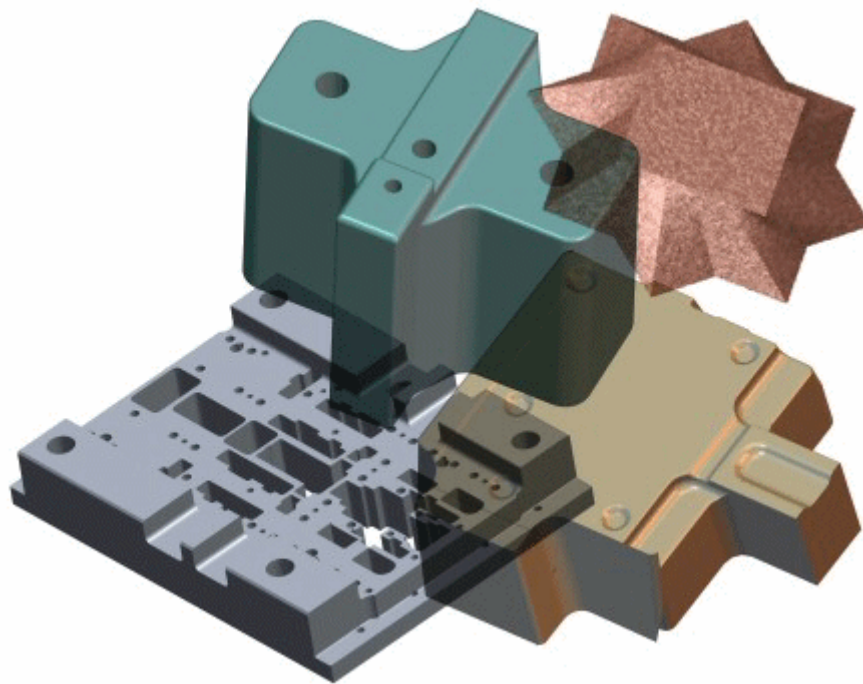




camworks
powered by SolidWorks

CAMWorks 2007 Wire EDM Tutorial



teksoft
Software Solutions
Simple | Powerful | Productive

License Agreement

In receiving a software product described herein, you are granted a nonexclusive license to use that software on one computer. TekSoft, Inc. retains full title to and ownership of the product. Use of the product constitutes acceptance of this agreement. Duplication of the software product for any purpose, other than backup protection, including duplication for any commercial purpose, is prohibited by law.

TekSoft, Inc. reserves the right to revise and improve products as it sees fit, and to revise the specifications and information contained herein without prior notice. Due to continuing product development, specifications and capabilities described in this manual are subject to change without notice.

Warranty

TekSoft, Inc. will replace defective software for a period of 90 days from date of delivery, at no charge. TekSoft, Inc. makes no other warranty, either express or implied, including, but not limited to, any implied warranties of merchantability or fitness for a particular purpose, regarding any product described herein. In no event shall TekSoft, Inc. be liable to anyone for special, collateral, incidental, or consequential damages in connection with or arising out of the purchase or use of these products. The sole and exclusive liability of TekSoft, Inc., regardless of form of action, shall not exceed the purchase price of the materials described herein.

Trademarks

The following and other product names and corporate references herein are registered or proprietary trademarks of their respective owners.

TekSoft™ is a trademark of TekSoft, Inc.

CAMWorks™ is a trademark of Geometric Software Solutions Co. Ltd.

SolidWorks® is a registered trademark and FeatureManager™ is a trademark of SolidWorks Corporation.

Microsoft® Windows® is a registered trademark of Microsoft Corporation in the United States and/or other countries.

All other brands and names are property of their respective owners.

Copyright ©2006 TekSoft, Inc. All Rights Reserved.



TekSoft is a subsidiary of Geometric Software Solutions Co. Ltd.

CW60h October 2006

Table of Contents

CHAPTER 1 LEARNING WIRE EDM BASICS	1-1
Wire EDM 1	1-2
What You'll Learn.....	1-2
Steps to Generate EDM Toolpaths and NC Code.....	1-2
Step 1: Model Part in SolidWorks or Import Part.....	1-2
Step 2: Change to CAMWorks Feature Tree	1-3
Step 3: Define the Machine.....	1-4
Step 4: Define the Stock.....	1-6
Step 5: Define Machinable Features	1-7
Step 6: Generate Operation Plan/Adjust Operation Parameters	1-8
Step 7: Generate Toolpaths	1-10
Step 8: Post Process Toolpaths.....	1-13
CHAPTER 2 LEARNING 2 AXIS CONTOUR CUTTING	2-1
2-Axis Contour 1	2-2
What You'll Learn.....	2-2
Extracting Machinable Features and Customizing AFR.....	2-2
Generating a 2 Axis Contour Operation and Toolpaths.....	2-4
Simulating and Post Processing Toolpaths	2-5
2-Axis Contour 2	2-6
What You'll Learn.....	2-6
Inserting 2.5 Axis EDM Features.....	2-6
Generating Operations and Toolpaths.....	2-8
Changing the Machining Order and Post Processing.....	2-9
2-Axis Contour 3	2-10
What You'll Learn.....	2-10
Inserting a Part Setup	2-10
Inserting Vertical and Tapered Open Profile Features.....	2-11
2 Axis Contour 4	2-14
What You'll Learn.....	2-14
Changing the Processing Order.....	2-14
CHAPTER 3 LEARNING 4 AXIS CONTOUR CUTTING	3-1
4-Axis Contour 1	3-2
What You'll Learn.....	3-2
Defining the Machine, Stock and Machining Direction	3-2
Defining a 4 Axis Die Feature Interactively	3-3
Generating a 4 Axis Contour Operation and Toolpaths.....	3-4

Table of Contents

- 4-Axis Contour 2 3-6
 - What You'll Learn..... 3-6
 - Automatically Extracting Sync Curves to Define a 4 Axis Feature 3-6
 - Generating the 4 Axis Contour Operation and Toolpaths..... 3-8
- 4-Axis Contour 3 3-9
 - What You'll Learn..... 3-9
 - Using Sketches to Define a 4 Axis EDM Feature..... 3-9
 - Changing the Leadin Point..... 3-11
- 4-Axis Contour 4 3-12
 - What You'll Learn..... 3-12
 - Picking Sync Curves to Define a 4 Axis EDM Feature 3-12

INDEX

Chapter 1 Learning Wire EDM Basics

This chapter provides an opportunity to learn CAMWorks Wire EDM through a step-by-step tour of the features and functions.

The exercises in this chapter are intended to show you how to use CAMWorks and may not correspond to actual machining practices.

Wire EDM 1

What You'll Learn

Steps to Generate EDM Toolpaths and NC Code

The following steps are used to generate wire EDM toolpaths and NC code:

1. Model the part or open the part file in SolidWorks.
2. Change to the CAMWorks Feature tree.
3. Set the machine type and machine parameters.
4. Define the stock shape and material.
5. Define machinable features.
6. Generate the operation plan and adjust operation parameters.
7. Generate toolpaths.
8. Post process the toolpaths.

The next series of exercises show you how to generate toolpaths on a SolidWorks part model. In order to give you a general understanding of how to use CAMWorks, you work with a part that was previously modeled in SolidWorks. When you define the operations and toolpaths, you will follow steps that are not explained in depth. This is done to show you the basics of generating toolpaths from start to finish without getting into the details at this time. Sample parts are provided for the exercises in this manual. When you install CAMWorks, these files are installed automatically.

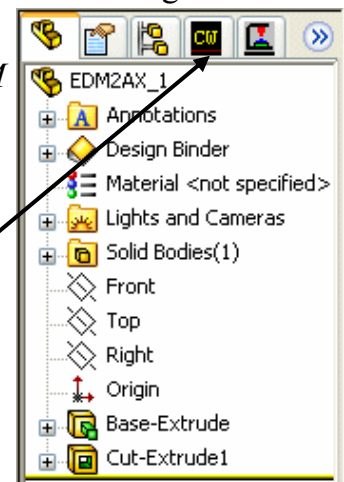
Step 1: Model Part in SolidWorks or Import Part

A part is a solid that is created with SolidWorks or imported into SolidWorks from another CAD system via an IGES, Parasolid, SAT file, etc. This exercise uses an existing SolidWorks part.

1. Open the part file **EDM2AX_1.SLDPRT** in the *\Examples\EDM* folder inside the CAMWorks folder (e.g., *\Program Files\CAMWorksxxxx\Examples\EDM*).


The FeatureManager design tree lists the features, sketches, planes and axes in the part.

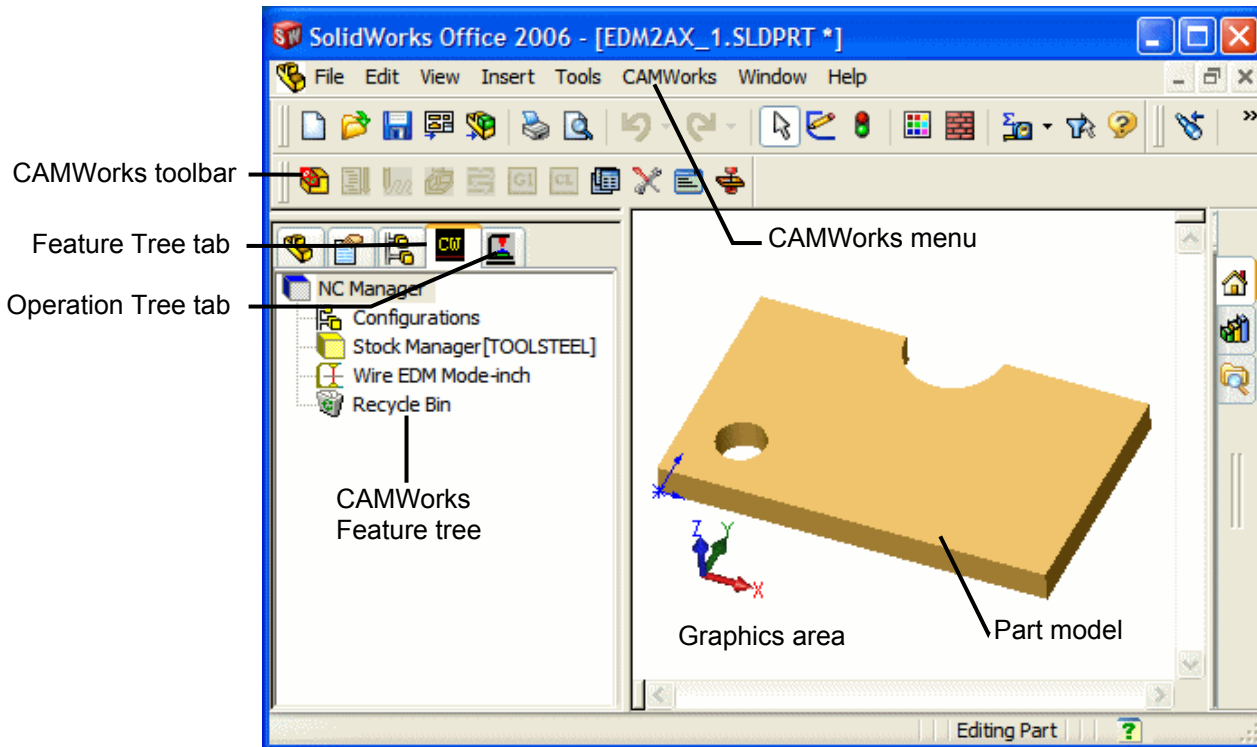
The tabs at the bottom or top are for moving between the SolidWorks trees and the CAMWorks trees.



Step 2: Change to CAMWorks Feature Tree





1.  Click the CAMWorks Feature Tree tab. The CAMWorks Feature tree displays.

 The Machine item may indicate that a mill machine is currently selected. Step 3 on page 1-4 explains how to change the machine to an EDM machine.



CAMWorks Machining Trees

The CAMWorks trees provide an outline view of the machining information for the model. Initially, the CAMWorks Feature tree shows the Configurations, NC Manager, Stock Manager, Machine and Recycle Bin items. As you follow the steps to generate an NC program, this tree expands to include Part Setups and machinable features. The tabs are for moving between the SolidWorks trees and the CAMWorks trees.

-  **Configurations** - Multiple CAMWorks datasets are supported in part mode. Each dataset is called a configuration. You can use configurations to support multiple machines and SolidWorks configurations.
-  **Stock Manager**
The stock is the material from which the part will be machined. You can define the stock as a rectangular bounding box or as an extruded sketch. You can also specify the type of material.
-  **Machine**
The Machine item defines the machine type that will be performed on the part (i.e., mill, turn, mill-turn or EDM). For mill and turn machines, the tool definitions and the machine controller (post processor) are also specified. The machines are set up in the Technology Database.
-  **Recycle Bin**
The Recycle Bin in the CAMWorks Feature tree is used to store machinable features that you do not intend to machine.

CAMWorks Menu

1. Click CAMWorks on the SolidWorks menu bar. The CAMWorks menu lists the CAMWorks commands. The commands are explained in the CAMWorks online Help.
2. Right click on the NC Manager in the tree. This is a shortcut menu. The right-click shortcut menus provide access to a variety of commands.

CAMWorks Toolbar

The CAMWorks toolbar provides access to the main CAMWorks commands found on the CAMWorks menu. Clicking a toolbar button is the same as selecting a command from the NC Manager level, regardless of the active item in the tree.

1. Locate the CAMWorks toolbar and click the Options button.
2. In the Options dialog box, click each tab to view the options and settings that you can change in CAMWorks.
3. Click the Help button at the bottom of the dialog box. Each tab is explained in the online Help.
4. Click the Close button in the upper right corner of the Help window to close the window.
5. Click Cancel to close the Options dialog box.



Step 3: Define the Machine

The Machine item in the tree identifies the type of machine: mill, turn or wire EDM. The machine type defines the machinable feature set that can be recognized automatically and defined interactively.

The icons that display in the tree identify the current machine:



An alternative machine can be selected at any time to output different G-code programs for alternative machine tools. If the machine type changes, then all features and operations will be deleted.

Define the machine:

1. Right click the Machine item in the CAMWorks Feature tree.

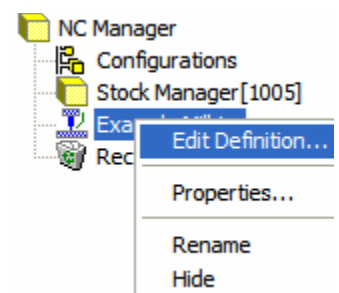
The shortcut menu displays. Right-click shortcut menus display commands that are appropriate for the item that is highlighted in the tree.

2. Select Edit Definition on the shortcut menu.



Did You Know ...

As an alternate to right clicking items in the tree and selecting the Edit Definition command, you can double click the following items: Stock Manager, Machine, Setup, Features and Operations.



The Machine dialog box displays the Machine tab. Milling is the default machining type that is set when CAMWorks is first installed. The default machining type is specified in the Technology Database. Before using CAMWorks for milling and turning to machine your parts, make sure you define the machine tools available in your facility. EDM machines are not currently defined in the Technology Database.


3. In the Available machines list, highlight Wire EDM Mode-in and click the Select button.

4. Click OK.

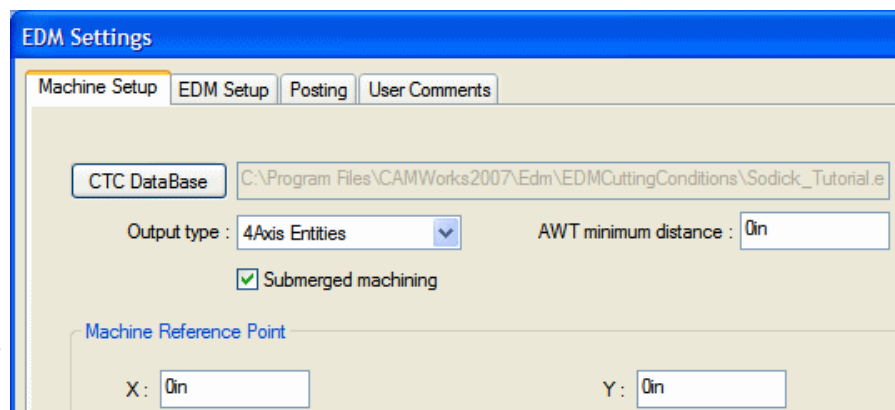
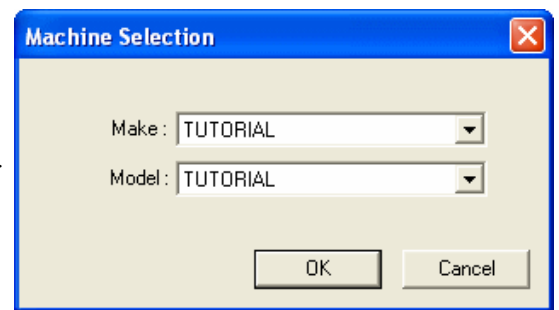
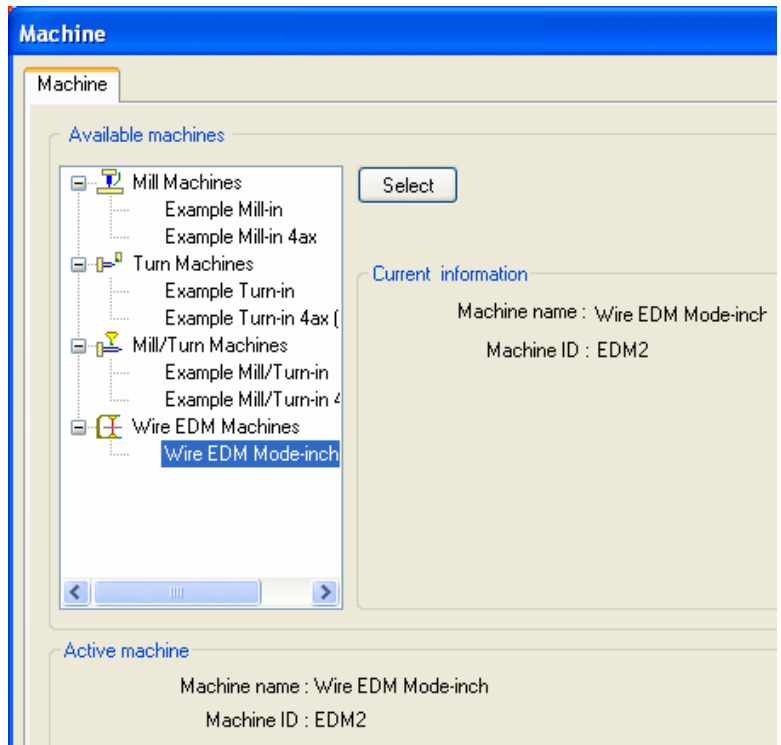
5.  Right click Wire EDM Mode-in in the tree and click Select Machine on the shortcut menu.

6. In the Machine Selection dialog box, select TUTORIAL for the Make and TUTORIAL for the Model, then click OK.

Use this selection for the exercises in this manual. When you use CAMWorks to machine your own parts, select your machine Make and Model from the drop down list.

7.  Right click Wire EDM Mode-in and select Edit Settings on the shortcut menu. The EDM Settings dialog box displays. On the Machine Setup tab, the path to the CTC Database is identified. This database has been set up for the TUTORIAL machine. The cutting condition database sets machine-specific parameters for power settings, etc.

8. Click the EDM Setup tab. This tab allows you to set the wire diameter, maximum taper angle, etc.
9. Click the down arrow next to the Wire diameter and select 0.008.



- Click the Posting tab.

The Posting page identifies the post processor. This post processor has been set up for the TUTORIAL machine and does not need to be changed.

The options on this page provide information used to generate the NC program, including the processing order.

- For the Die process order, use By Complete Feature.
With this option, the NC program will be generated in the following processing order: for each feature, cut the rough pass, glue stop cut off, then all skim passes.

The EDM Settings dialog box, Posting tab, shows the following configuration:

- Post paths:**
 - Post: C:\Program Files\CAMWorks2007\Edm\Posts\Sodick_Tutorial.pst
 - Hole Post: C:\Program Files\CAMWorks2007\Edm\Posts\Sodick_K1CN.hpst
- Process order:**
 - Punch: By Complete Feature
 - Die: By Complete Feature
 - ☐ Process by setup
- Output options:**
 - Program number: 1000
 - Line #: 1
 - Line # increment: 1
 - Subprogram start #: 7000
 - Subprogram increment #: 1
 - ☒ Use subprograms
 - ☒ Output comments
 - ☒ Output cutting conditions
 - ☐ Output cuts to separate files

Step 4: Define the Stock

The stock is the material from which the part will be machined. The default stock is the smallest bounding box that the part will fit into. You can change the stock definition by offsetting the bounding box from the part or by defining the stock from a sketch and a depth.

- Double-click Stock Manager in the Feature tree. The Manage Stock dialog box displays.
For this exercise, use the default bounding box and the default Material.
- Click OK to close the Material Selection dialog box.
- Click OK to close the Manage Stock dialog box.

The Manage Stock dialog box shows the following configuration:

- Parts:** NC Manager
- Stock type:** Bounding Box (selected)
- ☐ Use optimized bounding box
- Stock:** Stock [Bounding Box]
- Bounding box offset:**
 - X+: 0in, X-: 0in
 - Y+: 0in, Y-: 0in
 - Z+: 0in, Z-: 0in
 - Buttons: Get default, Set default
- Stock size:** X: 5in, Y: 3in, Z: 0.4in
- Material:** 1005
- Stock group:** Others
- Buttons: OK, Cancel, Help

Step 5: Define Machinable Features

In CAMWorks, machining can be done only on machinable features. You use the following two methods to define machinable features:


- Automatic Feature Recognition (AFR)
Automatic Feature Recognition analyzes the part shape and attempts to define Die (Pocket) and Open Profile machinable features for Wire EDM.
- Interactively created features
If AFR does not recognize a feature you want to machine, you need to define the feature using the Insert 2.5 Axis EDM Feature and Insert 4 Axis EDM Feature commands. Using the interactive method, you can define Die (Pocket), Punch (Boss) and Open Profile machinable features.


The idea of AFR is to analyze the part for features that can be machined. This process is much the same as what you would do if you were to pick up a part that you had to machine. You would look it over, take measurements, and begin deciding how to define areas or features to machine and what machining processes you would need.


CAMWorks is not machining the SolidWorks features directly. It creates a separate list of Machinable Features instead. This is because design features are typically not the same as manufacturing features.

Using Automatic Feature Recognition (AFR)

Define machinable features automatically:

1.  Click the CAMWorks Options button on the CAMWorks toolbar.
2. Click the EDM Features tab, make sure Perimeter feature is not checked, then click OK.
The *Extract machinable features* section on this tab allows you to control whether or not AFR finds certain feature types and how it classifies them.

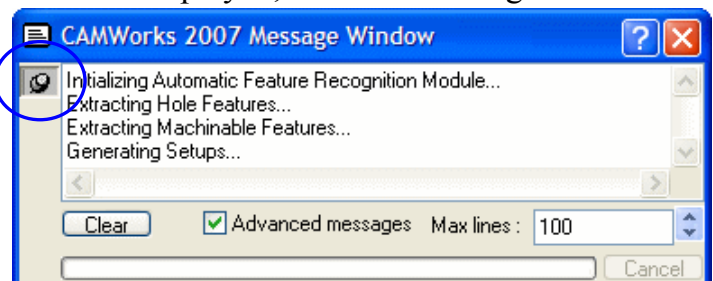
3.  Click the Extract Machinable Features button on the CAMWorks toolbar or right click NC Manager in the tree and select Extract Machinable Features on the shortcut menu.

4.  If the CAMWorks Message Window is not displayed, click the Message Window button on the CAMWorks toolbar.

This window reports the progress and status of the current process. Generating Setups is the last item.

You can control whether this window displays temporarily or permanently by clicking the Pushpin button.

CAMWorks generates the Part Setup and machinable features. The items display in the CAMWorks Feature tree.

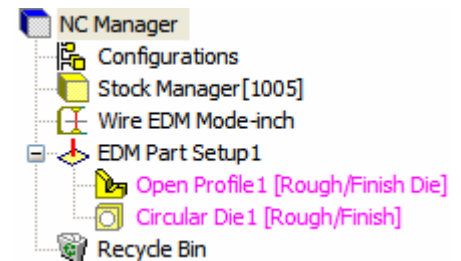


The Part Setup is the 2 axis plane that the tool movement will be based on. It has an origin location, and X, Y, and Z direction vectors. The Part Setup is created automatically; however, you can move the origin, and you can change the direction and angles of the X and Y axes.

A Part Setup is created for each different tool orientation. There is only one Part Setup for this part because all features can be machined using a single setup orientation. For each Part Setup, the machinable features are listed in the order in which they were recognized.

The Feature tree allows you to:

- Rename, suppress, and delete machinable features
- Change machinable feature parameters
- Set the default order in which the features are machined
- Insert additional 2.5 Axis and 4 Axis EDM features
- Search for a feature based on item name
- Hide or show feature display in graphics area
- Generate an Operation Plan and find the first operation for a feature




Did You Know ...

New features, interactively inserted features that have no operations and features that could not produce operations because the feature conditions have not been defined in the Technology Database display in a different color. You can set the color on the Display tab in the Options dialog box.

Step 6: Generate Operation Plan and Adjust Machining Parameters

An Operation Plan contains information on how each machinable feature is to be machined and how the NC code will be output. When Generate Operation Plan is run, operations for each machinable feature are created automatically. In some situations, the operations defined for a feature may not be sufficient and additional operations may be required. You can insert operations manually using the Insert Operation command. This command is explained in the CAMWorks online Help.

1.  Click the Generate Operation Plan button on the CAMWorks toolbar or right click NC Manager or Part Setup1 in the Feature tree and select Generate Operation Plan.

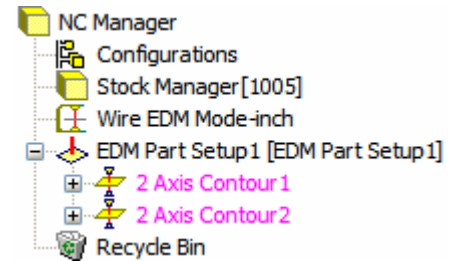
Note that clicking the toolbar button is the same as selecting the command from the NC Manager level, regardless of the active item in the tree.

CAMWorks generates the operation plan for all the machinable features in Part Setup1. The operations are listed in the CAMWorks Operation tree, which displays automatically.

The CAMWorks Operation tree provides an outline view of the operations for the machinable features. Initially, operations are listed under the Part Setup in the same order as the machinable features. At the top of the tree is the NC Manager. The Stock Manager and Machine items function the same as in the CAMWorks Feature tree.

The CAMWorks Operation tree allows you to:

- Insert, rename, suppress, and delete operations
- Drag and drop copy operations and features
- Change operation parameters
- Change the machining order
- Generate toolpaths
- Simulate toolpaths
- Post process the toolpaths
- Hide or show toolpath display
- Search based on item name



To the left of each toolpath operation is a plus sign (+). Clicking a plus sign displays the name of the Machinable Feature that this operation is going to machine. These Machinable Feature items can be used to view geometric information.



Did You Know ...

If an operation displays in a color instead of black, then toolpaths have not been generated. This occurs when you insert a new operation interactively, you insert a new feature interactively and generate operations for the new feature, or CAMWorks cannot generate the toolpath for an operation because of an error in the toolpath algorithm or a parameter is not correct. You can set the color on the Display tab in the Options dialog box.

Generate Operation Plan generates one cutting operation per machinable feature. Each operation contains machining parameters that affect how the toolpath is created and specific parameters that will be output to the NC program. These parameters can be edited before post processing the part.

Edit operation parameters using the Edit Definition command on the shortcut menu:

1. Double click 2 Axis Contour1 in the Operation tree.

The 2 Axis Contour dialog box displays. This dialog box gives you access to all the parameters used to create the toolpath.

2. On the EDM tab, set the following:
 - Number of skims = 2
 - Glue Stop = No Glue Stop
3. On the Leadin/Out tab, set the Leadin and Leadout to Parallel with a Length of 0.10in, then click OK.


Note that if you click the Save button before clicking OK, the current settings on the EDM and Leadin/Out tabs become the defaults for newly inserted or generated operations on this part or other parts.

4. Double-click on 2 Axis Contour2 in the tree.

5. On the EDM tab, set the following:
 - Number of skims = 2
 - Glue stop = Use Glue Stop
 - Stop distance = 0.1in.
6. On the Leadin/Out tab, set the Leadin and Leadout to Perpendicular with a Length of 0.1in, then click OK.

Step 7: Generate Toolpaths

CAMWorks calculates toolpaths using the operation parameters and the feature's size and shape.

1.  Click the Generate Toolpath button on the CAMWorks toolbar or right click the Part Setup1 or NC Manager items in the Operation tree and select Generate Toolpath on the shortcut menu.

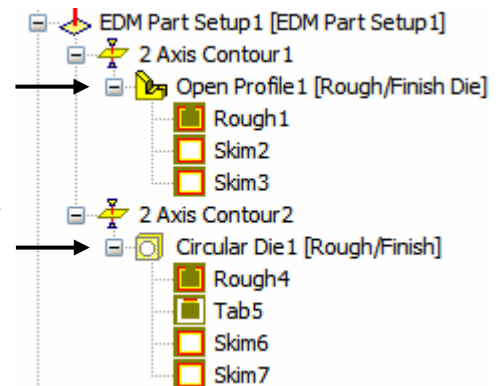
Clicking the toolbar button is the same as selecting the command from the NC Manager level, regardless of the active item in the tree.

CAMWorks calculates the toolpaths for each operation in the Part Setup.

You can also generate toolpaths for each operation individually by right clicking on an operation and clicking Generate Toolpath on the shortcut menu.

2. Click the + sign next to 2Axis Contour1 and 2 Axis Contour2 in the tree, then click the + sign next to Open Profile1 and Die1.

Under each machinable feature, the toolpath passes that were generated for that feature are listed. Notice a Tab cut is generated for the second operation since you set the Glue Stop to Use and set a Stop distance.




3. Click the first operation in the tree.

The toolpaths for that operation display on the part.

4. Click each toolpath pass under the operation. The toolpath for that pass displays.

Note that the toolpaths for each cut are on center. Compensation for each pass will be handled by the cutter compensation at the machine.

CAMWorks provides the ability to simulate the toolpath showing the tool movement and the resulting shape of the part.

5.  Click the Simulate Toolbar button of the CAMWorks toolbar or right click Part Setup1 in the tree and select Simulate Toolpath.

The Simulate Toolpath toolbar displays. The toolbar controls allow you to:






- Customize the display of the stock, tool and tool holder (wireframe, translucent, shaded, or no display).
- Run the simulation for all or selected operations.

- When simulating an operation, the simulation can be for the current operation or for all previous operations up to the selected operation.
- Pause the simulation and dynamically change the orientation of the part using zoom, pan, rotate, etc.
- Control the simulation speed by dragging the Simulation Speed Control slider up or down.

If you want to simulate only the toolpaths for a given operation, you can right click that operation, then select Simulate Toolpath.

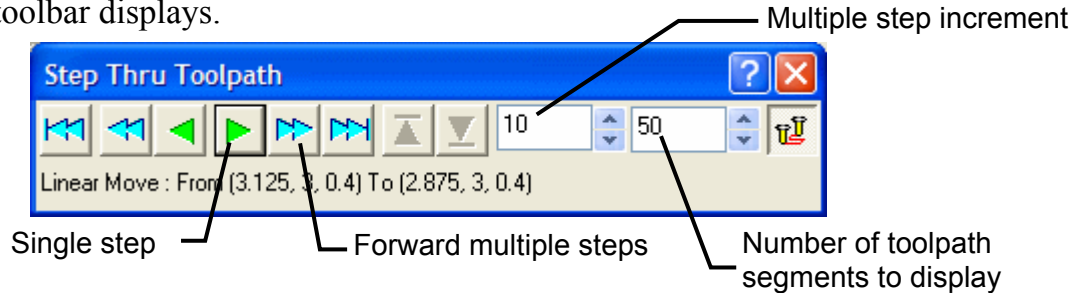
- Set the toolbar buttons as shown in the figure.




-  Click the Run button.
The simulation runs.
-  Click the Select Chips to Remove button. This button is enabled when the simulation is complete and provides two methods for removing loose chips.
- Pick the two loose stock chips to remove, then press the D (delete) or R (remove) key on the keyboard. The loose chips are removed in the graphics area.
-  Click the Select Chips to Remove button to exit the function.
- Click the Goto Start button and Run the simulation again.
-  Click the Select Chips to Remove button.
- Hold down the Ctrl key and pick the stock piece you want to keep, then press the D or R key on the keyboard.
If there are many loose chips to remove, picking the stock piece may be faster than picking each chip.
-  Click the X button in the upper right corner to close the simulation and return to the SolidWorks display.

CAMWorks also provides the ability to step through the toolpath using the Step Thru Toolpath command.

1. Right click an operation in the tree and select Step Thru Toolpath. The Step Thru Toolpath toolbar displays.




2. Click the Single Step button (single arrow) several times.
The active toolpath node is highlighted.
3. Set the Step Increment to 3 and click the Forward Multiple Steps button (double arrow) several times.
4. Click the Goto End button. The toolpaths are completed.
5.  Click the X button in the upper right corner to exit.



Did You Know ...

CAMWorks provides an option to display the XYZ position of the tool in the Message Window during Simulation and Step Thru Toolpath. To activate this option, select the Options command on the CAMWorks menu or the NC Manager shortcut menu. On the General tab, select the Message Window option. On the Simulation tab, select the Cutter coordinates in Message Window option.

Save the part with the machinable feature data:

1.  Click the Options button the CAMWorks toolbar.
Note that you can also click CAMWorks on the menu bar or right click NC Manager in the tree and select the Options command.
2. On the General tab, make sure the Save/Restore part option is checked, then click OK.
If this option is checked, when you save and close a part document, the machining data is saved. When the part document is opened again, the machining data is restored along with the part design information.
If this option is not checked, when you save and close a part document that contains at least one CAMWorks Setup, a message indicates that Save/Restore is disabled. If you click Yes, CAMWorks saves all CAMWorks data before closing the file. If you click No, CAMWorks closes the part and discards any new CAMWorks data since the last save.
3. Click Save As on the File menu.
4. In the Save As dialog box, type **cwedm1** and click the Save button.


SAVE FREQUENTLY!

- When you open a file, you are actually working on a copy of the file. The original is still stored on disk. Periodically saving your file ensures that your latest work is retained.
- CAMWorks provides an Auto save option on the General tab in the Options dialog box for automatically saving your CAMWorks data.
- Frequent saves prevent having to redo a time-consuming model or CAM operation. If a power failure occurs, you will lose whatever you have been working on.

Step 8: Post Process Toolpaths

Post processing is the final step in generating the NC program file. This step translates generalized toolpath and operation information into NC code for a specific machine tool controller. CAMWorks creates NC code for each toolpath in the order the operation appears in the Operation tree and the Process order specified on the Posting tab in the EDM Settings dialog box. When you post process a part, CAMWorks creates the NC program file. This is a text file that you can read, edit and print using a word processor or text editor.

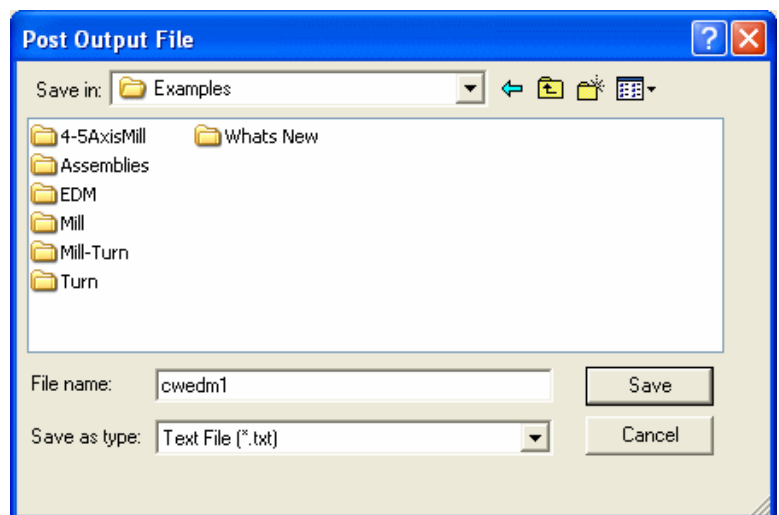
In this exercise, you post process all the operations and generate the NC program:

1.  Click the Post Process button on the CAMWorks toolbar or right click NC Manager in the Operation tree and click Post Process on the shortcut menu.

The Post Output File dialog box displays so you can name the NC program file.

If you want the file in another location, you can change the folder.

If the Post Process command is grayed out on the CAMWorks menu or on a shortcut menu, make sure that you have generated the toolpaths.



2. Click the down arrow to the right of the Save as type box.

CAMWorks provides a list of commonly used extensions that you can select. For this exercise, use the *txt* extension.

3. Click Save.

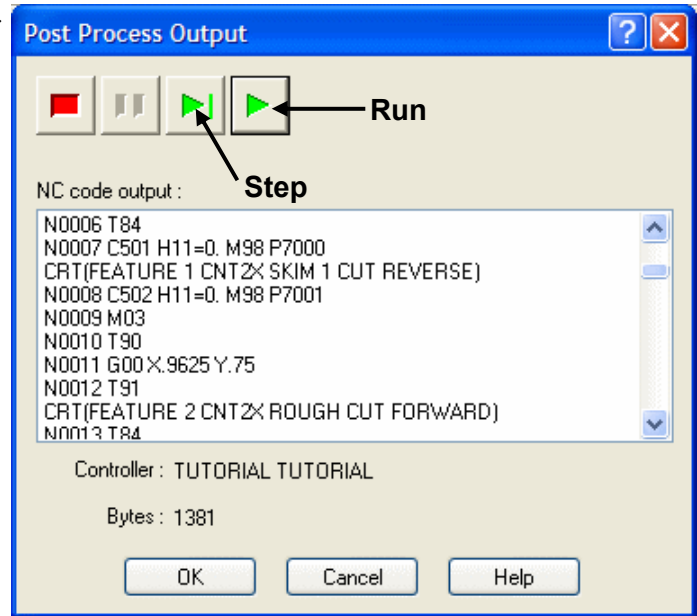
You do not have to type the extension. Naming the post output file the same as the part file is the most common way of saving parts and NC programs. Both files can have the same name because they have different extensions.

The Post Process Output dialog box displays.

4. Click the Step button on the control bar at the top.

CAMWorks starts to generate the NC program and the first line of NC code displays in the NC code output view box. The post processing mode is set to post process one line of code at a time (Step mode).

5. Click the Step button. The next line of NC code displays.
6. Click the Run button. Post processing continues until it is completed.
7. When the post processing is finished, view the code using the vertical scroll bar.
8. Click OK to close the Post Process Output dialog box.



Chapter 2 Learning 2 Axis Contour Cutting

This chapter provides an opportunity to learn CAMWorks Wire EDM through step-by-step exercises. We recommend that you go through the exercise in Chapter 1 first.

The exercises in this chapter are intended to show you how to use CAMWorks Wire EDM and may not correspond to actual machining practices.





2-Axis Contour 1

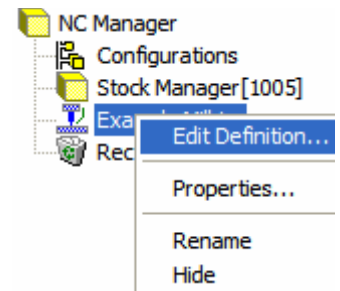
CAMWorks is a feature-based machining system. Feature-based machining provides numerous benefits because the definition of the feature enables a higher level of automation when creating machining operations and associated toolpaths. Adding to this benefit, CAMWorks provides both an automatic method of extracting features called Automatic Feature Recognition (AFR), and an interactive method using the Insert 2.5 Axis EDM Feature function. A CAMWorks file can contain both types of features.

This exercise demonstrates some of the features that are identified automatically by the Extract Machinable Features command and explains how to control the types of features found by AFR.




What You'll Learn

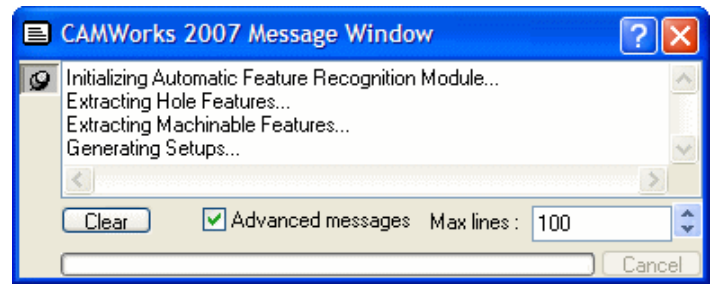
Extracting Machinable Features and Customizing AFR

1. Open the part file **EDM2AX_2.SLDPRT** in the *\Examples\EDM* folder inside the CAMWorks folder (e.g., *\Program Files\CAMWorksxxx\Examples\EDM*).
2.  Click the CAMWorks Feature Tree tab.
3. Double click the Machine item in the CAMWorks Feature tree. The Machine dialog box displays.
4. In the Available machines list, highlight Wire EDM Mode-mm, click the Select button, then click OK.
5.  Right click Wire EDM Mode-mm in the tree and click Select Machine on the shortcut menu.
6. In the Machine Selection dialog box, select TUTORIAL for the Make and TUTORIAL for the Model, then click OK.
7.  Right click Wire EDM Mode-mm in the tree and select Edit Settings on the shortcut menu.
8. On the EDM Setup tab, make sure the Wire diameter is set to 0.20mm.
9. On the Posting tab, make sure the Process order for Die is set to By Complete Feature, then click OK.
With this option, the NC program will be generated in the following machining order: for each feature, cut rough pass, glue stop cut off, then all skim passes.
10.  Right click Stock Manager in the Feature tree and select Edit Definition on the shortcut menu.
The Manage Stock dialog box displays.
11. For this exercise, click OK to use the default bounding box and material.

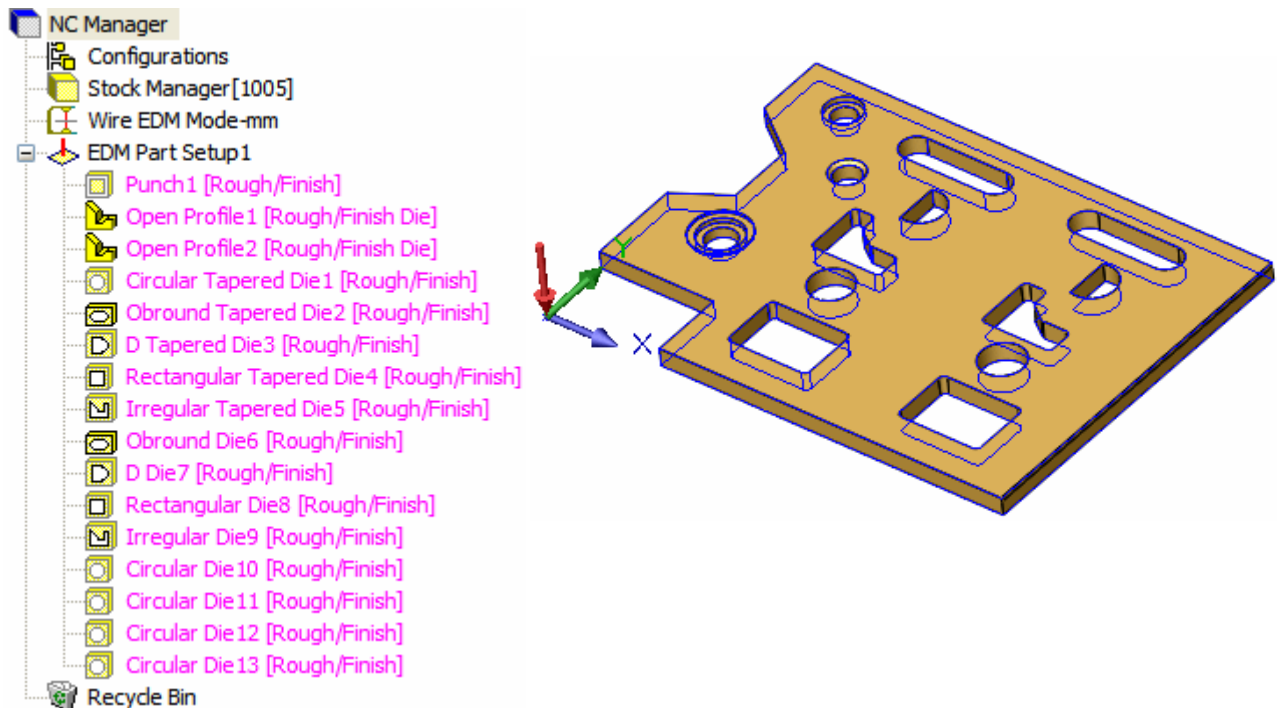


CAMWorks allows you to control the types of features that are created by AFR:

1.  Click the Options button on the CAMWorks toolbar.
2. On the EDM Features tab, make sure the Perimeter punch feature is checked and click OK.
3.  Click the Extract Machinable Features button on the CAMWorks toolbar or right click NC Manager in the tree and select Extract Machinable Features on the shortcut menu.
4.  If the CAMWorks Message Window is not displayed, click the Message Window button on the CAMWorks toolbar. This window reports the progress and status of the current process. Generating Setups is always the last item. When you see it, you know the AFR process is almost complete.



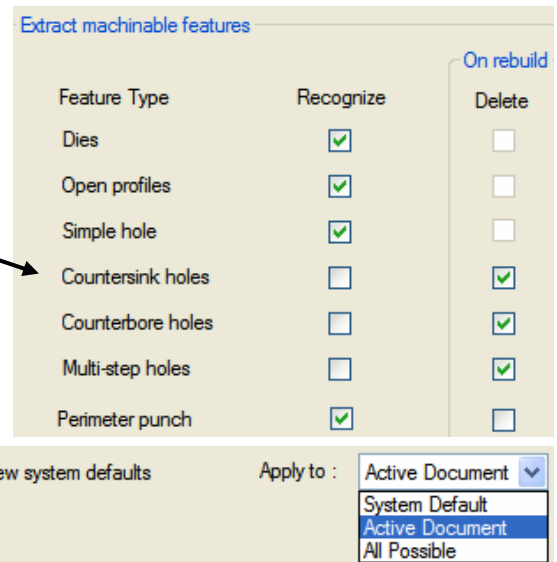
CAMWorks generates the Part Setup and machinable features.



5. Click each feature item in the tree to see the corresponding feature on the part.
AFR has generated all possible Open Profile and Die features. However, you may not want to use Wire EDM to machine all of these features, for example the countersink, counterbore and multi-step holes. You can delete these features individually by highlighting the features and pressing the Delete key or you can customize AFR to omit recognizing these features.

6.  Click the Options button on the CAMWorks toolbar.

7. On the EDM Features tab, remove the check marks from the Countersink holes, Counterbore holes and Multi-step holes options and check the On Rebuild Delete option for these features types.



8. Optionally, change the Apply to: setting.

This option allows you to select how the settings on this tab will be applied:

- Active Document - the part that you are currently working on
- System Default - all new parts that are created (note that the values that displayed when you opened this dialog box will be retained for the current part)
- All Possible - the current part as well as all new parts



Did You Know ...

CAMWorks data is saved in part and assembly template documents. For new part/assembly documents, the CAMWorks settings in the selected template will be used, not the CAMWorks System Defaults.

Templates allow you to maintain as many different documents for parts or assemblies as you need. For example, you can create separate part templates for Mill, Turn and Wire EDM with the machine type stored.

If you want the System Defaults to apply, disable the Save/Restore part or Save/Restore assembly option on the General tab in the CAMWorks Options dialog box, then save the template with a new name.


9. Click OK.

10.  Click the Extract Machinable Features button on the CAMWorks toolbar or right click NC Manager in the tree and select Rebuild.


The Circular Die features for the countersunk, counterbore and multi-step holes are now in the Recycle Bin.

Generating a 2 Axis Contour Operation and Toolpaths





Generate operations and toolpaths, then post process:

1.  Click the Generate Operation Plan button on the CAMWorks toolbar or right click NC Manager or Part Setup1 in the Feature tree and select Generate Operation Plan on the shortcut menu.



When you select Generate Operation Plan, operations are created for every selected feature. Operations are not generated for suppressed features and features in the Recycle Bin.

2.  Click the Generate Toolpath button on the CAMWorks toolbar or right click the Part Setup1 or NC Manager items in the Operation tree and select Generate Toolpath on the shortcut menu.

Simulating and Post Processing Toolpaths

1.  Click the Simulate Toolpath button on the CAMWorks toolbar or right click NC Manager or Part Setup1 in the tree and select Simulate Toolpath.
2. Set the display:
 - Stock, Tool and Holder = Shaded Display
 - Target Part = No Display
3.  Click the Run button.
4.  Click the Select Chips to Remove button.
5. Hold down the Ctrl key and pick the stock piece you want to keep, then press the D or R key on the keyboard.
The loose chips are removed in the graphics area.
6.  Click the X button in the upper right corner to cancel the simulation and return to the SolidWorks display.

Post process the part and view the code:







7.  Click the Post Process button on the CAMWorks toolbar or right click NC Manager in the Operation tree and click Post Process on the shortcut menu.
8. In the Post Output File dialog box, click Save.
9.  Click the Run button in the Post Process Output dialog box.
10. When the post processing is finished, use the vertical scroll bar to view the code and notice the machining order.
11. Click OK to close the dialog box.

2-Axis Contour 2

What You'll Learn

Automatic Feature Recognition may not be able to find all features on a part or may not recognize all the features that you want to machine. When this occurs, you can define these features interactively. In this exercise, AFR finds only the irregular die in the center of the part. You interactively insert 2.5 Axis EDM features to cut the two dies on the sides and the outside of the part.

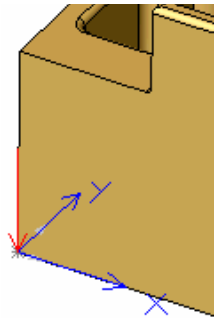
Inserting 2.5 Axis EDM Features

1. Open the part file **EDM2AX_3.SLDPRT** in the *\Examples\EDM* folder inside the CAMWorks folder (e.g., *\Program Files\CAMWorksxxx\Examples\EDM*).
2.  Click the CAMWorks Feature Tree tab.
3. Right click the Machine item in the CAMWorks Feature tree and select Edit Definition on the shortcut menu.
The Machine dialog box displays.
4. In the Available machines list, highlight Wire EDM–in, click the Select button, then click OK.
5.  Right click Wire EDM–in in the tree and click Select Machine on the shortcut menu.
6. In the Machine Selection dialog box, make sure TUTORIAL is selected for the Make and TUTORIAL for the Model, then click OK.
7.  Right click Wire EDM–in in the tree and select Edit Settings.
8. On the Posting tab, make sure the Process order is By Complete Feature for both Die and Punch, then click OK.
9. With this option, the NC program will be generated in the following machining order: for each Die feature, cut rough pass, glue stop cut off, then all skim passes. For each Punch feature, the processing order is rough cut leaving glue stop, 0 - 6 skim passes, glue stop cut off.
10.  Right click Stock Manager in the Feature tree and select Edit Definition on the shortcut menu.
11. In the Manage Stock dialog box, click OK to use the default bounding box and material.
12.  Click the Options button on the CAMWorks toolbar.
13. On the EDM Features tab, make sure the Perimeter punch feature is not checked and click OK.
14.  Click the Extract Machinable Features button on the CAMWorks toolbar. AFR finds only the Irregular Die in the center of the part.

15. Click Part Setup1 in the tree and notice the Z machining direction. The red arrow on the triad indicates the negative Z direction. The direction is correct for this part.

When necessary, you can reverse the direction by right clicking Part Setup1 in the tree, selecting Edit Definition and clicking the Reverse Direction button in the Part Setup dialog box.

Note that the SolidWorks axis does not control CAMWorks machining. However, it is used as a reference for certain CAMWorks functions (for example, offsetting the stock).



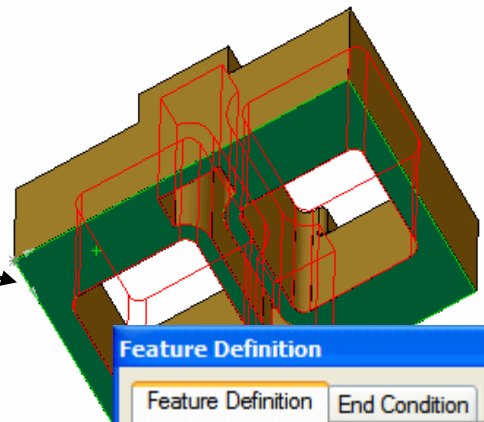
Insert 2.5 Axis EDM features interactively:

1. Right click Part Setup1 in the tree and select Insert 2.5 Axis EDM Feature on the shortcut menu. The Feature Definition dialog box displays.



2. Make sure Die is the selected Feature type.

3. Rotate the part and pick the bottom face.



Three Die features are shown in the Selected entities for features box.

4. Click Die – 2 and Die – 3 to view the features on the part.
5. Click Die – 1. This is the Irregular Die feature that has already been found by AFR so you can delete it.
6. Highlight Die – 1 in the list and press the Delete key.
7. Click the End Condition tab.

The End Condition page provides options to define the end condition for the features you are inserting. The default Upto Stock defines the depth to the material face. This amount is calculated automatically based on the position of the stock.

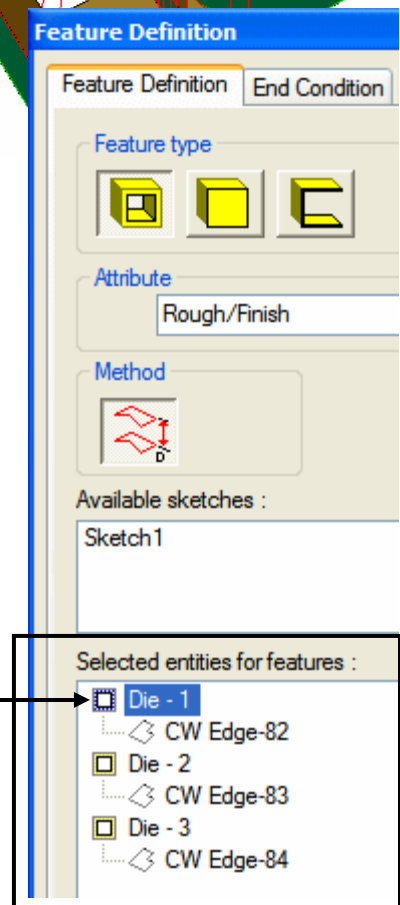
8. Click Insert to use the default End Condition and insert the features.

Rectangular Die2 and Rectangular Die3 are listed in the Feature tree.

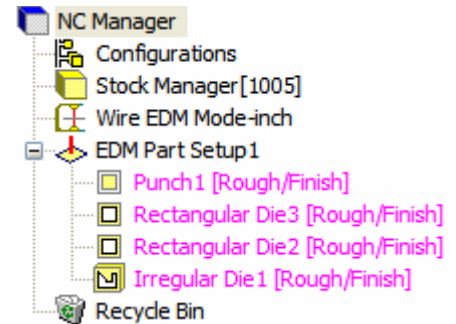
Define the feature to machine the outside of the part:



9. Click the Punch button for the Feature type.




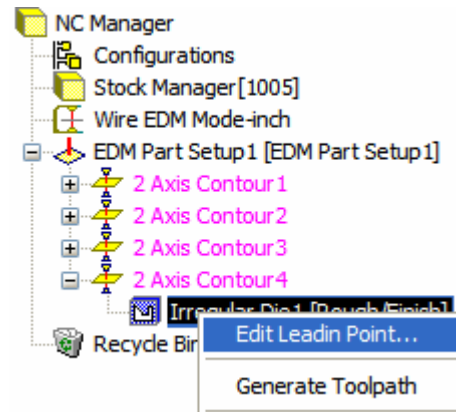
10. Pick the same face on the bottom of the part.
Punch – 1 is listed in the Selected entities for feature box.
11. Click the End Condition tab. You can use the default Upto Stock.
12. Click Insert. Punch1 displays in the tree.
13. Click Close to close the Feature Definition dialog box.



Generating Operations and Toolpaths

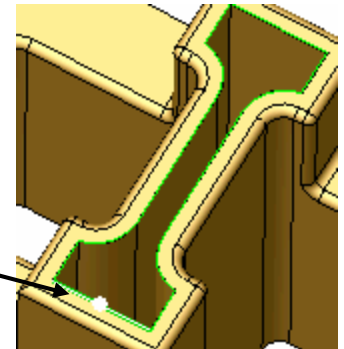
Generate operations and toolpaths:


1.  Click the Generate Operation Plan button on the CAMWorks toolbar or right click NC Manager or Part Setup1 in the Feature tree and select Generate Operation Plan on the shortcut menu.
2. Expand 2 Axis Contour4 in the Operation tree (click the + sign).
3. Right click Irregular Die1 and select Edit Leadin Point on the shortcut menu.



The Lead in Location dialog box displays. You can set the leadin/leadout location to be at the start-, mid- or end-point of a segment. The current location is indicated on the feature. Rotate the part and notice that this location may gouge the part.

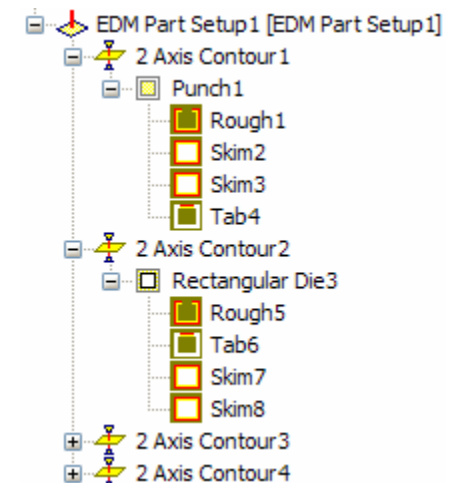
4. Leave the Lead in location set to Mid-point and pick the segment shown in the figure.
5. Click OK to close the dialog box.



6.  Click the Generate Toolpath button on the CAMWorks toolbar or right click the Part Setup1 or NC Manager items in the Operation tree and select Generate Toolpath on the shortcut menu.
7. Click the + sign next to 2 Axis Contour 1 in the tree to expand the operation, then expand Punch1.

The processing order is rough cut, skim passes, then glue stop cut off. You set the Process order for Punch to By Complete Feature on the Posting tab in the EDM Settings dialog box and the code for all Punch features will be output based on this order.

8. Click the + sign next to 2 Axis Contour 2 in the tree to expand the operation, then expand Rectangular Die3.






The processing order is rough cut, glue stop cut off, then skim passes. You set the Process order to By Complete Feature on the Posting tab in the EDM Settings dialog box and the code for all Die features will be output based on this order.

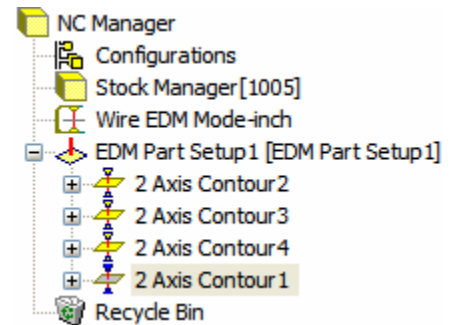
Changing the Machining Order and Post Processing

CAMWorks processes the punches and dies according to the order of the items in the Operation tree. The operation for the Punch feature is listed first in the tree, so you can reorder the operations before post processing.

1. Highlight 2 Axis Contour1 in the tree and drag it on top of 2 Axis Contour4.

The operations for the Die features will now be post processed before the operation for the Punch feature.

2.  Click the Options button on the CAMWorks toolbar.
3. Click the File Output tab. The File Output tab contains an option to automatically open the G-code file when you close the Post Process Output dialog box.
4. Check the Open G-code file in editor option.
5. Browse and select the editor: if you know the location of the editor you use to modify NC program files, click the Browse button, open the folder and select the program. Otherwise, select Notepad.exe in the \Windows or \WINNT folder and click Open, then click OK.
6.  Click the Post Process button on the CAMWorks toolbar or right click NC Manager in the Operation tree and click Post Process on the shortcut menu.
7. In the Post Output File dialog box, type a name for the file and click Save.
You do not have to type the extension.
8.  Click the Run button in the Post Process Output dialog box.
9. When the post processing is finished, click OK.
The program file opens automatically in the editor you selected.
10. Notice that G-code has been generated for the Die features and then the Punch feature.
11. Close the editor.








2-Axis Contour 3


What You'll Learn

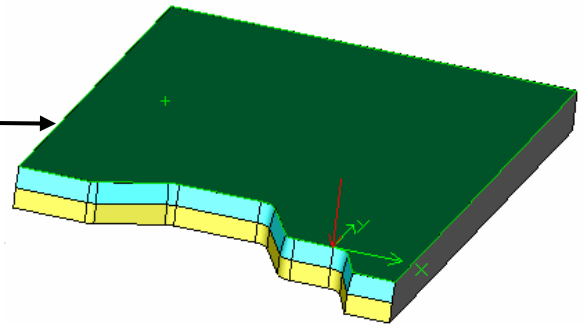
In this exercise, the only features you want to machine are the vertical and tapered open profiles at the end of the part. AFR does not find these features, so you need to insert a Part Setup, then define the 2 features interactively using the Insert 2.5 Axis EDM command.

Inserting a Part Setup

1. Open the part file **EDM2AX_4.SLDPRT** in the *\Examples\EDM* folder inside the CAMWorks folder (e.g., *\Program Files\CAMWorksxxx\Examples\EDM*).
2.  Click the CAMWorks Feature Tree tab.
3. Right click the Machine item in the CAMWorks Feature tree and select Edit Definition on the shortcut menu.
The Machine dialog box displays.
4. In the Available machines list, highlight Wire EDM Mode-mm, click the Select button, then click OK.
5.  Right click Wire EDM Mode-mm in the tree and click Select Machine on the shortcut menu.
6. In the Machine Selection dialog box, make sure TUTORIAL is selected for the Make and TUTORIAL for the Model, then click OK.
7.  Right click Wire EDM Mode-mm in the tree and select Edit Settings on the shortcut menu.
8. On the Posting tab, set the Process order for Die to By Complete Feature, then click OK.
With this option, the NC program is generated in the following machining order: Rough cut, glue stop cut off, 0 - 6 full skim passes.
9.  Right click Stock Manager in the Feature tree and select Edit Definition on the shortcut menu.
10. In the Manage Stock dialog box, click OK to use the default bounding box and material.
11.  Click the Extract Machinable Features button on the CAMWorks toolbar or right click NC Manager in the tree and select Extract Machinable Features on the shortcut menu.

You should run AFR even when you know AFR cannot find any features on a part. Running AFR sets up information for associativity that is necessary for the Rebuild function.


12.  Right click Stock Manager in the tree and select Insert Part Setup on the shortcut menu.
13. Pick the face on the top of the part, then click OK in the Part Setup dialog box.
Part Setup1 is listed in the tree.



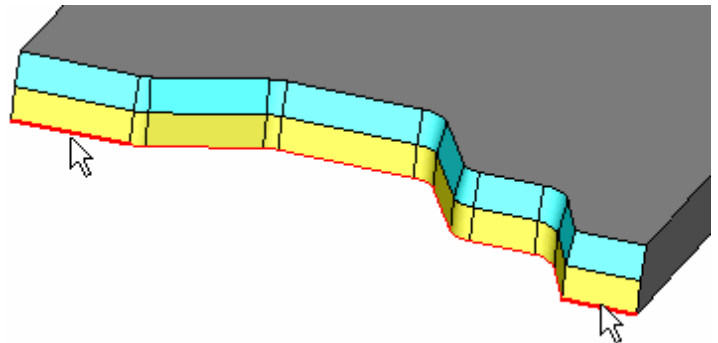
Inserting Vertical and Tapered Open Profile Features

The yellow faces define a vertical wall. The blue faces define a tapered wall. First you define the vertical open profile feature, then the tapered open profile feature.

1. Right click Part Setup1 in the tree and select Insert 2.5 Axis EDM Feature on the shortcut menu.

2.  Select Open Profile for the Feature type.

3. Pick the lower edge on the left, then hold down the Ctrl key and pick the lower edge on the right.



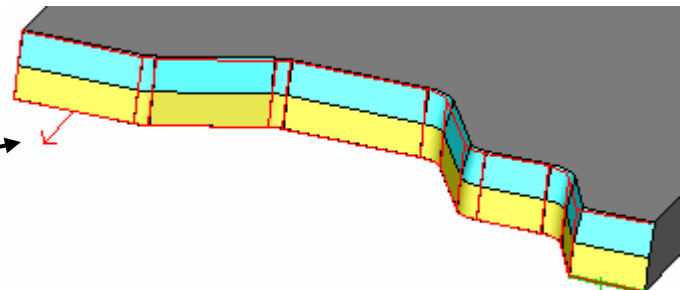
When you define an Open Profile Feature from multiple edges, you can chain select between the first and last edges of the feature.

- If the Shift key is pressed, the chaining direction is based on the fewest number of edges between the first and last selected edges.
- If the Ctrl key is pressed, the chaining direction is based on the greatest number of edges between the first and last selected edges.

Notice the arrow indicates which side of the feature to cut on. In this case, you want to cut on the other side.

4. Click Open Profile – 1 in the Selected entities for features list, then click the Flip direction check box.

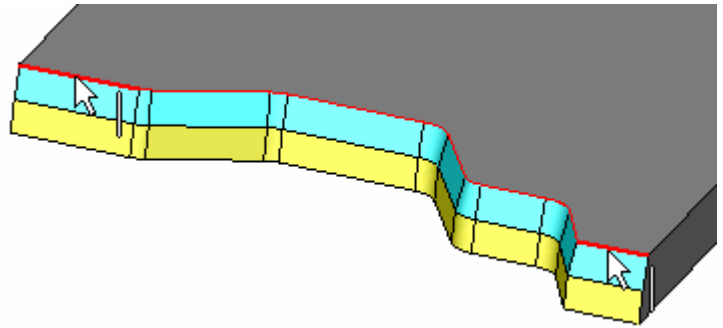
The arrow direction is reversed.





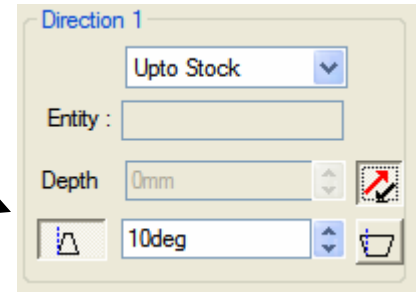
5. Click the End Condition tab.
This feature is vertical and no changes need to be made on this tab.
6. Click the Insert button.

Open Profile 1 is listed in the Feature tree. Next, you define the tapered open profile feature.

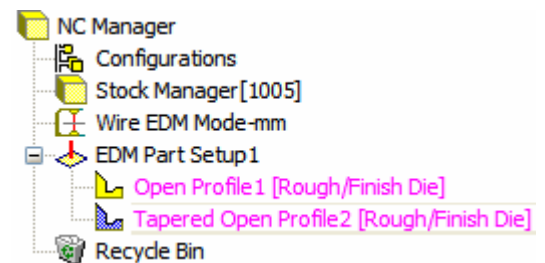
7. Pick the top edge on the left side.
8. Hold down the Ctrl key and pick the top edge on the right.
9. Click Open Profile – 1 in the Selected entities for features list, then click the Flip direction check box.
10. Click the End Condition tab.
11. Zoom in on the right side of this feature. There is a 10-degree taper.




12.  Click the Taper On/Off button to turn it on.
13. Set the degrees to 10 and press the Tab key.
14. Rotate the part and notice that the feature outline indicates the taper is going inward.
15.  Click the Reverse Taper button to reverse the taper so it is going outward. This button functions as a toggle.
16. Click Insert, then click Close.
17. Drag Tapered Open Profile2 after Open Profile1 in the Feature tree.









This is the order the features will be processed when the NC code is generated.



Generate operations and toolpaths:

1.  Click the Generate Operation Plan button on the CAMWorks toolbar or right click NC Manager or Part Setup1 in the Feature tree and select Generate Operation Plan on the shortcut menu.
2. Double-click 2 Axis Contour1 in the Operation tree.
The 2 Axis Contour dialog box displays. This dialog box gives you access to all the parameters used to create the toolpath.
3. On the EDM tab, set the following:
 - Number of skims = 2
 - Glue Stop = No Glue Stop
 - Process = Die
4. On the Leadin/Out tab, set the Leadin and Leadout to Parallel, then click OK.
5. Right click on 2 Axis Contour2 in the tree and select Edit Definition on the shortcut menu.
6. Make the same changes as for 2 Axis Contour1, then click OK.

7.  Click the Generate Toolpath button on the CAMWorks toolbar or right click the Part Setup1 or NC Manager items in the Operation tree and select Generate Toolpath on the shortcut menu.
8.  Click the Simulate Toolpath button of the CAMWorks toolbar or right click Part Setup1 in the tree and select Simulate Toolpath.
9. Set the display:
 - Stock, Tool Holder and Target Part = Translucent Display
 - Tool = Shaded Display
10.  Click the Run button.
11.  Click the X button in the upper right corner to cancel the simulation and return to the SolidWorks display.
12.  Click the Post Process button on the CAMWorks toolbar or right click NC Manager in the Operation tree and click Post Process on the shortcut menu.
13. In the Post Output File dialog box, click Save.
14.  Click the Run button in the Post Process Output dialog box.
15. When the post processing is finished, click OK.




The program file may open automatically in an editor if you enabled this option in the previous exercise.
16. View the code, then close the editor.

2 Axis Contour 4

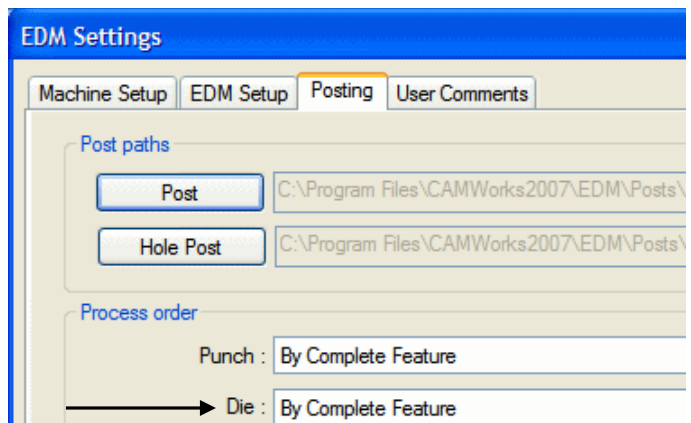
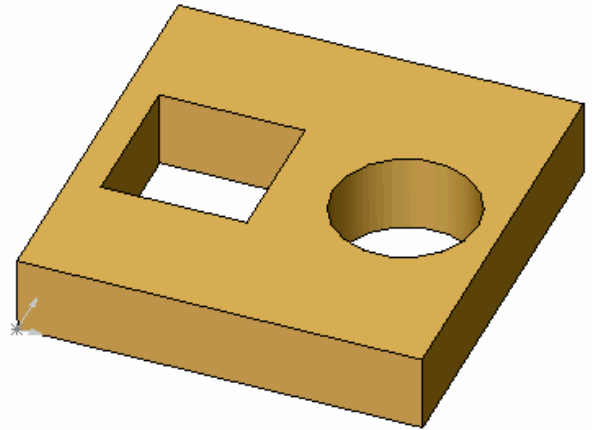
What You'll Learn





Post processing translates generalized toolpath and operation information into NC code for a specific machine tool controller. CAMWorks creates NC code for each toolpath in the order the operation appears in the Operation tree and the Process order specified on the Posting tab in the EDM Settings dialog box. In this exercise, you generate operations and toolpaths for two die features, then post process using each of the Process order options for dies and compare the code that is generated. For more information on the Process order options, see the Posting Tab topic in the Wire EDM online Help.

Changing the Processing Order



1. Open the part file **EDM2AX_5.SLDPRT** in the *\Examples\EDM* folder inside the CAMWorks folder (e.g., *\Program Files\CAMWorksxxxx\Examples\EDM*).
2.  Click the CAMWorks Feature Tree tab.
3. Right click the Machine item in the CAMWorks Feature tree and select Edit Definition on the shortcut menu.
The Machine dialog box displays.
4. In the Available machines list, highlight Wire EDM Mode-mm, click the Select button, then click OK.
5.  Right click Wire EDM Mode-mm in the tree and click Select Machine on the shortcut menu.
6. In the Machine Selection dialog box, make sure TUTORIAL is selected for the Make and TUTORIAL for the Model, then click OK.
7.  Right click Wire EDM Mode-mm in the tree and select Edit Settings on the shortcut menu.
8. On the Posting tab, make sure the Process order for Die is By Complete Feature, then click OK.






With this option, the NC program is generated in the following machining order: Rough cut leaving glue stop, glue stop cut off, 0 - 6 full skim passes.



9.  Right click Stock Manager in the Feature tree and select Edit Definition on the shortcut menu.
10. In the Manage Stock dialog box, click OK to use the default bounding box and material.
11.  Click the Extract Machinable Features button on the CAMWorks toolbar or right click NC Manager in the tree and select Extract Machinable Features on the shortcut menu.
Rectangular Die1 and Circular Die2 are listed in the tree.
12.  Click the Generate Operation Plan button on the CAMWorks toolbar or right click NC Manager or Part Setup1 in the Feature tree and select Generate Operation Plan on the shortcut menu.
13. Right click 2 Axis Contour1 in the Operation tree and select Edit Definition on the shortcut menu.
14. On the EDM tab, set the following:
 - Number of skims = 2
 - Glue Stop = Use Glue Stop
 - Stop distance = 2mm
15. On the Leadin/Out tab, set the Leadin and Leadout to Perpendicular with a Length of 6mm, then click OK.
16. Right click 2 Axis Contour2 in the tree and select Edit Definition.
17. Make the same changes on the EDM and Leadin/Out tabs.
18.  Click the Generate Toolpath button on the CAMWorks toolbar or right click the Part Setup1 or NC Manager items in the Operation tree and select Generate Toolpath on the shortcut menu.

Post process the Die features and view the code:

1.  Click the Options button on the CAMWorks toolbar.
2. Click the File Output tab.
The File Output tab contains an option to automatically open the G-code file. You can browse and select the editor that you want to open the G-code file when you close the Post Process Output dialog box.
3. Check the Open G-code file in editor option.
4. If you know the location of the editor you use to modify NC program files, click the Browse button, open the folder and select the program. Otherwise, select Notepad.exe in the \Windows or \WINNT folder and click Open, then click OK.
5.  Click the Post Process button on the CAMWorks toolbar or right click NC Manager in the Operation tree and click Post Process on the shortcut menu.

6. In the Post Output File dialog box, type **die complete** for the File name and click Save.
You do not have to type the extension.
7.  Click the Run button in the Post Process Output dialog box.
8. When the post processing is finished, click OK.
The program file opens automatically in the editor you selected.
9. View the code and notice the following processing order:
 - Rough cut leaving glue stop
 - Glue stop cut off
 - 0 - 6 full skim passes
10. Close the editor.
11.  Right click Wire EDM Mode–mm in the tree and click Edit Settings on the shortcut menu.
12. On the Posting tab, set the Process order for Die to Rough / Glue Stop / All Skims, then click OK.
13.  Post process the part again. Name the file **die rough-tab-all skims**.
14. View the code and notice the following processing order:
 - Cut all rough passes from first feature to last
 - Glue stop cut off on each feature from last to first
 - Perform all skim passes on each feature from first feature to last
15. Close the editor.
16.  Right click Wire EDM Mode–mm in the tree and click Edit Settings on the shortcut menu.
17. On the Posting tab, set the Process order for Die to Rough / Glue Stop / Skim Passes, then click OK.
18.  Post process the part again. Name the file **die rough-tab-skim pass**.
19. View the code and notice the following processing order:
 - Cut all rough passes from first feature to last
 - Glue stop cut off on each feature from last to first
 - Skim pass one on each feature from first to last
 - Skim pass two on each feature from last to first
 - Etc.

Chapter 3 Learning 4 Axis Contour Cutting

This chapter provides an opportunity to learn 4 Axis Contour cutting through step-by-step exercises. We recommend that you go through the exercise in Chapter 1 first.

The exercises in this chapter are intended to show you how to use CAMWorks Wire EDM and may not correspond to actual machining practices.




4-Axis Contour 1

Automatic Feature Recognition (AFR) is a powerful tool, but it cannot always recognize every feature on every part, nor does it currently support features where the side walls of the feature are not all vertical or not all constant taper. To machine these areas, you need to define features interactively using the Insert 4 Axis EDM Feature command. The Insert 4 Axis EDM Feature command allows you to define 4 axis features. These include Pocket (Die), Boss (Punch) and Open Profile features.



In this exercise, you insert a 4 Axis Die feature.

What You'll Learn

Defining the Machine, Stock and Machining Direction

1. Open the part file **EDM4AX_1.SLDPRT** in the *\Examples\EDM* folder inside the CAMWorks folder (e.g., *\Program Files\CAMWorksxxx\Examples\EDM*).
2.  Click the CAMWorks Feature Tree tab.
3. Double click the Machine item in the CAMWorks Feature tree.
The Machine dialog box displays.
4. In the Available machines list, highlight Wire EDM Mode—in, click the Select button, then click OK.
5.  Right click Wire EDM Mode—in in the tree and click Select Machine on the shortcut menu.
6. In the Machine Selection dialog box, select TUTORIAL for the Make and TUTORIAL for the Model, then click OK.
7.  Right click Wire EDM Mode—in in the tree and select Edit Settings on the shortcut menu.
8. On the Machine Setup tab, set the AWT minimum distance to 0.0in.
This parameter defines the minimum clearance required for the auto threader to work properly when threading in an open cavity next to the part profile previously machined by a rough cut. This would occur in automatic machining when re-threading the wire to perform the skim cuts in a die cavity.
9. On the EDM Setup tab, make sure the Wire diameter is set to 0.01in.
10. On the Posting tab, make sure the Process order for dies is set to By Complete Feature, then click OK.


With this option, the NC program will be generated in the following machining order: cut rough pass, glue stop cut off, then all skim passes.

11.  Double click Stock Manager in the Feature tree.
The Manage Stock dialog box displays.
12. For this exercise, click OK to use the default bounding box and material.
13.  Click the Extract Machinable Features button on the CAMWorks toolbar.
You should run AFR even when you know AFR cannot find any features on a part. Running AFR sets up information for associativity that is necessary for the Rebuild function.
14. Right click Stock Manager in the Feature tree and select Insert Part Setup on the shortcut menu.
The EDM Part Setup in the Feature tree is used to define the Z machining direction. AFR creates the Part Setups necessary to machine the features that are recognized automatically. In this exercise, AFR is not used since there are no 2 Axis EDM features on the part. You need to insert a Part Setup interactively.
15. Pick the top face and click OK.


Defining a 4 Axis Die Feature Interactively

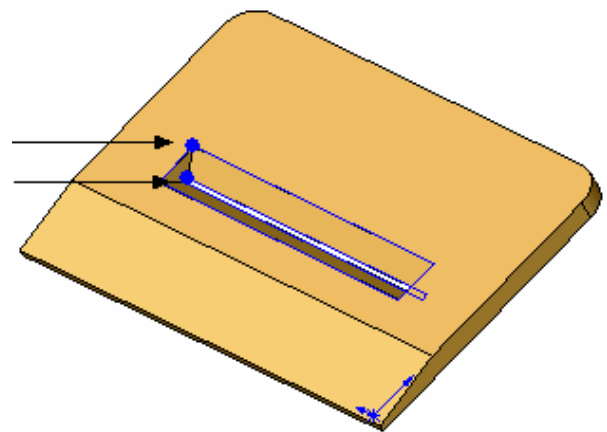
Define a 4 Axis EDM feature interactively:

1. Right click Part Setup1 and select Insert 4 Axis EDM Feature on the shortcut menu.
The Insert 4 Axis EDM Feature dialog box displays.

2.  Make sure Die is the selected Feature type.
3. For Curve-1, pick the top edge of the rectangular pocket as shown in the figure.
4. For Curve-2, pick the lower edge.






The start point on each curve is identified.

5.  If the start points are not as shown in the figure, click the Move Start Point button for one of the curves to change the position. The Move Start Point buttons are used when Curve 1 and Curve 2 contain the same number of entities.
6. Click Insert, then click Close.
Rectangular 4-Axis Die1 is listed in the tree.





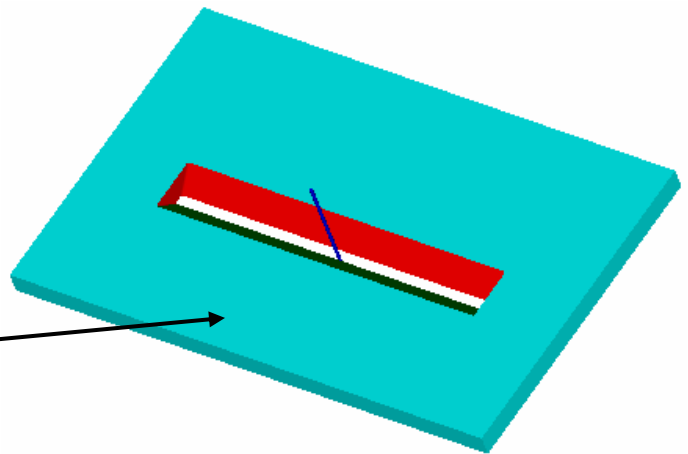
Generating a 4 Axis Contour Operation and Toolpaths

Generate operations and toolpaths, then post process:

1.  Click the Generate Operation Plan button on the CAMWorks toolbar or right click NC Manager or Part Setup1 in the Feature tree and select Generate Operation Plan on the shortcut menu.
2. Double click 4 Axis Contour1 in the Operation tree.
3. On the Leadin/out tab, set the Leadin and Leadout to Perpendicular with a .02in Length, then click OK.
4.  Click the Generate Toolpath button on the CAMWorks toolbar or right click the Part Setup1 or NC Manager items in the Operation tree and select Generate Toolpath on the shortcut menu.
5. Click the operation in the tree to view the toolpath.
6.  Click the Simulate Toolpath button on the CAMWorks toolbar or right click NC Manager or Part Setup1 in the tree and select Simulate Toolpath.
7. Set the display:
 - Stock and Tool = Shaded Display
 - Holder = No Display
 - Target Part = No Display
8.  Click the Run button.
9.  Click the Select Chips to Remove button.


This button is enabled when the simulation is complete and provides two methods for removing loose chips.
10. Hold down the Ctrl key and pick the stock piece you want to keep, then press the D or R key.

The loose chips are removed in the graphics area.
11.  Click the Select Chips to Remove button to exit the function.
12. Click the Goto Start button and Run the simulation again.
13.  Click the Select Chips to Remove button.





14. Pick the loose stock chip to remove and press the D or R key.

If there are many loose chips to remove, picking the stock piece may be faster than picking each chip.

15.  Click the X button in the upper right corner to cancel the simulation and return to the SolidWorks display.

Post process the part and view the code:

1.  Click the Post Process button on the CAMWorks toolbar or right click NC Manager in the Operation tree and click Post Process on the shortcut menu.
2. In the Post Output File dialog box, click Save.
3.  Click the Run button in the Post Process Output dialog box.
4. When the post processing is finished, use the vertical scroll bar to view the code.
5. Click OK to close the dialog box.





4-Axis Contour 2


What You'll Learn

When defining 4 Axis Features, sync curves are required when the number of entities in Curve 1 is not equal to the number of entities in Curve 2. Sync curves must be a single linear segment and can be selected from edges or 3d sketch objects. The sync curves are used to define sync points on the feature. As the wire cuts around the two curves defining the feature, the wire will arrive at the end points of each sync curve at the same time. Sync curves are similar to the guide curves used to create a loft feature in SolidWorks. If the number of entities in Curve 1 and Curve 2 is equal, then the sync curves are ignored.

In the part below, the number of entities in Curve 1 is not equal to the number of entities in Curve 2 of the 4 Axis feature. In this exercise, you insert the feature and automatically extract sync curves.

Automatically Extracting Sync Curves to Define a 4 Axis Feature


1. Open the part file **EDM4AX_2.SLDPRT** in the *\Examples\EDM* folder inside the CAMWorks folder (e.g., *\Program Files\CAMWorksxxxx\Examples\EDM*).
2.  Click the CAMWorks Feature Tree tab.
3. Double click the Machine item in the CAMWorks Feature tree.
The Machine dialog box displays.
4. In the Available machines list, highlight Wire EDM Mode-in, click the Select button, then click OK.
5.  Right click Wire EDM Mode-in in the tree and click Select Machine on the shortcut menu.
6. In the Machine Selection dialog box, make sure TUTORIAL is selected for the Make and TUTORIAL for the Model, then click OK.
7.  Right click Wire EDM Mode-in in the tree and select Edit Settings.
8. On the Posting tab, make sure the Process order for Punch is set to By Complete Feature, then click OK.
The processing order is rough cut leaving glue stop, 0 - 6 skim passes, then glue stop cut off.
9.  Double click Stock Manager in the Feature tree.
10. In the Manage Stock dialog box, click OK to use the default bounding box and material.


11.  Click the Extract Machinable Features button on the CAMWorks toolbar.
You should run AFR even when you know AFR cannot find any features on a part. Running AFR sets up information for associativity that is necessary for the Rebuild function.
12. Right click Stock Manager in the Feature tree and select Insert Part Setup on the shortcut menu.
13. Pick the top face on the part and click OK.

Insert the 4 Axis EDM feature interactively:

1. Right click Part Setup1 in the tree and select Insert 4 Axis EDM Feature on the shortcut menu.

The Insert 4 Axis EDM Feature dialog box displays.

2.  Click the Punch button for the Feature type.
3. For Curve-1, pick the top face of the part.
4. For Curve-2, rotate the part and pick the bottom face.

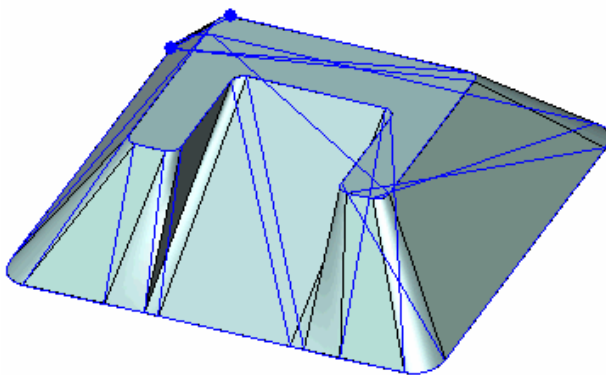
5.  Click the Automatically extract sync curves button. CAMWorks extracts the sync curves.



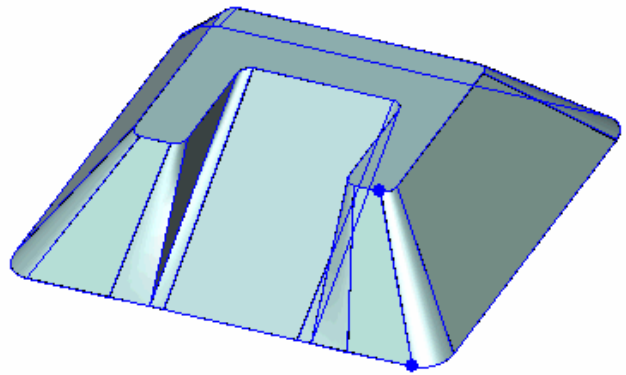
Did You Know ...

CAMWorks cannot extract sync curves automatically when a sketch is used for either Curve 1 or Curve 2.

6. Click Insert, then click Close.




Before Automatically Extracting Sync Curves




After Automatically Extracting Sync Curves


Generating the 4 Axis Contour Operation and Toolpaths

Generate operations and toolpaths:

1.  Click the Generate Operation Plan button on the CAMWorks toolbar or right click NC Manager or Part Setup1 in the Feature tree and select Generate Operation Plan on the shortcut menu.
2. Right click 4 Axis Contour1 in the Operation tree and select Edit Definition.
3. On the EDM tab, set the Glue stop to No Glue Stop, then click OK.

The Glue stop option controls whether a tab is created automatically. When No Glue Stop is selected, a tab is not created and the NC program will not contain a glue stop.

4.  Click the Generate Toolpath button on the CAMWorks toolbar or right click the Part Setup1 or NC Manager items in the Operation tree and select Generate Toolpath on the shortcut menu.
5. Click the operation in the tree to view the toolpath.
6. Click 4 Axis Contour1 in the tree, then click 4-Axis Punch1 to expand it. Notice that there is no Tab cut.

7.  Click the Simulate Toolpath button on the CAMWorks toolbar.


8. Set the display:
 - Stock and Tool = Shaded Display
 - Holder = No Display
 - Target Part = No Display

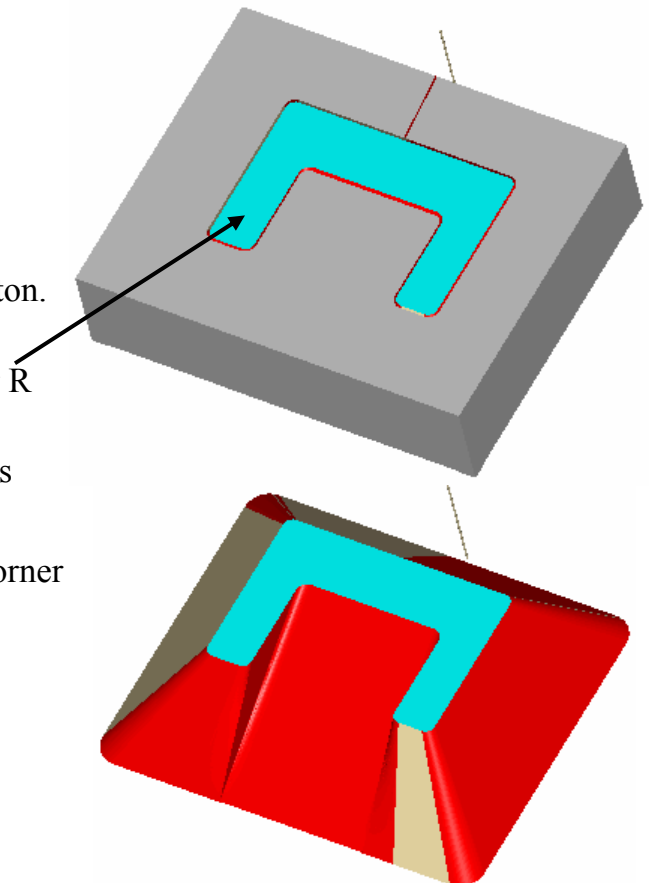
9.  Click the Run button.

10.  Click the Select Chips to Remove button.

11. Hold down the Ctrl key and pick the stock piece you want to keep, then press the D or R key.

The loose chips are removed in the graphics area.

12.  Click the X button in the upper right corner to cancel the simulation and return to the SolidWorks display.








4-Axis Contour 3

What You'll Learn

Faces and loops can be used to define a 4 Axis EDM feature if the faces/loops are on a plane that is parallel to the setup axis. In this exercise, since the top face on the part is not parallel, you select a sketch to define the top of the 4 Axis EDM feature.

Using Sketches to Define a 4 Axis EDM Feature

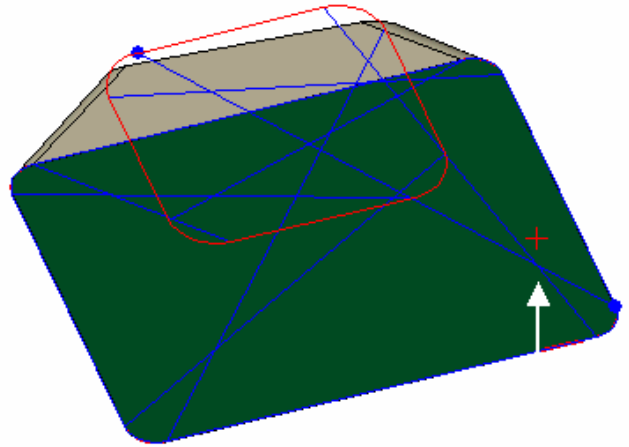
1. Open the part file **EDM4AX_3.SLDPRT** in the *\Examples\EDM* folder inside the CAMWorks folder (e.g., *\Program Files\CAMWorksxxx\Examples\EDM*).
2.  Click the CAMWorks Feature Tree tab.
3. Right click the Machine item in the CAMWorks Feature tree and select Edit Definition on the shortcut menu.
4. In the Available machines list, highlight Wire EDM Mode—in, click the Select button, then click OK.
5.  Right click Wire EDM Mode—in in the tree and click Select Machine on the shortcut menu.
6. In the Machine Selection dialog box, make sure TUTORIAL is selected for the Make and TUTORIAL for the Model, then click OK.
7.  Right click Wire EDM Mode—in in the tree and select Edit Settings on the shortcut menu.
8. On the Posting tab, set the Process order for Punch to By Complete Feature, then click OK.
9.  Right click Stock Manager in the Feature tree and select Edit Definition on the shortcut menu.
10. In the Manage Stock dialog box, click OK to use the default bounding box and material.
11.  Click the Extract Machinable Features button on the CAMWorks toolbar.
You should run AFR even when you know AFR cannot find any features on a part. Running AFR sets up information for associativity that is necessary for the Rebuild function.
12. Right click Stock Manager in the Feature tree and select Insert Part Setup on the shortcut menu.
13. Pick the Front reference plane and click the Reverse direction option.
14. Click OK.

Insert the 4 Axis EDM feature interactively:

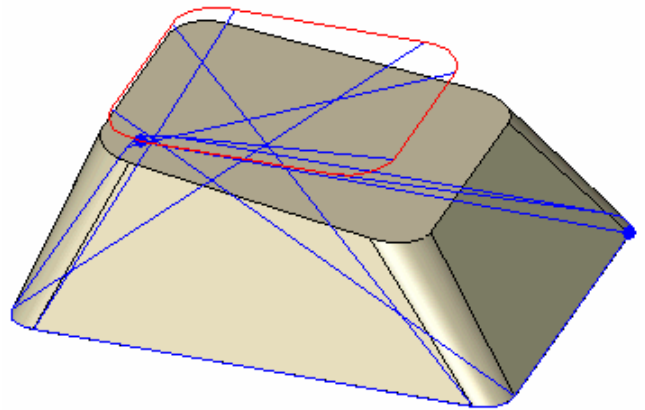
1. Right click Part Setup1 in the tree and select Insert 4 Axis EDM Feature on the shortcut menu.




2. Click the Punch button for the Feature type.
3. Pick Sketch2 for Curve-1.
4. For Curve-2, rotate the part and pick the bottom face anywhere on the right side as shown in the figure.



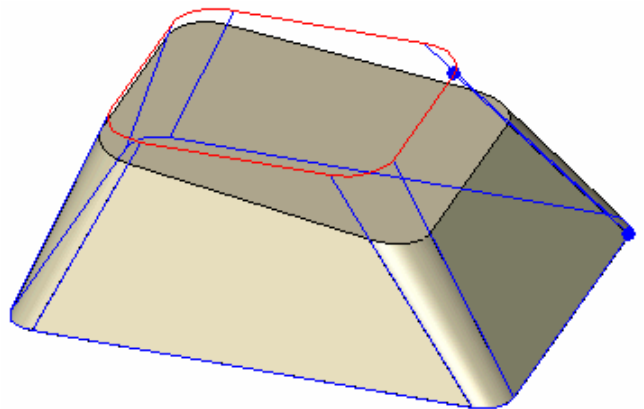
5. Rotate the part and notice the top and bottom start points are not lined up.



6.  Click the Move Start Point Location button under Curve-1 until the points line up.



The start point is used when Curve 1 and Curve 2 contain the same number of entities. Each time you click the button, the start position for Curve 1 or Curve 2 is moved from end point to end point.

7. Click Insert, then click Close.



Changing the Leadin Point

Generate the operation and toolpath:

1.  Click the Generate Operation Plan button on the CAMWorks toolbar or right click NC Manager or Part Setup1 in the Feature tree and select Generate Operation Plan on the shortcut menu.
2.  Click the Generate Toolpath button on the CAMWorks toolbar or right click the Part Setup1 or NC Manager items in the Operation tree and select Generate Toolpath on the shortcut menu.
3. Click the + sign next to 4 Axis Contour1 in the tree to expand the item.
4. Right click 4-Axis Punch1 under the operation and select Edit Leadin Point on the shortcut menu.




The Lead in Location dialog box displays and a round marker on the feature identifies the current leadin point.

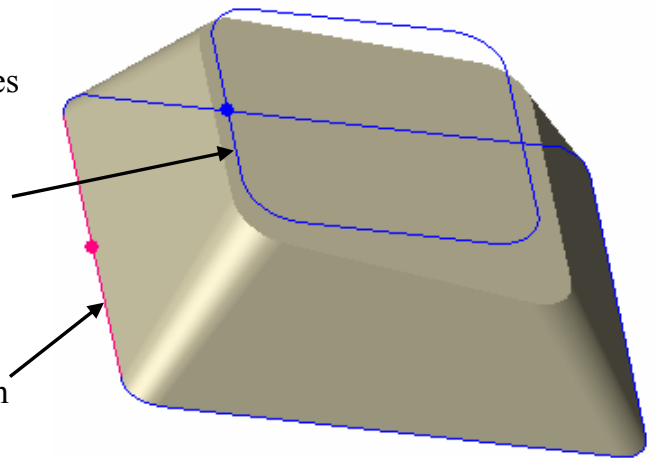
5. Use the current settings in the Lead in Location dialog box and pick the segment shown in the figure.

The entry point marker displays at the midpoint of the selected edge.

6. Change the Select profile option to Bottom and pick the segment shown in the figure.
7. Click OK.

The toolpath is automatically updated.

8.  Click the Simulate Toolpath button of the CAMWorks toolbar or right click Part Setup1 in the tree and select Simulate Toolpath.
9. Set the display:
 - Stock, Tool Holder and Target Part = Translucent Display
 - Tool = Shaded Display
10.  Click the Run button.
11.  Click the X button in the upper right corner to cancel the simulation and return to the SolidWorks display.








4-Axis Contour 4

What You'll Learn

In exercise 4-Axis Contour 2, you used the automatic synchronization to insert a 4 Axis EDM feature that had an unequal number of entities in the top and bottom curves. In this exercise, you insert a 4 Axis EDM feature that has an unequal number of entities in the top and bottom curves. Because you use sketches for the curves, however, you pick each of the sync curves instead of using the automatic function.

Picking Sync Curves to Define a 4 Axis EDM Feature

1. Open the part file **EDM4AX_4.SLDPRT** in the *\Examples\EDM* folder inside the CAMWorks folder (e.g., *\Program Files\CAMWorksxxxx\Examples\EDM*).
2.  Click the CAMWorks Feature Tree tab.
3. Right click the Machine item in the CAMWorks Feature tree and select Edit Definition on the shortcut menu.
4. In the Available machines list, highlight Wire EDM Mode-in, click the Select button, then click OK.
5.  Right click Wire EDM Mode-in in the tree and click Select Machine on the shortcut menu.
6. In the Machine Selection dialog box, make sure TUTORIAL is selected for the Make and TUTORIAL for the Model, then click OK.
7.  Right click Wire EDM Mode-in in the tree and select Edit Settings on the shortcut menu.
8. On the Posting tab, set the Process order for Punch to By Complete Feature, then click OK.
9.  Right click Stock Manager in the Feature tree and select Edit Definition on the shortcut menu.
10. In the Manage Stock dialog box, click OK to use the default bounding box and material.
11.  Click the Extract Machinable Features button on the CAMWorks toolbar.
You should run AFR even when you know AFR cannot find any features on a part. Running AFR sets up information for associativity that is necessary for the Rebuild function.
12. Right click Stock Manager in the Feature tree and select Insert Part Setup on the shortcut menu.
13. Pick the top face and click OK.

Insert the 4 Axis EDM Feature interactively:

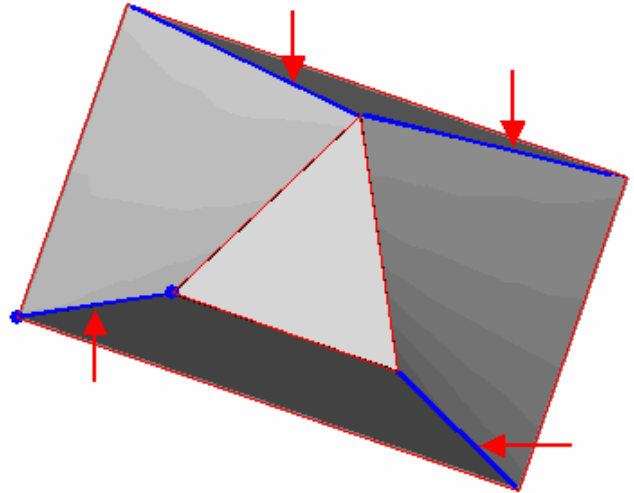
1. Right click Part Setup1 in the tree and select Insert 4 Axis EDM Feature on the shortcut menu.



2. Click the Punch button for the Feature type.

3. In the Insert 4 Axis EDM Feature dialog box, pick Sketch2 for Curve-1.
4. Pick Sketch1 for Curve-2.
5. Click in the Sync Curves list box to set the focus.
6. Pick the four segments shown in the figure to define the sync points.






In most cases, the order you pick the segments does not matter; however, picking the segments in sequential order is recommended.



7. Click Insert, then click Close.

Note that you can also insert this feature by picking the top and bottom faces or edges and using automatic synchronization. Sketches were used for demonstration purposes.

Generate the operation and toolpath:

1.  Click the Generate Operation Plan button on the CAMWorks toolbar or right click NC Manager or Part Setup1 in the Feature tree and select Generate Operation Plan on the shortcut menu.
2.  Click the Generate Toolpath button on the CAMWorks toolbar or right click the Part Setup1 or NC Manager items in the Operation tree and select Generate Toolpath on the shortcut menu.
3.  Click the Simulate Toolbar button of the CAMWorks toolbar or right click Part Setup1 in the tree and select Simulate Toolpath.
4. Set the display:
 - Stock, Tool Holder and Target Part = Translucent Display
 - Tool = Shaded Display
5.  Click the Run button.
6.  Click the X button in the upper right corner to cancel the simulation and return to the SolidWorks display.

Index

2 Axis Contour dialog box 1-9, 2-12, 2-15

2.5 Axis EDM feature
 defining automatically 1-7, 2-2, 2-15
 defining interactively 2-6, 2-11

4 Axis EDM feature
 inserting Die 3-3
 inserting Punch 3-7, 3-10, 3-13
 using sketches 3-10

A

ATW minimum distance 3-2
 Auto save option 1-12
 Automatic Feature Recognition 1-7, 2-2

C

CAMWorks
 machining trees 1-3
 menu 1-4
 Message Window 1-7, 2-3
 toolbar 1-4

E

Edit Leadin Point command 2-8, 3-11
 Edit Settings command 3-2, 3-6, 3-9, 3-12
 EDM Settings dialog box 1-5, 2-2, 2-6, 2-10, 2-14,
 3-2, 3-6, 3-9, 3-12
 Extract Machinable Features command 1-7, 2-2, 2-6,
 2-10, 2-15, 3-3

F

Feature Definition dialog box 2-7, 2-11
 Feature tree 1-8
 file, NC program 1-13

G

Generate Operation Plan command 1-8, 2-4, 2-8,
 2-12, 2-15, 3-4, 3-8, 3-11, 3-13
 Generate Toolpath command 1-10, 2-5, 2-8, 2-13,
 2-15, 3-4, 3-8, 3-11, 3-13
 glue stop option 3-8

I

Insert 2.5 Axis EDM Feature command 2-6, 2-11
 Insert 4 Axis EDM Feature command 3-3, 3-7, 3-10,
 3-13
 Insert 4 Axis EDM Feature dialog box
 automatically extract sync curves 3-7, 3-13
 Move Start Point button 3-3, 3-10

M

machinable features
 customizing AFR 2-3
 defining 2.5 Axis EDM feature interactively 2-6,
 2-11
 defining 4 Axis EDM feature interactively 3-3, 3-7,
 3-10, 3-13
 defining automatically 1-7, 2-2, 2-6, 2-10, 2-15
 features recognized automatically 1-7, 2-2
 interactively inserting Die feature 2-7, 3-3
 interactively inserting Open Profile feature 2-11
 interactively inserting Punch feature 2-7, 3-7, 3-10,
 3-13

Machine

 editing settings 3-2, 3-6, 3-9, 3-12
 identifying type 1-3, 1-4, 2-2, 2-6, 2-10, 2-14, 3-2,
 3-6, 3-9, 3-12
 selecting make/model 1-5, 2-2, 3-2
 setup options 1-5
 Machine dialog box, Machine tab 1-5
 Machine Selection dialog box 1-5, 2-2, 2-6, 2-10,
 2-14, 3-2, 3-6, 3-9, 3-12
 Machining order, setting 1-6, 2-2, 2-6, 2-10, 2-14,
 3-2, 3-6, 3-9, 3-12
 Manage Stock dialog box 1-6

N

NC program
 file 1-13
 generating 1-13
 generating 2 files 2-6
 setting machining order 2-14
 steps to generate 1-2

O

Operation tree 1-8
 operations
 changing machining parameters 1-9, 2-8, 2-12,
 2-15, 3-4
 editing leadin point 2-8, 3-11
 editing leadin/leadout 1-9, 2-12, 2-15, 3-4
 generating 2 Axis Contour 1-8, 2-4, 2-8, 2-12, 2-15
 generating 4 Axis Contour 3-4, 3-8, 3-11, 3-13
 generating toolpaths 1-10, 2-5, 2-8, 2-13, 2-15, 3-4,
 3-8, 3-11, 3-13

Index

- glue stop option 3-8
- post processing 1-13, 2-5, 2-9, 2-14, 3-5
- simulating toolpaths 1-10, 2-5, 2-13, 3-4, 3-8, 3-11, 3-13
- Options dialog box
 - EDM Features tab 2-4
 - File Output tab 2-9, 2-15
 - General tab 1-12
- P**
- Part Setup 1-7
- Part Setup dialog box 2-11
- Part Setup, inserting 2-10, 3-3, 3-7, 3-9
- parts used in exercises
 - edm2ax_1.sldprt 1-2
 - edm2ax_2.sldprt 2-2
 - edm2ax_3.sldprt 2-6
 - edm2ax_4.sldprt 2-10
 - edm2ax_5.sldprt 2-14
 - edm4ax_1.sldprt 3-2
 - edm4ax_2.sldprt 3-6
 - edm4ax_3.sldprt 3-9
 - edm4ax_4.sldprt 3-12
- Post Output File dialog box 1-13, 2-5, 3-5
- post process
 - generating NC program 1-13, 2-5, 2-9, 2-13, 3-5
 - selecting machining order 1-6, 2-2, 2-6, 2-10, 2-14, 3-2, 3-6, 3-9, 3-12
 - viewing code 1-14, 2-9, 2-15
- Post Process command 1-13, 2-5, 2-14, 3-5
- Post Process Output dialog box 1-14, 2-5, 3-5
- Punch feature, inserting interactively 2-7
- R**
- Recycle Bin 1-4
- S**
- Save/Restore part option 1-12
- shortcut menus 1-4
- Simulate Toolpath command 1-10, 2-5, 2-13, 3-4, 3-8, 3-11, 3-13
- Step Thru Toolpath command 1-12
- steps to generate NC program 1-2
- Stock 1-3, 1-6
- T**
- toolpaths
 - generating 1-10, 2-5, 2-8, 2-13, 2-15, 3-4, 3-8, 3-11, 3-13
 - post processing 1-13, 2-5, 2-9, 3-5
 - setting machining order 2-14
 - simulating 1-10, 2-5, 2-13, 3-4, 3-8, 3-11, 3-13
 - stepping through 1-12
- W**
- wire diameter, setting 1-5
- workpiece *See* Stock