sort by planes
- **Tool Plane** to sort by tool planes
- **Reserved** to sort by reserved status

*Figure 10-5* *These are sample work planes.*



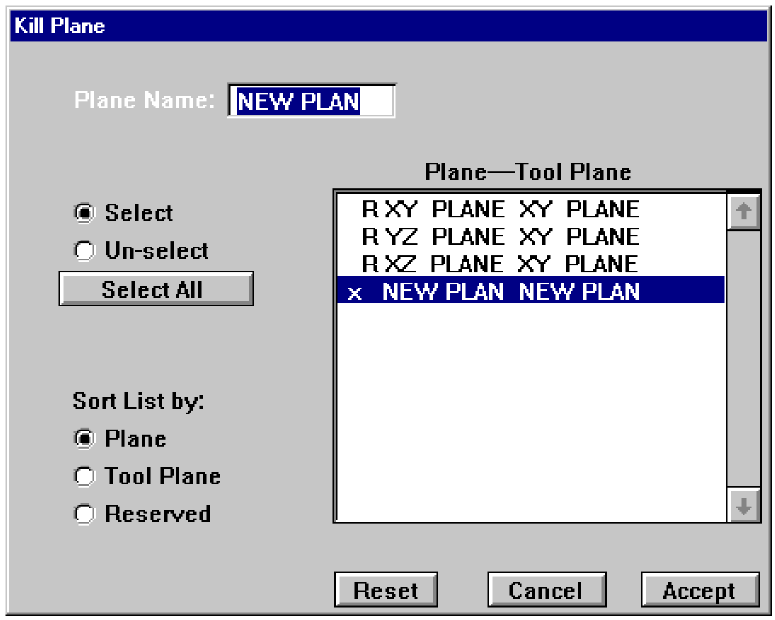
## Killing a Plane



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.

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

*__Figure 10-6__*
*Set the values on the Kill Plane dialog box.*




2. Set the **Plane Name** input field to the plane name to delete.
3. Turn on the **Select** option switch to mark individual, unreserved planes to delete; otherwise, turn on the **Un-select** option switch to keep the status of a previously selected, named plane in the list unchanged.
4. Select the **Select All** button to mark all unreserved, unused planes for deletion.
5. Set the **Sort List by** option switch to sort the plane information associated with the model by **Plane**, **Tool Plane**, or **Reserved** status.

## Points to Remember

- ❑ All basic ID and OD turning operations should be assigned to the ZX work plane.
- ❑ You can also use Define Plane to change the parameters for an existing nonreserved work plane, which changes the geometry associated with it.
- ❑ You cannot delete a reserved work plane.
- ❑ When you define and accept a work plane, it becomes the active work plane.
- ❑ Use Kill Plane to delete work planes that have no elements associated with them to free memory so you can add more elements to the model.
- ❑ You cannot kill reserved work planes.

# 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. How many synchronized turrets does the Wait command control the timing of?**

a) 0
b) 1
c) 2
d) as many as you want

_______________ **2. You cannot remove Wait commands after you insert them.**

a) true
b) false

_______________ **3. What input fields must you set when you select a milling tool as the active tool?**

a) Clear
b) Level
c) Prof_Top
d) all of the above

_______________ **4. What must you change to perform wrapping, and milling or off-center drilling operations on a part?**

a) work planes
b) tool planes
c) tools
d) layers

______________ **5. More than one island can exist inside the profile for pocketing.**

a) true
b) false

______________ **6. How many islands can exist inside the profile for pocketing multiple areas?**

a) 0
b) 1
c) 2
d) as many as you want

______________ **7. How many islands can exist for Open Profile?**

a) 0
b) 1
c) 2
d) as many as you want.

______________ **8. Which plane should basic ID and OD turning operations be assigned to?**

a) XY
b) YZ
c) ZX
d) kill plane

______________ **9. How do you delete a reserved work plane?**

a) Use Kill Plane.
b) Select the Undo button.
c) Press the Delete button on your keyboard.
d) You cannot delete it.

______________ **10. How do you delete a work plane?**

a) Use Kill Plane.
b) Select the Undo button.
c) Press the Delete button on your keyboard.
d) You cannot delete it.

# 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 Turning 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 1-23, *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 1-26

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 Clearance,* page 2-8

___C___ **4. What type of plane determines the orientation of geometry?**
*Lesson: Overview,* page 2-11

___B___ **5. 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: Overview,* page 2-15

___B___ **6. Using Status interrupts the current task.**
*Lesson: Checking Clearance,* page 2-8

___A___ **7. How do you know if tools require groups if there are no active groups?**
*Lesson: Using the Group Arrow Icon,* page 2-19

## 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 3-21

___A___ **4. You cannot remove active steps from a job.**
*Lesson: Removing Steps and Tools,* page 3-20

___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 3-27

_____**A**_____ **6. Once material information is set up, how do you access it from Production Turning?**
*Lesson: Opening a Material Librarian File*, page 3-36

_____**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 3-40

## 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 4-50

_____**D**_____ **5. How many elements in a model can you view modeling data for?**
*Lesson: Using Element Data*, page 4-33

_____**B**_____ **6. Viewing element data affects the geometry database.**
*Lesson: Overview*, page 4-33

_____**A**_____ **7. Changing the color of elements:**
*Lesson: Using Color Change*, page 4-52

_____**B**_____ **8. Elements to be trimmed or extended should be on different work planes.**
*Lesson: Trimming and Extending Geometry*, page 4-36

_____**B**_____ **9. How many groups does trimming by group impact?**
*Lesson: Trimming a Group of Elements*, page 4-38

_____**B**_____ **10. How many deleted groups can be recovered with the Undo button?**
*Lesson: Deleting Elements*, page 4-45

## Generating and Verifying Toolpath

___D___ **1. How do you tell SmartCAM to generate cuts with an offset when you use linear roughing?**
*Lesson: Performing Linear Roughing*, page 5-4.

___A___ **2. Use the Rough toolbox to create roughing toolpath to remove a large volume of material using multiple cutting moves.**
*Lesson: Using the Rough Toolbox*, page 5-3

___B___ **3. How do the roughing passes that contour roughing creates relate to the contour of the existing part profile?**
*Lesson: Performing Contour Roughing*, page 5-7

___C___ **4. What must the active step contain to use groove roughing?**
*Lesson: Performing Groove Roughing*, page 5-10

___D___ **5. What feature do you use to verify roughing toolpath?**
*Lesson: Overview*, page 5-19

## 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 Turning Answers

C 1. **How many synchronized turrets does the Wait command control the timing of?**
*Lesson: Working with Wait Commands,* page 7-9

B 2. **You cannot remove Wait commands after you insert them.**
*Lesson: Working with Wait Commands,* page 7-9

D 3. **What input fields must you set when you select a milling tool as the active tool?**
*Lesson: Learning Basic Milling Terminology,* page 8-2

A 4. **What must you change to perform wrapping, and milling or off-center drilling operations on a part?**
*Lesson: Changing Work Planes,* page 8-2

A 5. **More than one island can exist inside the profile for pocketing?**
*Lesson: Pocketing,* page 9-2

A 6. **How many islands can exist inside the profile for pocketing multiple areas?**
*Lesson: Pocketing Multiple Areas,* page 9-5

A 7. **How many islands can exist for Open Profile?**
*Lesson: Cutting an Open Profile,* page 9-10

C 8. **Which plane should basic ID and OD turning operations be assigned to?**
*Lesson: Overview,* page 10-1

D 9. **How do you delete a reserved work plane?**
*Lesson: Defining a Work Plane,* page 10-2.

A 10. **How do you delete a work plane?**
*Lesson: Killing a Plane,* page 10-5

# Glossary

**Accept**

Select this action button to accept entries into the active control panel.

**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.

**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.

**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.

**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.

**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.

**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.

**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.

**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.

**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.

**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.

**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.

**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.

**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.

**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.

**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.

**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.

**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.

**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.

**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.

**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.

**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.

**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.

**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.

**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.

**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.

**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.

**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.

**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

## A



## B



## C

## D

## E



## F

## J



## K



## L



## M

## Z