

**Getting Started with
FeatureCAM 2007**

Delcam USA

Information in this document is subject to change without notice. No part of this document may be reproduced or transmitted in any form or by any means, electronic, or mechanical, for any purpose, without the express written permission of Delcam USA. The software described in this document is furnished under a license agreement. The software may be used or copied only in accordance with the terms of the agreement.

© 1995 – 2006 Delcam USA. All rights reserved.

FeatureCAM, EZFeatureMILL FeatureTURN, EZFeatureTURN, FeatureMILL3D, FeatureTURN/MILL, FeatureWIRE and FeatureCAM and are trademarks of Delcam USA in the United States of America and other countries. Microsoft, MS, MS-DOS, Windows, Windows NT, Windows 2000, Windows XP and Windows 95 are registered trademarks of Microsoft Corporation. All other trademarks used herein are the property of their respective holders.

Restricted Rights Legend

The Program and Program Materials are provided with RESTRICTED RIGHTS. Use, duplication, or disclosure by the United States Government is subject to restrictions as set forth in subparagraph (c)(1)(ii) of the Rights in Technical Data and Computer Software Clause at DFARS 252.227-7013, Manufacturer is the Licensor: Delcam USA.

Permission to Copy for Licensed Users

Delcam USA grants permission for licensed users to print copies of this manual or portions of this manual for personal use only. Schools that are licensed to use FeatureCAM may make copies of this manual or portions of this manual for students currently registered for classes where FeatureCAM is used.

June 2006

Thirteenth Edition

Delcam USA

275 East South Temple, Suite 305

Salt Lake City, UT 84111

Fax 801-575-5017

Table of Contents

CHAPTER 1 - INSTALLATION

FEATURECAM PRODUCT LINE.....	1
INSTALLING THE DONGLE.....	2
INSTALLATION PROCEDURES.....	3
ENTERING TEMPORARY/EVALUATION CODES.....	3
Hints on Entering FeatureCAM Registration Codes.....	4
REGISTRATION.....	4
Filling Out Registration Forms.....	4
After Receiving Your Code.....	5
FEATURECAM MAINTENANCE RELEASES.....	5
EVALUATING PRODUCTS YOU HAVE NOT YET PURCHASED.....	6
INTERNET WEB SITE.....	7
FEATURECAM NETWORK LICENSING.....	7
FEATURECAM 2007 – WHAT’S NEW.....	7

CHAPTER 2 - STEPS FOR CREATING A PART

STARTUP: NEW PART DIALOG BOX.....	10
 STEP1: STOCK.....	10
 STEP 2: GEOMETRY.....	10
 STEP 3: CURVES.....	10
 Closed Boundary.....	11
 Open Boundary.....	11
 STEP 4: FEATURES.....	11
 STEP 5: TOOLPATHS.....	12
 STEP 6: NC CODE.....	14
 Optimize Feedrates.....	14
 Display.....	14

	Save	14
	Remap Tooling	14
	CUSTOMIZE MANUFACTURING	15
	Manufacturing Preferences	15
	Material Database.....	16
	Configure Post Processor	16
	OPTIONAL STEPS	17
	Surfaces	17
	Solids	17
	AFR.....	17
	On-line Help	17
	Reference Section.....	18
	What's New Section	19
	Context Sensitive Help.....	19
	WRITTEN DOCUMENTATION.....	19
 CHAPTER 3 - INTERFACE TOUR		
	FEATURECAM INTERFACE COMPONENTS.....	21
	Standard Toolbar	22
	Menu Bar	23
	Steps Toolbox.....	23
	Graphics Window.....	23
	Manufacturing Results Window	23
	Geometry Dialog Bar	24
	Assistance Bar	24
	Status Bar	24
	Part View Toolbox	24
	SELECTING GRAPHICAL OBJECTS.....	25
	OPTIONAL TOOLBARS	26
	Advanced Toolbar	26
	Snap Mode Toolbar and Dialog Box	27
	Geometry Toolbar	29
	Steps Toolbar.....	29
	WARPING DIALOG BOXES.....	30

CUSTOMIZING TOOLBARS.....	30
Toolbars Tab.....	30
Commands Tab and Misc. Tab.....	31
DISPLAY MODE BAR AND SHADING MODE.....	31
COLORS.....	33
 CHAPTER 4 - MAKING YOUR FIRST 2 ½ D PART	
STOCK PREPARATION.....	35
THE FIRST TIME.....	35
CREATE A PART FILE.....	36
VIEWING.....	37
TOOLPATHS.....	38
ORDER OF MANUFACTURING OPERATIONS.....	39
PART DOCUMENTATION.....	40
CONTROLLING THE AUTOMATION.....	41
NC CODE.....	41
TOOL MAPPING.....	41
CHANGING POST PROCESSORS.....	42
SAVING NC CODE.....	42
SAVING THE FEATUREMILL PART.....	42
THE NEXT STEP.....	42
 CHAPTER 5 - FEATURES FROM CURVES	
SETTING THE SNAPS.....	43
BUILDING CIRCLES.....	43
DRAWING LINES.....	44
CHAINING A CURVE.....	45
FEATURE FROM A CURVE.....	45
SIMULATE THE PART.....	46
MODIFYING THE BOSS FEATURE TO CONTOUR AROUND THE PART.....	46
CHANGING THE PLUNGE POINT.....	47
THE NEXT STEP.....	47
 CHAPTER 6 - INTRODUCTION TO TURNING	
THE FIRST TIME.....	49
CREATE A PART FILE.....	50
MAKING FEATURES.....	51
ORDERING OF OPERATIONS.....	54
PART DOCUMENTATION.....	54
NC CODE.....	55
CHANGING POST PROCESSORS AND SAVING NC CODE.....	55
SAVING THE FEATURETURN PART.....	55
THE NEXT STEP.....	56
 CHAPTER 7 - ADVANCED TURNING AND MACHINE SIMULATION	
GETTING STARTED.....	57

SETUP MACHINE SIMULATION	58
ADD A SUBSPINDLE FEATURE	59
ADD A BAR FEED FEATURE.....	59
RUN SUBSPINDLE SIMULATION	60
SETUP SECOND TURRET	61
SYNCHRONIZE OPERATIONS.....	61
RUN SYNCHRONIZED MULTI-TURRET SIMULATION	62
THE NEXT STEP.....	62
CHAPTER 8 - CREATING A TURN/MILL PART	
GETTING STARTED	63
CREATING A TURNED FEATURE	64
CREATING A RADIAL PATTERN ON THE FACE.....	65
ENGRAVING ON THE FACE.....	65
CREATING A PATTERN OF WRAPPED SLOTS	66
SIMULATING A TURN/MILL PART.....	66
The Next Step.....	66
CHAPTER 9 - FEATURES FROM 3D CAD MODELS	
TOOL CRIBS	68
IMPORTING THE CAD MODEL	68
SHADE THE PART	69
AUTOMATIC FEATURE RECOGNITION.....	69
RECOGNIZE ALL HOLES USING ANOTHER METHOD	70
RECOGNIZING ALL POCKETS AUTOMATICALLY	70
CREATE A SLOT FEATURE USING THE PICK SURFACE  BUTTON	71
CREATE A SIDE FEATURE	71
SIMULATE THE ENTIRE PART	72
THE NEXT STEP.....	72
CHAPTER 10 - INTRODUCTION TO 3D	
INTRODUCTION TO 3D MODELING	74
Create the Bottle Surface.....	75
INTRODUCTION TO 3D MANUFACTURING.....	75
SIMULATING THE ENTIRE PART.....	76
THE NEXT STEP.....	76
CHAPTER 11 - ADVANCED 3D MILLING	
IMPORT THE PART	77
CREATE THE 3D STRATEGY.....	78
SIMULATING THE TOOLPATHS.....	79
Clean Up the Fillets.....	80
THE NEXT STEP.....	81
CHAPTER 12 - SOLID MODELING	
GETTING STARTED	85

CREATE THE INITIAL SOLID	86
HIDDEN LINE GRAPHICS	87
CREATE THE CURVED LOWER SURFACE.....	87
CREATE CONSTANT AND VARIABLE RADIUS FILLETS	88
ADD THICKNESS	89
CREATE A MOLD CAVITY	89
THE NEXT STEP	90
CHAPTER 13 - INTRODUCTION TO FEATUREWIRE	
GETTING STARTED	91
CREATING THE PROFILE	91
CREATING A WIRE EDM FEATURE.....	92
SIMULATING A WIRE EDM TOOLPATH.....	92
GENERATING NC CODE.....	93
ADD A TAPER ANGLE	93
CREATING THE SECOND PROFILE.....	94
CREATING A 4-AXIS WIRE EDM FEATURE	94
MODIFYING THE START POINT	95
MATCHING THE CURVES	96
SIMULATE THE PART	96
THE NEXT STEP	96
CHAPTER 14 - FEATUREMILL3D 2007 TRANSITION GUIDE	
OVERVIEW.....	97
BACKWARD COMPATIBILITY	98
Reading Files from Previous Versions.....	98
Using Version 2006 Technology in Version 2007.....	99
CONVERTING AN INCOMPATIBLE OPERATION.....	99
Step-by-Step Instructions for Converting an Operation.....	100
INCOMPATIBILITIES	102
Ignored Settings	103

Chapter 1

Installation

FeatureCAM Product Line

FeatureCAM products include:

- FeatureMILL2.5D – 2.5D design and toolpath generation for 2 and 3-axis mills.
- FeatureMILL3D – 3D surface modeling and 3-axis toolpath generation
- Solid Modeling – Solid modeling and tools for creating molds from solid models.
- FeatureRECOGNITION – 3D surface and solid import and the recognition of 2.5D features from solid models. Accelerates making 2.5D and turned parts from solid models.
- FeatureTURN – 2-axis design and toolpath generation for 2-axis lathes.
- FeatureTURN/MILL – Supports lathes with C and Y-axis milling capabilities.
- FeatureWIRE – 2 and 4axis wire EDM toolpath creation.
- 5-axis Positioning – Manufacture 2.5D features from 5-axis orientations.
- Tombstone – Multiple part manufacturing for horizontal or vertical milling machines with indexers.
- Machine Simulation – Modeling and simulation of a CNC machine.
- Multi-turret Turning- Use of multiple turrets for production turning.
- Network – Flexible product licensing allows sharing FeatureCAM licenses across a network.
- Native Import Modules – Import Catia, Unigraphics, Pro/E and STEP files.

Installing the Dongle

A dongle is supplied with FeatureCAM for copy protection. The dongle is a physical device that attaches to your computer's parallel printer port or a USB port. FeatureCAM checks to see if a dongle is present before it generates NC code for your parts or before saving a file. The dongle must remain connected for the entire time you are running FeatureCAM.



USB Dongle



Parallel Dongle

If you downloaded a copy of FeatureCAM over the Internet, or your product box is marked as *Evaluation Copy*, you do not have a dongle. You can draw and simulate designs, but you can't generate NC code, save, or export any files.

If the side of your product box says *Upgrade* then you purchased an upgrade version of FeatureCAM and you must use the dongle you received with your earlier FeatureCAM purchase.

If you have a dongle, install it on your computer by inserting it in the parallel printer port (or USB port if using a USB dongle) and screwing in the thumbscrews. The dongle is programmed for an initial period (see *Registration* below). If you have licensed FeatureCAM, then you are given a code to provide unlimited use when you register your copy.

You may have multiple devices attached to your parallel port. Many computers have a printer attached to that port. If you have any devices attached to the port, you will need to chain the devices in sequence and the FeatureCAM dongle should be the first in the stack. Note that you may not connect multiple FeatureCAM dongles to a single parallel port.

The dongle is a pass through device. You can attach your dongle to the computer and the other devices can hook up to the dongle. You may have to experiment with the order of devices a bit to find which sequence works best with your devices. Printers should probably be at the end of the device chain. In general, try to put the FeatureCAM dongle in as the first or second device in the sequence.

Installation Procedures

Use the following information to help you install the program and dongle (software key).

For information on installation issues not covered here, refer to the README.WRI and RELNOTES.WRI files on the FeatureCAM Installation CD. These files are also copied to the hard disk in the EZFM directory in a successful installation.

If you're upgrading an older copy, you can safely overwrite old versions of FeatureCAM. When you are prompted for the installation directory, simply use the same directory you did last time. Your part files and customizations are preserved and can be used with this version of FeatureCAM.

If you wish to continue to run your older copy of FeatureCAM, then install the new version in a different directory. Be aware that your old version may not be able to read files from your new version, but the new version will always be backwards compatible with older versions of files.

The following instructions are written for a CD ROM that is drive D. Substitute the actual drive letter where appropriate.

1. Attach the dongle, if you have one, to your printer port.
2. Put the CD in drive D.
3. If installing from CD, the installation screen should automatically be displayed.
4. If installing from disks or if the installation screen does not automatically come up, choose Run from the Start menu. Type d:\setup if installing from CD.
5. Follow the instructions on the installation screens.

Entering Temporary/Evaluation Codes

If you purchased an upgrade to an existing copy of FeatureCAM, your package will contain temporary codes for activating your dongle until you get a chance to register. You must enter these codes to run your new version. If you purchased a new package with a new dongle inside, you do not have to complete these steps.

Run FeatureCAM.

If the New Part Document Wizard comes up, click *Cancel*.

If you have any windows open in FeatureCAM, close them.

Select *Evaluation Options* from the *File* menu.

Click Next.

Click I have an evaluation code and click Next.

Enter your evaluation code in the space provided and click *Finish*.

Hints on Entering FeatureCAM Registration Codes

1. Evaluation codes are a collection of the digits 0-9 and the letters A-F.
2. There are no letter Os in the codes, only the number zero.
3. The case of the letters does not matter. Spaces are also ignored.

Registration

You must register your new copy of FeatureCAM even if it is an upgrade to an older version. If you're not sure whether you purchased an upgrade or not, look at the side of your FeatureCAM box. If *Upgrade* is checked, you purchased an upgrade to your existing copy of FeatureCAM.

1. If FeatureCAM is not already running, run FeatureCAM from the Start menu.
2. If you purchased a new copy of FeatureCAM, the *Registration* dialog box is displayed.
3. If you are updating an existing copy, close all the windows in FeatureCAM (without closing the application itself) and select *Evaluation Options* from the *File* menu.
4. If you did not purchase an upgrade, you will see *Must register* in the *Status* column of the components you purchased. Your dongle will initially work for a limited amount of time. You must register them to remove this limitation.
5. If you did purchase an upgrade, the products that you licensed in previous versions will have *Yes* in the evaluating column. Those components that are already enabled for the version you are installing will have a status of *Fully enabled*. These products will not need to be registered again.
6. Click *Next*.

Filling Out Registration Forms

Please fill out the registration form included in the FeatureCAM box and follow the instructions on it. If you do not have a copy of the form please contact our corporate office at the toll-free number 1-888-393-6455 ext 10, 1-801-575-6021 ext 10 (outside of the United States) or email a request to admin@featurecam.com.

After Receiving Your Code

1. Run FeatureCAM.
2. Click *Next* in the *Registration* dialog box.
3. Click *I have registered and have a code*.
4. Select the product(s) that you received codes for. Click *Next*.
5. Enter code(s).
6. Click *OK*.
7. You will see the message *Thank you for registering FeatureCAM*.

Your product is now registered and fully enabled.

FeatureCAM Maintenance Releases

If you have purchased and registered a version of a FeatureCAM product, you are allowed to install maintenance releases free of charge. A maintenance release is any product version with the same first number as the product you are licensed to run. For example Version 13.0.1.36 the first number is 13.

To check to see if a new version of your FeatureCAM product is available:

1. Make sure you are connected to the Internet and select *Check for new version* from the *Help* menu.
2. If a more current version is available you are offered the option of updating your version of the FeatureCAM.

The first time you run your maintenance release the *Registration* dialog box may be displayed. If this happens:

1. Click *Next*.
2. Click *Finish*.

Evaluating Products You Have Not Yet Purchased

Once you are running FeatureCAM, you can evaluate other FeatureCAM products.

1. Run FeatureCAM.
2. If the New Part Document Wizard comes up, click *Cancel*.
3. Select *Evaluation Options* from the *File* menu.
4. Check the *Evaluate* checkboxes for the product components you want to evaluate.
5. Click *OK*.

FeatureCAM is now running in evaluation mode. You cannot save parts or post, but you can create part models and generate toolpaths. You can tell FeatureCAM is in evaluation mode by the word, (*evaluation*), in the title bar.



The next time you run FeatureCAM, the *Registration* dialog box is displayed. To run the program normally:

1. Run FeatureCAM and click *Next*.
2. Select *Run the program* and click *Finish*.
3. If the New Part Document Wizard comes up, click *Cancel*.
4. Select *Evaluation Options* from the *File* menu.
5. Uncheck all the *Evaluate* checkboxes.
6. Click *OK*.

Internet WEB Site

The Internet WEB site <http://www.featurecam.com> contains useful information for FeatureCAM users including:

1. New product announcements.
2. A comprehensive database of technical tips.
3. Maintenance releases.
4. Evaluation copies of new software.
5. Local dealer information.
6. Tradeshows where FeatureCAM is being exhibited.

There is no charge for accessing this WEB site, so feel free to access the latest information on FeatureCAM.

FeatureCAM Network Licensing

If you want to have the option of running your licensed FeatureCAM products on more than one computer, you may be interested in the *FeatureCAM Network Licensing* option. If you have licensed this option, see the READMEN.htm file located in the D:\netlic directory of the CDROM.

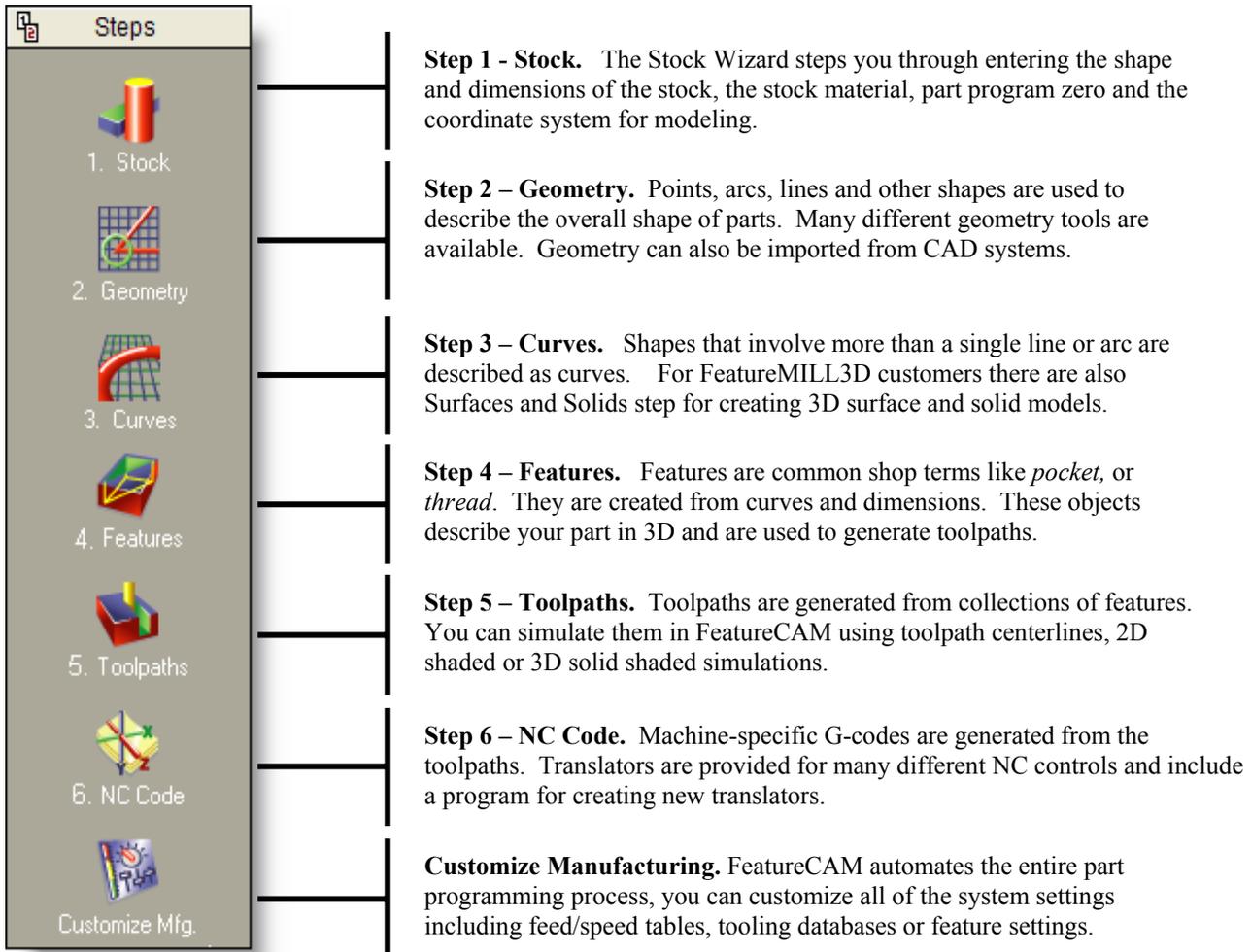
FeatureCAM 2007 – What's New

If you are an existing FeatureCAM user and wish to find out about new functionality and other changes to FeatureCAM, see the What's New section of the FeatureCAM help file.

Chapter 2

Steps for Creating a Part

The process for creating parts is laid out sequentially in the Steps toolbox. By using each step, you are guided through the process of creating part programs with FeatureCAM.



Startup: New Part Dialog Box

When you start FeatureCAM, the *New Part Dialog Box* is the first thing you see. In this dialog box you can choose to open a new part file, or continue working on an existing file.



Step 1: Stock

The stock is the initial material from which you will cut your part. It can be rectangular, circular, a more general extruded shape, or a solid model. The stock shape is used for 3D toolpath simulations and to control the extent of milling features like bosses. FeatureCAM has embedded feed/speed tables for many different stock materials, so that feeds and speeds can be automatically calculated based on the stock material. The Stock Wizard provides step-by-step instructions for specifying or modifying the general stock shape, specific stock dimensions, fixture IDs, part program zero, and the origin of your modeling coordinate system. To navigate through the wizard, enter the desired information on each page and click the *Next* button to move to the next page.

After completing this step, if you want to create holes, rectangular pockets, slots, circular pockets or thread mill features, you can skip to step 4.



Step 2: Geometry

Every part begins with an initial design. You may already have a part drawing, a part sketch or you may want to use FeatureCAM to draw your part design from scratch. The geometry step presents you with methods of creating points, lines, arcs and circles. You can also clip geometry. If you would like to create multiple objects, check the *Create more than one* option at the bottom of the dialog box before selecting the type of object you want to create.

After selecting the type of object you want to create, you are provided instructions in the yellow *Assistance* bar at the bottom of the screen. Type any numeric arguments that are required. Use the TAB key to move to the next field. For point coordinates you can type the individual X, Y and Z coordinates or use the mouse to graphically locate them in the *Graphics* window. As you move the mouse around the Graphics window, you'll notice that the cursor will snap to discrete locations like circle centers or line endpoints. This location can be controlled by the Snapping dialog, which is discussed in *Chapter 3*.



Step 3: Curves

This step groups arcs and lines into curves by automatically tracing smooth paths. If you want to make a feature from more than a single line, arc or circle, you must connect the geometry into a curve.



Closed Boundary

This option creates closed curves (curves that form a loop) by clicking on one piece of geometry. An attempt is made to create a closed loop by following smooth paths. If this method does not result in the correct pieces, click the *Clear pieces* button at the bottom of the screen and select the Curves step again and use the Open boundary command.



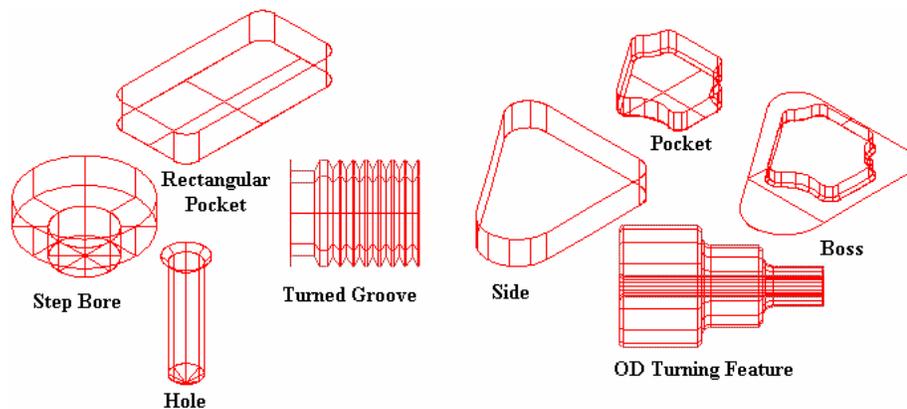
Open Boundary

This command creates curves by manually clicking on segments of the curves. To create an open curve (curves that do not form a loop), click on the beginning of the curve and then click on the end. If the resulting curve is not correct, click on a number of intermediate positions to guide the process of creating curves. Closed curves can also be created with this technique by clicking back to the initial piece of geometry.



Step 4: Features

Toolpaths are generated from features. For example, if you want a hole in your part, choose the Hole feature. If your part has a pocket, choose the Pocket feature. The Feature wizard walks you through the process of creating features. One category of features is those that can be created solely from dimensions. These features include holes, rectangular-shaped pockets, threads and round pockets (called Step Bores).



Other features require curves to describe their shape. They are listed under the *From Curve* category. These features include pockets, bosses, grooves, chamfers, rounds, sides and faces.

If you want to create a pattern of features, click the *Make a pattern from this feature* checkbox. Click *Next* to step through the wizard.

This wizard steps you through:

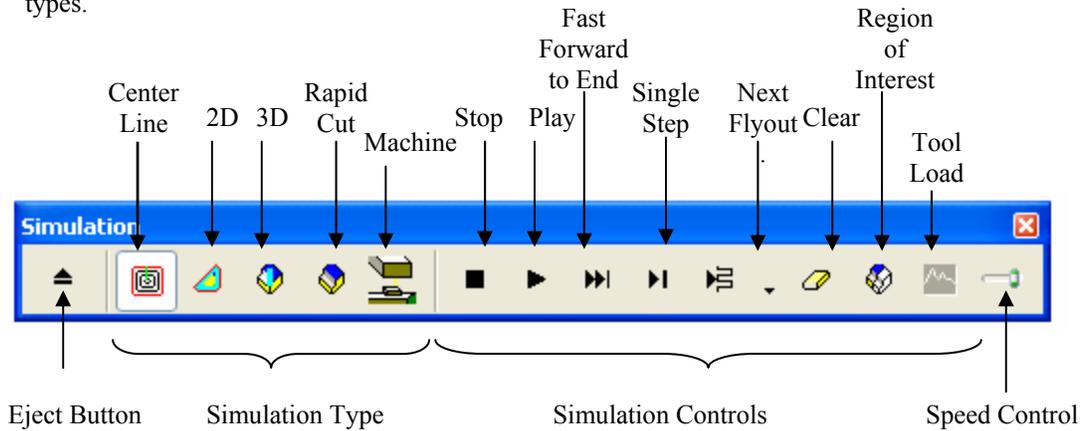
- Selecting the feature type
- Optionally entering pattern dimensions
- Positioning the feature
- Specifying the manufacturing strategy
- Reviewing the automatically selected tools
- Reviewing the automatically calculated feeds and speeds
- Overriding the tooling and feed/speed values
- Changing manufacturing attributes

To modify a feature either double-click on a feature or select the feature and click the *Properties* button to open the Properties dialog box for a feature. This dialog box presents all the screens of the New Feature Wizard in a single dialog box. The individual screens are separated into different tabs of the dialog box.

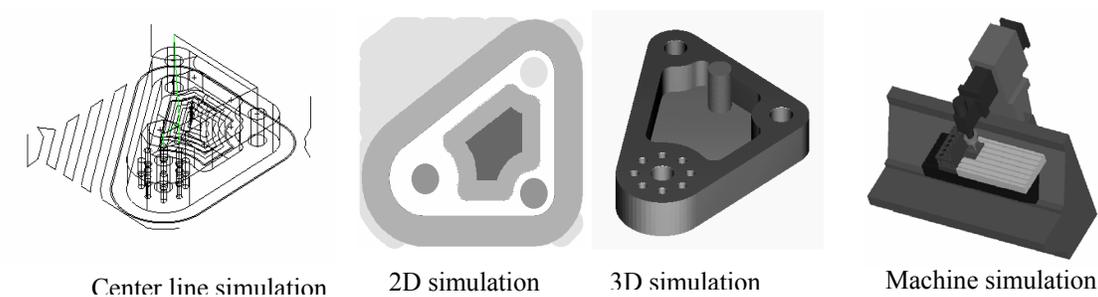


Step 5: Toolpaths

This step displays the *Simulation* toolbar. This toolbar has two sections, simulation types and simulation controls. Select the mode of simulation you would like to run from among the simulation types.



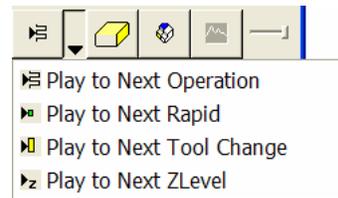
- *Centerline* simulation displays lines that represent the center of the tool.
- *2D* shows a color simulation from the top view. Each operation is displayed in a different color.
- *3D* shows a 3-dimensional solid shaded simulation.
- *Rapid cut* displays the final result without animating the tool. (FeatureMILL3D only)
- *Machine* simulation displays the entire machine. (Machine Simulation only)



Once you select a simulation type, use the simulation controls to control the simulation. Click the Play button to run the simulation. (After pressing the play button, this button turns into the Pause button. Click this button again to pause the simulation.) The Single Step button will display one toolpath move. The Fast Forward to End button, displays all the toolpaths for the part without animating the results. You can then view the toolpaths for each individual operation by clicking the Clear button and then clicking on an operation in the Operation List window.

The Play to Next Operation button will display one complete operation such as a spot drill or a pocket roughing operation. This button is actually a fly-out menu. By clicking on the triangle to the right of the button the following additional options are revealed:

- The Next Rapid button will simulate up to the next rapid move.
- The Next Tool Change button will simulate up to the next tool change.
- Play to Next Zlevel will display the next Z level of a Z level rough or finish operation.



Use the Erase button to erase the centerline toolpaths on the screen. The Region of Interest is used to limit the portion of the part that is simulated for 3D or Rapid Cut simulations. The Show tool load button indicates whether to display a graph of the tool load when the next 3D Simulation is performed. The Eject button removes the toolbar and the simulation from the screen. The speed control slider controls how fast simulations run. Move the slider to the right to speed up, or to the left to slow down.

If you see something in the simulation you want to change, you can override any of the tool choices, operation order, or the feeds and speeds if you wish. Continue to simulate and fine-tune the part until you have the settings exactly right.



Step 6: NC Code

You generate NC code in FeatureCAM using this step.



Optimize Feedrates

Use this option to even out the load on the tool by adjusting the feedrate of each NC block.



Display

This option displays the code in the Manufacturing Results window. You can also display the tooling lists and operations sheets by clicking the tabs located at the bottom of the Manufacturing Results window.



Save

The NC code is saved to disk using this command. You also have the option of saving additional documentation like tool lists, operations lists and part databases.



Remap Tooling

This option allows you to change the locations of the selected tools in the tool changer.



Customize Manufacturing

This step allows you to customize the tooling databases, feed/speed databases, and the default system parameters. These options allow you to fine tune FeatureCAM to your preferences.

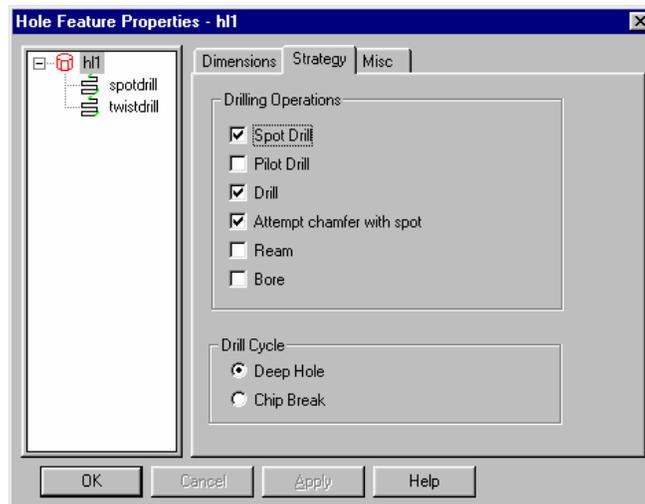


Manufacturing Preferences

Manufacturing preferences, or *attributes*, control the way a feature is manufactured. FeatureCAM has two different types of attributes, *default* and *feature*. Default attributes set the default settings for new features and are accessed by choosing the *Customize Mfg.* step and then selecting the *Establish my manufacturing preferences* option. Indicate your preference for settings including climb milling, step over values, canned cycles or ramping and these settings will be used for all parts that you create.

Feature attributes apply to specific features and override the default settings. For example you may set a default setting for climb milling, but override it for bi-directional roughing for a particular feature.

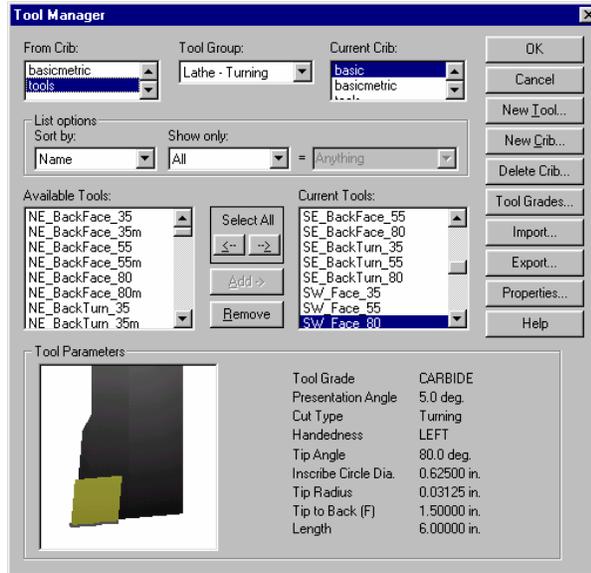
To set feature attributes, pick the feature graphically and click the Properties  button located at the bottom of the screen.





Tooling

FeatureCAM supports all the tool types in your shop including drills, taps, reamers, end mills, boring bars, and face mills. Each tool type is described by a series of dimensions.



Tools are grouped into Tool Cribs. FeatureCAM provides a comprehensive tooling database and creates a basic tool crib for Inch, or Millimeter tools, or both. FeatureCAM selects tools from the active tool crib to manufacture the part. You can pick the tool crib you want to use for each part or machine. You can also create custom tool cribs with the Tool Manager to reflect the tools owned by your shop or to exactly represent the tools preloaded in your tool changer.



Material Database

FeatureCAM comes standard with an integrated materials database. Feeds and speeds are stored in tables for each material. Each table contains the settings for different cutting operations. These tables can be customized to change the values or you can add tables for new materials.



Configure Post Processor

This option is available for customizing post processor settings or selecting a new post processor.

Optional Steps

 **Surfaces**

This step is available in FeatureMILL3D. This step launches the 3D surface wizard which steps you through creating surfaces from curves, primitive surfaces, surfaces from one surface, and surfaces from multiple surfaces.

 **Solids**

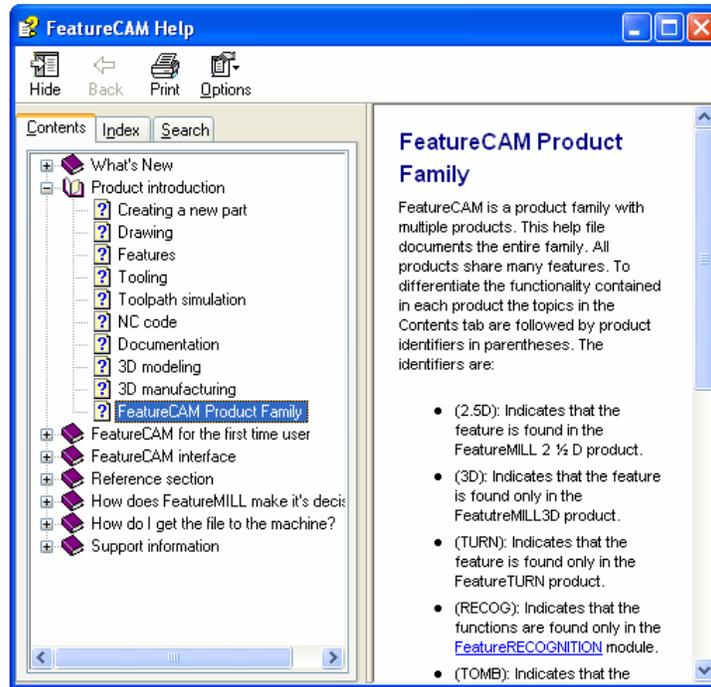
This step is available in the Solid Modeling option. The Solids step initiates the Solids wizard which helps you create solids using numerous techniques.

 **AFR**

This step is available in the FeatureRECOGNITION option. AFR stands for *automatic feature recognition*. This step automatically recognizes features from solid models.

On-line Help

FeatureCAM has an extensive on-line help system included with the program. All documentation other than the *Getting Started with FeatureCAM* guide is on-line. This documentation includes product introductions and reference manuals for all FeatureCAM products.



Select *FeatureCAM Help* from the Help menu to display on-line help.

Three different tabs are available.

- **Contents tab** provides an outline view of the help system. Double click on a book or a page to open it. From a page, click *Help Topics* to return to the outline view.
- **Index tab** allows you to access the help file using index topics. Type the topic name in the top box and click the *Display* button.
- **Search tab** provides search capabilities for all pages of the help file. Type the word you would like to find in the top box and then pick a topic in the bottom box. Click *Display* to view the page.

Help is a point and click interface to on-line information screens that are linked together. A link is any green underlined text or any portion of a diagram that turns the cursor into the pointer cursor. Click on a link to reveal a new screen.

Reference Section

The *Reference* section of the on-line help contains in-depth technical descriptions of FeatureCAM product features. It is organized in books that address major product topics. For example, the *Drawing*

book contains all the drawing tools contained in FeatureCAM; the *Making features* book contains a description of features broken down by individual products.

What's New Section

This section keeps you up to date on the latest additions to FeatureCAM.

Context Sensitive Help

While using FeatureCAM you can access the documentation for the current topic by either:

- Pressing F1 for help on your current activity.
- Clicking Help in the dialog boxes.
- Using the tool bar button, , for context sensitive help. Click this button and then click on a menu item, button or dialog box to receive more information.

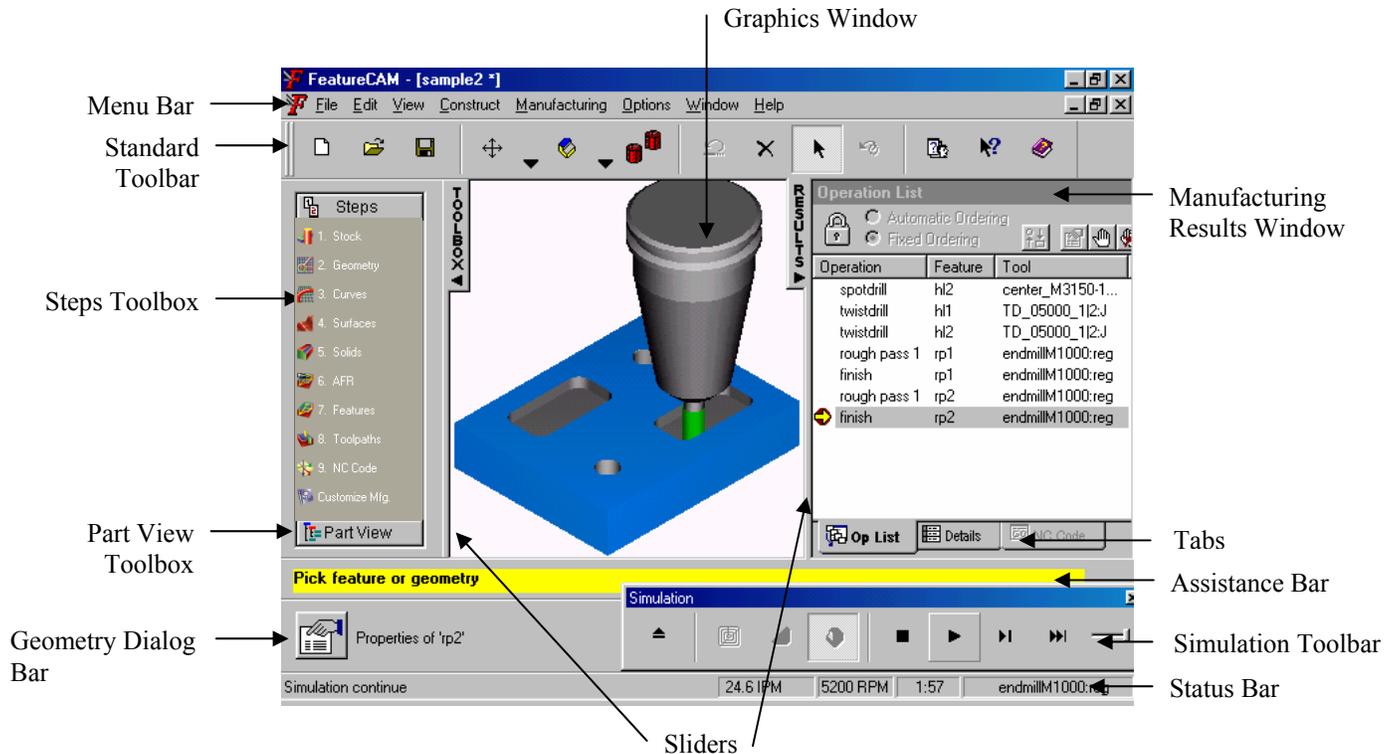
Written Documentation

Formatted versions of manuals can be accessed from the Help menu and viewed using Adobe Acrobat. These versions of the documentation are convenient for searching or printing for viewing off-line.

Chapter 3

Interface Tour

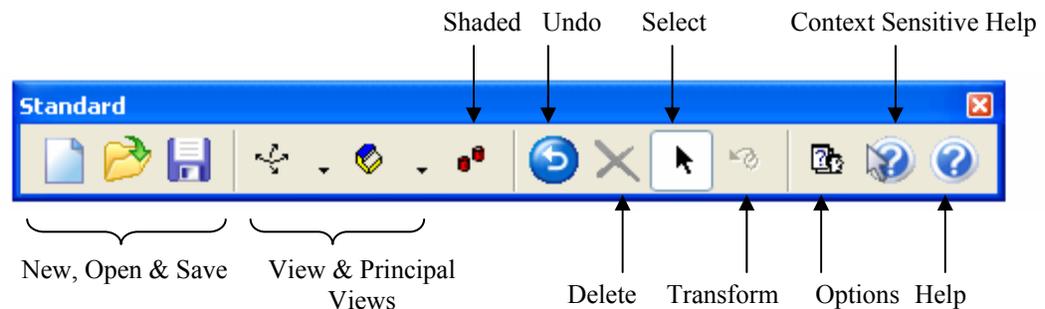
FeatureCAM Interface Components



The initial screen provides you with a clean interface for learning FeatureCAM. You can turn on the display of additional toolbars in the View menu under *Toolbars*. You can also control how much space the Graphics window uses compared to the Manufacturing results window by moving the Sliders from side to side.

Standard Toolbar

A getting started guide can't describe every button available in the new interface. But every button on a tool bar includes a tool tip. To view the tip, hold the cursor over the button until the tool tip appears. Tool tip describes the function the button activates. Besides the tool tip, the Assistance bar prompts you for the next step in the process. It senses your context and helps you design geometry and features. Refer to the Help system for detailed information.



File functions are controlled in the first group of three buttons, such as creating a new file, opening an existing file, or saving the current file.

View allows you to interactively change the view of the part. Select any of the options from the viewing fly-out. Your cursor shows the same icon as the viewing mode you selected. Viewing is performed interactively in FeatureCAM with the mouse. Rotation, panning and translation are possible. Interactive viewing is disabled while simulating toolpaths or in select or edit modes. See the *Fly-out Toolbars* section below.

Principal Views changes the view of a part to one of seven standard views. See the *Fly-out Toolbars* section below.

Shaded allows you to work in shaded mode for solid modeling.

Undo reverses the effect of the previous command. FeatureCAM remembers a long list of the commands that are performed, so you can select *Undo* multiple times to undo multiple commands.

Delete erases the object that is selected on the screen. The object is removed from the system.

Select puts you in Select mode. You can tell you are in select mode because the mouse appears as a standard arrow pointer in the part window. In select mode, you can click objects to select them.

Transform moves or copies the selected entity according to settings you make in the dialog box.

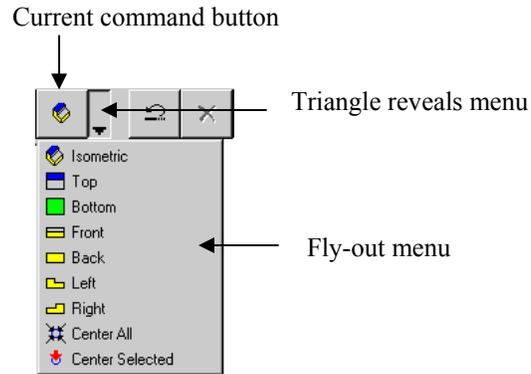
Context sensitive help accesses on-line help. Click the button, then select the button, field, dialog box or menu item you have questions about.

Options provides access to all the options settings in FeatureCAM.

Help brings up on-line help.

Fly-out Toolbars

A number of buttons in FeatureCAM toolbars are fly-out toolbars. Any button with a small black triangle in the lower right corner is a fly-out toolbar. By clicking on the triangle the menu is revealed. Click again to select a specific option. The last option you picked from the fly-out menu becomes the current command and is displayed as a button in the toolbar. To select this option again, just click on the button in the toolbar.



Menu Bar

The menu bar contains all the menus of the system. These menus contain all the commands available in FeatureCAM.

Steps Toolbox

The *Steps* toolbox contains an ordered list of steps for creating part programs. Each step is a wizard that presents a series of dialog boxes for each process. They are listed in the order in which you will use them during the process of creating a part program.

Graphics Window

The left-hand window is the graphics window. This window shows the graphical representation of the part and the graphical toolpath simulations.

Manufacturing Results Window

The right-hand window is the Manufacturing Results window. This window contains the

automatically generated documentation including tooling lists, setup sheets and the NC part programs. Selecting one of the tabs at the bottom of the window changes the content of this window.

Geometry Dialog Bar

The geometry dialog box is located under the graphics window and manufacturing results window. The point locations and parameters for geometry creation are entered here.

Assistance Bar

The Assistance bar displays step-by-step instructions for the current command.

Status Bar

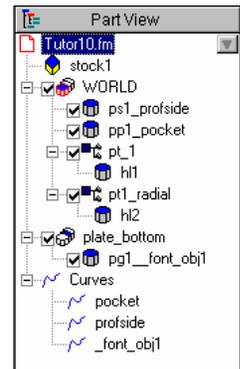
The Status bar shows your current drawing units, tool crib, and post processor settings as well as your keyboard status and information about the simulation when you run one.

The Status bar is not a toolbar, nor can it be docked. It is the bottom part of the FeatureCAM window and displays information about the computer, the state of the program, or information about what you are doing. You can toggle it on or off in the View menu under Toolbars.

Part View Toolbox

The Part View toolbox provides a hierarchical view of the part. Each setup is listed with each of the feature of the setup listed underneath. From this view you can select, show, hide or modify features. To navigate the part view:

1. If an object has a "+" in its left margin, click it to expand the view to reveal objects that are subordinate. Click on the "-" to collapse the view and hide the subordinate objects. Double clicking on the object also performs the same function.
2. Clicking on object in the tree view will select the object in the graphics window. This selection method is good for distinguishing between overlapping objects in the graphics window. Make sure you click on the name and not the check box. See item 4 below.
3. Clicking on an object also reveals the  button for that feature. Clicking on this button brings up menu of actions you can perform on the object. The actions available are dependent on the object type. Right clicking on the object in the tree view also reveals this menu.



4. Features have a checkbox. If you uncheck the object it will be excluded from the next toolpath generation.
5. By dragging a feature or setup, you can reorder these items.
6. Double-clicking on a feature displays its Properties dialog box for editing.

All objects are represented in the part view with the exception of geometry.

 Part icon represents the entire part. The part file name is also displayed in the tree view.

 Stock icon represents the part stock.

 Setup icon represents a setup. The features of a setup are listed underneath a setup. If you click on a setup, it changes the current setup and current UCS.

 Feature icon represents features. It can be a 2.5D, 3D or turn feature. The checkbox next to the feature can be used to exclude it from toolpath generation. The feature is not deleted; it is just ignored for toolpath generation.

 Curve icon represents a curve in the part model.

 Surface icon represents surfaces. Surface milling features are classified as features, not surfaces.

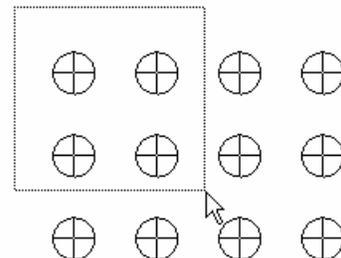
 Solid icon represents a solid. Individual design features are listed underneath.

 Turret icon represents a turret on a lathe. Selecting a specific turret displays the NC code for that turret.

Selecting Graphical Objects

Many commands require you to select object in the Graphics window. To select object you can:

1. Click the left mouse button on an object. This selects that object (by turning it red) and deselects any other object.
2. Hold down the SHIFT key and click the left mouse button on an object. This adds that object to the selected set of objects. This method allows you to select more than one object.
3. Click the left mouse button and drag the mouse before letting up on the button. This method is called *box select*. As you drag the mouse, a box is displayed in the Graphics window. All objects that are completely enclosed in the box are selected. If you hold down the SHIFT key while selecting, the objects are added to the selected set of objects.

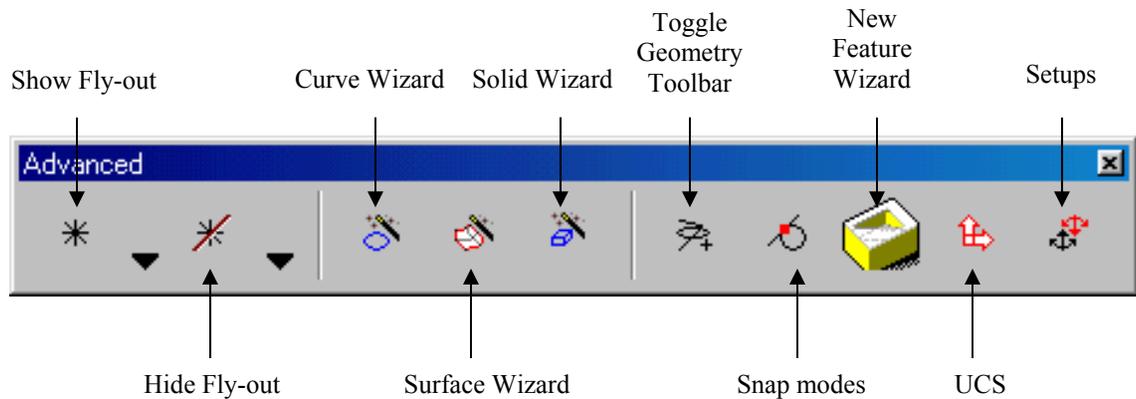


Note: you must be in Select mode (by clicking the Select  button) to perform a selection.

Optional Toolbars

While the Steps toolbox and the standard toolbar are all that is necessary to run FeatureCAM, there are additional toolbars that can be displayed by selecting the *Toolbars* option from the *View* menu.

Advanced Toolbar



Show Fly-out is a fly-out menu for displaying different categories of objects .

Hide Fly-out is a fly-out menu for hiding different categories of objects. This does not delete the objects. They are only removed from the display. These objects can be re-shown using commands in the Show Fly-out.

Curve Wizard provides a centralized interface for curve commands. The commands available in this wizard are also available in the Curves and Surfaces toolbar.

Surface Wizard presents step-by-step instructions for 3D surface construction. The commands available in this wizard are also available in the Curves and Surfaces toolbar.

Solid Wizard presents solid modeling commands available in the Solid Modeling option.

Toggle Geometry Toolbar toggles the display of the Geometry toolbar. This toolbar contains fly-outs for the creation of points, lines, arcs, fillets and dimensions.

Snap Modes displays the Snap Modes dialog. Use this dialog to set the locations for point snapping.

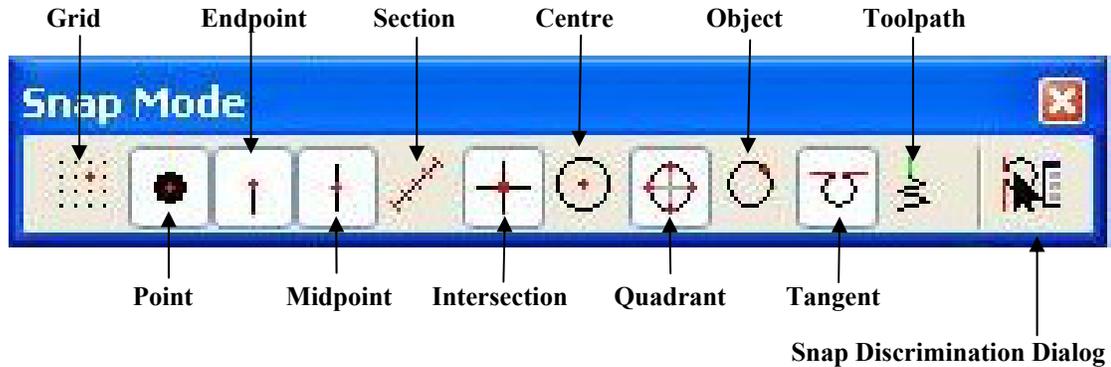
New Feature Wizard steps you through the process of creating part features.

UCS opens the User Coordinate System dialog box for creating or changing existing UCSs.

Setup brings up the Setup dialog box for creating or editing part program zero.

Snap Mode Toolbar and Dialog Box

Entering points by coordinates is exact but not always convenient or feasible. Picking points with the mouse is not always precise enough. The bridge between the methods of point selection is *snaps*. Snapping is a CAD concept that helps you position lines, points or shapes as you construct the geometry of a part. The Snap Mode bar or the Snap Mode Dialog box controls where the cursor snaps. What you have snapped to is displayed in the status bar.



Grid displays a grid and enables snapping to the grid. Selecting Snapping Grids from the Options menu controls the spacing of the grid. **Grid** snaps to a point on a coordinate system that is laid out on the stock. The grid size can be modified from the Options menu.

Point snaps to a point object.

Endpoint and **midpoint** snap to the ends and middle points of finite lines and arcs. Endpoint also applies to the corners of the stock.

Section snaps equal intervals of a finite line. The number of sections is controlled by the *Sections* parameter of the Snapping Grids dialog box.

Intersection snaps to the intersection of lines, arcs and circles.

Center snaps to circle centers. This setting also controls the display of circle and arc center points.

Quadrant snaps to the four points on a circle corresponding to 0° , 90° , 180° and 270° .

Tangent snaps the point so that the object you are creating will be tangent to the object you snapped to.

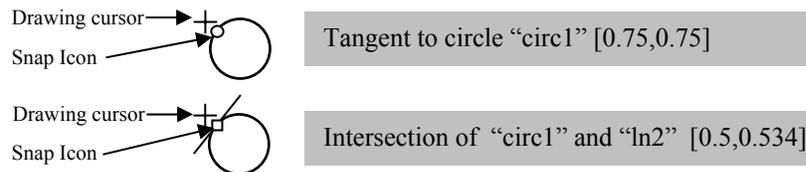
Object snaps to a point on another object. This includes lines of surfaces and is convenient for snapping points to locations on a surface.

Toolpath snaps to points on a centerline toolpath simulation.

Snap discrimination dialog displays a dialog box if the point you select could snap to more than one location.

Snapping Feedback

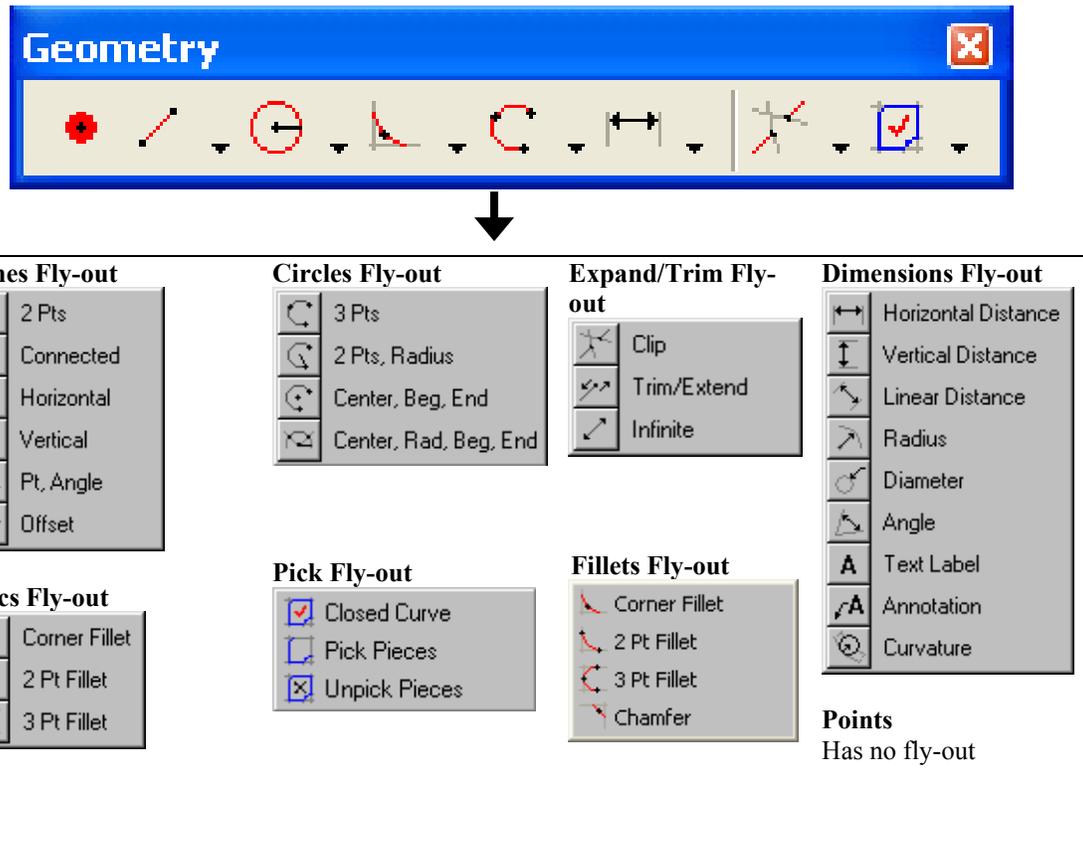
As you snap the cursor changes shape. A small snap icon shadows the cursor around the graphics window. The snap icon jumps to locations near the cursor. It also changes shape in relationship to what it snaps to. When snapping to a circle center, endpoint, midpoint, quadrant or section, the icon is a small box. When snapping tangent to a circle or arc, the icon is a small circle that can slide along the circle or arc while maintaining tangency. Snapping to a grid point, the icon is a small plus sign. When snapping to an object, the icon is a large dot. The following figure shows the cursors and icons of the Graphics window along with the messages displayed in the status bar.



You can have more than one active snap mode at a time. With snaps, it's easy to create any geometry tangent to circles or arcs, pick the exact center of circles and so on.

Geometry Toolbar

The Geometry Toolbar has a series of fly-out toolbars shown below.



Steps Toolbar

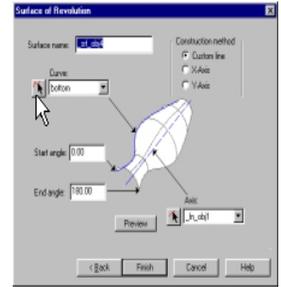


The Steps Toolbox is also available as a toolbar. It is useful to display this toolbar if you want to constantly view the Part View while still having the commands of the Steps Toolbox available. See Chapter 2 for a description of these buttons.

Warping Dialog Boxes

During graphical selections, certain dialog boxes will warp out of the way so that the dialog box does not cover the Graphics window. If you click a button like the Pick curve  button, the dialog box warps into a single small title bar as shown below.

The cursor changes into a Pick cursor . Select the object on the screen and the dialog will reappear. To cancel the pick, click on the red “X” in the small title bar. To disable dialog warping, uncheck *Warp Dialogs* in the *Options* menu.



Customizing Toolbars

The appearance and contents of the toolbars can be customized by selecting *Toolbars* from the *View* menu. The *Customize toolbars* dialog box is displayed.

Toolbars Tab

This dialog box has three different purposes.

Displaying/hiding toolbars - Check the checkbox in front a toolbar’s name to display it. Uncheck to hide it.

Changing the display of toolbars. - The appearance of the toolbars can be either flat or 3D. The size can be either large or small. Large buttons are recommended for low screen resolutions or touch screens.

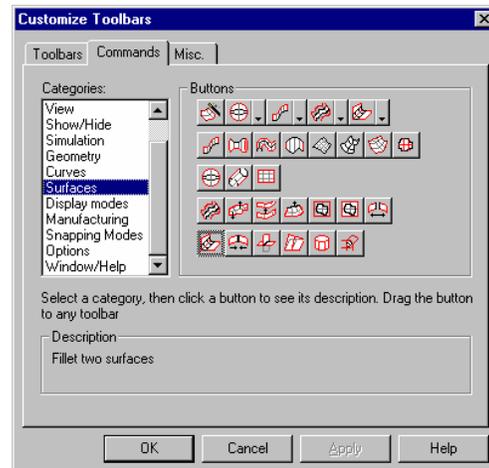
Creating/deleting toolbars - The *New* button will create a new toolbar. The *Delete* button will delete the toolbar that is selected in the list. You can only delete toolbars that you create. The *Reset selected* button will return the selected toolbar to its original state. The *Default toolbars* button will return all the toolbars to their original state and delete any user-defined toolbars.

Commands Tab and Misc. Tab

Clicking on the Commands tab puts the interface into a mode that allows you to rearrange the buttons of the interface. You can drag buttons from one toolbar to another or drag a button from a toolbar into the FeatureCAM window to remove it from the toolbar.

By clicking on one of the categories, buttons of that type are displayed. Click on a button to receive a description. You can drag one of these buttons onto any toolbar.

The Misc. tab controls the appearance of the Steps toolbox and the display of the status bar and assistance bars.



Display mode bar and shading mode

By default, everything that you create in FeatureCAM is displayed as a line drawing in the graphics window. Objects can also be viewed as a 2D cross section, 3D shaded or 3D hidden line drawings. See the chart below for the display modes that apply to specific object types and how to activate each type of graphics.

Lines	Line drawings apply to points, geometry, dimensions, curves, milling features, turning features, surfaces, solids, stock. It is the default display mode.
2D view	Applies to stock for a turning setup and turning features. This mode is toggled by clicking the 2D turned profiles  button in the display mode toolbar.
Shaded	<ul style="list-style-type: none"> 3D shading applies to milling features, turning features, surfaces, solids. There are a number of ways to shade and unshade objects. Toggle between line drawings and shaded graphics by clicking the Shade  button in the Standard toolbar. Shade specific objects on the screen by selecting the object and clicking the Shade selected  button in the Display mode toolbar. This is the only way to shade features. Click the Unshade selected  button to return the selected object to lines or click the Unshade all  button to return all objects to line drawings. These buttons are in the Display mode toolbar.

Hidden line	Hidden line graphics only apply to solids so it is best to hide all other objects and display only solids while in this mode. Toggle the Hidden line  button in the Display mode toolbar to clip and remove the hidden lines of a solid.
-------------	---

The following figures show a solid displayed as lines, hidden lines and shaded.



Display mode toolbar

The buttons of Display mode toolbar, control the current display mode.

The Display mode toolbar has the following buttons to help you visualize and work with surfaces in the model.



Shade selected shades the selected entities. This gives you a solid like view of the surfaces.



Unshade selected removes shading from the selected entities.



Unshade all unshades everything that is displayed.



Hide shaded isoline is a toggle switch to hide or show the lines of a surface or solid while it is shaded.



Hidden line mode will display all solids as a hidden line drawing. Note that only lines within the solid are hidden. It is best to hide everything and show only the solids to use this mode. Unclick the button to turn off the mode.



Show normals displays the normals for selected surfaces in the model.



Unshow normals hides the normals for selected surfaces in the model.

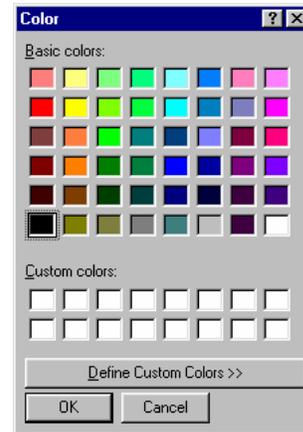


Turning 2D view toggles the view of a turning stage between 2D or 3D line drawing view.

Colors

Different colors are different for each object type. For example geometry is black and objects you select turn red. You can change the default color for object types by selecting *Coloring* from the *Options* menu and then selecting *Default colors*. The *Color* menu comes up. Click on the new color and click *OK*.

You can also change the color of any object, regardless of the object type, by selecting *Coloring* from the *Options* menu and then selecting *Change selected*.



Chapter 4

Making Your First 2 ½ D Part

NOTE: You must have FeatureMILL2.5D to perform this tutorial.

This chapter is a step-by-step tutorial for building a simple part and introducing you to:

- Creating features
- 3D viewing
- Generating toolpaths
- 3D toolpath simulation
- Automatic tool selection, feed/speed calculation
- Dynamically generated setup documentation

Stock Preparation

It is assumed that you have already prepared your stock. If you want to use FeatureCAM for this task, you will want to use the Face feature to face the top of the part and use a Side feature with a line as the profile to square an edge.

The First Time

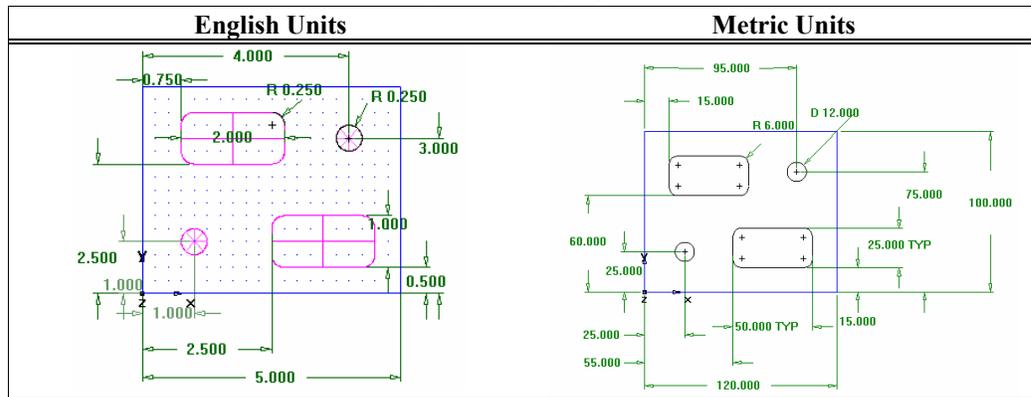
1. Begin FeatureCAM by selecting *FeatureCAM* from the *Start* menu.

The first time you run FeatureCAM, the tool database is initialized. You'll be prompted for some information concerning your tool preferences and then the tool information is created based on your answers.

Create a Part File

1. If the New Part Document Wizard is displayed, click *New File*, and click *Next*. Click *Milling Setup* and click *Inch* or *Millimeter* depending on your preference and click *Finish*.
2. If the New Part Document is not displayed, click *New Part Document*  button. Select *Milling Setup* and click *Inch* or *Millimeter* depending on your preference and click *OK*.
3. In the Dimensions dialog box, enter 1 (25mm) for thickness, 4(100mm) for width, 5(120mm) for length and click *Finish*. Next time you use FeatureMILL it will remember the stock dimensions that you entered last.
4. Click *OK* in the Stock Properties dialog box

For reference, here is a dimensioned drawing of the sample part.



4. Click the *Features* step  from the Steps toolbar.
5. Select *Hole* and click *Next*.
6. Enter 0.5 (12mm) for the *Diameter* and click *Next*.
7. Enter 1.0 (25mm) for the *X* location of the hole center and 1.0 (25mm) for the *Y* location. Click *Next*.

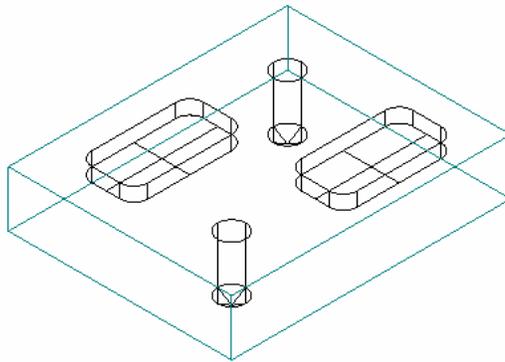
The *Strategies* page is presented. This page provides control over the types of operations that will be used to cut the feature. The default operations for a hole are to spot drill and drill the hole. If the hole had a chamfer, the default would be to cut the chamfer with the spot drill operation.

8. Accept the default strategy settings by clicking *Next*.

The *Operations* page shows a summary of the operations that will be created to cut the feature. The names of the tools that have been automatically selected and the feeds and speeds are also displayed.

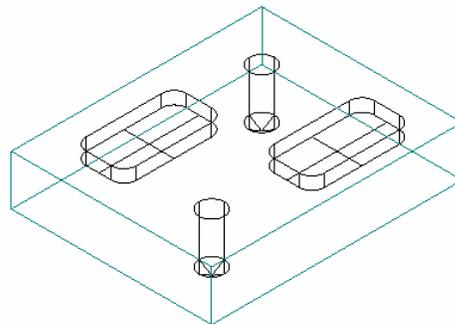
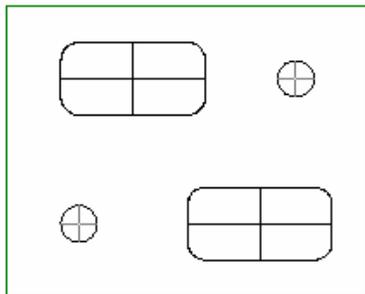
9. Click on the triangle next to the *Finish* button and select the *Finish* option.

10. Create a rectangular pocket feature by clicking the *Features* step  again from the Steps toolbar.
11. Select *Rectangular Pocket* and click *Next*.
12. Accept the default dimensions by clicking *Next*.
13. Enter 0.75 (15mm) for *X*, 2.5 (60mm) for *Y* and 0 for *Z*. Click *Next*.
14. The *Strategies* page indicates that rough and finish operations will be created. Click *Finish* to accept.
15. Use the *Features* step  to create a second 0.5 (12mm) diameter hole at *X*=4 (95mm) and *Y*=3 (75mm).
16. Use the *Features* step  to create another pocket just like the first, except positioned at *X*=2.5 (55mm), *Y*=0.5 (15mm).



Viewing

1. So far you have been viewing the part from the top view. Click the *Isometric* view button  from the standard toolbar to get a 3-dimensional view.



Order of Manufacturing Operations

The Op. list tab on the right side of the window shows all of the operations that the features are translated into. If you notice that there's a yellow warning sign  next to an operation, this is FeatureCAM's way of letting you know that there might be a problem with that operation. You can ignore the warnings for now.

Follow these steps to see how to control the order of manufacturing in FeatureMILL.

1. Notice on the top of the Op. list tab window the *Automatic Ordering* radio button is selected. This means that automatic rules are being applied to order the operations. Click the *Ordering options*



button.

2. Make sure that *Minimize tool changes* is the only one checked and click *OK*.
3. Click on the *Steps* toolbox and click the *Toolpaths* step . Click the 3D button and click the play button. If the *Ordering* dialog box appears click *OK*. You'll notice in the simulation that it performs all the spot-drills, then the drills, and then the rough and finish milling for the pockets.
4. Click the *Stop*  button when simulation is complete to exit simulation mode.
5. Click the *Ordering options* button again. Check *Do finish cuts last*, uncheck everything else and click *OK*. Notice that the order of operations has changed in the Operation List.
6. Click the *Play* button again. You'll notice in the simulation that the finish cuts for the two pockets now are cut last.
7. Click the *Stop*  button when simulation is complete.
8. Click the *Ordering options* button again. Uncheck everything and click *OK*. Now the operations will have the same order as the features in the Part View.
9. Click on the *Part View* tab located above the *Steps* toolbox. This view shows you a list of all the setups and features you have created. See *Chapter 3* for details on the part view.
10. Click *rect_pock2*, the last name in the Part View list, and drag it up ahead of *hole2*.
11. Click the *Play* button again. The second pocket is now cut as the third feature.
12. Click the *Stop*  button when simulation is complete.

So far the order of the operations is determined automatically by a set of rules. You can also specify an exact ordering of operations.

13. Click the *Manual Ordering* radio button below the *Automatic Ordering* radio button. In the *Fixed Operation Ordering* dialog box that appears click *OK*.

14. Select an operation from the list and drag it either up or down.
15. Click the *Play* button again. You'll notice that the simulation now performs the operations in the order that you had specified.
16. Click the *Stop*  button when simulation is complete.
17. Click the *Automatic Ordering* radio button and click OK
18. Click the *Eject* button to erase the simulation and remove the Simulation toolbar.

Part Documentation

Not only did the simulation model the manufacturing of the part, it also generated complete tool and operations lists. The tools selected are based on the tool cribs you design so you're always sure to simulate what you can produce with the tools at hand. The information is already computerized and can be printed for use as an operator's checklist.

1. Click on the *Details* tab in the *Results* window. The operation sheet looks similar to this:

Op: 1 hole1 (spotdrill)
Op: 2 hole1 (drill)
Op: 3 rect_pock1 (rough1)
Op: 4 rect_pock1 (rough2)
Op: 5 rect_pock1 (finish)
Op: 6 rect_pock2 (rough1)
Op: 7 rect_pock2 (rough2)
Op: 8 rect_pock2 (finish)
Op: 9 hole2 (spotdrill)
Op: 10 hole2 (drill)

You can review this list using the scroll bars, or print it by selecting Print from the File menu. Set the checkboxes for what you want to print and click *OK*

2. Click on the *Tool List* radio button at the top of the *Details* tab to show the Tooling list. The Tool list contains all of the tools used to create the part based on the crib you have selected. The Tool list looks similar to this:

Crib: basic
Summary:
Slot 1: center_5 D 0.1875 in.
Slot 2: TD_05000_1|2:J D 0.5000 in.
Slot 3: endmill0500:reg D 0.5000 in.
Slot 4: endmill0375:reg D 0.3750 in.

Controlling the Automation

1. Click on the Part View and select *hole1* from the list. Click the *Properties*  button at the bottom of the screen.
2. Click the *Strategy* tab.
3. Uncheck the Spot Drill checkbox and click OK.
4. Click the *Toolpaths* step . Click *3D*  and click the play button. Notice there is no spot drilling for that hole this time. If you look through the operations list, notice there is now only one spotdrill operation listed. FeatureMILL can optimize the part manufacturing process for you, but you control the level of automatic optimization you want.
5. Click *Eject*.

NC Code

The purpose of FeatureMILL is to generate NC code to manufacture parts with CNC machines. After you have simulated the part, you can generate NC code. Before you can generate NC code for a part, you have to run a simulation to calculate toolpaths. If you are starting this part of the tutorial without having just run the simulation described above, simulate your part now.

3. Click the *NC Code* step  in the Steps toolbar.
4. Click the *Display the NC code*  button in the dialog box to generate the NC code.

NOTE: If you are running an evaluation copy of FeatureMILL, a dongle, a device that attaches to the computer printer port, is necessary to generate NC code, or save or export files and is not available in the demonstration version. When you purchase FeatureMILL, a dongle is provided to you.

Tool Mapping

1. Click the *NC Code* step  in the Steps toolbar. Click the *Re-map the tools* button  from the dialog box. A dialog box is displayed shows that the tools are in the following order:

TD_0500_1 2:J	1
Endmill0500:reg	2
Endmill0375:reg	3
Center_5	4

2. In this dialog box you can modify the location of the tools in the tool changer. To move the center

drill to the 5th position in the tool changer, click on *Center_5* in the table.

3. Change the *Tool* from a 4 to a 5 in the *Tool number* field at the bottom of the dialog box and click *Set*.

Note that you cannot change the number directly in the table.

4. Click *OK*.

Changing Post Processors

1. Follow the remaining steps to change your post processor and save the NC code.
2. From the *Manufacturing* menu, select *Post Process*.
3. Click *Browse* to view the post processors.

The default directory is c:\Program Files\FeatureCAM\M-lbry\Inch for English units. If you need a metric post, browse up one directory level to M-lbry. Then browse the Metric directory.

4. Select your processor and click *Open*. Click *OK* in the Post Options dialog box. Your processor is now selected. Run a simulation of the part to regenerate the NC code.

NOTE: Clicking *Close* will not change your post processor.

Saving NC Code

1. Click the *NC Code* step  in the Steps toolbar. Click the *Save NC* button  from the dialog box.
2. Accept the default filename and directory and click *OK*.

You are ready to machine the part.

Saving the FeatureMILL Part

1. Select the *Save* button  from the Standard toolbar.
2. Select the directory and then type your part name.

The Next Step

You're done with this tutorial and have seen the basic functions of FeatureMILL. The next section expands on this part by building a curve around the features. The curve is created with the layout and drawing tools.

Chapter 5

Features From Curves

NOTE: You must have FeatureMILL to perform this tutorial. It is recommended that you complete Chapter 4 before starting this tutorial.

This tutorial builds on the one you just completed and introduces you to:

1. Creating geometry (lines, arcs, fillets, etc.)
2. Creating irregularly-shaped features
3. Contouring around a part and adjusting where the tool plunges
4. Open your sample part from the last chapter or if you skipped the earlier tutorial, open the sample2.fm file or the mtrc_gsg.fm file for metric users. If you installed in the default directory, these files should be in C:\Program Files\FeatureCAM\Examples\Tutorials\Features from Curves.

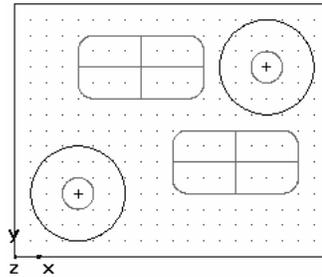
Setting the Snaps

1. Select *Snapping Modes* from the *Options* menu.
2. Check if the following buttons are enabled, if not, click on it to enable them and turn the others off: *Grid points*, *End points*, *Intersections*, *Circle centers*, *Tangent to objects*. A white box will surround a button when it's turned on.
3. Click *OK*.

Building Circles

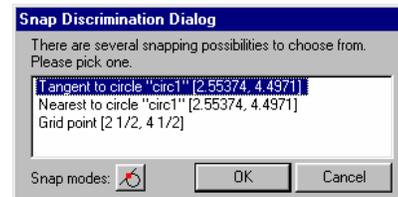
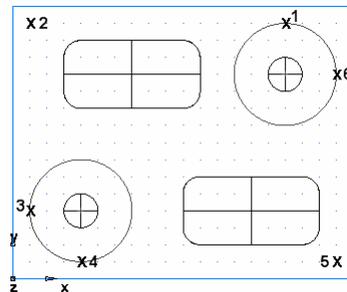
1. The *Principal Views* button  in the Standard toolbar is a fly-out toolbar. Click the triangle next to the button and then click on the *Top* button in the fly-out toolbar. It is usually easier to draw geometry in this view. (For more information about the fly-out toolbar options, refer to Chapter 3, Interface Tour.)
2. Select the *Geometry* step  from the Steps Toolbox. Check the *Create more than one* checkbox at the bottom of the dialog box and click the *Circle from Rad, Center*  button. You can move your mouse over an icon to read its label.

3. Set R in the dialog bar (below the graphics window) to 0.75 inch (20mm).
4. Click near the center of one hole. The snap jumps to the grid at the hole's center. If you miss the hole center, click Undo (or select Undo in the Edit menu) and try again.
5. Repeat this action for the other hole. Your part looks similar to the figure below.

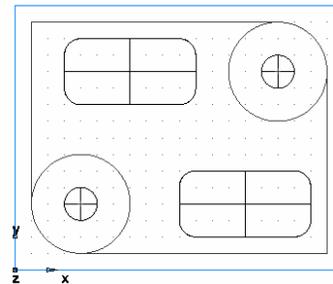


Drawing Lines

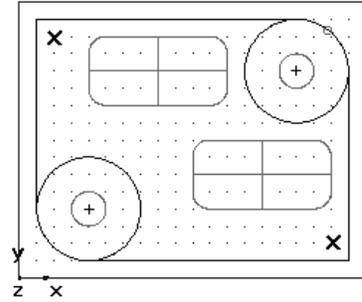
1. Select the *Geometry* step  from the Steps Toolbox. Check the *Create more than one* checkbox at the bottom of the dialog box and click the *Line from 2 pts* button .
2. For point 1 in the diagram below, click the top of the right circle. If the Snap Selection dialog box appears, select *Tangent to circle* and click *OK*.



3. For the second point click on the upper left grid point, point 2, as shown with Xs above.
4. Continue building lines from point 2 to point 3, from point 4 to point 5, and from point 5 to point
5. The completed lines look like this:

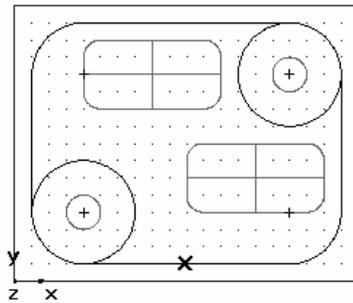


6. Select the *Geometry* step  from the Steps Toolbox. Check the *Create more than one* checkbox at the bottom of the dialog box and click the *Corner Fillet*  button.
7. Set *R* in the dialog bar to 0.75 inch (20mm)
8. Click the point marked near the upper left corner. The diagram on the right shows where to click to create the fillets. A corner fillet appears automatically trimming the lines to fit.
9. Click the point marked near the lower right corner. A corner fillet appears automatically trimming the lines to fit.



Chaining a Curve

1. Select the *Curves* step  from the Steps Toolbar.
2. Select *Chain into a closed boundary*  and click on the bottom line as shown.



FeatureMILL automatically chains a closed curve and chooses a smooth path so you don't have to chain all curves manually.

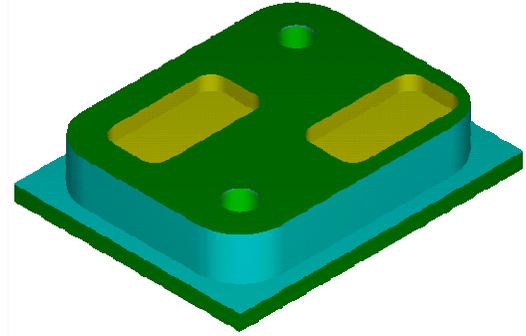
Feature from a Curve

1. With the curve still highlighted, click the *Features* step  from the Steps toolbox.
2. Select *Boss* and click *Next*.
3. The curve you selected is already selected in this dialog box. Keep on clicking *Next* until you get to the Dimensions page of the dialog.
4. Set the *Height* to 0.75 (20mm).
5. Click *Finish*.

Simulate the Part

You are now ready to simulate your part.

1. Click *Isometric*  from the Principal views fly-out.
5. Click the *Toolpaths* step  from the Steps toolbar and click the *3D simulation*  button and then the play button.
6. Click *OK* in the *Ordering* dialog box and the 3D simulation is displayed in the graphics window.
7. Click the *Eject*  button to clear the screen.



Modifying the Boss Feature to Contour Around the Part

If you watched the simulation carefully, you saw that the boss feature performed both a roughing and finishing operation. To simply contour around the outer boundary of the part, you'll need to eliminate the roughing operation. The operations that are initially created for a feature are specified on the *Strategies* page of the New Feature wizard, but you can easily modify the strategy for any existing feature.

1. Pick the Boss feature and click the *Properties* button at the bottom of the screen.
2. The *Properties* dialog box comes up. This dialog box contains the pages of the *New Feature* wizard as separate tabs.
3. Click on the *Strategy* tab. Notice that the *Rough* and *Finish* checkboxes are checked. Uncheck the *Rough* checkbox, and click *OK*.
4. Simulate the toolpaths again using the *Toolpaths* step .

Changing the Plunge Point

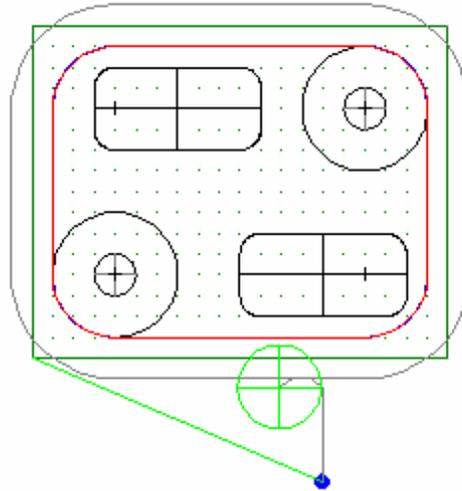
The tool ramped on and off the part at the location that you chained the curve. In this case, the tool is plunging on the stock. To change the plunge point for this operation:

1. Click on the triangle next to the *Principal Views* button  then click on the *Top* button in the fly-out toolbar
2. Pick the Boss feature and click the *Properties* button at the bottom of the screen.
3. The left-hand side of the dialog box contains the tree view. It shows the feature with its operations listed underneath. By clicking on an operation in the tree view, you can modify an operation.
4. Click on the *finish* operation in the tree view. The tabs of the dialog box change to represent information specific to the operation. Click on the *Milling* tab. This tab contains the various attributes for changing the toolpaths that are generated for the operation.

5. Click on the *Plunge point(s)* attribute.
6. Click the *Pick location*  button that appears at the bottom of the dialog box.
7. Click at a location below the part.

Note that if you pick a location that is too close to the part, the plunge point is ignored that the toolpaths will remain unchanged.

8. Click *OK*.
9. Click the *Toolpaths* step , click the centerline simulation  button and the play button to view the changes in the toolpath.



The Next Step

You have completed this Tutorial.

Chapter 6

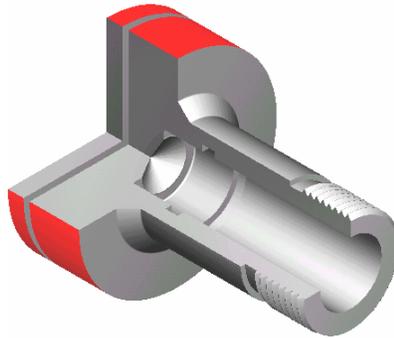
Introduction to Turning

NOTE: You must have FeatureTURN or FeatureTURN/MILL to perform these tutorials.

If you want to evaluate FeatureTURN:

1. Select *FeatureCAM* from the *Start* menu.
2. From the *File* menu, choose *Evaluation Options* and make sure *Turning* is checked.
3. Click *OK*.

The example is a simple turned part designed to introduce you to several of the drawing and machining tools available in FeatureTURN. The tutorial walks you through creating the part and introduces you to FeatureTURN's automation and ease of use.

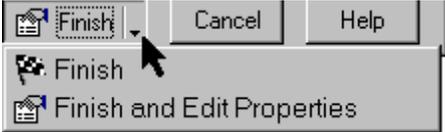


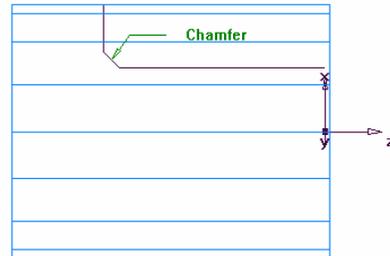
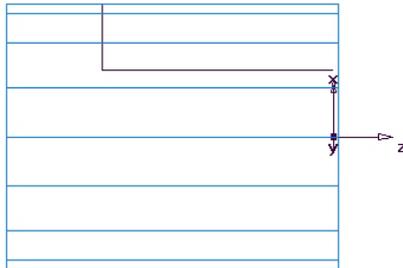
The First Time

1. Select *FeatureCAM* from the *Start* Menu.

The first time you run FeatureCAM, the material and tool database is initialized. You'll be prompted for some information concerning your tool preferences and then the tool information is created based on your answers.

Create a Part File

1. If the New Part Document Wizard is displayed, click *New File*, and click *Next*. Click *Turn/Mill* and click *Inch* or *Millimeter* depending on your preference and click *Finish*.
2. If the New Part Document Wizard is not displayed Click *New Part Document*  button. Click *Turn/Mill* and the *Inch* or *Millimeter* radio button depending on your preference. Click *OK*.
3. A Turning setup will default to a round stock. Once in stock properties set the *OD* to 4 (100mm), *Length* to 5 (125mm), and *ID* to 0.
4. Click on the triangle next to the Finish button and select the Finish and Edit properties and set *Z* to 0.0625 (1.5mm) and click *OK*.
 
5. Select *Turning input modes* from the *Options* menu. Make sure that *3D* is checked.
6. Select *Set tool crib* from the *Manufacturing* menu. Click on the *Tools* crib and click *OK*.
7. Click on the triangle of the *View* fly-out  on the Standard tool bar and choose *Center All* to bring your entire drawing into view.
8. Select the *Geometry* step  from the Steps Toolbox. Check the *Create more than one* checkbox at the bottom of the dialog box and click the *Line from 2 pts* button . On the bottom of the screen, type in these points to create two lines that define the OD profile: ($X=2(50mm)$, $Y=0$, $Z=-3.5(-88mm)$), ($X=1(25mm)$, $Y=0$, $Z=-3.5(-88mm)$). Press ENTER. Now you should see a line drawn. Create a second line with the values ($X=1(25mm)$, $Y=0$, $Z=-3.5(-88mm)$), ($X=1(25mm)$, $Y=0$, $Z=0$) and press ENTER.

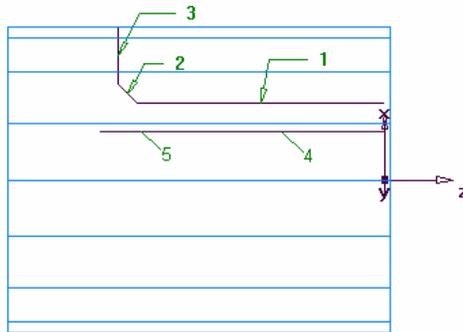


9. Select the *Geometry* step  from the Steps Toolbox and select *Chamfer*  from the list of *Fillet* commands. Enter *width=0.25(6mm)* and *height=0.25(6mm)* at the bottom of the screen and click your mouse close to the chamfer location as shown in the figure and the chamfer will snap into place. Click your mouse button to place the chamfer on your drawing. Notice how the chamfer automatically trims your lines.

Select the *Geometry* step  from the Steps Toolbox and click the Line from 2 pts button . Use the following values: ($X=0.625(16mm)$, $Y=0$, $Z=0$), ($X=0.625(16mm)$, $Y=0$, $Z=-3.75(-94mm)$). Press the ENTER key.

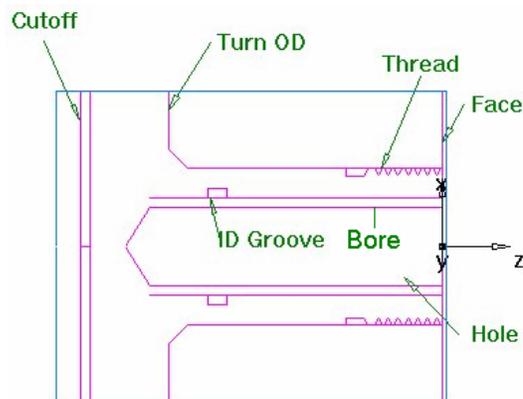
10. Select the *Curves* step  from the Steps Toolbar. Click the *Chain into open boundary* button . Click on locations 1,2,3 as shown in the figure below. Each line segment will change color when it is selected. Name this curve "Turn" and press the ENTER key.

11. Select the *Curves* step  from the Steps Toolbar. Click the *Chain into open boundary* button . Chain the Bore ID curve by clicking on locations 4 and 5. Label it "Bore" and press the ENTER key. Your drawing should look like the one below.



Making Features

The *2D Turned Profiles*  button is located on the display mode toolbar and will simplify the screen. Annotations have been added below to show the 7 features that make up the part.



1. Click *Features step*  from the Steps Toolbar.

Note: For Turn/Mill users only, an extra form will pop up asking the user to choose *Turning* or *Turn/Mill*. If this appears select *Turning*.

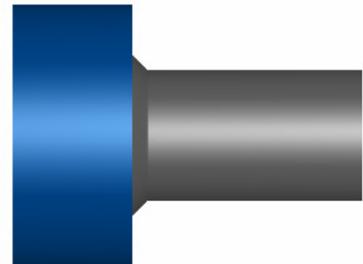
2. On the New Feature form, select *Turn* and click *Next*.
3. You will now pick the curve graphically. Click on the Pick Curve button . The dialog box will warp to reveal the Graphics window beneath. Click on the curve you named “Turn” earlier. Since this pick could be either the curve or the line you may see the Select dialog box. This box helps to clarify a pick. In this case, select “Turn” from the list and click *OK* and then click *Next* in the *New Feature – Curve* dialog box.
4. Accept the default strategy settings by clicking *Finish*.
5. Create a Facing Feature by clicking the *Features step*  from the Steps Toolbar.
6. Click *Face* and click *Next*.
7. Set *Outer Diameter* to 4 (100mm), *Inner Diameter* to 0, *Thickness* to 0.0625 (1.5mm). Note that a face feature automatically knows the *OD* and *ID* values from the stock properties. Click *Finish* to accept the default values for the rest of the dialog boxes.
8. Click the *Toolpaths step*  in the Steps Toolbox. The *Simulation* toolbar is displayed.



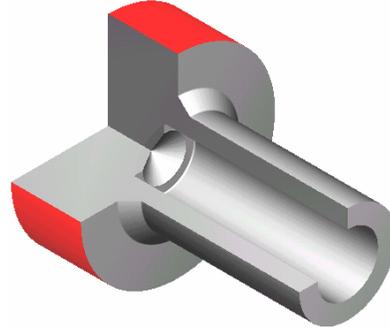
9. Click the *3D* button and the Play button. Click *OK* in the Ordering dialog if it appears. The following simulation is displayed. Click the *Eject* button to return to the drawing view.
10. Use the *Features step*  from the Steps Toolbar to create a hole. On the *Dimensions* page set the *Diameter* to 1.0 (24mm), *Depth* to 3.75 (94mm). Click *Next* and set *Z* to 0 and click *Finish*.
11. Create a bore feature by using the same process you used to create the turn feature. Use the curve you named “Bore”.
12. Click on the triangle of the *Principal views* fly-out



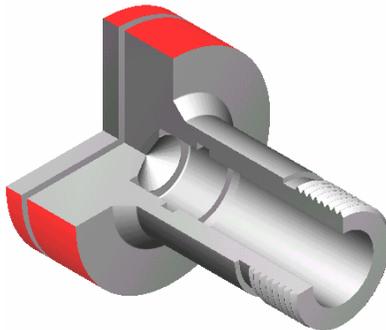
on the Standard tool bar and click on the *Isometric* button from the fly-out to see an isometric view of your part.



13. Click the *Toolpaths* button  in the Steps toolbox and view a 3D simulation by clicking the *3D* button and the *Play* button. Click the *Eject* button to clear the screen
14. Return to the *Principal Views* fly-out and select *Top* to return to the original view.
15. Click the *Features* step  and select the *Groove* radio button in the *From Dimensions* group. On the Dimensions page set *Location* to *ID*, *Orientation* to *X-axis*, *Diameter* to 1.25 (31mm), *Depth* to 0.125 (3mm), *Width* to 0.250(6mm), Leave the other settings at 0. Click *Next* and set *Z* to -3 (-75mm). Click *Finish*.
16. Use the *Features* step  and select the *Thread* radio button in the *From Dimensions* groups. On the Dimension page, check *Get the dimensions from a standard thread*, click *OD* and select the 2.0000-4.5UNC (M50-15 for metric) thread. Click *Next*.
17. On the *Dimensions* page set *Thread* to *Right Hand*, *Thread Length* to 1.0 (24mm). Click *Finish*.
18. Click the *Toolpaths* step  in the Steps toolbox and view a 2D simulation by clicking the *2D* button and the *Play* button. Click the *Eject* button to clear the screen.
19. Use the *Features* step  to make a *Cutoff* Feature. On the Dimensions page set *Diameter* to 4(100mm), *Inner Diameter* to 0, *Width* to 0.122 (3mm), Click *Next* and set *Z* to -4.5 (-112mm). Click *Finish*.



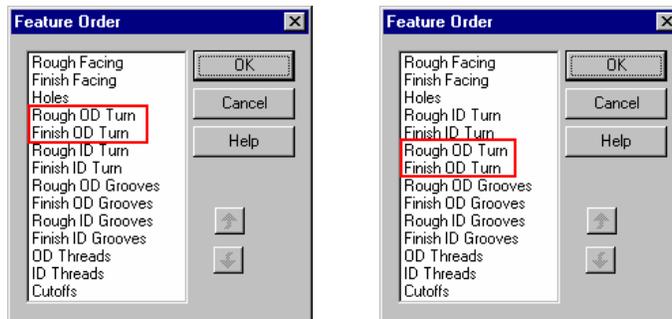
The sample part is complete. Watch a 2D simulation followed by a 3D simulation to view the finished part.



Ordering of Operations

You can control the ordering of operations by modifying the *Turn Operation Template*. This template lists the manufacturing order of the various turning features.

1. Click the *Ordering option*  button on the *Op. List* tab in the Results window.
2. The *Automatic ordering option* dialog box comes up. Click on the *Use template* radio button and click the *Edit template* button.
3. The default order is shown on the left. Click on *Rough OD Turn* and click the down arrow until it is located under the *Finish ID Turn* operation. Click on *Finish OD TURN* and click the down arrow until it is located under *Finish ID Turn*. Click *OK* twice.



4. Simulate the toolpaths and you'll notice that the OD roughing and finishing now happen after the hole is drilled.

Part Documentation

FeatureTURN will automatically generate a Manufacturing Operation sheet and a Tool list. The Manufacturing Operation sheet appears by default in the Results window. It contains information such as machining time, tools selected, feeds and speeds, and horsepower requirements for the various operations. A Tool list is also created that gives all the specification and slot assignments for the tools used to manufacture your part.

1. Click on the *Details* tab at the bottom of the Results window. Each operation of the process plan is listed in order.
2. Click on the *Tool List* radio button at the top of the *Details* tab. Now the tool list is presented in the Results window. This document provides a summary of all the tools required for the job as well as details of each tool.

You can print this documentation by selecting *Print* from the *File* menu.

NC Code

After you have simulated the part, you can generate NC code. FeatureTURN comes with many post processors and the ability to create custom post processors as well.

Before you can generate NC code for a part, you have to run a simulation to calculate toolpaths. If you are starting this part of the tutorial without having just run the simulation described above, simulate your part now.

1. Click the *NC code* step  in the Steps toolbar.
2. Click the *Display the NC code*  button in the dialog box to generate the NC code.

NOTE: If you are running an evaluation copy of FeatureTURN, a dongle, a device that attaches to the computer printer port, is necessary to generate NC code, or save or export files and is not available in the demonstration version. When you purchase FeatureTURN, a dongle is provided to you.

Changing Post Processors and Saving NC Code

1. If you are running FeatureTURN, default post processor is for a Bridgeport machine. Follow the remaining steps to change your post processor and save the NC code.
2. From the *Manufacturing* menu, select *Post Process*.
3. Click *Browse* to view the post processors.
4. The default directory is c:\Program Files\FeatureCAM\T-lbry\Inch for English units. If you need a metric post, browse up one directory level to M-lbry. Then browse the *Metric* directory.
5. Select your processor and click *Open*. Click *OK* in the Post Options dialog box. Your processor is now selected. Run a simulation of the part to regenerate the NC code.

NOTE: Clicking *Close* will not change your post processor.

6. Click the *NC code* step  in the Steps toolbar. Click the *Save NC* button  from the dialog box.
7. Accept the default filename and directory and click *OK*.

You are ready to machine the part.

Saving the FeatureTURN Part

1. Select the *Save* button  from the Standard toolbar.
2. Select the directory and then type your part name and click *OK*.

The Next Step

You have completed this tutorial.

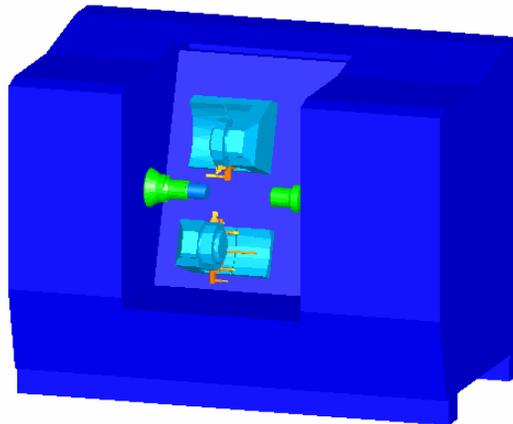
Chapter 7

Advanced Turning and Machine Simulation

NOTE: You must have FeatureTURN, Multi-turret Turning and Machine Simulation to perform this tutorial.

This chapter introduces you to:

- Creating subspindles
- Using Machine simulation
- Setting up a second turret
- Synchronizing operations



Getting Started

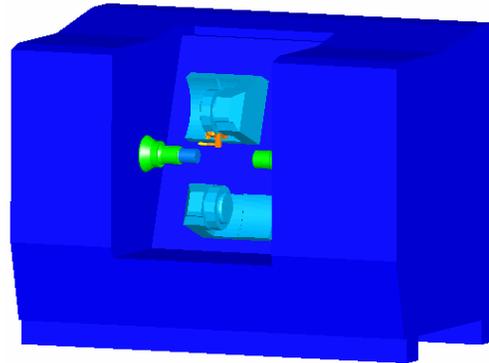
All files for this tutorial are located in \Examples\Tutorials\Multi-turret Turning\ where you

installed FeatureCAM. If you installed the program in the default directory, these files should be in C:\Program Files\FeatureCAM\Examples\Tutorials\Multi-turret Turning\.

1. Open the tutorial file *twinturret_tutorial.fm*

Setup Machine Simulation

1. Select *Toolbars* from the *View* menu. Make sure that *Advanced* option is checked and click *OK*.
Click the *Setups*  button from the *Advanced* toolbar.
2. Select *Setup1* from the drop-down list and click *Edit*. Keep clicking *Next* until you get to the *Setup - Simulation information* page.
3. Click the *Browse*  button and locate c:\Program Files\FeatureCAM\Examples\Tutorials\Multi-turret Turning\TURNING_INDEX_G200.md.
4. Select the file and click *Open*.
5. Click *Finish* and then click *Close* in the *Setups* dialog box.
6. From the *Manufacturing* menu, select *Post Process*.
7. Click *Browse* to view the post processors.
8. Go to the directory c:\Program Files\FeatureCAM\Examples\Tutorials\Multi-turret Turning\.
9. Select the *Moriseiki with Mitsubishi Mill-turn upper and lower example.CNC* processor and click *Open*.
10. Enter X = 18 and Z =0 for the *Tool Change Location*. Click *OK*. Your processor is now selected and the tool change location is set.
11. Click the *Toolpaths* step  in the *Steps* Toolbox to bring up the simulation toolbar.
12. Click the *Machine simulation*  button and then click the *Single Step*  button. Click *OK* if the *Ordering* dialog box appears. You should now see the machine displayed on your screen.
13. Click the right mouse button in the graphics window to bring up the *View* fly-out menu. Use the viewing options to get a good view of the machine.



14. Click the Play button. You should now see the machine parts moving and cutting the part.

You can use the options in the View fly-out menu during the simulation to get a better view without pausing the simulation.

15. Click the *Eject* button.

Add a Subspindle Feature

So far we have been dealing with a single spindle. Now we'll introduce a second spindle, or a subspindle, to our setup.

1. Click on the Part View above the Steps Toolbar.
2. Notice that we have a number of spindle features already created:
 - main_off
 - sub_off
 - sub_open_and_drop
 - sub_pos_in
 - sub_close
 - main_on_sync
 - sub_on_sync
3. Select *Setup1* from the list.
4. Select the *Features*  step from the Steps Toolbar.
5. If the New Feature – Type dialog box is displayed, select *Turning* and click *Next*.
6. Select *Subspindle* and click *Next*.
7. Select *Sub Spindle* and *Position the spindle* radio buttons and click *Next*.
8. Check the *Send the subspindle home* checkbox and click *Finish*. If the Sub Spindle Properties dialog box appears click *OK*.
9. Click on Part View and notice the new subspindle feature added to the end of the list under the label *Setup1*.
10. Select the new subspindle feature from the Part View list and right click. Select *Rename* and type in “sub_home” as the new name. Now all the subspindle moves are complete.

Add a Bar Feed Feature

1. Select *Setup1* from the Part View and select the *Features*  step from the Steps Toolbar.
2. If the New Feature – Type dialog box is displayed, select *Turning* and click *Next*.
3. Select *Bar Feed* and click *Next*.

4. Enter 3.0 for *Diameter* and 4.2 for *Feed Amount*. Click on the down arrow next to the Finish button and select the *Finish and Edit Properties* option.
5. Enter -4.2 for the *Z* location and click *OK*.

Run Subspindle Simulation

1. Click on Part View and check all the boxes that are unchecked in all the setups. All the checkboxes in the Part View should now be checked.
2. Click on the *Op List* tab at the bottom of the Results window.
3. Point your mouse over the Operations column header and click the right mouse button. This will bring up a fly-out menu.



4. Check the *Show Setup* option. This will display a new column in the Operation List called *Setup Name*. This column lets you know which spindle will be used to run an operation.

Notice that all the features listed under *Setup1* in the Part View have the setup name of *Main spindle* in the Operation List. These features will be cut on the main spindle of the machine. Features listed under *Setup2* will be cut on the subspindle.

5. Hold down the CTRL key and select the last two operations in the Operation List. While still holding down the CTRL key, drag the lines up and place them right below the *cutoff* operation.

6. Click the Toolpaths step  in the Steps Toolbox. Click the *Machine simulation*  button and click Play. You should now see the machine parts moving and cutting the part on both the main spindle and the subspindle.

Notice that the operations were simulated twice. This is done to show how production turning works. This feature will become more apparent at the end of this tutorial.

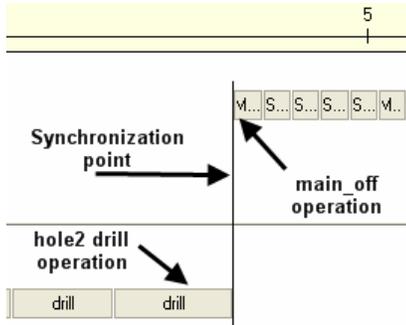
7. Click the *Eject* button.

Setup Second Turret

1. Click on the *Op List* tab in the Results window.
2. Point your mouse over the Operations column header and click the right mouse button. Check the *Show Turret* option. This will display the turret number to the left of the tool name.

Setup Name	Tool
Main spindle	P1 SW_Turn_80_RH
Main spindle	P1 SW_Turn_80_RH
Main spindle	* P1 TD_10000_1:T new
Main spindle	* P1 WN_B_Small_80_RH
3. Shift select all the operations in the Operations List that are on the Sub Spindle (select the first subspindle operation, hold down the SHIFT key and select the last operation).
4. Click the right mouse button and select the *Set opens to 'Lower turret'* option. Notice that now the turret number for those operations has changed from *P1* to *P2*. These operations will now run on the machine's second or lower turret.
5. Click the *Turrets* tab in the Results window. Here you can see the turret and spindle that each operation is associated with and the total time for cutting the part.
6. Click on the Timeline that appears at the top of the Turrets window and drag it left and right while holding down the left mouse button to make it shorter or longer.

Synchronize Operations

1. Select the last hole *drill* operation block in the Sub Spindle section (You may need to extend the timeline to view the operation).
2. Hold down the CTRL key and select on the *main_off* operation block in the Main spindle section (You can move your mouse over an operation block to display the name of the operation)
3. Click the right mouse button and select the option *Set sync for 'radial_pattern1.hole2.drill' before 'main_off.Main spindle off'*.
4. Notice that now the operations have been shifted and a new synchronization point is set. The *main_off* operation will now happen after the drill operation.
 
5. Select the *bore2 finish* operation in the Main spindle section.
6. Hold down the CTRL key and select the *bore1 rough pass 1* operation block in the Sub Spindle section.
7. Click the right mouse button and select the option *Set sync for 'bore2.finish' before 'bore1.rough*

pass 1'.

Now there should be a total of two synchronization points set.

Run Synchronized Multi-Turret Simulation

1. Click the Toolpaths step  in the Steps Toolbox. Click the *Machine simulation*  button and click Play.
2. Notice that now both turrets are running in a synchronized fashion based on the synchronization points that we set earlier.
3. Click the *Eject* button

The Next Step

You have completed this tutorial.

Chapter 8

Creating a Turn/Mill Part

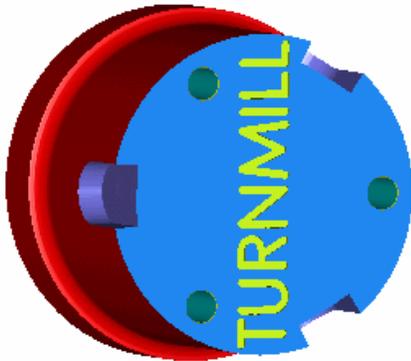
NOTE: You must have licensed the Turn/Mill option to run this tutorial.

This tutorial is only available in inch units.

In this tutorial you will learn to:

- Create parts for lathes with milling capabilities
- Mix turning and milling features
- Create milling features on the OD and face of the part
- Accurately simulate a turn/mill part

The following part will be created in this tutorial:



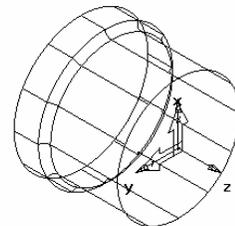
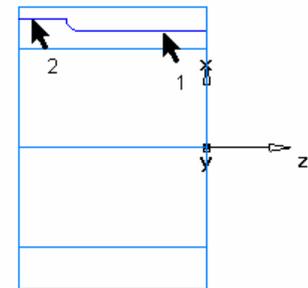
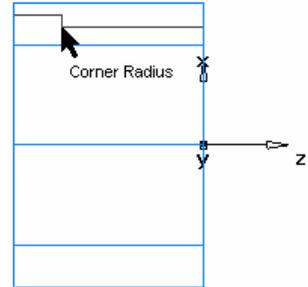
Getting Started

1. Start FeatureCAM.
2. Click *New Part Document*  button.
3. Click on *Turn/Mill*, click *Inch* and click *OK*.

4. In the stock dialog enter 3 for the *OD*, 2 for the *Length* and 0 for the *ID* and click Finish. (if this dialog box does not display automatically, click the Stock step from the Steps toolbox). Click *OK* in the Stock Properties dialog box if it appears.
5. From the *Options* menu select *Turning Input Modes* and check *Diameter*.

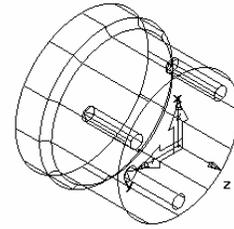
Creating a Turned Feature

1. Select the *Geometry* step  from the Steps Toolbox and click the *Connected Lines* button  .
2. On the bottom of the screen, type in these points to create two lines that define the OD profile: ($D1=2.5, Z1=0$), ($D2=2.5, Z=-1.5$). Press ENTER. Type the values ($D2=2.75, Z=-1.5$). Press ENTER. Type ($D2=2.75, Z=-2.0$) and press ENTER.
3. Select the *Geometry* step  from the Steps Toolbox and click the *Corner fillet* button  .
4. Enter 0.125 as the radius at the bottom of the screen and click at the location labeled as *Corner radius* in the figure.
5. Select the *Curves* step  from the Steps Toolbar. Click the *Chain into open boundary* button  . Click on the locations labeled 1 and 2 in the figure. Click ENTER.
6. Select the *Features* step  from the Steps Toolbar.
7. Click *Turning* and click *Next*.
8. Click *Turn* and click *Finish*.
9. Click *Isometric*  from the Principal Views fly-out.
10. Click the 2D turned profiles  button in the display mode toolbar to turn on 3D line drawing view. For details on the Display mode toolbar, see (Chapter 3, Interface Tour).



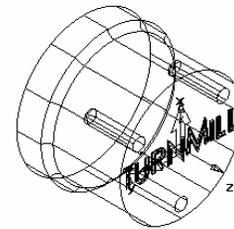
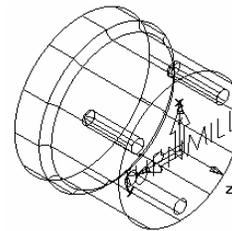
Creating a Radial Pattern on the Face

1. Select the *Features* step  from the Steps Toolbar.
2. Click *Turn/Mill* and click *Next*.
3. Click *Hole* and *Make pattern from this feature* and click *Next*.
4. Enter *1.0* for the *Depth*, *0.0* for the *Chamfer* and *0.25* for the *Diameter* and click *Next*.
5. Click *Radial* and click *Next*.
6. Enter *3.0* for the *Number*, *2.0* for the *Diameter*, *120* for the *Spacing Angle*, *60* for the *Angle* and click *Axial* for the *Feature Orientation* and click *Finish*.



Engraving on the Face

1. Select the *Curves* step  from the Steps Toolbar and select the *Create curves using curve wizard*  button.
Create curves using curve wizard  button.
2. Click *Other methods*, click *Text* and click *Next*.
3. Enter *TURNMILL* as the *Text*, click *Linear* as the *Type*. Enter *(0.0, -0.045, 0.0)* as the *X, Y and Z* locations. Enter *-90* as the *Angle*. Enter *Center* as the *Justification*, Enter *0.4* and *0.4* as the *X and Y Scaling*.
4. Click *Font* and select *Machine Tool Gothic* for the font type and *72* for the font size. Click *OK* and then *Finish*.
5. Select the text string in the graphics window and click the *Features* step  from the Steps Toolbar.
6. Select *Turn/Mill* and click *Next*.
7. Select *Groove* and click *Next* until you get to the *Dimensions* page of the dialog box.
8. On the *Dimensions* page, enter *0.0625* as the *Width*, click *Simple* and *Face*, and enter *0.02* as the *Depth*.
9. Click *Finish*.



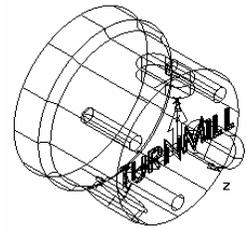
Creating a Pattern of Wrapped Slots

1. Select the *Features* step  from the Steps Toolbar.
2. Click *Turn/Mill* and click *Next*.
3. Click *Slot* and *Make pattern from this feature* and click *Next*.
4. Set *Length* to 1.0, *Width* to 0.5, *Depth* to 0.25 and click *Next*.
5. Select *Radial* and click *Next*.
6. Set *Number* to 3, *Diameter* to 2.5, *Spacing Angle* to 120, *Angle* to 120 and *Feature Orientation* to *Radial*. Click *Next*.
7. Set *Z* to 0.25.
8. Click the triangle located in the *Finish* button and click *Finish and Edit Properties*. This brings up the properties dialog box for the pattern and allows us to change some additional attributes.
9. Click *Wrap feature around Z axis* and click *OK*.



Simulating a Turn/Mill Part

1. Click the *Toolpaths* step  in the Steps Toolbox. The Simulation toolbar is displayed.
2. Click the *3D simulation*  button and click *Play* . Notice how the toolpaths are accurately simulated including the part rotations.
3. Click *Eject*  at the completion of the simulation



The Next Step

You have completed this tutorial. If you want to see other examples of turn/mill parts, look in the [examples/turnmill](#) directory.

Chapter 9

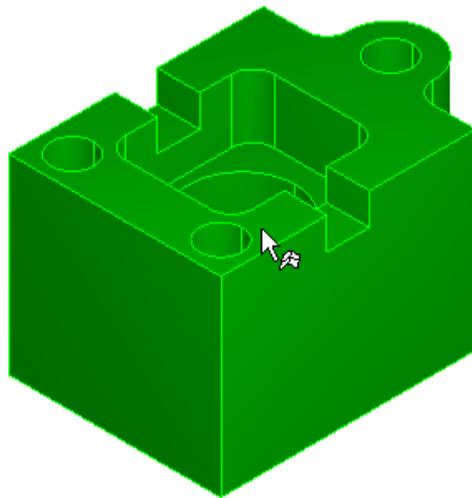
Features from 3D CAD Models

NOTE: You must have FeatureMILL with the FeatureRECOGNITION option or FeatureMILL3D to perform this tutorial.

This tutorial is only available in metric units. The part will be recognized twice, first using the AFR wizard and second using other feature recognition techniques.

In this tutorial you will learn to:

- Import 3D models
- Dynamically shade 3D models
- Create features directly from 3D solid models using both automatic feature recognition and more interactive methods.



Tool Cribs

You must have metric tooling loaded to perform this tutorial. If you do not have metric tools in your database:

1. Exit FeatureCAM by selecting *Exit* from the *File* menu.
2. Click on the *Start* menu. Click on *Programs* (or *All Programs*). Select *InitDB* from the FeatureCAM program group. Or you can run *Initdb.exe* from the program directory where FeatureCAM is installed.
3. Select *On my local computer* and click *Next*.
4. Select *Both* and click *Next*.
5. Answer the questions of the wizard and click *Finish* at the end.

Importing the CAD Model

1. Start FeatureCAM.
2. Click *New Part Document*  button.
3. Click on *Milling Setup*, click *Millimeter* and click *OK*.
4. In the *Dimensions* dialog box, click *Finish*. Click *OK* if the *Stock Properties* dialog appears.
5. Select *Import/export options* from the *File* menu. Check *Import SAT, SLDPRT, MOD, DXF & X_T as solids*. Check *Center stock automatically* and click *OK*.
6. Select *Import* from the *File* menu and browse to *C:\Program Files\FeatureCAM\Examples\Tutorials\3D models*.
7. Select files of type *ACIS* and select *Tutor9.sat* and click *Open*.
8. Select *Use the wizard* option in the *Import Results* dialog box and keep clicking *Next* until you get to the *Stock Dimensions* page.
9. Select *Compute stock size from the size of the part* and click *Finish*. The stock is now resized.
10. Click on the *Part* view. Click on the “+” next to the *Solids* category. Notice that *solid1* is listed. This is the solid model you just imported. Click back onto the *Steps* toolbox.
11. Click *Isometric*  from the *Principal Views* fly-out. You’ll notice that the stock has been resized but the setup is located at the center of the stock.

12. Click the *Setups*  button from the *Advanced* toolbar.
13. Click *Edit* and click *Next*.
14. Select *Align to Stock Face* and click *Next*.
15. Click the  button on the lower left end of the stock picture.
16. Click *Finish* and click *Close*. The stock has now been aligned to the stock face.
17. Select *Set tool crib* from the *Manufacturing* menu. Click on the *tools* crib and click *OK*. Notice at the bottom of the window, in the Status bar the current tool crib has been changed to *tools*.

Shade the Part

1. Click the Shade  button.
2. This shaded view makes it easier to see the cavities in the model. Remember, at this point the part model contains only surfaces.
3. Toggle the Shade button to turn off the shading.

Automatic Feature Recognition

This entire part can be automatically recognized using the automatic feature recognition (AFR) wizard.

1. Click the *AFR* step  from the Steps Toolbar.
2. Click the *Next* button twice.
3. The features that were just recognized are listed. Click the *Finish* button.
4. Click the Toolpaths step  in the Steps Toolbox. The Simulation toolbar is displayed.
5. Click the 3D simulation  button and click *Play* . If the Ordering dialog box appears, click *OK*.
6. Click the *Eject*  button.

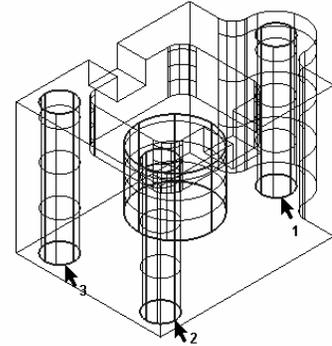
Note that all features of the model were automatically recognized and the part is ready to cut. With many parts automatic feature recognition is all you need to program your part, but with others, you may need to use some of the other feature recognition techniques.

For the sake of demonstration you will remove the features you just created and reprogram them using additional feature recognition techniques.

7. Click the Undo  button to remove the features you created so far.

Recognize All Holes Using Another Method

1. Select the *Features* step  from the Steps Toolbar.
2. Click *Hole* and *Extract with FeatureRECOGNITION* and click *Next*.
3. Click *Recognize and construct multiple holes* and click *Next*.
4. The holes that are automatically recognized are shown on the screen. Three holes were recognized along with the circular pocket in the middle. We want to ignore the center pocket, so select the three other holes with the mouse and click *Finish*.



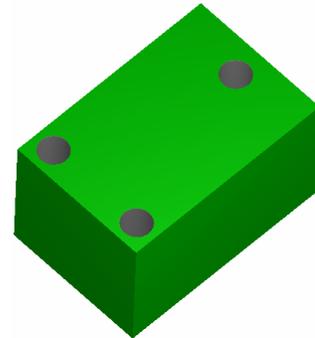
5. Click *OK* if the *Pattern Properties* dialog box appears. The three holes have now been recognized.

6. Click the *Ordering options*  button in the Op. list tab.
7. Click on *Cut higher operations first*, *Minimize tool changes* and *Minimize rapid distance*. Click *OK*.

8. Click the *Toolpaths* step  in the Steps Toolbox. The Simulation toolbar is displayed.

9. Click the *3D simulation*  button and click *Play* .

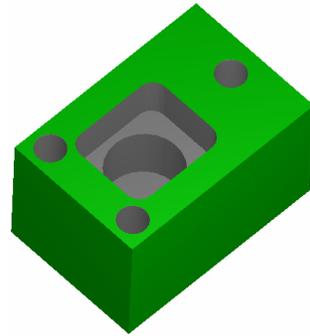
10. Click the *Eject*  button.



Recognizing All Pockets Automatically

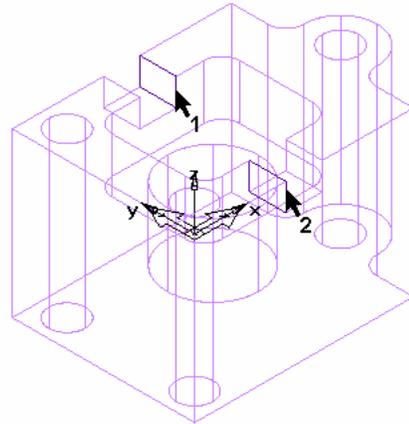
1. Select the *Features* step  from the Steps Toolbar.
2. Click *Pocket* and *Extract with FeatureRECOGNITION* and click *Next*.
3. Click *Automatic recognition* and click *Next*.

4. Click *Select all* and click *Finish*.
5. Click the Toolpaths step  in the Steps Toolbox. The Simulation toolbar is displayed.
6. Click the 3D simulation  button and click *Play* .
7. Click the *Eject*  button.



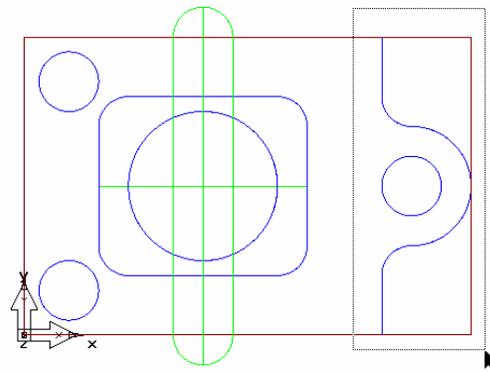
Create a Slot Feature Using the Pick Surface Button

1. Select the *Features* step  from the Steps Toolbar.
2. Click *Slot* and *Extract with FeatureRECOGNITION*.
3. Click *Next*.
4. Click the *Pick surface*  button and click on location 1. If the *Select* dialog box comes up, select the name that corresponds to the surface at location 1 and click *OK*. (The surfaces are highlighted in the graphics window as you select the names.)
5. Click the *Pick surface* button again and click on location 2. Select the surface from the *Select* dialog box and click *OK*.
6. Click *Next* and review the extracted slot dimensions and click *Finish*.

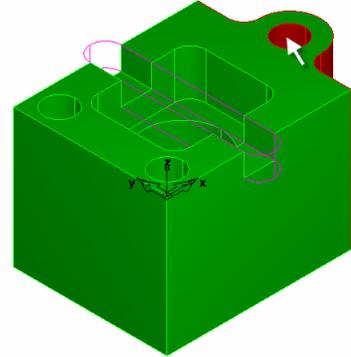


Create a Side Feature

1. Select *Top* from the Principal Views fly-out.
2. Drag-select the region shown below.

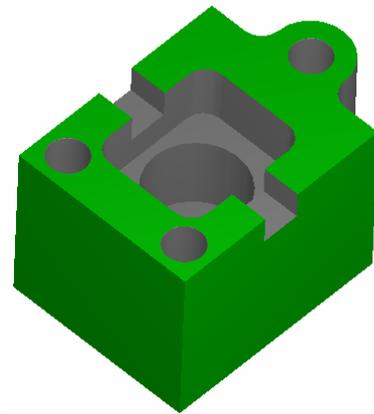


3. Select *Isometric* from the Principal Views fly-out.
4. Click the Shade  button to turn on shading. The shaded view will make it easier to select faces on the surface.
5. Hold down the SHIFT key and click on the side of the hole pattern that is selected as shown in the image. This will remove the hole pattern from the selected group.
6. Select the *Features* step  from the Steps Toolbar.
7. Click *Side* and *Extract with FeatureRECOGNITION* and click *Next*.
8. Click *Select side surfaces* and click *Next*.
9. Click *Next*.
10. If the blue arrow is pointing to the inside of the part, click the *Reverse side*  button until the blue arrow points to the outside of the part.
11. Click *Finish*.



Simulate the Entire Part

1. Click the *Toolpaths* step  in the Steps Toolbox. The Simulation toolbar is displayed.
2. Click the *3D simulation*  button and click *Play* .
3. Click *Eject*  at the completion of the simulation.



The Next Step

The features for the entire part have been created directly from the surfaces of the solid model. Using FeatureCAM's feature-based manufacturing automation, this part can be created quickly and easily.

Chapter 10

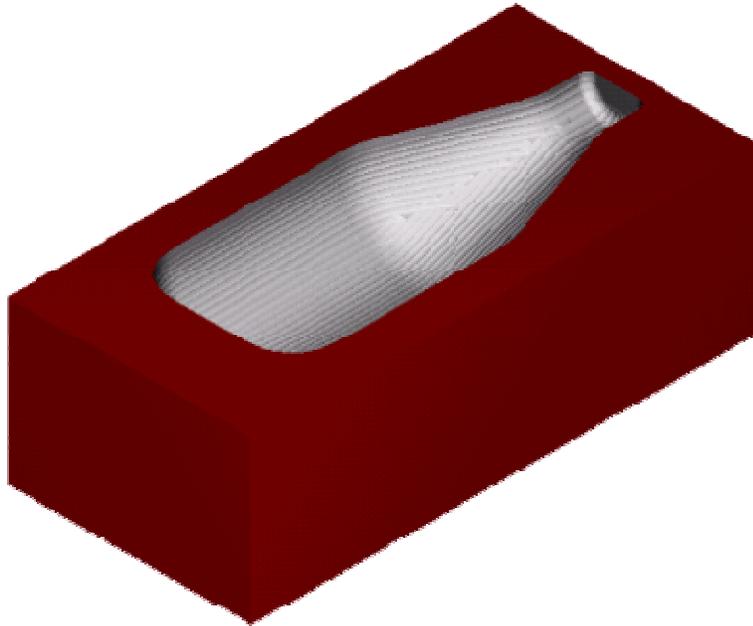
Introduction to 3D

NOTE: You must have FeatureMILL3D to perform the examples in this chapter. These examples are only specified in inch units. You must have the *basic* tool crib installed.

This example introduces:

- Modeling 3D surfaces
- Manufacturing surfaces using surface milling features
- Manufacturing operations
- Tool selection
- 3D manufacturing attributes

In this example you will model and create manufacturing toolpaths for the following part.

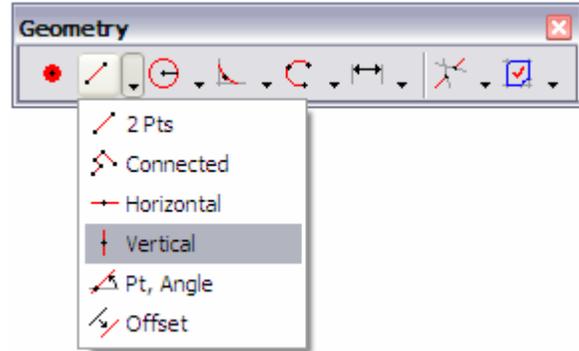


Introduction to 3D Modeling

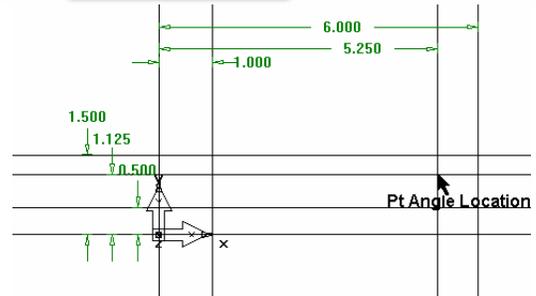
1. Start FeatureCAM.
2. Click *New Part Document*  button.
3. Click on *Milling Setup*, click *Inch* and click *OK*.
4. In the *Dimensions* dialog box, click *Block* and set *Width* to 3, *Length* to 6.25 and *Thickness* to 2.0.

5. Click *Finish*.

6. Select *Toolbars* from the *View* menu. On the *Toolbars* tab, make sure that *Geometry* and *Advanced* are checked and click *OK*.



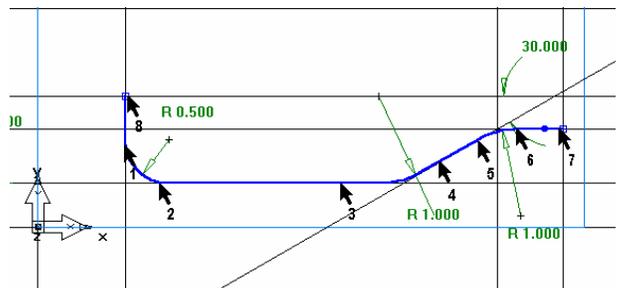
7. Click on the lines fly-out menu and select *Vertical*.
8. On the bottom of the screen, type in *1.0* for X, *0.0* for Z and press the ENTER key.
9. Create two more lines at X= 5.25 and X = 6.0.
10. Click on the lines fly-out and select *Horizontal*. Create horizontal lines at Y= 0.5, 1.125, and 1.5.



11. Click on the lines fly-out menu and select *Line at Angle thru Pt.*. Type *30* as the angle and click at the intersection labeled *Pt Angle Location* in the figure.

12. Click on the arc fly-out menu and select *Arc from Radius, 2Pts*. Enter *0.5* as the radius and click on locations 1 and 2.

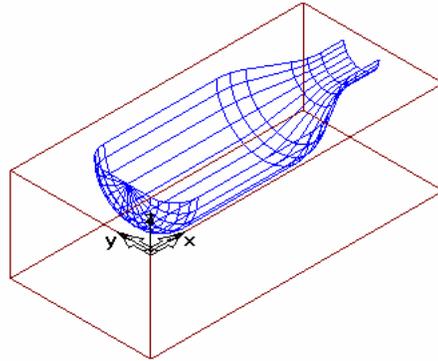
13. Create another arc of radius *1.0* and click at location 3 and 4. Create another arc of radius *1.0* at locations 5 and 6.



14. Select the *Curves* step  from the *Steps* Toolbox. Select *Chain pieces into an open...*  button. Click at the locations 7 and 8.

Create the Bottle Surface

1. Click the *Surfaces* step  from the Steps Toolbox.
2. Click *Surface of Revolution* and click *Next*.
3. Click the *Pick curve*  button and select the curve you chained.
4. Enter *0.0* for *Start angle*, and *180* for *End angle*.
5. Click the *Pick line*  button and select the top horizontal line (the one at $Y=1.5$).
6. Click *Finish*.
7. Click *Isometric*  from the Principal Views fly-out.



To control how the part is displayed you can change the values in the Viewing Options dialog box.

8. Select *Viewing...* from the Options menu.
9. Uncheck the *Show surface boundaries only* checkbox and hit *Apply*. You should now see your part drawn with lines in the interior of the surface. For large models, this option will make the display of the part slower.
10. Enter *20* for *Wireframe* and hit *Apply*. The surface will now be drawn with more lines. Decreasing the value of *Surface fineness* will improve the display quality but can slow down the graphics.
11. Select *Hide All Geometry* from the *Hide*  fly-out.

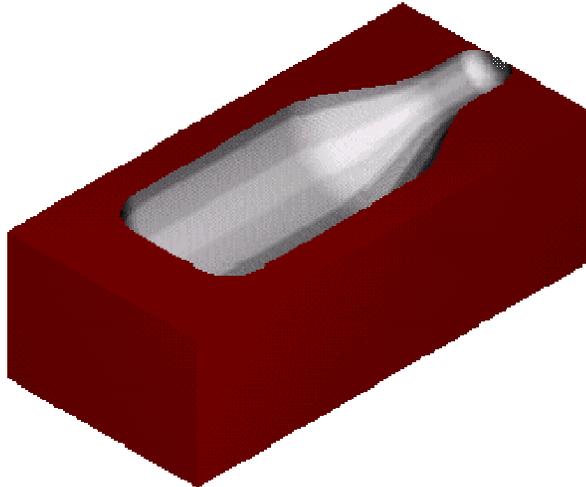
Introduction to 3D Manufacturing

1. Click on the *Select*  button.
2. Select the surface.
3. Click the *Features* step .
4. Select the *Surface Milling* radio button.
5. Click *Next* twice to get to the New Strategy page of the dialog box.

6. Click *Choose rough, semi finish, and finish ...* and click *Next*.
7. Click *Z level rough* and click *Next*.
8. Select *None* and click *Next*.
9. Select *Isoline* and click *Finish and Edit*.
10. Click on *Z level* in the tree view.
11. Select the *Strategy* tab and check *Classify slices as pocket*.

Simulating the Entire Part

1. Click the *Toolpaths* step  in the Steps Toolbox. The Simulation toolbar is displayed.
2. Click the *3D simulation*  button and click *Play* .
3. Click *Eject*  at the completion of the simulation



The Next Step

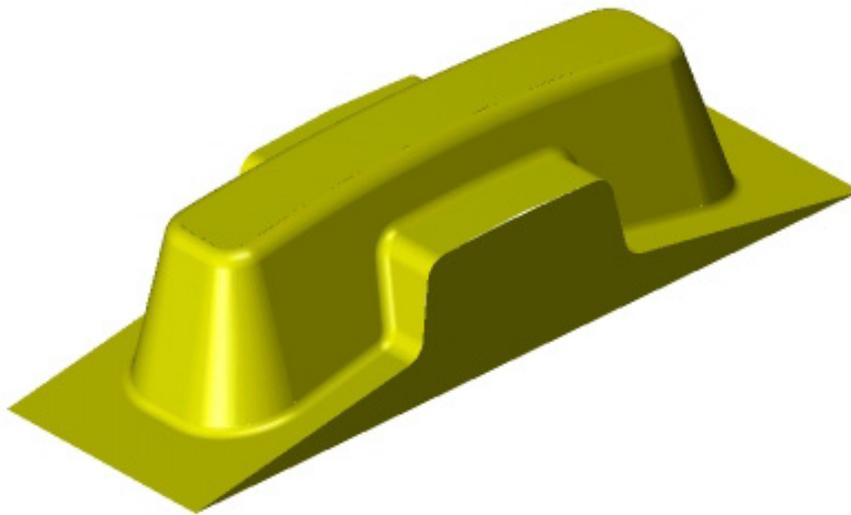
You have completed this tutorial and can move to the next chapter for more advanced 3D manufacturing techniques.

Chapter 11

Advanced 3D Milling

NOTE: You must have FeatureMILL3D to perform this tutorial.

In this example you will import a 3D IGES surface model and create 4 different types of toolpaths for manufacturing the part.



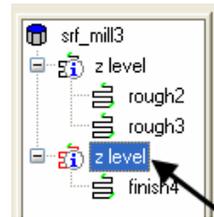
Import the Part

1. Click the *New Part Document*  button.
2. Click *Milling Setup, Inch*, and click *OK*.
3. Click *Finish* in the *Dimensions* dialog box. If the *Stock Properties* dialog box appears click *OK*.
4. Select *Import/Export Options* from the *File* menu.
5. Check *Center stock automatically* and click *OK*.

6. Select *Import* from the *File* menu.
7. Select *IGES* as *Files of type*.
8. Select *phone.igs* from *c:\Program Files\FeatureCAM\examples\3D* and click *Open*.
9. Click *No* in the pop-up window.
10. Select the *Use the wizard* option in the *Import Results* dialog box and keep clicking *Next* until you get to the *Stock Dimensions* page.
11. Click *Compute stock size from the size of the part*, and click *Finish*.

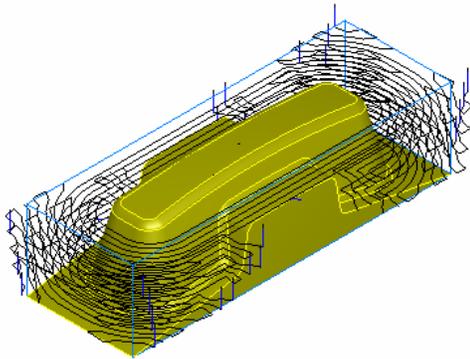
Create the 3D Strategy

1. Click on the *Isometric*  button from the *Principal views* fly-out.
2. Select *Select all* from the *Edit* menu.
3. Select the *Features* step  from the *Steps Toolbar*.
4. Click *Surface Milling* and click *Next* twice.
5. Click *Choose rough, semi finish, and finish ...* and click *Next*.
6. Click *Z level rough*.
7. Check *Flat surface support* and click *Next*.
8. Select *Z Semi* and click *Next*.
9. Select *Z Level* and click on the triangle next to the *Finish* button and select the *Finish and Edit Properties* option.
10. Select the second z-level operation in the list.
11. Check the *Interleave spiral paths* checkbox and click *OK*.
12. Click on *Manufacturing* and select *Post Process*.
13. Change the *Tool Change Location Z* to be 2.0
14. Press *OK*.

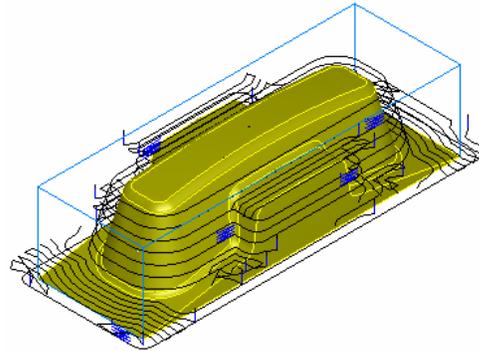


Simulating the Toolpaths

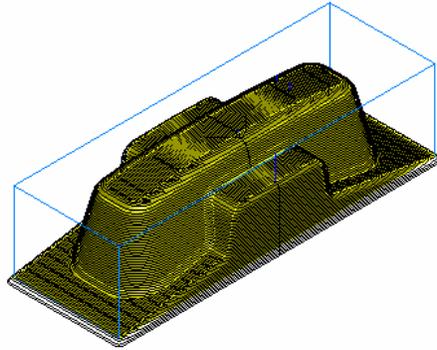
1. Click the *Toolpaths* step  in the Steps Toolbox. The Simulation toolbar is displayed.
2. Click the *Centerline simulation*  button and click Play .
3. Click the *Erase*  button to temporarily clear the screen.
4. Click on the *Op List* tab in the Results window.
5. Select the first operation in the Operations List and click the *Preview toolpaths*  button.
6. Click the *Play Selected*  button from the simulation toolbar to see the toolpath for this operation.
7. Click on the different operations and click the *Play Selected* button to see the different simulations.



Z Rough



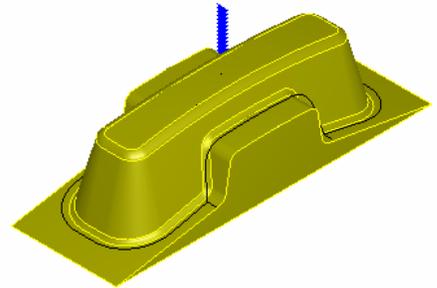
Z Semi Finish



Z Level Interleave

Clean Up the Fillets

1. Click on Part View. Select *srf_mill1* from the list and click the *Properties*  button at the bottom of the screen
2. Click the *Process* tab.
3. Click the *Add new operation*  button. Click *Pencil* and click *Next*.
4. Click the *Finish* radio button
5. Click on the *finish3* process in the tree view
6. Select tooling and then select a 0.25 diameter Ball endmill.
7. Click the *Finish* button at the bottom of the dialog box.
8. Click the *Preview toolpaths*  button in the Operations List portion of the Manufacturing Results window. The dialog box will warp.
9. Click the *Play selected*  button in the simulation toolbar. A single toolpath is created along the fillet. Note that this operation takes some time to calculate.
10. Click the *Stop*  button.

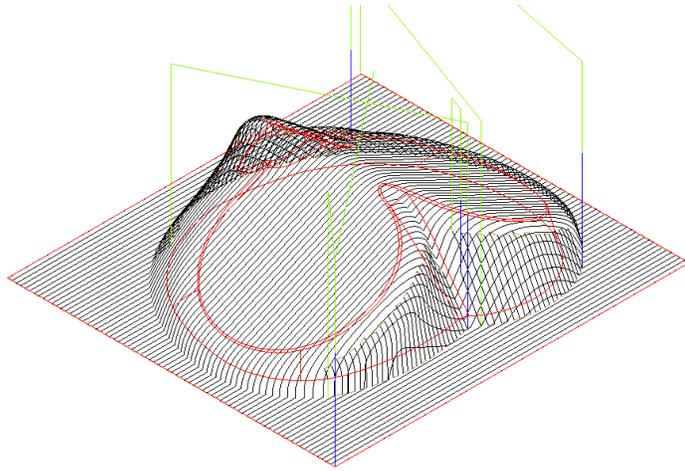


The Next Step

You have completed this tutorial. In addition the machining strategies used above, FeatureCAM has the following finishing strategies:

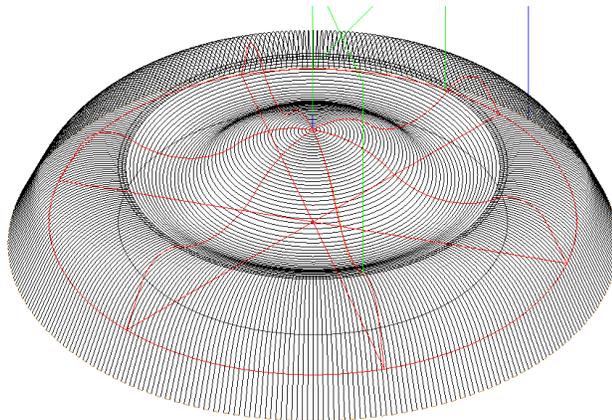
Parallel Machining

The Parallel machining strategy machines the part in a series of parallel passes at a specified angle and stepover. A second perpendicular pass can be applied to improve surface finish in steep areas.



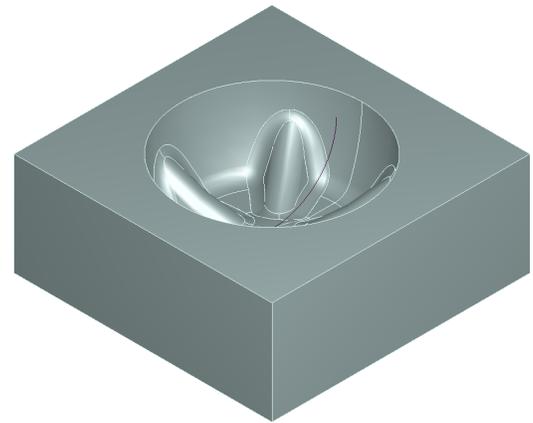
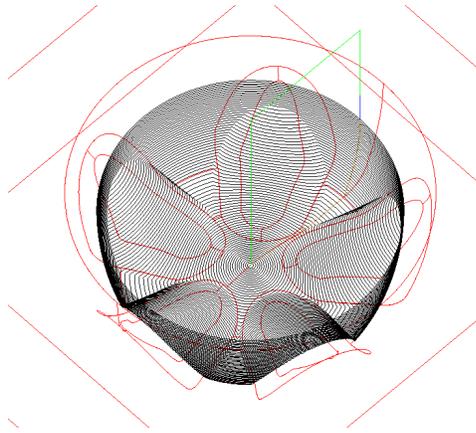
Radial and Spiral Machining

The Radial machining strategy machines a part with a series of passes which radiate from a center point. Spiral machining machines the part with a single continuous spiral pass. In the example to the right the outer area has been machined with the Radial strategy, the center area using Spiral.

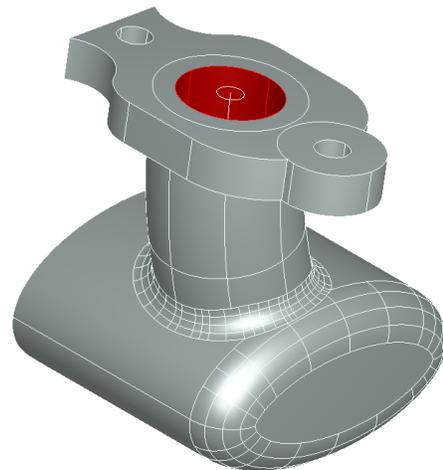
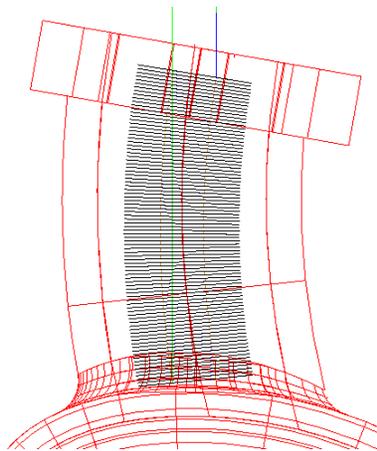


Flowline Machining

In this strategy a control surface is used to guide the direction of machining. This enables the production of toolpaths with an exceptional level of control over surface finish. With suitable cutters it is possible to use the flowline strategy to machine undercuts without the use of a 5 axis machinetool.



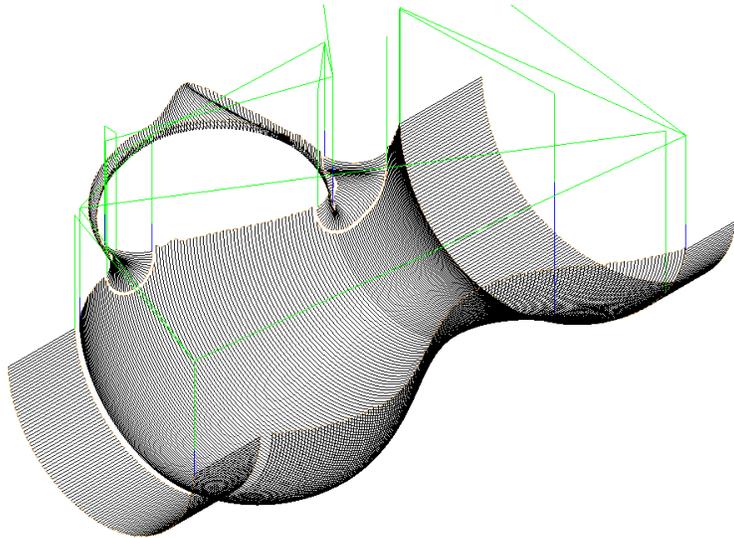
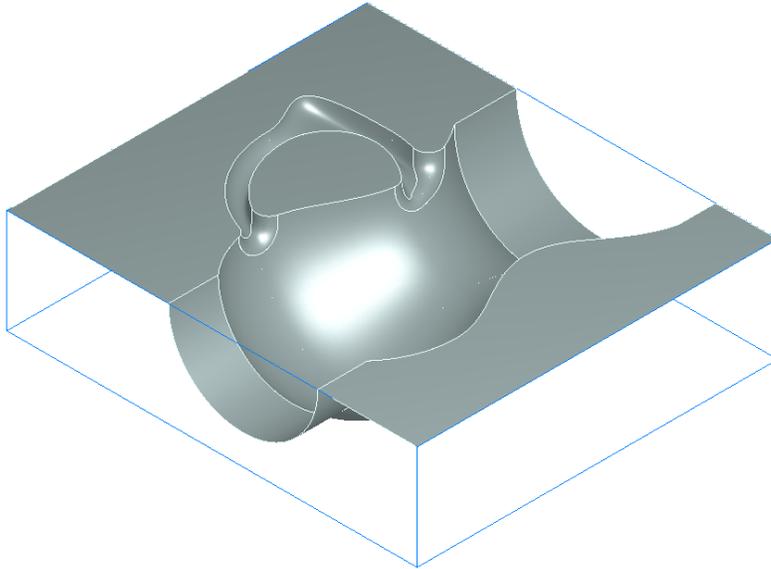
Flowline machining of a blowmold part



Flowline machining of an undercut feature

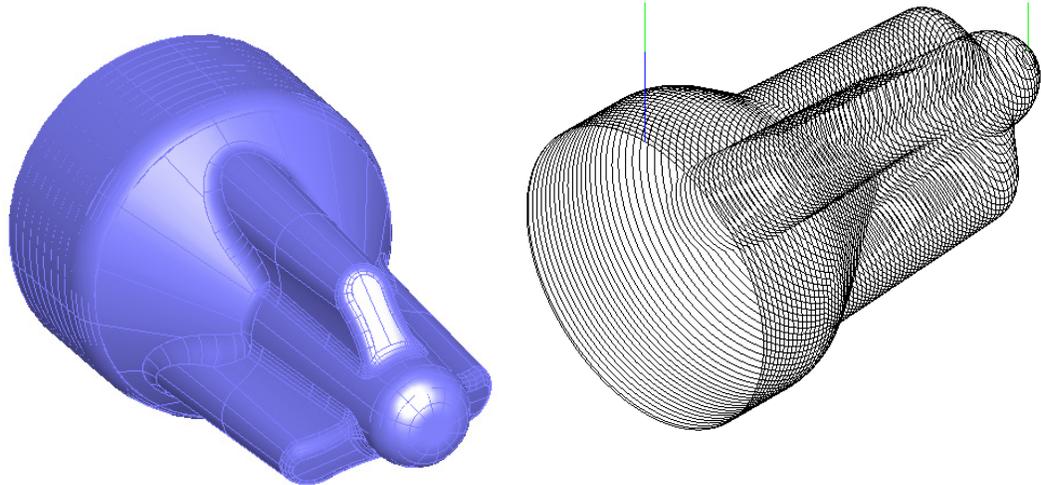
Isoline Machining

In this strategy each surface on a part is machined in a user defined direction. This gives a great level of control over the surface finish and tool loading. The example below shows a jug mold which has been machined using the Isoline strategy. Note how the cutting direction on each surface has been chosen to give the best finish.



Four Axis Machining

FeatureCAM rotary machining allows the user to generate four axis toolpaths for finish machining. The tool moves in 3 axes whilst the workpiece rotates around a rotary axis. Rotary moves may be indexing or continuous.



For further information about all of the above strategies please download the advanced 3D machining tutorials from our website:

www.featurecam.com

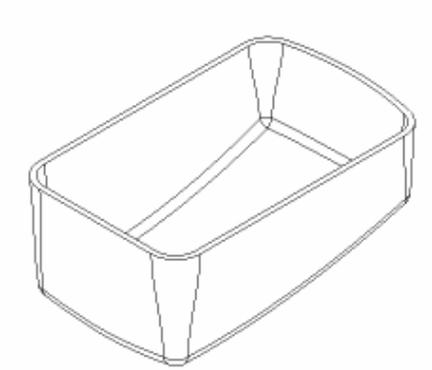
Chapter 12

Solid Modeling

Note: You must have the Solid Modeling Option to perform this tutorial.

This tutorial introduces you to the solid modeling capabilities of FeatureCAM by creating a model of the simple computer speaker shown below. The tutorial is available only in metric units and introduces you to:

- Extrude design features
- Hidden line graphics
- Loft design features
- Filleting solid models
- Core/cavity separation of a solid model
- Creating manufacturing features from solids

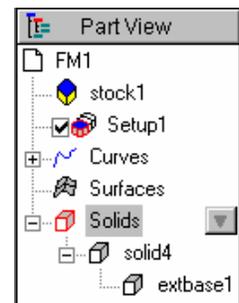
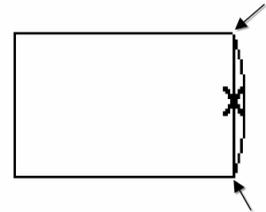


Getting Started

1. Click *New Part Document*  button.
2. Click on *Milling Setup*, click *Millimeter* and click *OK*.
3. Click *Finish* in the *Dimensions* dialog box.
4. If the stock dialog box is displayed, click *OK*.
5. Turn on the necessary toolbars by selecting *Toolbars* from the *View* menu. Check the *Steps*, *Display Mode* and *Solid* checkboxes on the *Toolbars* tab and click *OK*.

Create the Initial Solid

1. Click on the Curves step from the Steps Toolbar and click on the *Curve wizard* .
2. Select *Other methods* as the method and *Rectangle* as the constructor. Click *Next*.
3. Click the *Use corner, width and height* radio button and enter 0.0 and 0.0 for the X and Y Corner coordinates.
4. Set *Width* to 100 and *Height* to 65. Check *Create as arcs and lines* and click *Finish*.
5. Click on the vertical line on the right and click the *Delete*  button.
6. Select *Snapping modes* from the *Options* menu. Make sure the *End points* button is pressed and click *OK*. (A pressed button will have a white square around it)
7. Select the *Geometry*  step from the *Steps* toolbar. Click the *Arc from Radius, 2 Pts*  button.
8. Enter a *Radius* of 105 and click on the two open end points where the deleted vertical line connected.
9. Select the *Curves* step  from the Steps Toolbar.
10. Select *Chain into a closed boundary*  and click on the arc you just created. Click the *Create* button at the bottom of the screen.
11. Click the *Solid Wizard*  button from the Advanced toolbar.
12. Click *Define a custom shape* and click *Next*.
13. Select *From curves* as the method and *Extrude* as the constructor and click *Next*.
14. Enter -50 as the *Vector Z* coordinate and -5 as the *Draft Angle* and click *Finish*.
15. Click on the *Part view* on the left side of the screen. The *Solid* category now has a "+" next to it. Click on the "+" to reveal the solid, called *solid4*. That is the name of the solid we are creating. The number following the *solid* name may be different for you.
16. Click on the "+" next to *solid4*. This reveals the features that make up

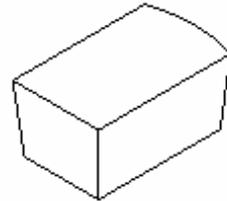
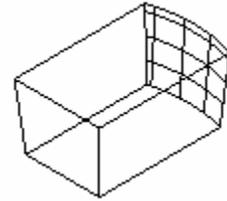


the solid. There is only one feature called *extbase1*, a base feature is the initial feature for a solid.

Hidden Line Graphics

With solids there are additional graphics options available. By default, you are in line drawing mode. With solids, hidden line graphics are available.

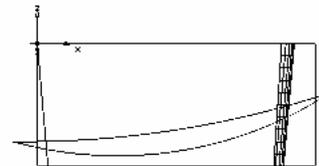
1. Click the Isometric button  from the Principles Views Flyout.
2. Select *Hide stock* from the Hide fly-out in the Advanced toolbar.
3. The default view of your solid model is shown in the top figure. In this figure it is impossible to tell if there is a top on the solid.
4. Click the *Hidden Line*  button. It is located in the Display toolbar. The graphics shown in the bottom figure are displayed. In this figure it is clear that there is a top on the solid.
5. While hidden line mode provides nice graphics, it is considerably slower than simple line drawings and sometimes hides portions of the model you need to pick, so click the *Hidden Line*  button again to toggle the line drawing mode.



Note that hidden line graphics are only available for solids.

Create the Curved Lower Surface

1. Select *Show stock* from the Show fly-out in the Advanced toolbar.
2. Double-click on the stock and click the *Resize...* button.
3. Click *Compute stock size from the size of the part*, and click *Next*. Click *Move stock* and click *Finish*.
4. Select the *Front*  button from the *Principal Views* flyout.
5. Select the *Geometry* step  from the Steps toolbar. Click the *Arc from Radius, 2 Pts*  button.
6. Set the *Radius* to 400. Set *X1* to 120, *Y1* to 0 and *Z1* to -20. Set *X2* to -10, *Y2* to 0 and *Z2* to -40,



and click the *Create* button.

7. Create a second arc by entering *160* as the *Radius* and clicking on the right and left endpoints of the first arc. If your arc curves up instead of down, click on the arc and click the *Options* button at the bottom of the screen.

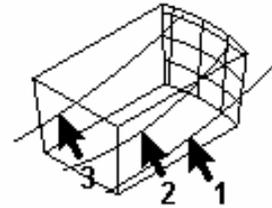
NOTE: If the cursor is still in the arc create mode, click on the *Select*  button in the Standard toolbar. Now you are able to click on and select the arc.

8. Select the top arc (the one you created first) and click the *Transform*  button.
9. Click *Translate* and the *Copy* radio button. Enter *70* in the Y coordinate of *XYZ distance* and click *OK*.
10. Select the bottom arc and click the *Transform*  button.
11. Click *Translate* and the *Move* radio button and transform it 35 in the Y direction and click *OK*.

12. Select *Isometric*  from the Principal views fly-out.

13. Click the *Solid Wizard*  button from the Advanced toolbar.

14. Select *From curves* as the method and *Lofted* as the constructor and click *Next*.

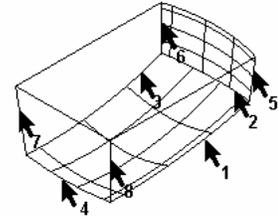


15. Since we want to use this design feature to remove material, click the *As Cut* radio button and click the pick curve  button.
16. Select the arcs in the order shown in the figure. If the selected arcs appear out of order in the curve list, use the arrows to move the arcs up or down. Click *Finish*.
17. The curves are used to create a lofted surface and then the surface is used to cut the solid. Note that an additional design feature, called *loftcut1* is now listed in the part view.

Create Constant and Variable Radius Fillets

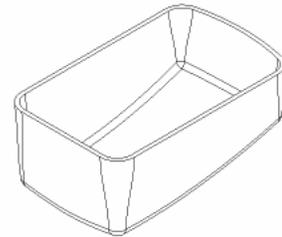
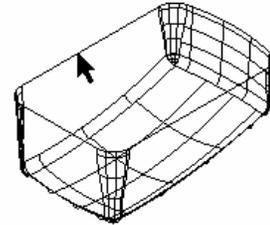
1. Click the *Solid Wizard*  button from the Advanced toolbar.
2. Select *Shape Modifiers* as the method and *Fillet Edges* as the constructor and click *Next*.
3. Enter *5* as the *Radius*, click the *Pick curve*  button and pick the edges 1-4 shown in the figure.

- Click *Variable* and enter 5 as the *Begin radius* and 10 as the *End radius* and pick the edges 5-8 in the figure. This will create a variable radius fillet on these edges. As you click each edge an arrow is displayed to show the edge direction. The first radius will be applied to the tail of the arrow and the second to the head of the arrow.
- Click *Finish*. Note that the selected edges are filleted and the surrounding surfaces have been trimmed. The corners where the fillets meet are automatically filleted.



Add Thickness

- Click the *Solid Wizard*  button from the Advanced toolbar.
- Select *Shape modifiers* as the method and *Shell* as the constructor and click *Next*.
- Enter an *Offset distance* of -2 and click the *Pick surface*  button.
- Click on the top surface as shown in the figure. This surface will not be offset. If the *Select* dialog box comes up, pick the name that corresponds to the top surface. (The surfaces are highlighted in the graphics window as you select the names.)
- Click *Finish*.
- All faces of the solid except the top surface are now offset by 2 mm to provide thickness to the part. Use hidden line graphics to display the graphics shown on the right.



At this point, the solid model of the speaker is complete.

Create a Mold Cavity

Now the challenge is to create a mold from the solid model. For parts that can be created with two mold halves, FeatureCAM's core/cavity feature is a useful tool.

- Click the *Solid Wizard*  button from the Advanced toolbar.
- Select *Manufacturing* as the method and *Select Core/Cavity* as the constructor and click *Next*.
- Click the *Bottom* radio button.
- Click the *Make solid from result*. This will create a new solid that contains the cavity faces.

5. Click *Finish*. Note that a new solid, called *sheet1*, is listed in the part view.
6. Right click on the *solid1* in the parts view and select *Hide Selected*. You can now see the cavity.

The Next Step

You have completed this tutorial.

Chapter 13

Introduction to FeatureWIRE

NOTE: You must license FeatureWIRE to perform this tutorial.

In this tutorial you will learn the basics of creating wire EDM toolpaths using FeatureWIRE. This tutorial is only available in inch units. Specifically you will learn how to:

- Set up your material and wire thickness.
- Create wire EDM features.
- Specify a wire EDM cutting strategy.
- Simulate wire EDM toolpaths.

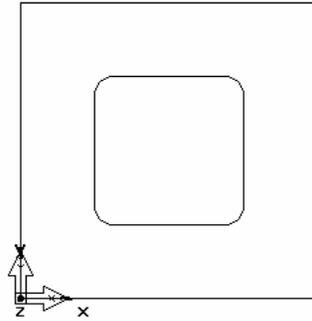
Getting Started

1. Begin FeatureCAM by selecting FeatureCAM from the *Start* menu.
2. Click *New Part Document*  button.
3. Click on *Wire EDM Setup*, click *Inch* and *OK*.
4. Set the stock *Width* and *Length* to *4.0* and the thickness to *0.5* and click *Finish*.
5. If the Stock Properties dialog box appears click *OK*.

Creating the Profile

1. Click the *Curves* step  from the Steps toolbox.
2. Click the *Create curves using the Curve wizard*  button.
3. Click *Other methods* and click *Rectangle* and click *Next*
4. Click *Use corner, width and height*.

5. Set the corner point to $1,1,0$.
6. Set the corner radius to 0.5 and set the *Width* and *Height* to 2.0 . Click *Finish*.

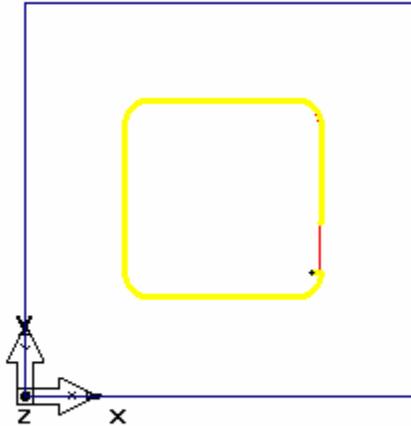


Creating a Wire EDM Feature

1. Click the *Features*  step from the Steps toolbox.
2. Click the *Die* radio button in the *2 Axis* group and click *Next*.
3. Click the *Pick curve*  button and select the curve you just created. Click *Next* until you get to the Dimensions page.
4. Enter 0.5 as the *Thickness* and click *Next* twice to get to the *Strategies* page.
5. Select *Retract* as the operation and check the *Cutoff* and *Contour* checkboxes.
6. Click *Finish*. If the Die properties page appears click *OK*.

Simulating a Wire EDM Toolpath

1. Click the *Toolpaths*  step. The Simulation toolbar is displayed.
2. Click the 2D simulation  button in the Simulation toolbar. Move the speed control slider  to the middle so that the simulation is slowed down.
3. Click the *Play to Next Operation*  button. You'll notice that the Retract operation makes two passes. If you need to slow down the simulation, drag the *Sim Speed* control  to the left.



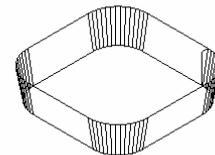
4. Click the *Play to Next Operation*  button again. The Cutoff operation now takes two passes. (Slow down the simulation if you need to).
5. Click the *Play to Next Operation*  button once more. The final contour operation is now simulated.
6. Click the *Eject*  button.

Generating NC Code

1. Click the *NC Code* step  in the Steps toolbar.
2. Click the *Display the NC code*  button in the dialog box to generate the NC code.

Add a Taper Angle

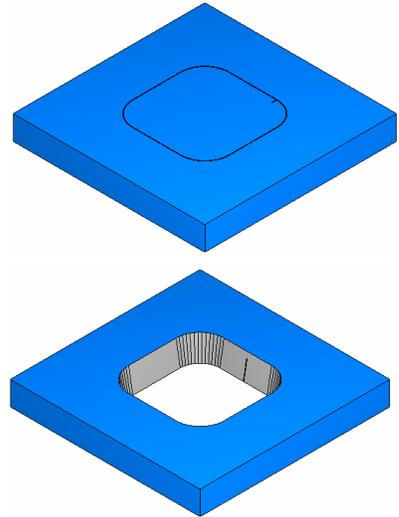
1. Click on the Part View. Select the *die1* feature from the list and click the *Properties*  button at the bottom of the screen.
2. Click the *Constant* radio button. Select *Left* as the taper type and enter *10* as the taper angle in the *deg.* field and click *Apply*.
3. Click the *Isometric* view button  from the Standard toolbar to get a 3-dimensional view.
4. Set the taper type to *Right* and click *Apply*. View the new shape in *Isometric* view.



5. Change the taper type back to *Left* and click *OK*.
6. Click the *Toolpaths*  step. Click *3D*  and click the Play button.

Since this is a die feature, it is assumed that the region inside of the curve will be discarded.

7. Click the *Select*  button from the Standard toolbar.
8. Click on the stock in the region inside of the curve. FeatureCAM will now delete that part of the stock.
9. Click the *Eject*  button.
10. Click on the Part View and right-click the *die1* feature to bring up a fly-out menu. Select the *Hide Selected* option. Now the feature will not be displayed on the screen.



Creating the Second Profile

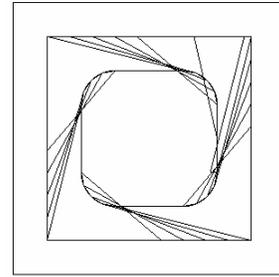
1. Click the *Curves* step  from the Steps toolbox.
2. Click the *Create curves using the Curve wizard*  button.
3. Click *Other methods* and click *Rectangle* and click *Next*.
4. Click *Use corner, width and height* and set the corner point to $0.5, 0.5, 0.0$
5. Set the *Width* and *Height* to 3.0 . Click *Finish*.

Creating a 4-axis Wire EDM Feature

1. Select the *Top*  button from the *Principal Views* fly-out.
2. Click the *Features*  step from the Steps toolbox.
3. Click the *Die* radio button in the *4 Axis* group and click *Next*.
4. Click the *Pick curve*  button and select the curve that you just created. Click *Next*.

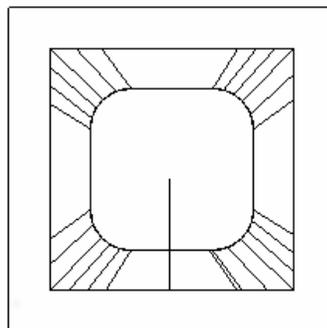
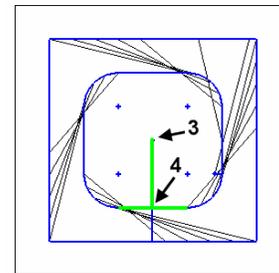
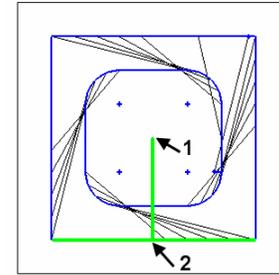
5. Click the *Pick curve*  button and select the curve that you created first. Keep clicking *Next* until you get to the *Dimensions* page.
6. Enter *0.50* as the *Thickness* and click *Finish*. If the Die properties page appears click *OK*.

The feature is twisted because the start points were not set.



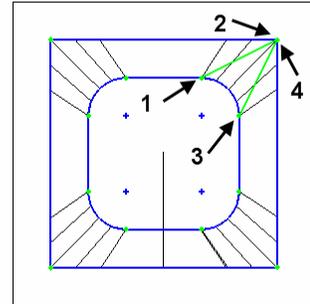
Modifying the Start Point

1. Click on Part View. Select the *die4x1* feature from the list and click the *Properties*  button.
2. Click on the *Start* tab.
3. Click the *Pick location*  button for the *Upper Curve Start/End Point*.
4. In the graphics window, click on a point in the center of the stock.
5. Click on the bottom horizontal line of the *outer curve*. You should notice a green line between the point and the edge of the curve.
6. Click the *Pick location*  button this time for the *Lower Curve Start/End Point*.
7. Click in the center of the stock and click on the bottom horizontal line of the *inner curve*.
8. Click *OK*. The feature now should no longer be twisted.



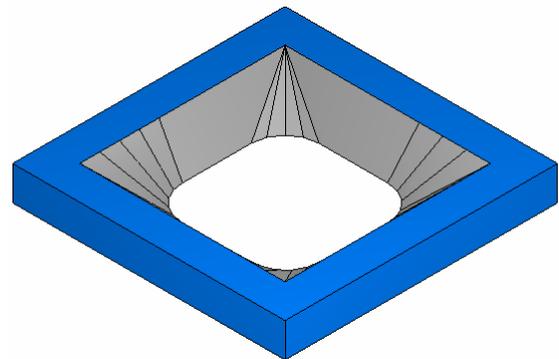
Matching the Curves

1. Click on Part View. Select the *die4x1* feature from the list and click the *Properties*  button.
2. Click the Dimensions tab and click the *Match Curves...* button.
3. Click the *Pick lower curve point*  button.
4. Select the point on the lower curve at location 1 then select the point on the upper curve at location 2. You should see a green line between the two points.
5. Hold down the shift key and select the other point on the lower curve at location 3 then select again the corner point at location 4. Now you have matched both points on the lower curve to the single point on the upper curve.
6. Click *Apply* and then *OK* in the Match Curves dialog box.
7. Click *OK* in the Die Properties dialog box.



Simulate the Part

1. Click on the *Part View* and uncheck the box next to the *die1* feature. When it's unchecked the feature will not be shown in the simulation.
2. Click the *Isometric* view button  from the Standard toolbar.
3. Click the *Toolpaths*  step. Click *3D*  and click the play button.
4. Click the *Select*  button from the Standard toolbar.
5. Click on the region inside of the curve on the die feature.
6. Click the *Eject*  button.



The Next Step

You have completed this tutorial.

Chapter 14

FeatureMILL3D 2007

Transition Guide

Overview

FeatureMILL3D 2007 has been extensively upgraded from FeatureCAM 2006. This upgrade resulted in many new roughing and finishing options in FeatureMILL3D 2007 as well as increased reliability and performance. We're pleased that we were able to include these new capabilities without significantly changing FeatureMILL3D's user interface. In order to achieve these improvements a few compromises had to be made relative to previous versions. In a few cases, strategies and options from previous versions had to be modified. Often there are new strategies in FeatureMILL3D 2007 that are equivalent to the previous functionality. In other cases, users may find that converting to a new strategy in FeatureMILL3D 2007 will do a better job than the previous technology.

This chapter is for users of previous versions of FeatureMILL3D software. It is intended to:

1. Provide alternatives and suggestions for using the new technology available in FeatureMILL3D 2007.
2. Explain the automatic conversion process for files from previous versions.
3. Show how FeatureMILL3D version 2006 technology can still be used within version FeatureMILL3D 2007.
4. Explain the manual conversion process for files from previous versions that contain incompatible options.

Here are some important points to remember about FeatureMILL3D 2007:

- Version 2007 contains many enhancements and new strategies.
- As much as possible, the user interface and strategies in FeatureMILL3D 2007 are compatible with previous versions.
- As a convenience, all the version 2006 technology is still available in version 2007, if necessary, for files saved by previous versions of FeatureCAM.

- Version 2006 technology will not be supported indefinitely. Users are strongly encouraged to convert any incompatible operations from existing files to use the new technology wherever possible.

Backward Compatibility

FeatureMILL3D 2007 uses a new machining library to compute surface milling tool-paths. This library completely replaces the old version 2006 machining library, but there are options from the old library that are not supported. To provide full backward compatibility, FeatureMILL3D 2007 includes both the new and the old libraries. The old library is provided to support files from previous versions when necessary, but will not be supported in the future.

Reading Files from Previous Versions

When a file from a previous version of FeatureCAM is read, each surface milling feature is examined. Each operation within a surface milling feature (e.g.: a parallel finish, or a Z rough) is checked for any incompatible settings. If any incompatible operations are found a warning dialog appears.



Each incompatible operation is labeled so that it will use the version 2006 machining library to compute tool-paths. This label is visible in the operation list right-click menu (see next section). All compatible operations in all the surface milling features will automatically use the new machining library. All new features and operations created in FeatureMILL3D 2007 will only use the new machining library. A detailed list of the incompatibilities is included later in this document.

Once the file is saved in version 2007, the warning will not appear when reading the file back into version 2007 although it may still contain incompatible operations marked to use the 2006 machining library.

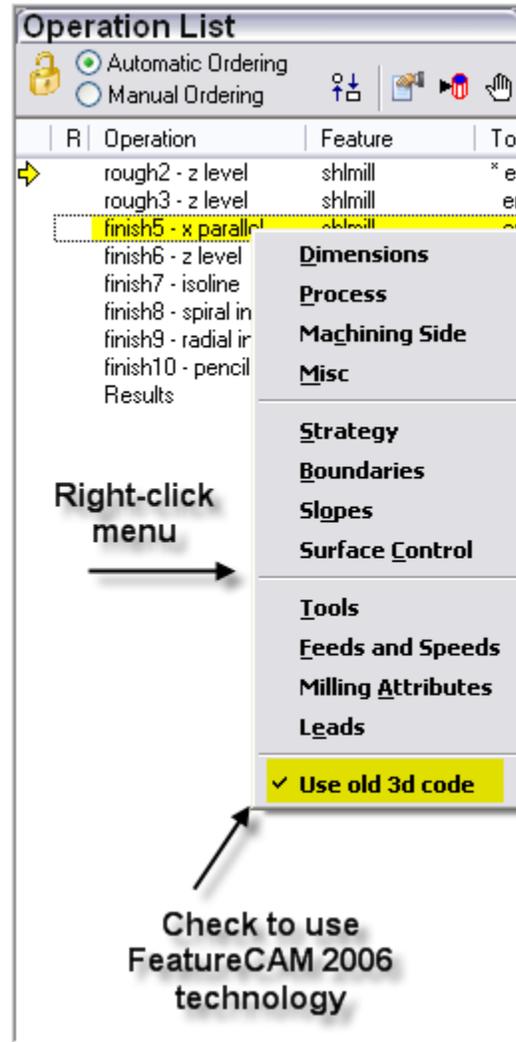
Using Version 2006 Technology in Version 2007

The version 2006 machining library is accessed via the operation list right-click menu. In the operation list window, if you select and right click on a surface milling operation you will see the menu on the right:

In this case, a file from a previous version has been read into FeatureMILL3D 2007 with an incompatible X parallel operation (e.g.: it uses the **scallop height** option). After seeing the warning dialog above, the user can right click on the surface milling operations in the operations list to see which are incompatible. The menu will have the **Use old 3d code** item checked for any incompatible operations.

Version 2006 Interface

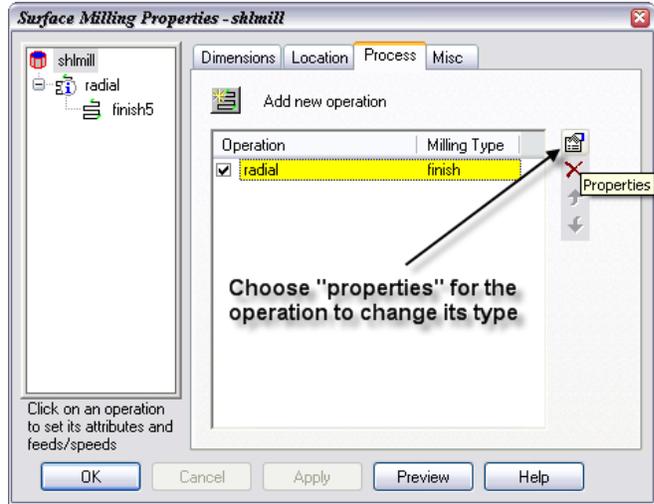
There are numerous differences between the surface milling user interface in version 2006 and 2007. The training and help files provided with version 2007 provide the documentation for version 2007. For any surface milling operations using the version 2006 machining library, the *version 2006 user interface* is used for the feature operation pages (strategy, milling, etc.). There is no additional documentation for the version 2006 user interface. Be aware that these differences can be subtle and the only way you can be sure about an operation is the right-click operations menu.



Converting an Incompatible Operation

In many cases, an incompatibility may not be significant and the user may wish to convert the operation to use the version 2007 machining library. In many cases, this is as simple as un-checking the **Use old 3d code** item on the right-click operation list menu. *The operation will retain all the compatible settings and ignore the rest.* In other cases, the user will have to use the feature Process page to change the type of an operation to a version 2007 operation.

For instance, radial inward is incompatible with version 2007 since only outward radial patterns are now supported. The user must use the process **Properties** to change to a *version 2007* radial operation and then also make sure the **Use old 3d** item is un-checked in the right-click operation list menu. This process also applies to the case where you wish to convert an old operation to a new operation with a different strategy, but retain some of your common settings (step-over, tolerance, etc.)



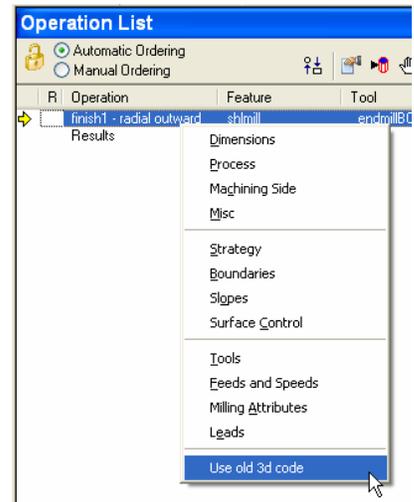
Step-by-Step Instructions for Converting an Operation

For converting an operation to use the version 2007 machining library, there are two cases:

Simple Conversion

In most cases, the operation can just be switched to use the version 2007 machining library using the operation list right-click menu:

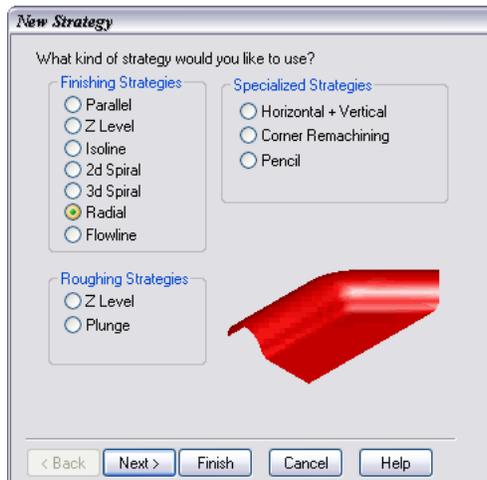
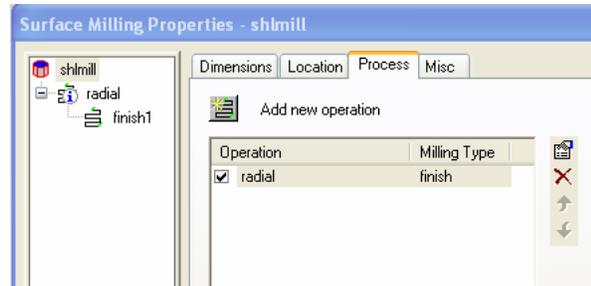
1. Make sure the *Results* window is visible on the right-hand side of the FeatureCAM window.
2. Click the *Op List* tab.
3. Select the operation in the list and right-click the mouse to bring up the fly-out menu.
4. Un-check *Use old 3d code*.



Converting an Operation by Changing Strategy

In other cases, you may need to change the strategy of an operation to use the version 2007 machining library. See table below for the cases where this is necessary. In this case the steps are:

1. Open the surface milling properties dialog box.
2. Select the feature in the tree view.
3. Click the *Process* tab.
4. Select the operation to convert (“radial”) from the list box.
5. Click the *Properties*  button. This will bring up the *New Strategy* dialog.
6. Select the “Radial” radio button and click *Finish*.



7. Go to the *Operation List* in the Results window.
8. Select the operation in the list and right-click the mouse to bring up the fly-out menu.
9. Make sure *Use old 3d code* is un-checked

Incompatibilities

This section lists all the incompatibilities between 2006 and 2007 and provides suggestions for alternative solutions using the 2007 machining library. The following table lists *all the operations that will not be automatically converted to use the new 2007 machining library* when reading a file saved from a previous version. The first column describes the operation from the previous version. The “Version 2007 technology” column gives some suggestions on how to convert the operation to use the new version 2007 machining library if applicable.

Version 2006 technology	Version 2007 technology
Plunge roughing	This is an operation that is still fully supported, but it uses version 2006 technology within version 2007. New plunge roughing operations can be created in version 2007. The operations will be labeled with use old 3d code in the operation list right-click menu.
Parallel roughing (or finish with z increment set)	Z increments for parallel are not available via the parallel strategy. Instead, version 2007 provides a zig-zag option for Z roughing which is equivalent. However, no automatic conversion is provided. Use the operation properties to change into a Z rough operation. New Z roughing provides many improvements.
Spiral, isoline, radial roughing (or finish with z increment set)	Z rough pocket roughing is fairly similar to spiral roughing. Isoline and radial roughing are rarely used and have no real equivalent in 2007.
Radial in	Version 2007 radial operation only supports radial out (unidirectional or bi-directional). Use operation properties to convert to a new 2007 radial operation. If radial in is absolutely required, use version 2006 library or convert to a flowline milling using a surface of revolution. (Also note that if you set the “radius start” and the “radius end” attributes appropriately, the resulting V2007 tool-path is similar to a v2006 “radial in”).
Z level finish or rough with bottom up option	There is no equivalent of this option in version 2007.
Z level finish with start points set	Not supported.
Z level finish with new technology	Un-check Use old 3d code to convert to new Z finish operation and then use internal only edge boundary or user defined boundaries. This is superior to

NOT checked. This is an older version of Z level technology no longer supported.	old Z slicing technology.
Z rough open pocket option	Not supported. Un-check use old 3d code and use Z rough boss option. Set an appropriate stock curve for equivalent machining with 2007.
Independent pre-drill operation. A pre-drill operation that does not precede a Z rough operation.	Pre-drilling is only supported as an option of Z roughing (or new zig-zag Z roughing). If a Z roughing operation is present, the pre-drilling is converted to use the new machining library. If your old file used pre-drilling with parallel roughing, for example, delete the pre-drilling operation, convert the parallel roughing to a new Z roughing operation (see above) and use the pre-drilling option on the Leads tab.
Parallel finish with scallop-height setting.	Not supported in 2007. Z finish and remachining support scallop-height step-overs. Another option is using parallel with the Add perp. Remach Pass option checked on the strategy tab.
Spiral with uphill only or downhill only set.	Not supported in 2007. Only parallel supports the uphill/downhill options.
Spiral with start point set	Not supported.
4 th axis wrapping	Not supported. 4 th axis wrapping for surface milling has very limited application. Version 2007 supports simultaneous 4-axis rotary milling which is superior to wrapping. No automatic conversion is provided. Use unwrapped surfaces and convert to new 4 th axis surface milling operation.
Parallel + 90 remachining. This combination has 2 separate operations.	The separate remachining operation is not supported. The combination <i>is</i> supported by using a single parallel operation that makes two perpendicular parallel passes. Convert to a version 2007 parallel operation by editing the operation on the Process page. Then uncheck the use old 3d code item on the operation list right-click menu. Then check the Add perp. remach. Pass on the operations strategy page. This is equivalent to the version 2006 combination operation.

Ignored Settings

The following settings are also no longer available in version 2007, but operations with these settings are not marked as incompatible when read from an old file. Instead, the operations will use the new machining library but these settings will be ignored. There is no warning dialog in this case. If these

settings are required, the operation can be converted back to use the version 2006 machining library by checking the **Use old 3d code** on the operation list right-click menu.

- **Edge rollover none** – **Edge rollover** has been changed to **Edge boundary** in version 2007. Edge boundary **Internal only** and **all** are fairly equivalent to the version 2006 edge rollover options. There is no equivalent to the old **edge rollover none**. Version 2007 does have a new option which is edge boundary **none** which means that no restrictions whatsoever are applied to the tool-paths regarding edge contact. The tool-paths will cut the silhouettes of the part (as long as they are not restricted by the stock or other means).
- **First feed** and **Number of first feed paths** – These options for controlling the feed rate are no longer available. Use lead-in's and ramps.
- **Plunge points** – This setting is no longer available. For Z roughing, use the pre-drilling option.
- **Retract gap** – This setting is no longer required with the new machining library. Use the **stepover rapid distance** to control retracts and plunges.
- **Improved** – Numerical improvement is no longer supported. The new machining library provides better tool-path point spacing and better accuracy than the old machining library.
- Z interleaved **spiral max slope** – This setting is no longer available. It is automatically set. It might be provided in future versions.

Index

- 2D simulation, 12
- 3D milling, 77
 - Pencil milling, 80
- 3D simulation, 12
- 3D solid simulation, 12
- 3Dmilling roughing, 78
- A**
- Arcs fly-out, 29
- B**
- Block preparation, 35
- C**
- Centerline simulation, 12
- Chaining, 45
- Circles, 43
- Circles fly-out, 29
- Curve wizard, 26
- Curves step, 9, 10
- D**
- Default attributes, 15
- Design
 - Solids, 85
 - Surface, 75
- Dimensions fly-out, 29
- Display mode toolbar, 32
- Dongle, 2
- E**
- Engraving, 65
- Evaluating FeatureCAM, 6
- F**
- Feature recognition, 67
- FeatureCAM product line, 1
- FeatureRECOGNITION, 67
- Features
 - 2.5D milling, 36
 - 2.5D milling from curves, 43
 - 3D Milling, 77
 - Editing, 41
 - Recognizing, 67
 - Turn/mill, 65
 - Turned, 51
- Features step, 9, 11
- FeatureTURN, 49, 57
- FeatureTurn/Mill, 63
- Feed optimization, 14
- Fillets, 88
- Fly-out toolbars, 23
- G**
- Geometry step, 9, 10
- Geometry toolbar, 29
- Graphics window, 23
- Grooves
 - OD Turned, 53
 - Turned face, 65
- H**
- Help
 - Context sensitive, 19
 - On-line, 17
- Hidden line, 32
- Hide fly-out menu, 26
- Holes
 - Recognizing, 70
 - Turn/mill, 65
- I**
- IGES import, 77

Index

- Import
 - IGES, 77
 - SAT, 68
 - Solid, 68
- Installation, 3

- L**
- Lines, 44
- Lines fly-out, 29
- Live tooling, 63

- M**
- Manufacturing preferences, 15
- Manufacturing results window, 23

- N**
- NC code
 - Generating, 41
 - Saving, 42
- NC code step, 9, 14

- O**
- Optimize feedrates, 14
- Order of operations, 54

- P**
- Part view toolbox, 21, 24
- Pencil milling, 80
- Plunge point, 47
- Pockets
 - Recognizing, 70
- Post processor
 - Editing, 16
 - Selecting, 42

- R**
- Region of interest, 12
- Registration, 4
- Remapping tooling, 14

- S**
- SAT import, 68
- Saving NC code, 42
- Security key, 2
- Selecting Graphical Objects, 25
- Setups button, 27
- Shading, 31
- Show fly-out menu, 26
- Simulation Toolbar, 12
- Snap discrimination dialog, 44
- Snap feedback, 28
- Snap mode toolbars, 27
- Snap modes, 26, 27
- Solid wizard, 26
- Standard toolbar, 22
- Steps, 9
 - Curves, 9, 10
 - Features, 9, 11
 - Geometry, 9, 10
 - NC code, 9, 14
 - Stock, 9, 10
 - Toolpaths, 9, 12
- Steps toolbar, 29
- Stock preparation, 35
- Stock step, 9, 10
- Surface modeling, 75
- Surface wizard, 26

- T**
- Text, 65
- Threads, 53
- Tool load, 12
- Tool mapping, 14, 41
- Tool numbers, 14
- Tool pockets, 14
- Tool slots, 14
- Toolbars
 - Customizing, 30
 - Display mode, 32
 - Fly-out, 23
 - Geometry, 29
 - Simulation, 12
 - Snap mode, 27
 - Standard, 22
 - Steps, 29
- Toolpaths step, 9, 12
- Trademark information, iv
- Transform, 22
- Trial versions, 6

Turn/mill, 63
Turning, 49, 57

U

Undo, 22

V

Variable radius fillets, 88

W

- Window
 - Graphics, 23
 - Manufacturing results, 23
- Wizards
 - Curve, 26
 - Solid, 26
 - Surface, 26