CAMWorks 2022

Wire EDM Tutorial

Disclaimer

Geometric Americas, Inc. makes no warranties, either expressed or implied with respect to this manual. Geometric Americas, 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.

Trademarks

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

CAMWorks® is a registered trademark of Geometric Americas, Inc.

SOLIDWORKS® is a registered trademark of Dassault Systèmes SOLIDWORKS Corp.

FeatureManager™ is a trademark owned jointly by DS SOLIDWORKS and Bentley Systems, Inc.

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

Geometric
An HCL Technologies Company

TABLE OF CONTENTS

<u>NOTE</u>

Customizing Wire EDM module to suit your facility's requirements

The **_Wire EDM_** module of CAMWorks is used for the purpose of Wire EDM machining. Before you can machine parts using this module, you need to customize the Wire EDM Machines, Models and Post files to suit your facility's requirements.

Please refer the Appendix section of this manual for the following:

- List of EDM Post files and their Folder Locations (Appendix A).
- Reassigning Post Processors to existing EDM Machine Makes/ Models (Appendix B).
- Adding EDM Machine Makes and Models (Appendix C).
- Restoring customized EDM Post files after upgrading CAMWorks (Appendix D).

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.

The exercise parts are installed when you install CAMWorks and are in the *\Examples\Tutorial_Parts\EDM* folder. This folder is inside the CAMWorks data folder. (Drive:*\CAMWorksData\CAMWorks202x\Examples\Tutorial_Parts\EDM*)

IMPORTANT!   **CAMWorks uses a set of knowledge-based rules to assign machining operations to features. The Technology Database contains the data for the machining process plans and can be customized for your facility's machining methodology. When you do these exercises, your results may not be exactly the same as described in the steps and illustrated in the figures. This is because the machining sequences and operations data in your Technology Database may be different from the database used to produce the documentation.**
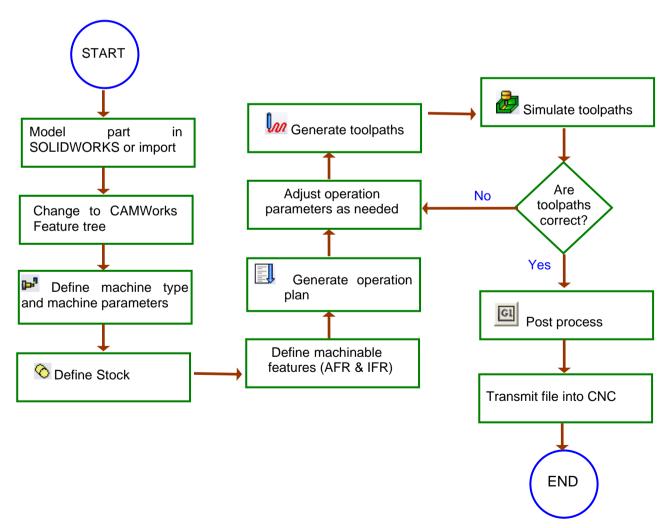
Wire EDM 1

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/CAMWorks Solids.
2. Change to the CAMWorks Feature tree.
3. Define the Machine type and machine parameters.
4. Define the Stock definition.
5. Define machinable features and adjust feature parameters.
6. Generate the Operation Plan and adjust operation parameters.
7. Generate Toolpaths.
8. Simulate Material Removal.
9. Post Process the toolpaths.

The following series of exercises show you how to generate finish toolpaths on a SOLIDWORKS or CAMWorks Solids 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 tutorials in this manual. When you install CAMWorks, these files are installed automatically.

Topics covered in this tutorial:

- Using the CAMWorks Command Manager or Workflow toolbar to execute commands
- Editing the Machine Definition
- Editing the Stock Definition
- Changing Default Settings using Options dialog box
- Defining Machinable Features Automatically
- Generating Operation Plan
- Generating Toolpaths
- Simulate Toolpaths
- Step Through Toolpath
- Post Process Toolpaths

Step 1: Model Part in SOLIDWORKS/CAMWorks Solids or Import Part

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

Open the part file **EDM2AX_1.SLDPRT** in the following folder.

Drive:\CAMWorksData\CAMWorks202x\Examples\Tutorial_Parts\EDM

EDM2AX_1.SLDPRT

Viewing the FeatureManager Design Trees

The FeatureManager design tree displays the list of the features, sketches, planes and axes related to the part.

To use CAMWorks, you need to move between SOLIDWORKS/CAMWorks Solids trees and the CAMWorks Feature trees. Different tabs are provided to access the SOLIDWORKS/CAMWorks Solids trees and the CAMWorks Feature trees. Click the 📌 *Pin* button to continuously view this Tree area.

If the CAMWorks tabs [🔳, 🔧, 🔩] are not visible, you can expand the size of the tree. Position the cursor on the line that divides the tree area from the graphics area. When the cursor changes to a bar, drag the bar to the right until the tabs display.
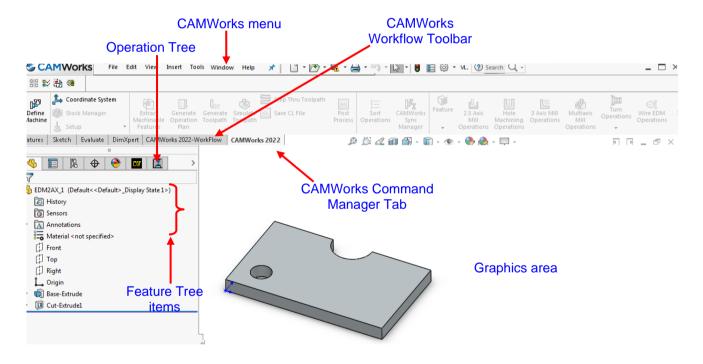
Step 2: Change to CAMWorks Feature Tree

🔳 Click the *CAMWorks Feature Tree* tab.

When the CAMWorks Feature tree is displayed, it initially lists Configurations, Stock Manager, Machine and Recycle Bin items.

> 📝 **The icons that display for the 'Machine' are indicative of the machine which is currently selected. Step 3 of this tutorial explains how to select an EDM machine (shown in the images on the next page).**

CAMWorks Machining Trees

The CAMWorks machining trees provide an outline view of the machining information for the model. Initially, the CAMWorks Feature tree shows the Configurations, CAMWorks 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/CAMWorks Solids trees and the CAMWorks trees.

- 🔳 **Configurations**

Multiple CAMWorks datasets are supported. Each dataset is called a configuration. You can use configurations to support multiple machines and SOLIDWORKS configurations.

- **Stock Manager**

  The turn stock is the material from which the part will be machined. You can define the stock as a cylinder (for bar stock) or as a closed sketch (for a forging or casting) and specify the type of material.

- **Machine**

  Turn   Mill   Mill-Turn   Wire EDM

  The Machine item defines the machine tool that will be used to machine the part. 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 is used to store machinable features that you do not intend to machine.
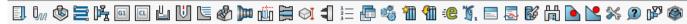
CAMWorks Menu and CAMWorks NC Manager

1. Click *CAMWorks* on the SOLIDWORKS/CAMWorks Solids menu bar.

2. The CAMWorks NC Manager item is present in both the Feature tree and the Operation tree. Right click on the *CAMWorks NC Manager* in the tree. A list of executable commands is displayed on the context menu. These right-click context menus provide access to a variety of commands. The commands displayed on the context menu for *CAMWorks NC Manager* item in the Feature tree is different from that in the Operation tree.

3. The CAMWorks commands are explained in the CAMWorks Context-based Help.

CAMWorks Command Manager

Click *CAMWorks* on the SOLIDWORKS/CAMWorks Solids menu bar. This action displays the CAMWorks Command Manager. The CAMWorks Command Manager provides access to the main CAMWorks commands. The commands are explained in the CAMWorks Context-based Help.

CAMWorks Command Manager

Customization of CAMWorks Command Manager

Command Manager is a context-sensitive toolbar that can be dynamically updates based on the toolbar you want to access. It provides access to the main CAMWorks commands found on the CAMWorks menu. By default, it has toolbars embedded in it.

The CAMWorks Command Manager can be customized. Right click anywhere on the CAMWorks Command Manager and select *Customize* from the RMB context menu. The *Customize* dialog box will be displayed. Use the *Toolbars, Commands, Menus, Keyboard shortcut, Mouse gesture and Options* tab of this dialog box to customize the Command Manager as per your requirements.

Alternative Access to CAMWorks Commands

All the commands executed from the CAMWorks Command Manager can also be alternatively accessed from the RMB context menu of the CAMWorks NC Manager. This is a context menu. To execute the command, right click on the CAMWorks NC Manager item in the tree and select the desired command from the RMB context menu. In addition to the Command Manager commands, this right click context menu also provides access to a variety of commands.

CAMWorks Workflow Toolbar

The CAMWorks Workflow toolbar contains all the major commands of CAMWorks in a separate toolbar. When executed in sequential order, they define the workflow in CAMWorks. The workflow includes, setting up of the machine, stock and subsequent features, operations and toolpaths.

CAMWorks Options

1.   Click the *Options* button on the CAMWorks Command Manager.

    OR

    Click *CAMWorks* on the SOLIDWORKS/CAMWorks Solids menu bar and select *Options* from the dropdown menu.

Options button on the CAMWorks Command Manager

The Options dialog box is displayed. This dialog box contains various tabs to customize settings and options related to Saving data, Feature recognition, Display, Simulation, Updation and Rebuild of CAMWorks data and File locations.

2. Click on each different tab in this dialog box and click the *Help* button. Each tab is explained in the Context-based Help.

3. [x] To close the Context-based Help, click the *Close* button in the upper right corner of the Help window to close the window.

4. Click *OK/Cancel* to close the Options dialog box.

Step 3: Define the Machine

The machine definition specifies the type of machining that will be done for the current model and the associated machine tool for proper generation of the NC program. Based on the selected machine, the Extract Machinable Features command will recognize mill, turn or Wire EDM features. The machine can be changed to produce NC code for an alternate machine tool of the same type. If the machine type changes, then all features and operations will be deleted. The icons that display in the tree identify the machine:

<div align="center">

Turn   **Mill**   **Mill-Turn**   **Wire EDM**

</div>
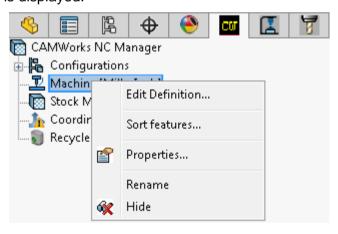
1. Right click the *Machine* item in the CAMWorks Feature tree and select the *Edit Definition* from the context menu.

   OR

   Double click the *Machine* item in the Feature tree to edit the machine definition.

   The *Machine* dialog box is displayed.

<div align="center">

'Edit Definition' on the context menu

</div>
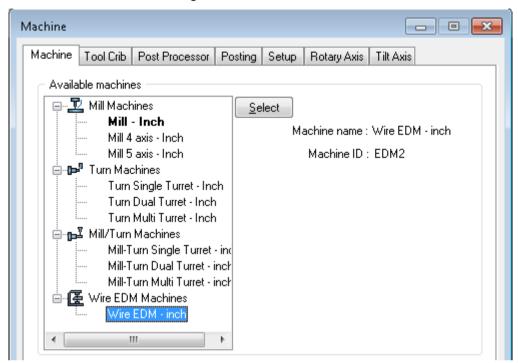
Did You Know: **In the Feature and Operation trees, instead of right-clicking items and selecting Edit Definition on the context menu, you can double-click the item to open the corresponding dialog box for editing the Stock Manager, Machine, Setups, Features and Operations.**

In the Available machines list, make sure *Wire EDM–Inch* is selected as Active machine.
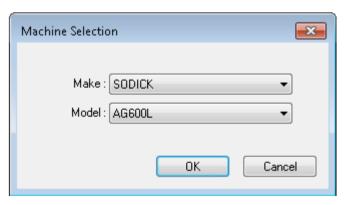
This is the machine used for all the tutorials for EDM machining in this manual. When you use CAMWorks to machine your own parts, select the machine tool you want to use.

2. Click *OK* to close the *Machine* dialog box.

Selecting the Machine

3. ⬛ Right click *Machine [Wire EDM–Inch]* in the CAMWorks Feature tree and click *Select Machine* on the context menu.

4. In the Machine Selection dialog box, select *SODICK* for the Make and *AG600L* for the Model from dropdown list.

Machine Selection Dialog Box
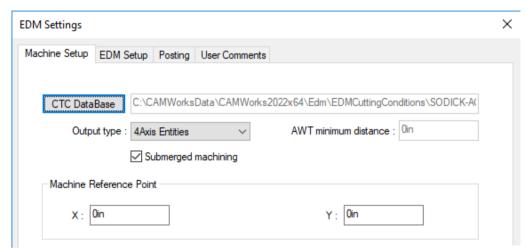
5. Click *OK* to close the dialog box.

📝 **Contact your CAMWorks Reseller for more information on obtaining and/or customizing machines for your facility.**

---

6.   Right click *Machine [Wire EDM–Inch]* and select *Edit Settings* on the context menu.

     The EDM Settings dialog box is displayed.

7. Observe that in the Machine Setup tab, the path to the *CTC Database* is identified.

     This database has been set up for the SODICK machine. The cutting condition database sets machine-specific parameters for power settings, etc.

Machine Setup Tab on EDM Settings Dialog box

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** from the list.

10. Click the *Posting* tab.
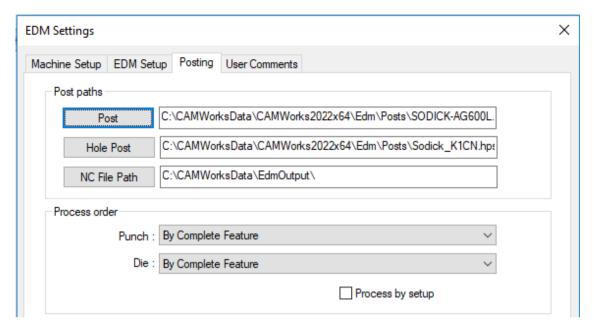
     This tab identifies the *post processor*. This post processor has been set up for the SODICK machine and will be used for machining in this tutorial (does not need to be changed). The parameters on this tab provide information used to generate the NC program.

11. Make sure *By Complete Feature* option is selected from dropdown list for Punch and Die Process Order.

     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, and then all skim passes.

     -    **For each Punch feature:** The order is rough cut leaving glue stop, 0 - 6 skim passes, and then Glue stop cut off.

12. Click *OK* to apply the changes and close the dialog box.
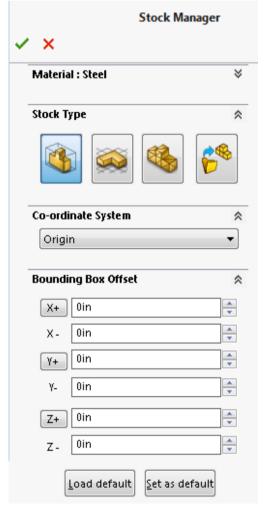
EDM Settings dialog box

Step 4: Define the Stock

The stock is the material from which the part will be machined. The default stock is the smallest cube (bounding box) that the part will fit into. Typically, this is not the size of the stock you will be using. You can change the stock definition either by offsetting the bounding box from the part or by defining the stock from a sketch and a depth *(extruded sketch)* or from an STL file.

1. Double click *Stock Manager* in the Feature tree.

   The Stock Manager dialog box is displayed.

2. For this tutorial, leave the stock set as a default *Bounding Box*.

3. Click the *Material* button, select *Steel* from dropdown list.

4. Click *OK* to apply the changes and close the Material dialog box.

5. Click *OK* to close the Manage Stock dialog box.

Stock Manager dialog box

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 *New 2.5 Axis EDM Feature* and *New 4 Axis EDM Feature* commands. Using the interactive method, you can define Die (Pocket), Punch (Boss) and Open Profile machinable features.

How AFR analyses a Solid Part

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 does not machine the SOLIDWORKS/CAMWorks Solids features directly. It creates a separate list of Machinable Features instead. This is because design features are typically not the same as manufacturing features and may have several areas that need to be machined in different ways with different tools.

Using Automatic Feature Recognition (AFR)

Define machinable features automatically:

1. Click the *CAMWorks Options* button on the CAMWorks Command Manager.

2. Click the *EDM Features* tab and make sure the *Perimeter Punch* feature option is not checked.

3. Click *OK* to apply the changes and close the dialog box.

   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.

4. In the Feature tree, click the *Extract Machinable Features* button on the CAMWorks Command Manager.
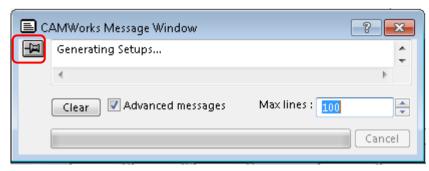
   OR

   Right click *CAMWorks NC Manager* in the tree and select *Extract Machinable Features* on the context menu.

   The *CAMWorks Message Window* displays automatically to report the progress and status of the process.

5. Click the *Message Window* button on the CAMWorks Command Manager, if the CAMWorks Message Window is not displayed.

CAMWorks Message Window
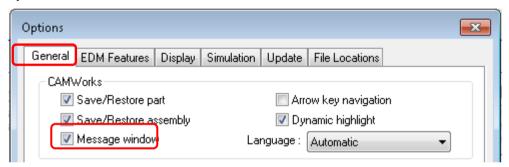
6. You can control whether this window displays temporarily or permanently by clicking the Pushpin button on Message Window.

   OR

   Click on the *CAMWorks Options* button on the CAMWorks Command Manager.

   The *Options* dialog box is displayed.

7. Check the *Message Window* option on the *General* tab of this dialog box to view this window continuously.

General Tab of CAMWorks Options dialog box

8. Click *OK* to close the Options dialog box.

   CAMWorks generates the EDM Part Setup and machinable features. The items display in the CAMWorks Feature tree.
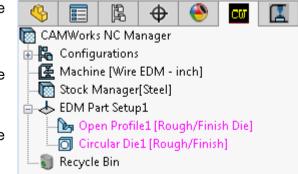
   The EDM Part Setup is the 2 axis plane that the tool movement will be based on. It has an origin location, and X, Y, and Z direction vectors. The EDM Part Setup is created automatically; however, you can move the origin, and change the direction and angles of the X and Y axes.

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

   The Feature tree allows you to:

   - Copy, rename, suppress, combine and delete machinable features

   - Change machinable feature parameters

   - Change the order in which the features are machined

Features recognized by AFR

- Insert 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: Features display in a different color when they have no operations generated. Once operations are successfully generated, they display in another color. If operations are not generated for certain features (this occurs when feature conditions have not been defined in the Technology Database), then they retain their original color indicating non-generation of operations. 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 based on information in the Technology Database. In some situations, the operations defined for a feature may not be sufficient and additional operations may be required. You can insert operations interactively using the New Operation command. This command is explained in the CAMWorks Context-based Help.

Click the *Generate Operation Plan* button on the CAMWorks Command Manager.

OR

Right click *CAMWorks NC Manager* or *EDM Part Setup1* in the Feature tree and select *Generate Operation Plan* on the context menu.

> **Note:**
>
> - **If you execute the Generate Operation Plan command from the Command Manager or the CAMWorks NC Manager level, then operations will be generated for all prismatic features in the tree, regardless of the active item in the tree.**
>
> - **If you execute the Generate Operation Plan command from the EDM Part Setup level, then operations will be generated only for those prismatic features listed under the given EDM Setup.**
>
> - **If you execute the Generate Operation Plan command at the feature level (by right-clicking on a feature listed in the Feature tree and executing Generate Operation Plan command from the RMB context menu), then operations will be generated only for the selected feature.**

CAMWorks generates the operation plan for all the machinable features in EDM Part Setup1. The operations are listed in the CAMWorks Operation tree, which displays automatically. The Operation tree can also be accessed by clicking the CAMWorks Operation Tree tab.
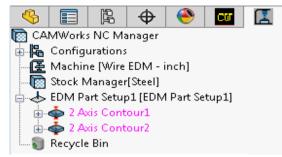
The CAMWorks Operation tree provides an outline view of the operations for the machinable features. Operations are listed under the EDM Part Setup in the same order as the machinable features. At the top of the tree is the CAMWorks NC Manager. The Stock Manager and Machine
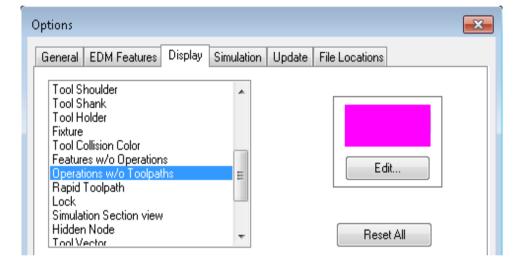
items are same as in the CAMWorks Feature tree. You can change the stock size and shape to produce G-code.

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.

Operations generated on executing GOP command

Display tab of Options dialog box

Did You Know: If an operation displays in a color instead of **black**, then toolpaths have not been generated for that particular operation. 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 for operations without toolpaths on the Display tab in the Options dialog box.

Adjusting Operation Parameters

Generate Operation Plan command generates one cutting operation per each 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 generating the toolpath and post processing the part.

Edit operation parameters:
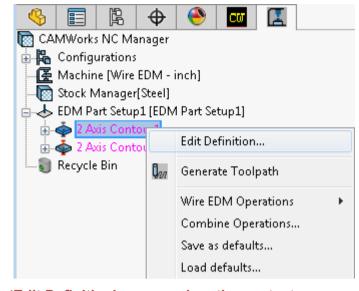
1. Double click *2 Axis Contour1* in the Operation tree.

    OR

    Right click on *2 Axis Contour1* and select *Edit Definition* on the context menu.

    The 2 Axis Contour dialog box is displayed. This dialog box gives you access to all the parameters used to create the toolpath.
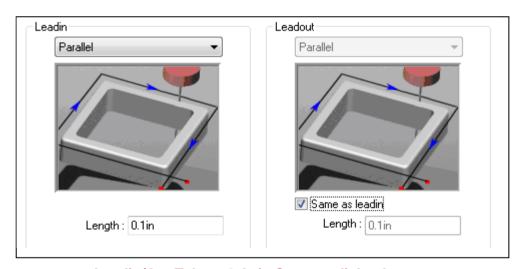
2. Click on the *EDM* tab and set the following parameters:

    - Number of skims to **2**

    - Glue Stop to ***No Glue Stop***

'Edit Definition' command on the context menu

3. Click on the *Leadin/Out* tab,

    - In Leadin group box, set Leadin to *Parallel* with length **0.1in.**

    - In the Leadout group box, make sure the *Same as Leadin* is checked.

        When the *Same as Leadin* is checked, CAMWorks set the parameters of Leadout as same as Leadin. When *Same as Leadin* is not checked, you can set the different required parameters for Leadin and Leadout.

Leadin/Out Tab on 2 Axis Contour dialog box

4. Click *OK* to apply the changes and close the dialog box.

5. Double click *2 Axis Contour2* in the Operation tree.

    OR
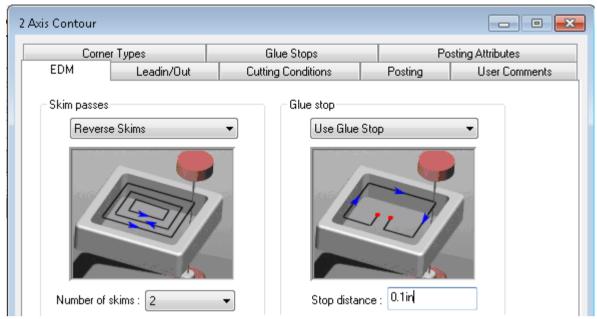
    Right click on *2 Axis Contour2* and select *Edit Definition* on the context menu.

    The 2 Axis Contour dialog box is displayed.

6. Click on the *EDM* tab and set the following parameters:

    - Number of skims = **2**

- Glue stop = **Use Glue Stop**
- Stop distance = **0.1in**

EDM Tab on 2 Axis Contour dialog box

7. Click on the *Leadin/Out* tab,

- In Leadin group box, set Leadin to *Perpendicular* with length **0.10in**
- In the Leadout group box, make sure the *Same as Leadin* is checked.

8. Click *OK* to apply the changes and close the dialog box.

Step 7: Generate Toolpaths

CAMWorks calculates toolpaths using the operation parameters and the feature's size and shape. After toolpath is generated for a particular operation, then that operation is displayed in black-colored font in the Operation tree.

1. 📝 Click the *Generate Toolpath* button on the CAMWorks Command Manager.

OR

Right click the *EDM Part Setup1* or *CAMWorks NC Manager* in the Operation tree and select *Generate Toolpath* on the context menu.

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

Note:
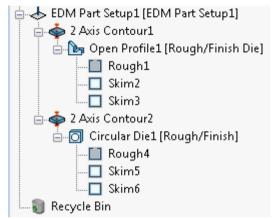
Just like the Generate Operation Plan command, the scope of the Generate Toolpath command too depends from which level the command is executed.

- **Executing the Generate Toolpath command from the CAMWorks NC Manager level generates toolpaths for all the operations.**

- **Executing the Generate Toolpath command at EDM Part Setup level generates toolpaths only for operations listed under the given EDM setup.**

- **You can also generate toolpaths for each operation individually by right-clicking on an operation and executing Generate Toolpath command on the context menu.**

2. Click the (⊞) plus sign next to *2Axis Contour1* and *2 Axis Contour2* in the Operation tree, then click the (⊞) plus sign next to *Open Profile1* and *Circular Die1*.

3. Highlight the first operation in the Operation tree. This action displays the toolpath for that operation on the part.

4. Highlight the each toolpath pass under the operation. The toolpath for that corresponding operation is displayed.

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

Operation tree after generation of Toolpaths

Step 8: Simulate Toolpath

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

1. Click the *Simulate Toolpath* button of the CAMWorks Command Manager.

   OR

   Right click *EDM Part Setup1* in the tree and select *Simulate Toolpath* on the context menu.

   The Toolpath Simulation toolbar is displayed.

   The toolbar controls allow you to:
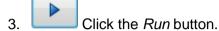
   - Run the simulation in *Tool* or *Turbo* mode.
   - Display the simulated part, the design part and a comparison of the two by clicking on the *Show Difference* button .
   - Customize the display of the stock, tool and tool holder (*wireframe, translucent, shaded, or no display*) in Tool mode.
   - Note that these options are disabled in Turbo mode.
   - 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 using *Pause* button in both *Tool* and *Turbo* mode.
   - Dynamically change the orientation of the part using zoom, pan, rotate, etc.
   - Control the simulation speed by dragging the Simulation Speed Control slider

   left or right.

If you want to simulate only the toolpaths for a given operation, you can right click that operation, and then select *Simulate Toolpath* in context menu.
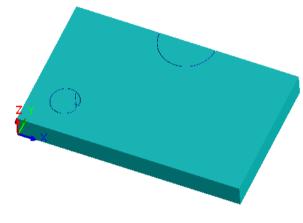
2. Set the simulation mode to *Tool mode* by clicking on the *Tool mode* button .

3. Click the *Run* button.

The simulation runs in Tool mode.

4. Click the *Select Chips and Press Ctrl+D Key* button. This button is enabled when the simulation is complete. CAMWorks provides two methods for removing loose chips. Follow the steps:

i. Pick the two loose stock chips to remove, then press the *Ctrl+D* key on the keyboard.

ii. The loose chips are removed in the graphics area.

Simulation Mode

iii. Click the *Select Chips and Press Ctrl+D Key* button to exit the function.

iv. Click the *Goto Start* button and click run the simulation again.

v. Click the *Select Chips and Press Ctrl+D Key* button.

vi. Hold down the *Ctrl* key and pick the stock piece you want to keep, then press the *Ctrl+D* key on the keyboard.

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

5. Click the *Close* button in the upper right corner to close the simulation and return to the SOLIDWORKS/CAMWorks Solids display.

Step Through Toolpath

CAMWorks also provides the ability to visualize the tool motion and verify tool positions using the Step Thru Toolpath command.

1. **Executing the Step Thru Toolpath command:** Both the *Simulate Toolpath* and *Step Thru Toolpath* commands can be executed at the global, setup and operation levels.

- **Global level:** To step through the toolpaths of all the operations in the Operation tree, right click on the *CAMWorks NC Manager* in the Operation tree and select *Step Thru Toolpath* command in the context menu.

OR
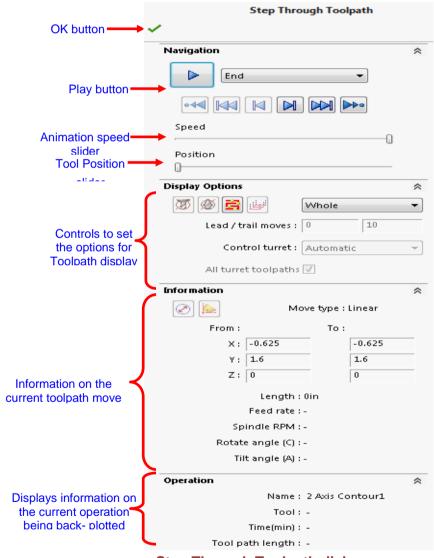
Select *Step Thru Toolpath* command on the Command Manager.

- **Setup Level:** Right click on the *EDM Part Setup1* and select *Step Thru Toolpath* command in the context menu.

- **Operation level:** Right click on the desired operation in the Operation tree and select *Step Thru Toolpath* in the context menu.

On execution of the Step Thru Toolpath command using one of the above methods, the *Step Thru Toolpath* dialog box is displayed.

2. Use controls given in the *Display Options* group box to set the options as to how the toolpath will be displayed during the Step Through process.

3. In the *Play Tool Until* dropdown list, leave the end condition set to the default *End.*

4. Use the *Animation speed* slider to adjust the animation speed.

5. Click the *Play* button ▶ to continuously back-plot the tool till the end condition specified in the *Play Tool Until* field is reached.

OK button

Play button

Animation speed slider

Tool Position slider

Controls to set the options for Toolpath display

Information on the current toolpath move

Displays information on the current operation being back-plotted
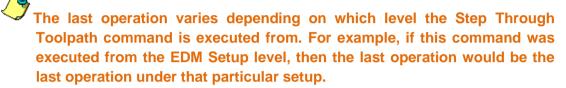
Step Through Toolpath dialog

6. During animation, the *Play* button display changes to *Pause* button ⏸. Click the *Pause* button to pause the animation while it is in progress.

7. When the animation has not yet begun or when it is paused, the *Forward Single Step, Goto End of Current Toolpath* and *Goto End* buttons are active.

   - Use the *Forward Single Step* button to move one toolpath record forward each time the button is clicked.

   - Use the *Goto End of Current Toolpath* button to move the tool position to the last toolpath record of the current operation.

   - Use the *Goto End* button to move the tool position to the last toolpath record of the very last operation.

     The last operation varies depending on which level the Step Through Toolpath command is executed from. For example, if this command was executed from the EDM Setup level, then the last operation would be the last operation under that particular setup.

8. When the animation has been completed or when it is paused, the *Reverse Single Step, Goto Start of current Toolpath* and *Goto Start* buttons are active.

   - Use the *Reverse Single Step* button to move one toolpath record backwards each time the button is clicked.

   - Use the *Goto Start of Current Toolpath* button to move the tool position to the first toolpath record of the current operation.

   - Use the *Goto Start* button to move the tool position to the first toolpath record of the very first operation.

9. If you are unsure about the use of any parameter within this dialog box, click on the *Help* button ? at the upper right corner. This action will display the Context-based Help for Step Thru Toolpath command.

10. Click the *OK* button in the upper left corner to close the dialog box.

Save the part with the machinable feature data

1. Click the *Options* button the CAMWorks Command Manager.

   OR

   Select the *Options* command from the RMB context menu of the CAMWorks NC Manager in the tree.

   The *Options* dialog box is displayed.

2. On the *General* tab, make sure the *Save/Restore part* option is checked.

3. Click *OK* to apply the settings and close the dialog box.

   If this *Save/Restore part* option is checked, when you save and close a part document, the machining data is saved and restored with the part design information when the part document is reopened.

   If this option is not checked, when you save and close a part document that contains at least one Setup, a message indicates that Save/Restore is disabled. If you click Yes, CAMWorks

saves all machining data before closing the file. If you click No, CAMWorks closes the part and discards any new machining data since the last save.

4. Select *Save As* on the File menu. Browse to the location where you wish to save the file.

5. In the *Save As* dialog box, assign the desired file name 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 9: 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. These are text file that you can read, edit and print using a word processor or text editor.

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

1. Click the *Post Process* button on the CAMWorks Command Manager.

   OR

   Right click *CAMWorks NC Manager* in the Operation tree and click *Post Process* on the context menu.

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

   Typically, the NC program file is stored in the folder that contained the last part that was opened. If you want these files in another location, you can change the folder location.
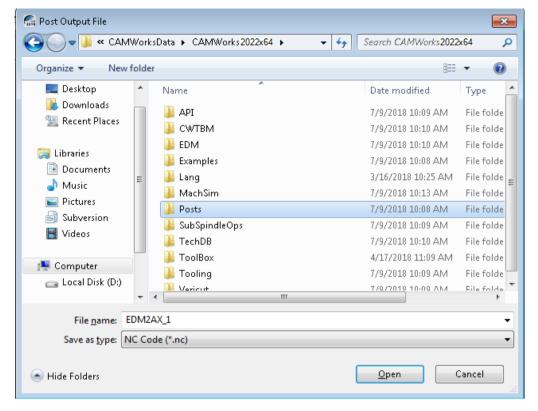
   If the Post Process command is grayed out on the CAMWorks Command Manager or on any context menu, make sure that you have generated the toolpaths.

2. In the *Post Output File* dialog box, 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.
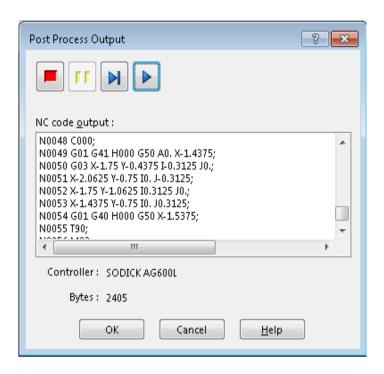
Post Output File Dialog box

3. In the *File name* textbox, type the suitable file name, and then click *Save* button.

    The *Post Process Output* dialog box is displayed.

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 is displayed.

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 dialog box.

Post Process Output dialog box

2. LEARNING 2 AXIS CONTOUR CUTTING

This chapter provides an opportunity to learn CAMWorks Wire EDM through step-by-step exercises.

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

Before you begin with the tutorials given in this chapter, ensure that you are through with the concepts explained in Chapter 1.

The tutorial parts are installed when you install CAMWorks and are in the *\Examples\Tutorial_Parts\EDM* folder on your computer. This folder is inside the CAMWorks data folder.

(Drive:*CAMWorksData\CAMWorks202x\Examples\Tutorial_Parts\EDM*).

IMPORTANT!   CAMWorks uses a set of knowledge-based rules to assign machining operations to features. The Technology Database contains the data for the machining process plans and can be customized for your facility's machining methodology. When you do these exercises, your results may not be exactly the same as described in the steps and illustrated in the figures. This is because the machining sequences and operations data in your Technology Database may be different from the database used to produce the documentation.

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 New 2.5 Axis EDM Feature function. A CAMWorks file can contain both types of features.

This tutorial 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.

Open the Part and Defining the Machine and Stock
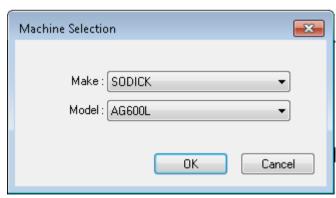
1. Open the part file **EDM2AX_2.SLDPRT** in the following folder.

   Drive:\CAMWorksData\CAMWorks202x \Examples\Tutorial_Parts\EDM

2. Click the *CAMWorks Feature Tree* tab.

3. Double click the *Machine* item in the CAMWorks Feature tree.

   The Machine dialog box is displayed.

4. In the Available machines list, highlight *Wire EDM-Metric*, and click the *Select* button.

5. Click *OK* to apply changes and close the Machine dialog box.

EDM2AX_2.SLDPRT

6. Right click *Machine [Wire EDM-Metric]* in the tree and select the *Select Machine* on the context menu.

7. In the Machine Selection dialog box, select *SODICK* for the Make and *AG600L* for the Model from dropdown list.

8. Click *OK* to close the dialog box.

9. Right click *Machine [Wire EDM-Metric]* in the Feature tree and select *Edit Settings* on the context menu.
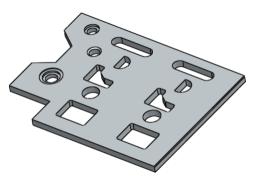
   The EDM setting dialog box is displayed.

10. On the EDM Setup tab, make sure the Wire diameter is set to **0.20mm**.

Machine Selection Dialog Box

11. On the Posting tab, make sure the Process order for Die is set to *By Complete Feature.*

12. Click *OK* to close the dialog box.

   - 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, and then all skim passes.

13. Double click *Stock Manager* in the Feature tree.

The Stock Manager dialog box is displayed.

14. ✅ Click *OK* to use the default Bounding box and Material.

Extracting Machinable Features and Customizing AFR

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

1. ✂ Click the *Options* button on the CAMWorks Command Manager.

2. On the EDM Features tab, make sure the Perimeter punch feature is checked.

3. Click *OK* to close to dialog box.

4. 🗒 Click the *Extract Machinable Features* button on the CAMWorks Command Manager.
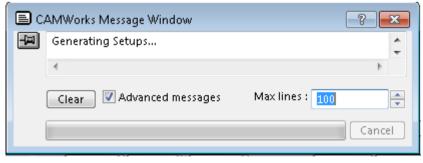
   OR

   Right click *CAMWorks NC Manager* in the tree and select *Extract Machinable Features* on the context menu.

   The *CAMWorks Message Window* is displayed automatically to report the progress and status of the process.

5. 📄 Click the *Message Window* button on the CAMWorks Command Manager, if the CAMWorks Message Window is not displayed.

6. CAMWorks generates the EDM Part Setup and machinable features. The items display in the *Feature* tree 🔳.
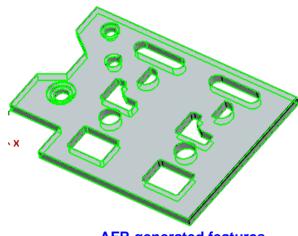
Features recognized by AFR

CAMWorks Message Window

7. Highlight the first feature item in the Feature tree, hold down the *Shift* key, then highlight the last feature item.

AFR generated features

This action displays all features on the part (in the graphics area).

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.

8. Click the *Options* button on the CAMWorks Command Manager to display Options dialog box.

9. On the EDM Features tab, remove the check marks from the features type option in the Extract machinable features group box:

   - Countersink holes
   - Counterbore holes
   - Multi-step holes options

10. Check the On Rebuild Delete option for these same features types as shown in the image on next page:

   - Countersink holes
   - Counterbore holes
   - Multi-step holes

11. Optionally, change the *Apply to* setting.
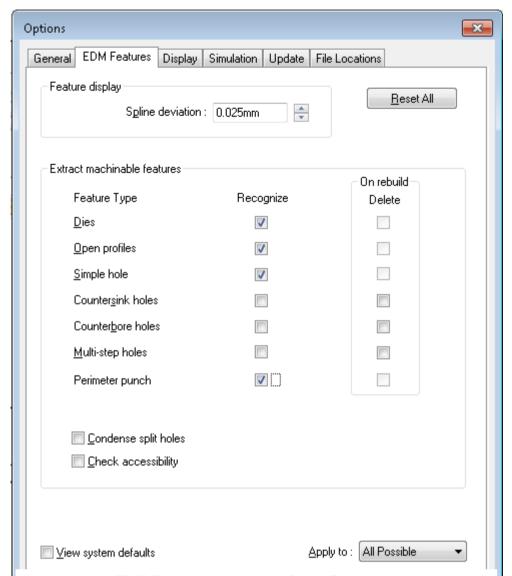
   This option allows you to select how the settings on this tab will be applied:

   - **Active Document** - The part that you are currently working on.
   - **System Default** - All new parts that are created.
   - **All Possible** - The current part as well as all new parts.

     The values that displayed when you opened this dialog box will be retained for the current part.

12. Reset the *Active Document* option in Apply to setting.

EDM Features tab on Options dialog box

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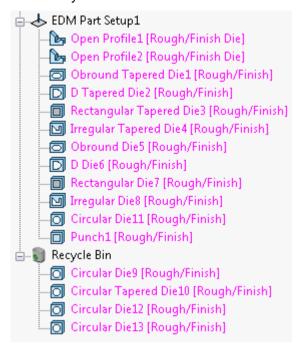
If you want the System Defaults to apply, disable the Save/Restore part or Save/Restore assembly option on the General tab in the CAMWorks Options dialog box, then save the template with a new name.

13. Click *OK* to close the Options dialog box.

14. Click the *Extract Machinable Features* button on the CAMWorks Command Manager.

    OR

    Right click *CAMWorks NC Manager* in the tree and select *Rebuild* on the context menu.

Note that the Circular Die features for the Countersunk holes, Counterbore holes and Multi-step holes are now move in the Recycle Bin.

Recognized operations

Generating a 2 Axis Contour Operation and Toolpaths

1. Click the *Generate Operation Plan* button on the CAMWorks Command Manager.

   OR

   Right click *CAMWorks NC Manager* or *EDM Part Setup1* in the Feature tree and select *Generate Operation Plan* on the context menu.

   When you select *Generate Operation Plan*, operations will be 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 Command Manager.

   OR

   Right click the *EDM Part Setup1* or *CAMWorks NC Manager* items in the Operation tree and select *Generate Toolpath* on the context menu.

Simulating and Post Processing Toolpaths

Simulation Toolpath

1. Click the *Simulate Toolpath* button on the CAMWorks Command Manager.

   OR

   Right click *CAMWorks NC Manager* or *EDM Part Setup1* in the tree and select *Simulate Toolpath* on the context menu.

The Toolpath Simulation dialog box is displayed.

2. Set the display parameters:

   - Stock: *Shaded Display*

   - Tool: *Shaded Display*

   - Tool Holder : *Shaded Display*

   - Target Part : *No Display*

3.   Click the *Run* button.
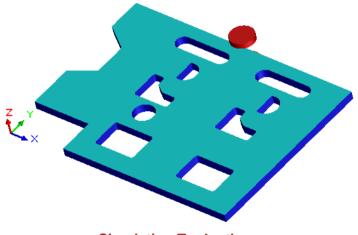
4. Click the *Select Chips and Press the Ctrl+D Key* button.

5. Hold down the *Ctrl* key and pick the stock piece you want to keep, and then press the *Ctrl+D key* on the keyboard.

   The loose chips are removed in the graphics area.

6. Again, click the *Select Chips and Press Ctrl+D Key* button to exit the function.

Simulation Toolpath

7. Click the *Close* button in the upper right corner to cancel the simulation and return to the SOLIDWORKS/CAMWorks Solids display.

Post Process Toolpath

1.   Click the *Post Process* button on the CAMWorks Command Manager.

   OR

   Right click *CAMWorks NC Manager* in the Operation tree and click *Post Process* on the context menu.

   The *Post Output File* dialog box is displayed.

2. Browse to the folder where you wish to save the file.

If you are running CAMWorks in Demo mode, the Post Process functionality will be disabled.

3. If you do not wish to save the file with the default file name, assign another file name in the *File name* field.

4. Click *Save* to save the file.

   The *Post Process Output* dialog box is displayed.

5. Click the *Play* button.

   This command generates the NC code. The generated NC code can be viewed in the *NC code output* area of the dialog box by using the vertical scroll.

6. After viewing the code, click *OK* to close the dialog box.

2 Axis Contour 2

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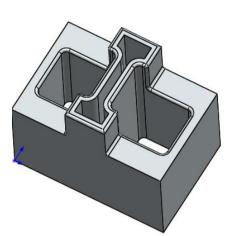
Automatic Feature Recognition may not be able to find all features on a part or may not recognize all the features that you want to machine. When this occurs, you can define these features interactively. In this tutorial, AFR finds only the irregular die in the center of the part. You can interactively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

Open the Part and Defining the Machine and Stock

1. Open the part file **EDM2AX_3.SLDPRT** in the following folder.

    Drive:\CAMWorksData\CAMWorks202x\Examples\Tutorial_Parts\EDM

2. Click the *CAMWorks Feature Tree* tab.

3. Double click the *Machine* item in the CAMWorks Feature tree.

    The Machine dialog box is displayed.

4. In the Available machines list, make sure *Wire EDM–Inch* is selected as Active machine.

5. Click *OK* to close the dialog box.

6. Right click *Machine [Wire EDM–Inch]* in the tree and click *Select Machine* on the context menu.

7. In the Machine Selection dialog box, make sure *SODICK* is selected for the Make and *AG600L* for the Model from dropdown list.

EDM2AX 3.SLDPRT

8. Click *OK* to close the dialog box.

9. Right click *Machine [Wire EDM–Inch]* in the tree and select *Edit Settings* on the context menu.

10. On the Posting tab, make sure *By Complete Feature* is selected from dropdown list for both Punch and Die Process order.

11. Click *OK* to apply changes and close the dialog box.

    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.

12. Double click *Stock Manager* in the Feature tree.

13. In the Stock Manager dialog box, click *OK* to use the default Bounding box and Material.
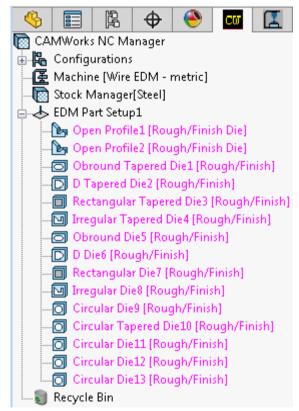
Extract Machinable Features

1. Click the *Options* button on the CAMWorks Command Manager.

2. On the EDM Features tab, make sure the *Perimeter Punch* feature is not checked and click *OK*.

3. Click the *Extract Machinable Features* button on the CAMWorks Command Manager.
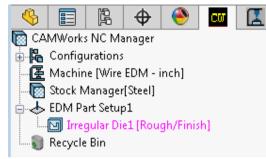
   AFR finds only the Irregular Die in the center of the part.

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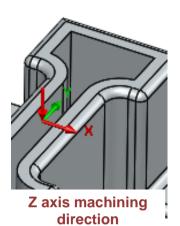
   The red arrow on the triad indicates the negative Z direction. The direction is correct for this part.

   When necessary, you can reverse the direction by double clicking *EDM Part Setup1* in the tree and clicking the *Reverse Setup Direction* option in the EDM Setup dialog box.

   The SOLIDWORKS/CAMWorks Solids axis does not control CAMWorks machining. However, it is used as a reference for certain CAMWorks functions (for example, offsetting the stock).

AFR Generated Features

Z axis machining direction

Inserting 2.5 Axis EDM Features Interactively
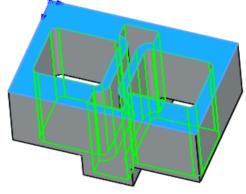
1. Rotate the part so that the bottom face is visible.

2. Right click *EDM Part Setup1* in the Feature tree and select *2.5 Axis EDM Feature* on the context menu.

   The 2.5 EDM Feature dialog box is displayed.

3. On the Feature Definition tab, make sure *Die* is the selected as Feature type.

4. Pick the bottom face of the part.

   Three Die features are displayed in the Selected Entities group box.

5. Click *Die – 2* and *Die – 3* to view the features on the part.

6. Click *Die – 1*. This is the Irregular Die feature that has already been found by AFR so you can delete it.

7. Highlight *Die – 1* in the list and press the *Delete* key on keyboard.

   Now, only two Die features are available in the Selected Entities group box.

Pick the bottom face of the Part

8. 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.

9. ✔ Click *OK* 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:

10. Once again right click *EDM Part Setup1* in the tree and select *2.5 Axis EDM Feature* on the context menu.

    The 2.5 EDM Feature dialog box is displayed.
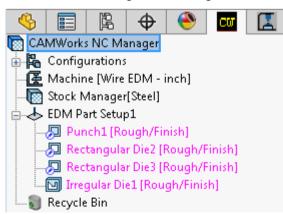
11. 🔲 Click the *Punch* button for the Feature type.

12. Pick the same face on the bottom of the part.

    Punch – 1 is listed in the *Selected Entities list box* for feature.
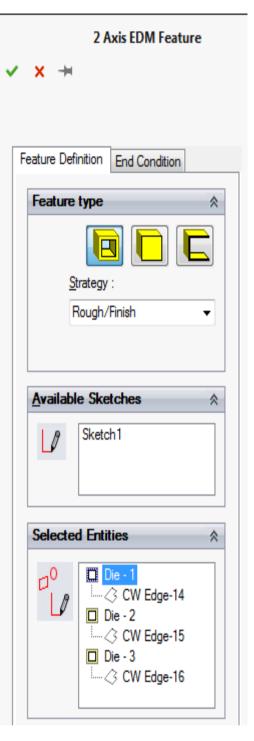
13. Click the *End Condition* tab. You can use the default *Upto Stock*.

14. ✔ Click *OK* to use the default End Condition and insert the features.

    Punch1 is listed in the Feature tree.

15. Use mouse to drag and drop the features in the order as shown in the right side image.

Re-arranged order of features

2 Axis EDM Feature

Feature Definition | End Condition

Feature type

Strategy :
Rough/Finish

Available Sketches

Sketch1

Selected Entities

Die - 1
— CW Edge-14
Die - 2
— CW Edge-15
Die - 3
— CW Edge-16

2 Axis EDM Feature Dialog Box

Generating Operations Plan and Toolpaths

1. 📄 Click the *Generate Operation Plan* button on the CAMWorks Command Manager.

   OR
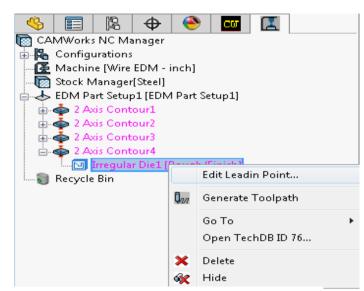
   Right click *CAMWorks NC Manager* or *EDM Part Setup1* in the Feature tree and select *Generate Operation Plan* on the context menu.

2. Rotate the part so that the top face is visible.

3. Click plus sign (⊞) next to *2 Axis Contour4* to expand in the Operation tree.

4. Right click *Irregular Die1* and *select Edit Leadin Point* on the context menu.

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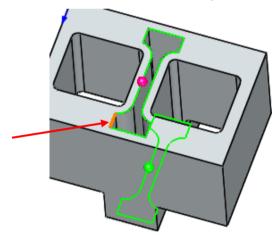
   You can set the leadin/leadout location to be at the start-point, mid-point or end-point of a segment. The current location is indicated on the feature. Rotate the part and notice that this location may gouge the part.

Select *Edit Leadin Point* on the context menu

5. Leave the *Lead in location* set to *Mid-point* and pick the segment shown in the image.

Pick the segment
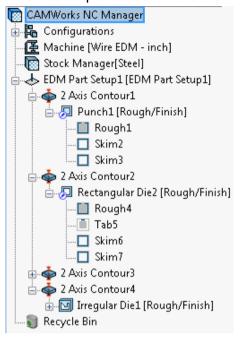
6. Click *OK* to close the dialog box.

7. 🔧 Click the *Generate Toolpath* button on the CAMWorks Command Manager.

8. Click the (⊞) plus sign next to *2 Axis Contour1* in the Operation tree to expand the operation, and then expand *Punch1*.

   The processing order is rough cut and then skim passes. 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.

9. Click the (⊞) plus sign next to *2 Axis Contour 2* in the Operation tree to expand the operation, and then expand *Rectangular Die2*.
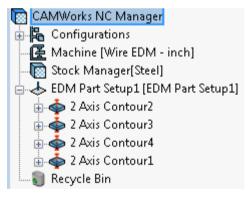
The processing order is rough cut, glue stop cut off, then skim passes. You set the Process order to *By Complete Feature* on the Posting tab in the EDM Settings dialog box and the code for all Die features will be output based on this order.

Operation tree after generated toolpath

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.

Re-arranged order of features

1. Highlight *2 Axis Contour1* in the tree and drag it on bottom 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 Command Manager.

   The Options dialog box is displayed.

3. Click the *File Locations* 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. In the Post processing group box, 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\WINNT folder, click *Open*, and then click *OK*.

Browse the location of editor

6. Click the *Post Process* button on the CAMWorks Command Manager.

    OR

    Right click *CAMWorks NC Manager* in the Operation tree and click *Post Process* on the context menu.

7. In the Post Output File dialog box, type a name for the file and click *Save*.

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. Click the button to close the editor.

```
EDM2AX_3.nc - Notepad

File  Edit  Format  View  Help

(MACHINE SODICK AG600L );
(PART EDM2AX_3 );
(CREATED-THU. 01/05/2017 07:12PM );
(MAXIMUM TAPER = 0. );
( C    =   ON  OFF IP   HRP MAO SV     V    SF   C PIK CTRL WK   WT   WS   WP    P
TP 0. (ENTER TAPER SIZE HEIGHT);
TN -2.5 (ENTER TAPER OPPOSITE HEIGHT);
G26 RA0.000   (ENTER PROGRAM ROTATION ANGLE);
N0001 (MAIN PROGRAM);
N0002 G54 G90;
N0003 G00 X-0.75 Y0.425;
N0004 T91;
N0005 G29;
N0006 T94;
N0007 T84;
CRT(FEATURE 1 CNT2X ROUGH CUT FORWARD);
CRT(2 AXIS CONTOUR2-RECTANGULAR DIE2 [ROUGH/FINISH]);
```

Opened the G-code file in Editor

2 Axis Contour 3

Topics covered in this tutorial:

In this tutorial, 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 New 2.5 Axis EDM command.

Open the Part and Defining the Machine and Stock
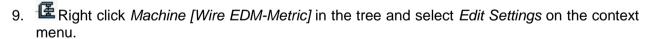
1. Open the part file **EDM2AX_4.SLDPRT** in the following folder.

   Drive:\CAMWorksdata\CAMWorks202x\Examples\Tutorial_Parts\EDM

2. ![icon] Click the *CAMWorks Feature Tree* tab.

3. Double click the *Machine* item in the CAMWorks Feature tree.

   The Machine dialog box is displayed.

4. In the Available machines list, highlight *Wire EDM–Metric*, click the *Select* button.

5. Click *OK* to close the Machine dialog box.

6. ![icon] Right click *Machine [Wire EDM-Metric]* in the tree and click *Select Machine* on the context menu.

7. In the Machine Selection dialog box, make sure *SODICK* is selected for the Make and *AG600L* for the Model from dropdown list.

8. Click *OK* to close the dialog box.

9. ![icon] Right click *Machine [Wire EDM-Metric]* in the tree and select *Edit Settings* on the context menu.

   The EDM setting dialog box is displayed.

10. On the Posting tab, set the Process order for Die to *By Complete Feature.*

11. Click *OK* to close the dialog box.

    With this option, the NC program is generated in the following machining order:

    - Rough cut, glue stop cut off, 0 - 6 full skim passes.

12. ![icon] Double click *Stock Manager* in the Feature tree.

13. ![icon] In the Stock Manager dialog box, click *OK* to use the default Bounding box and material.

EDM2AX_4.SLDPRT

Extract Machinable Features and Interactively Inserting a Part Setup

1. Click the *Extract Machinable Features* button on the CAMWorks Command Manager.

    OR

    Right click *CAMWorks NC Manager* in the tree and select *Extract Machinable Features* on the context 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.
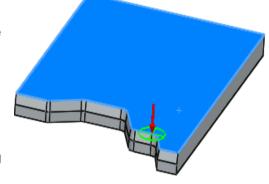
2. Right click *Stock Manager* in the Feature tree and select *Part Setup* on the context menu.

    The EDM Setup dialog box is displayed.

3. Pick the face on the top of the part.

    Face1 is listed in the Entity field.

4. Click *OK* to apply changes and close the dialog box.

    EDM Part Setup1 is listed in the tree.

Pick the top face

Inserting Vertical and Tapered Open Profile Features
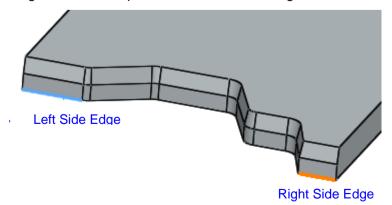
The yellow faces define a vertical wall. The blue faces define a tapered wall. First, you define the vertical open profile feature, then the tapered open profile feature.

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

    The 2.5 Axis EDM Feature dialog box is diaplayed.

2. Select *Open Profile* for the Feature type.

3. Pick the lower edge on the left side of solid part, then hold down the *Ctrl* key and pick the lower edge on the right of the solid part as shown in the image.

Left Side Edge

Right Side Edge

Pick the Lower Left and Right Side Edge

When you define an Open Profile Feature from multiple edges, you can select the chain 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 that the arrow direction indicates which side of the feature to cut on. In this tutorial, you will cut the feature from other side.
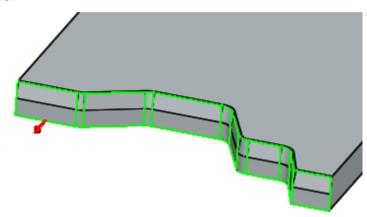
4. Click *Open Profile – 1* in the Selected Entities for features list, then click the *Flip direction to cut* check box in Feature type group box.

   The arrow direction is reversed.

5. Click the *End Condition* tab.

   This feature is vertical and no changes are required to be made on this tab.

6. ✔ Click *OK* button to apply the changes.

Indicating the reversed arrow direction
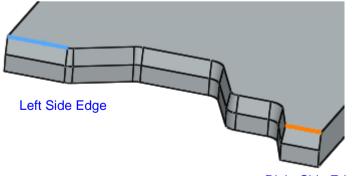
   Open Profile 1 is listed in the Feature tree.

7. Next, you will define the tapered open profile feature.

8. Right click *EDM Part Setup1* in the Feature tree and select *New 2.5 Axis EDM Feature* on the context menu.

   The 2.5 Axis EDM Feature dialog box is displayed.

9. Select *Open Profile* for the Feature type.

10. Pick the upper edge on the left side of solid part, then hold down the *Ctrl* key and pick the upper edge on the right of the solid part as shown in the image.

Left Side Edge

Right Side Edge
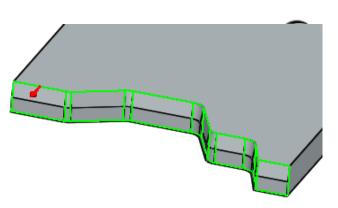
Pick the Upper Left and Right Side Edge

11. Click *Open Profile – 1* in the Selected Entities for features list, then click the *Flip direction to cut* check box in the Feature Type group box.
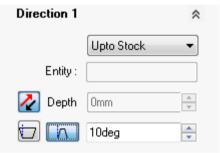
The arrow direction is reversed.

12. Click the *End Condition* tab.

13. Zoom in on the right side of the taper feature of the part. There is a 10-degree taper.

14. [△] Click the *Taper On/Off* button to turn it on.

15. Set the degrees to **10** and press the *Tab* key on the keyboard.

16. Rotate the part and notice that the feature outline indicates the taper is going inward.

17. [icon] Click the *Reverse Taper* button to reverse the taper so it is going outward. This button functions as a toggle.

18. [✔] Click *OK* button to apply the changes.

19. Tapered Open Profile 2 is listed in the Feature tree. This is the order the features will be processed when the NC code is generated.

Indicating the reversed arrow direction

Set the Taper Feature on End Condition tab

Inserted Vertical and Taper Open Profile Feature in Feature tree

Generate Operations Plan and Toolpaths

1. [icon] Click the *Generate Operation Plan* button on the CAMWorks Command Manager.

   OR

   Right click *CAMWorks NC Manager* or *EDM Part Setup1* in the Feature tree and select *Generate Operation Plan* on the context menu.

   The 2 Axis Contour1 and 2 Axis Contour2 operations are generated and listed in Operation tree.

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

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

3. On the EDM tab, set the following parameters:

Select Process as Die on the EDM tab

- Number of skims = **2**
- Glue Stop = **No Glue Stop**
- Process as = **Die**

4. Click on the *Leadin/Out* tab, set the Leadin to *Parallel*.

5. In the Leadout group box, make sure the *Same as Leadin* is checked.

   When the *Same as Leadin* is checked, CAMWorks set the parameters of Leadout as same as Leadin. When *Same as Leadin* is not checked, you can set the different required parameters for Leadin and Leadout.

6. Click *OK* to apply the changes and close the dialog box.

7. Double click *2 Axis Contour2* in the Operation tree.

8. Follow the steps from **3 to 5** and make the same changes as for 2 Axis Contour1.

9. Click *OK* to apply the changes and close the dialog box.

10. Click the *Generate Toolpath* button on the CAMWorks Command Manager.

    OR

    Right click the *EDM Part Setup1* or *CAMWorks NC Manager* items in the Operation tree and select *Generate Toolpath* on the context menu.

Simulation and Post process Toolpath

1. Click the *Simulate Toolpath* button of the CAMWorks Command Manager.

   OR

   Right click *EDM Part Setup1* in the tree and select *Simulate Toolpath* on the context menu.

2. Set the display parameters:

   - Stock: *Translucent Display*

   - Tool Holder: *Translucent Display*

   - Target Part: *Translucent Display*

   - Tool: *Shaded Display*

3. Click the *Run* button.

4. Click the button in the upper right corner to cancel the simulation and return to the SOLIDWORKS/CAMWorks Solids display.

5. Click the *Post Process* button on the CAMWorks Command Manager.

   OR

   Right click *CAMWorks NC Manager* in the Operation tree and click *Post Process* on the context menu.

6. In the Post Output File dialog box, type a name for the file and click *Save*.

---

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 if you enabled this option in the previous tutorial.

9. Notice that G-code has been generated for the Vertical and Tapered feature.

10. ✖ Click the button to close the editor.

2 Axis Contour 4

Topics covered in this tutorial:

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 Context-based Help.

Open the Part and Defining the Machine and Stock

1. Open the part file **EDM2AX_5.SLDPRT** in the following folder

   Drive:\CAMWorksData\CAMWorks202x\Examples\Tutorial_Parts\EDM

2. Click the *CAMWorks Feature Tree* tab.

3. Double click the *Machine* item in the CAMWorks Feature tree.

   The Machine dialog box is displayed.

4. In the Available machines list, highlight *Wire EDM–Metric*, click the *Select* button.

5. Click *OK* to close the Machine dialog box.

6. Right click *Machine [Wire EDM–Metric]* in the tree and click *Select Machine* on the context menu.

EDM2AX_5.SLDPRT
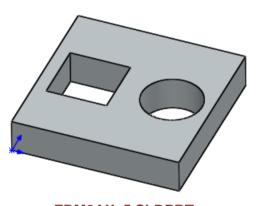
7. In the Machine Selection dialog box, make sure *SODICK* is selected for the Make and *AG600L* for the Model from the dropdown list.

8. Click *OK* to close the dialog box.

9. Right click *Machine [Wire EDM–Metric]* in the tree and select *Edit Settings* on the context menu.

   The EDM setting dialog box is displayed.

10. On the Posting tab, make sure the Process order for Die is *By Complete Feature*.

11. Click *OK* to close the dialog box.

    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.

12. Double click *Stock Manager* in the Feature tree.

13. In the Stock Manager dialog box, click *OK* to use the default Bounding box and Material.

Extract Machinable Features

1. ⚒ Click the *Options* button on the CAMWorks Command Manager.
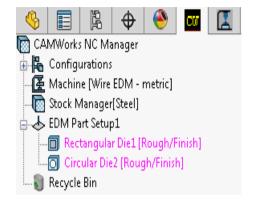
2. On the EDM Features tab, remove the check mark from *Perimeter punch* and click *OK*.

3. 📄 Click the *Extract Machinable Features* button on the CAMWorks Command Manager.

    OR

    Right click *CAMWorks NC Manager* in the tree and select *Extract Machinable Features* on the context menu.

    Rectangular Die1 and Circular Die2 are listed in the tree.

Features recognized by AFR

Generate Operation Plan and Toolpath

1. 📄 Click the *Generate Operation Plan* button on the CAMWorks Command Manager.

    OR

    Right click *CAMWorks NC Manager* or *EDM Part Setup1* in the Feature tree and select *Generate Operation Plan* on the context menu.
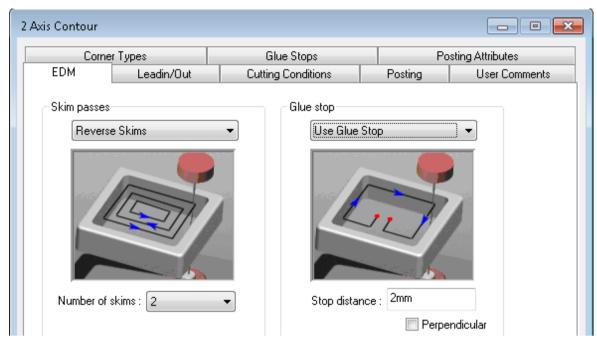
    The 2 Axis Contour1 and 2 Axis Contour2 operations are generated and listed in the Operation tree.

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

    The 2 Axis Contour dialog box is displayed.

3. On the EDM tab, set the following parameters:

    - Number of skims = **2**

    - Glue Stop = **Use Glue Stop**

    - Stop distance = **2mm**

EDM Tab on 2 Axis Contour dialog box

4. Click on the *Leadin/Out* tab, set the Leadin to *Perpendicular* with Length of **6mm**.

5. In the Leadout group box, make sure the *Same as Leadin* is checked.

   When the *Same as Leadin* is checked, CAMWorks set the parameters of Leadout as same as Leadin. When *Same as Leadin* is not checked, you can set the different required parameters for Leadin and Leadout.

6. Click *OK* to apply the changes and close the dialog box.

7. Double click *2 Axis Contour2* in the tree.

   The 2 Axis Contour dialog box is displayed.

8. Follow the steps from **3** to **5** and make the same changes as on the 2 Axis Contour1.

9. Click *OK* to apply the changes and close the dialog box.

10. Click the *Generate Toolpath* button on the CAMWorks Command Manager.

    OR

    Right click the *EDM Part Setup1* or *CAMWorks NC Manager* items in the Operation tree and select *Generate Toolpath* on the context menu.

Post Process the Die Features

1. Click the *Options* button on the CAMWorks Command Manager.

   The Options dialog box is displayed.

2. Click the *File Locations* tab.

The File Output tab contains an option to automatically open the G-code file when you close the Post Process Output dialog box.

3. In the Post processing group box, check the *Open G-code file in editor* option.

4. **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\WINNT folder, click *Open*, and then click *OK*.

Browse the location of editor

5. [G1] Click the *Post Process* button on the CAMWorks Command Manager.

   OR

   Right click *CAMWorks NC Manager* in the Operation tree and click *Post Process* on the context menu.

6. In the Post Output File dialog box, type a name for the file and click *Save*.

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. [✕] Click the button to close the editor.

11. [icon] Right click *Machine [Wire EDM–Metric]* in the Feature tree and click *Edit Settings* on the context menu.

12. On the Posting tab, set the Process order for Die to *Rough / Glue Stop / All Skims*, then click *OK*.

13. Follow the same steps from 5 to 8 and assigned the desired file name.

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. [✕] Click the button to close the editor.

16. ⬛ Again, right click *Machine [Wire EDM–Metric]* in the tree and click *Edit Settings* on the context menu.

17. On the Posting tab, set the Process order for Die to *Rough / Glue Stop / Skim Passes*, then click *OK*.

18. Follow the same steps from **5** to **8** and assigned the desired file name.

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.

3. LEARNING 4 AXIS CONTOUR CUTTING

This chapter provides an opportunity to learn 4 Axis Contour cutting through step-by-step exercises.

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

Before you begin with the tutorials given in this chapter, ensure that you are through with the concepts explained in Chapter 1.

The tutorial parts are installed when you install CAMWorks and are in the *Examples\Tutorial_Parts\EDM* folder on your computer. This folder is inside the CAMWorks data folder.

(Drive:*CAMWorksData\CAMWorks202x\Examples\Tutorial_Parts\EDM*).

IMPORTANT! **CAMWorks uses a set of knowledge-based rules to assign machining operations to features. The Technology Database contains the data for the machining process plans and can be customized for your facility's machining methodology. When you do these exercises, your results may not be the same as described in the steps and illustrated in the figures. This is because the machining sequences and operations data in your Technology Database may be different from the database used to produce the documentation.**

---

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 New 4 Axis EDM Feature command. The New 4 Axis EDM Feature command allows you to define 4 axis features. These include Pocket (Die), Boss (Punch) and Open Profile features.
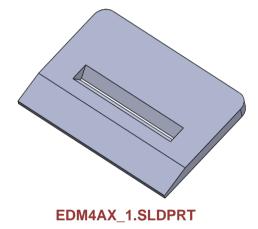
In this exercise, you insert a 4 Axis Die feature.

Open the Part and Defining the Machine and Stock

1. Open the part file **EDM4AX_1.SLDPRT** in the following folder.

   Drive:\CAMWorksData\CAMWorks202x\Examples\Tutorial_Parts\EDM

2. Click the *CAMWorks Feature Tree* tab.

3. Double click the *Machine* item in the CAMWorks Feature tree.

   The Machine dialog box is displayed.

4. In the Available machines list, highlight *Wire EDM–Inch*, click the *Select* button.

5. Click *OK* to close the dialog box.

6. Right click *Machine [Wire EDM–Inch]* in the tree and click *Select Machine* on the context menu.

7. In the Machine Selection dialog box, select *SODICK* for the Make and *AG600L* for the Model from the dropdown menu.

EDM4AX_1.SLDPRT

8. Click *OK* to close the dialog box.

9. Right click *Machine [Wire EDM–Inch]* in the tree and select *Edit Settings* on the context menu.

10. On the EDM Setup tab, make sure the Wire diameter is set to **0.01in**.

11. 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:

    - Cut rough pass, Glue stop cut off, and All skim passes

12. Double click *Stock Manager* in the Feature tree to open Stock Manager dialog box.

13. For this exercise, click *OK* to use the default Bounding box and Material.

Extract Machinable Features and Interactively Inserting Part Setup

1. Click the *Extract Machinable Features* button on the CAMWorks Command Manager.

    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.

2. Right click *Stock Manager* in the Feature tree and select *New Part Setup* on the context 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 tutorial, AFR is not used since there are no 2 Axis EDM features on the part. You need to insert a Part Setup interactively.

3. Pick the top face of the part (*highlighted in blue color*).

4. Click *OK* to insert the Part Setup.

    EDM Part Setup1 is listed in the Feature tree.

Pick the top face
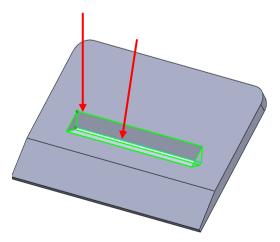
Defining a 4 Axis Die Feature Interactively

1. Right click *EDM Part Setup1* and select *New 4 Axis EDM Feature* on the context menu.

    The New 4 Axis EDM Feature dialog box is displayed.

2. Make sure *Die (Pocket)* is the selected Feature type.

3. Click on *Curve-1* field to set the focus.

4. Pick the top edge of the rectangular pocket as shown in the image.

5. Click on *Curve-2* field to set the focus.

6. Pick the lower edge of the rectangular as shown in image.

    The start point on each curve is identified.

7. If the start points are not as shown in the image, 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.

8. Click *OK* to insert and close the dialog box.

    4-Axis Die1 is listed in the tree.

Pick the top and lower edges

Generating a 4 Axis Contour Operation and Toolpaths

1. Click the *Generate Operation Plan* button on the CAMWorks Command Manager.

   OR

   Right click *CAMWorks NC Manager* or *EDM Part Setup1* in the Feature tree and select *Generate Operation Plan* on the context menu.

2. Double click *4 Axis Contour1* in the Operation tree.

3. Click on the *Leadin/Out* tab, set the Leadin to *Perpendicular* with Length of **0.02in**.

4. In the Leadout group box, make sure the *Same as Leadin* is checked.

   When the *Same as Leadin* is checked, CAMWorks set the parameters of Leadout as same as Leadin. When *Same as Leadin* is not checked, you can set the different required parameters for Leadin and Leadout.

5. Click *OK* to apply the changes and close the dialog box.

6. Click the *Generate Toolpath* button on the CAMWorks Command Manager.

   OR

   Right click the *EDM Part Setup1* or *CAMWorks NC Manager* items in the Operation tree and select *Generate Toolpath* on the context menu.

7. Click the operation in the tree to view the toolpath.

Simulation and Post Process Toolpath

1. Click the *Simulate Toolpath* button on the CAMWorks Command Manager.

   OR

   Right click *CAMWorks NC Manager* or *EDM Part Setup1* in the tree and select *Simulate Toolpath* on the context menu.

2. Set the display parameters:

   - Stock: *Shaded Display*

   - Tool: *Shaded Display*
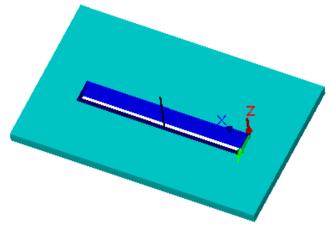
   - Tool Holder: *No Display*

   - Target Part: *No Display*

3. Click the *Run* button.

4. Click the *Select Chips and Press Ctrl+D Key* button. This button is enabled when the simulation is complete.

5. Pick the loose stock part to remove, and then press the *Ctrl+D key* on the keyboard.

   The loose chips are removed in the graphics area.

6. [icon] Click the *Select Chips and Press Ctrl+D Key* button to exit the function.\

Simulation Toolpath

7. [icon] Click the *Close* button in the upper right corner to close the simulation and return to the SOLIDWORKS/CAMWorks Solids display.

8. [icon] Click the *Post Process* button on the CAMWorks Command Manager.

   OR

   Right click *CAMWorks NC Manager* in the Operation tree and click *Post Process* on the context menu.

9. In the Post Output File dialog box, type the desired file name and then click *Save*.

10. [icon] Click the *Run* button in the Post Process Output dialog box.

11. When the post processing is finished, use the vertical scroll bar to view the code.

12. Click *OK* ✔ to close the dialog box.

4 Axis Contour 2

Topics covered ion this tutorial:

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/CAMWorks Solids. If the number of entities in Curve 1 and Curve 2 is equal, then the sync curves are ignored.
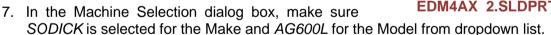
In the part below, the number of entities in Curve 1 is not equal to the number of entities in Curve 2 of the 4 Axis feature. In this tutorial, you insert the feature and automatically extract sync curves.

Open the Part and Defining the Machine and Stock

1. Open the part file **EDM4AX_2.SLDPRT** in the following folder.

    Drive:\CAMWorksData\CAMWorks202x\Examples\Tutorial_Parts\EDM

2. Click the *CAMWorks Feature Tree* tab.

3. Double click the *Machine* item in the CAMWorks Feature tree.

    The Machine dialog box is displayed.

4. In the Available machines list, highlight *Wire EDM–Inch*, click the *Select* button.

5. Click *OK* to close the dialog box.

6. Right click *Machine [Wire EDM–Inch]* in the tree and click *Select Machine* on the context menu.

7. In the Machine Selection dialog box, make sure *SODICK* is selected for the Make and *AG600L* for the Model from dropdown list.

EDM4AX 2.SLDPRT

8. Click *OK* to close the dialog box.

9. Right click *Machine [Wire EDM–Inch]* in the tree and select *Edit Settings* on the context menu.

10. On the Posting tab, make sure the Process order for Punch is set to *By Complete Feature.*

11. Click *OK* to close the dialog box.

    The processing order is rough cut leaving glue stop, 0 - 6 skim passes, then glue stop cut off.

12. Double click *Stock Manager* in the Feature tree.

13. In the Stock Manager dialog box, click *OK* to use the default Bounding box and Material.

Extract Machinable Features and Interactively Inserting Part Setup

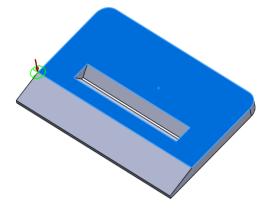1. Click the *Extract Machinable Features* button on the CAMWorks Command Manager.

    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.

2. Right click *Stock Manager* in the Feature tree and select *New Part Setup* on the context menu.

3. Pick the top face on the part *(highlighted in blue color)*.

4. Click *OK* to insert the changes.

    EDM Part Setup1 is listed in the Feature tree.

Pick the top face

Inserting and Automatically Extracting Sync Curves to Define a 4 Axis EDM Feature

1. Right click *EDM Part Setup1* in the tree and select *New 4 Axis EDM Feature* on the context menu.

    The New 4 Axis EDM Feature dialog box is displayed

2. Click the *Punch* button for the Feature type.

3. Click in the *Curve-1* field to set the focus, pick the top face of the part.

    Loop 1 is selected in the Curve-1 area.

4. Click in the *Curve-2* field to set the focus.

5. Rotate the part and pick the bottom face of the part.

6. Alternatively, you can pick any edge of the bottom face.
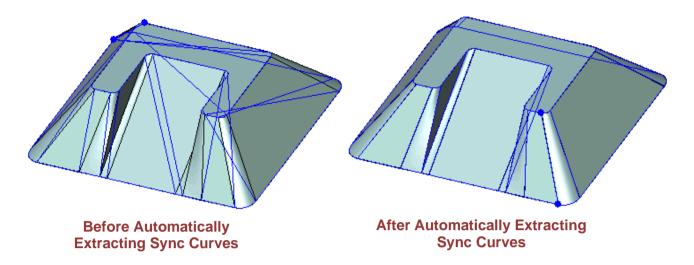
    Loop 2 is selected in the Curve-2 area.

7. 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 or both. In such a case, the 'Automatically extract sync curves' button is disabled.

8. Click *OK* to insert the changes.

Before Automatically Extracting Sync Curves

After Automatically Extracting Sync Curves

Generating the 4 Axis Contour Operation and Toolpaths

1. Click the *Generate Operation Plan* button on the CAMWorks Command Manager.

    OR

    Right click *CAMWorks NC Manager* or *EDM Part Setup1* in the Feature tree and select *Generate Operation Plan* on the context menu.

2. Double click *4 Axis Contour1* in the Operation tree.

    The 4 Axis Contour dialog box is displayed.

3. On the EDM tab, make sure Glue Stop is set to **No Glue Stop.**

4. Click *OK* to apply the changes.

    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.

5. Click the Generate Toolpath button on the CAMWorks Command Manager.

    OR

    Right click the *EDM Part Setup1* or *CAMWorks NC Manager* items in the Operation tree and select *Generate Toolpath* on the context menu.

6. Click the operation in the tree to view the toolpath.

7. Click (⊞) plus sign to expand next to *4 Axis Contour1* in the tree, then again click (⊞) plus sign to expand *4-Axis Punch1*.

8. Notice that there is no Tab cut.

4 Axis Contour 3
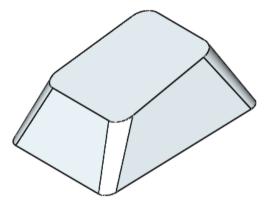
Topics covered in this tutorial:

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.

Open the Part and Defining the Machine and Stock

1. Open the part file **EDM4AX_3.SLDPRT** in the following folder.

   Drive:\CAMWorksData\CAMWorks202x\Examples\Tutorial_Parts\EDM

2. Click the *CAMWorks Feature Tree* tab.

3. Double click the *Machine* item in the CAMWorks Feature tree.
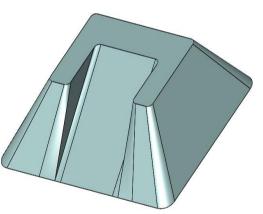
4. In the Available machines list, highlight *Wire EDM–Inch*, click the Select button.

5. Click *OK* to close the dialog box.

6. Right click *Machine [Wire EDM–Inch]* in the tree and click *Select Machine* on the context menu.

7. In the Machine Selection dialog box, make sure *SODICK* is selected for the Make and *AG600L* for the Model from dropdown list.

EDM4AX_3.SLDPRT

8. Click *OK* to apply the changes.

9. Right click *Machine [Wire EDM–Inch]* in the tree and select *Edit Settings* on the context menu.

   The Edit Settings dialog box is displayed.
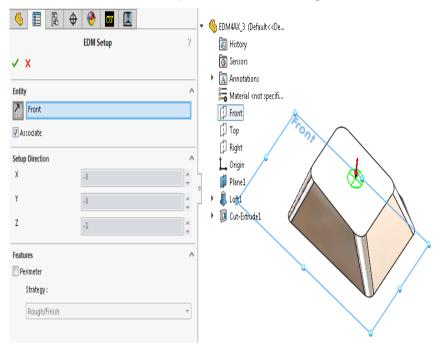
10. On the Posting tab, set the Process order for Punch to *By Complete Feature*.

11. Click *OK* to close the dialog box.

12. Double click *Stock Manager* in the Feature tree.

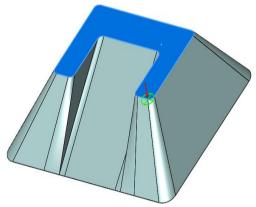13. In the Stock Manager dialog box, click *OK* to use the default Bounding box and Material.

Extract Machinable Features and Using Sketches to Define a 4 Axis EDM Feature

1. Click the *Extract Machinable Features* button on the CAMWorks Command Manager.

   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.

2. Right click *Stock Manager* in the Feature tree and select *Part Setup* on the context menu.

3. In the graphics area, double click on *EDM4AX_3* to expand and highlight the *Front* reference plane.

4. [icon] Click the *Reverse selected entity* button to reverse the machining direction.

5. [icon] Click *OK* to insert the Part Setup and close the dialog box.
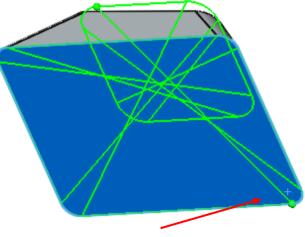
Inserting EDM Part Setup

Interactively Inserting the 4 Axis EDM Feature

1. Right click *EDM Part Setup1* in the tree and select *New 4 Axis EDM Feature* on the context menu.

2. [icon] Click the *Punch* button for the Feature type.

3. Pick *Sketch2* from the Available Sketches for Curve-1.

   This action will list the *Sketch2* in the Curve-1.

4. Rotate the part so that the bottom face is visible.

5. Click in the Curve-2 to set the focus, pick the bottom face anywhere on the right side of the part as shown in the image.

6. Rotate the part to the top direction again and notice the top and bottom start points are not lined up.

Pick the Right side of the part
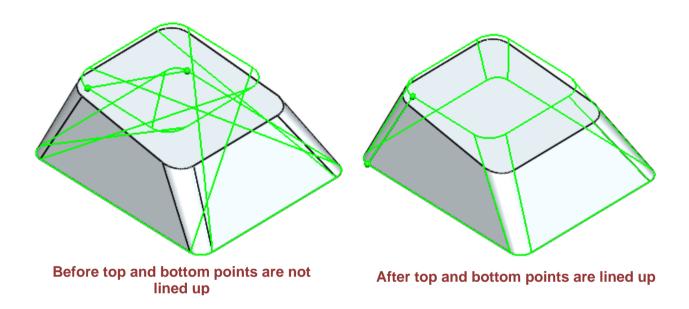
7. Click in the *Curve-1* to set the focus.

8. <u> > </u> Click the *Move Start Point Location* button under Curve-1 until the points line up as shown in the image below.

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

9. Click *OK* to insert the changes.

Before top and bottom points are not lined up

After top and bottom points are lined up

Generate Operation Plan and Toolpath

1. Click the *Generate Operation Plan* button on the CAMWorks Command Manager.

   OR

   Right click *CAMWorks NC Manager* or *EDM Part Setup1* in the Feature tree and select *Generate Operation Plan* on the context menu.

2. Click the *Generate Toolpath* button on the CAMWorks Command Manager.

   OR

   Right click the *EDM Part Setup1* or *CAMWorks NC Manager* items in the Operation tree and select *Generate Toolpath* on the context menu.

Changing the Leadin Point

1. Click the (⊞) sign next to 4 Axis Contour1 in the Operation tree to expand the item.

2. Right click *4-Axis Punch1* under the operation and select *Edit Leadin Point* on the context menu.

   The Lead in Location dialog box is displayed and a round marker on the feature identifies the current leadin point.
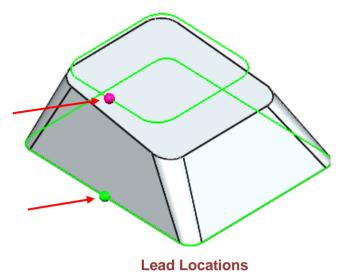
3. Rotate the part to adjust the lead locations as shown in the image.

4. Use the current settings in the Lead in Location dialog box and pick the top segment shown in the image.

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

5. Change the Select profile option to *Bottom* in the Select profile group box and pick the bottom segment shown in the image.

6. Click *OK* to apply changes and close the dialog box.

   The toolpath is automatically updated.
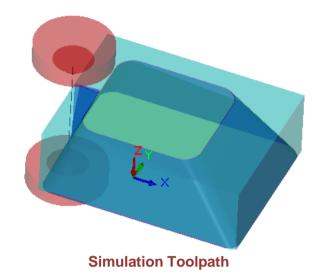
Lead Locations

Toolpath Simulation

1. Click the *Simulate Toolpath* button of the CAMWorks Command Manager.

   OR

   Right click *EDM Part Setup1* in the tree and select *Simulate Toolpath* on the context menu.

2. Set the display parameters:

   - Stock: *Translucent Display*

   - Tool Holder: *Translucent Display*

   - Target Part: *Translucent Display*

   - Tool : *Shaded Display*

3. Click the *Run* button.

4.   Click the button in the upper right corner to cancel the simulation and return to the SOLIDWORKS/CAMWorks Solids display.

Simulation Toolpath

4 Axis Contour 4
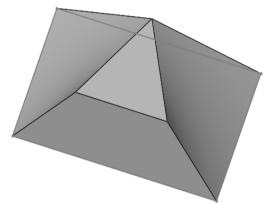
Topics covered in this tutorial:

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.

Open the Part and Defining the Machine and Stock

1. Open the part file **EDM4AX_4.SLDPRT** in the following folder.

   Drive:\CAMWorksData\CAMWorks202x\Examples\Tutorial_Parts\EDM

2. Click the *CAMWorks Feature Tree* tab.

3. Double click the *Machine* item in the CAMWorks Feature tree.

4. In the Available machines list, highlight *Wire EDM–Inch*, click the Select button.

5. Click *OK* to close the dialog box.

6. Right click *Machine [Wire EDM–Inch]* in the tree and click *Select Machine* on the context menu.

7. In the Machine Selection dialog box, make sure *SODICK* is selected for the Make and *AG600L* for the Model from dropdown list.

EDM4AX_4.SLDPRT

8. Click *OK* to close the dialog box.

9. Right click *Machine [Wire EDM–Inch]* in the tree and select *Edit Settings* on the context menu.

   The EDM Settings dialog box is displayed.

10. On the Posting tab, set the Process order for Punch to *By Complete Feature*.

11. Click *OK* to close the dialog box.

12. Double click *Stock Manager* in the Feature tree.

13. In the Stock Manager dialog box, click *OK* to use the default Bounding box and Material.

Extract Machinable Features and Inserting Part Setup
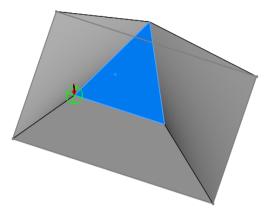
1. Click the *Extract Machinable Features* button on the CAMWorks Command Manager.

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.

2. Right click *Stock Manager* in the Feature tree and select *New Part Setup* on the context menu.

3. Pick the top face on the part.

4. ✔ Click *OK* to insert the changes.

5. EDM Part Setup 1 is listed in the feature tree.

Pick the top face

Interactively Inserting and Picking Sync Curves to Define a 4 Axis EDM Feature
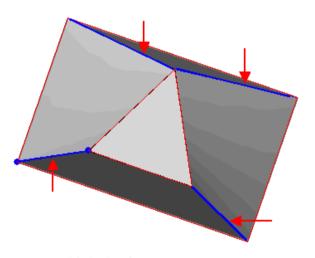
1. Right click *EDM Part Setup1* in the Feature tree and select *New 4 Axis EDM Feature* on the context menu.

2. 🟨 Click the *Punch* button for the Feature type.

3. Pick the *Sketch2* from the Available Sketches for Curve-1.

   This action will list Sketch2 in the Curve-1.

4. Pick *Sketch1* from the Available Sketches for Curve-2.

   This action will list Sketch1 in the Curve-2.

5. 📐 The Automatically extract sync curve button is disabled since a sketch was used for Curve-2. Click in the *Sync Curves* list box to set the focus.

6. Pick the four segments as shown in the image 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 *OK* to insert the changes.

Pick the four segments

📝 **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 Operation Plan and Toolpath

1. Click the *Generate Operation Plan* button on the CAMWorks Command Manager.

   OR

   Right click *CAMWorks NC Manager* or *EDM Part Setup1* in the Feature tree and select *Generate Operation Plan* on the context menu.

2. Click the *Generate Toolpath* button on the CAMWorks Command Manager.

   OR

   Right click the *EDM Part Setup1* or *CAMWorks NC Manager* items in the Operation tree and select *Generate Toolpath* on the context menu.

Simulate Toolpath

1. Click on the *Simulate Toolpath* button of the Command Manager.

   OR

   Right click *EDM Part Setup1* in the tree and select *Simulate Toolpath* command from the context menu.

2. Set the displays:

   - Stock : *Translucent Display*

   - Tool Holder: *Translucent Display*

   - Target Part: *Translucent Display*

   - Tool: *Shaded Display*

3. Click the *Run* button.

4. Click the button in the upper right corner to cancel the simulation and return to the SOLIDWORKS/CAMWorks Solids display.


APPENDIX A: EDM MACHINE SETUP UTILITY AND EDM POST FILES

EDM Machine Setup Utility
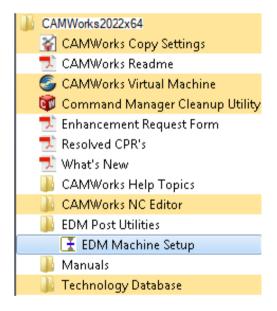
The *EDM Machine Setup* application is a separate software utility provided with the CAMWorks application. It is to be used for adding/editing EDM Machine Makes and Models. Use this *EDM Machine Setup* utility to edit the EDM Machine Makes and Models shipped along with CAMWorks as well as add custom EDM Machine Makes and Models along with corresponding post information.

Opening the EDM Machine Setup Utility
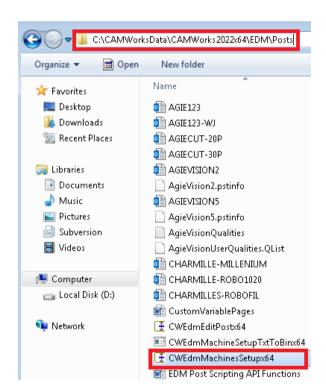
To open the *EDM Machine Setup* utility, perform any one of the following steps:

- Browse to the *EDM**Posts* folder within the CAMWorks data folder and double-click **CWEdmMachinesSetupx64.exe** file. The folder path to this file would be:

  Drive:\CAMWorksData\CAMWorks2022x64\EDM\Posts

  OR

- Alternatively, you can click on the **Windows Start menu** and select

  All Programs>>CAMWorks2022x64>>EDM Post Utilities>>EDM Machine Setup.

The above action will display the *EDM Machine Setup* dialog box.

Accessing the EDM Machine Setup Utility from the Windows Start menu

Location of CWEDMMachineSetupx64.exe

Reassigning Post Processors to existing EDM Machines

Refer: Appendix B: Reassigning Post Processors to existing EDM Machines Makes/Models.

Adding EDM Machine Makes and Models using EDM Machine Setup Utility

Refer: Appendix C: Adding EDM Machines and Models.

EDM Post Files & their folder locations

CAMWorks comes pre-installed with a set of EDM Machine Makes and Models and corresponding post-specific information. The following files are used for EDM post processing in CAMWorks:

1. Wire EDM post file (*.pst)

This is a unique post file for each EDM Machine Make/Model (example: Agie123_A.PST). Two versions of each EDM Post file is available viz. SQLite based and MS Access based.

Folder location of Post Files shipped with CAMWorks:

- Pre-installed SQLite-based Wire EDM Post files are located at:

  Drive:\CAMWorksData\CAMWorks2022x64\EDM\Posts

- Pre-installed MS Access based Wire EDM Post files are located at:

  Drive:\CAMWorksData\CAMWorks2022x64\EDM\Legacy-MSAccessDatabases\Posts

2. MachineSetup.bin

The *MachineSetup.bin* is a binary file that contains the list of EDM Machine Makes and Models and post-specific information used by CAMWorks.

Folder Location of the MachineSetup.bin file

When CAMWorks is installed, this file is installed in the *EDM\Posts* folder within the CAMWorks Data folder.

- For SQLite based EDM Post files, the folder location of this file will be:

  Drive:\CAMWorksData\CAMWorks2022x64\EDM\Posts

- For MS Access based EDM Post files, the folder location of this file will be:

  Drive:\CAMWorksData\CAMWorks2022x64\EDM\Legacy-MSAccessDatabases\Posts

Editing the MachineSetup.bin file

CAMWorks comes pre-installed with a set of EDM Machine Makes/Models and post-specific information. The *MachineSetup.bin* is a binary file that stores Mapping information on EDM Machine Makes, Models and corresponding post information used in CAMWorks. This *MachineSetup.bin* file cannot be edited directly. Whenever you add/modify any Machine Makes and Models or post-specific information using the EDM Machine Setup utility, then the *MachineSetup.bin* file will be automatically updated.

3. Post information files

These files are dependent on the EDM Machine Make and Model. These files are located in the same location as the *MachineSetup.bin file*.

4. MachineSetup.txt

The MACHINESETUP.TXT file is used by the EDM Machine Setup Utility to generate the MachineSetup.bin file. This file is located in the same location as the *MachineSetup.bin file*.

5. Wire EDM Cutting conditions file (*.brcc, *.ecc, *.sbcc or *.sdcc)

The *Cutting Conditions Database* is an optional file that is associated with certain Machine Makes and Models that support this database format. The Wire EDM Cuttings Conditions files refer to such CTC Databases. (Example: *Agie123_A.ecc)*.

Folder location of Wire EDM Cutting Conditions files shipped with CAMWorks:

- Pre-installed SQLite-based Wire EDM Cutting Conditions files are located at:

  Drive:\CAMWorksData\CAMWorks2022x64\EDM\EDMCuttingConditions

- Pre-installed MS Access based Wire EDM Cutting Conditions files are located at:

  Drive:\CAMWorksData\CAMWorks2022x64\EDM\Legacy-MSAccessDatabases\ EDMCuttingConditions

APPENDIX B: REASSIGNING POST PROCESSORS TO EXISTING EDM MACHINE MAKES/MODELS

To assign/reassign the EDM Post processor and/or CTC Database associated with an existing EDM Machine Make and/or Model, following are the steps:

Step 1: Copy the customized Wire EDM Post Processor Files

Copy the customized Wire EDM Post Processor files (*.pst) and any additional post information files that you wish to assign to the specific EDM machine in the applicable *Posts* folder within the CAMWorks Data folder.

- For SQLite based customized Wire EDM Post Processor files (*.pst), the folder location where the files are to be copied is:

  Drive:\CAMWorksData\CAMWorks2022x64\EDM\Posts

- For MS Access based customized Wire EDM Post Processor files (*.pst), the folder location where the files are to be copied is:

  Drive:\CAMWorksData\CAMWorks2022x64\EDM\Legacy-MSAccessDatabases\Posts

Step 2: (Optional) Copy the customized Wire EDM Cutting Conditions Files (CTC Database)

If the EDM Machine Make/Model you intend to add supports a Cutting Conditions Database, then copy the custom Wire EDM Cutting conditions file (*.brcc, *.ecc, *.sbcc, *.sdccc, etc.) to the applicable Cutting conditions folder.

- For SQLite based customized Wire EDM Cutting Conditions file, the folder location where the file is to be copied is:

  Drive:\CAMWorksData\CAMWorks2022x64\EDM\EDMCuttingConditions

- For MS Access based customized Wire EDM Cutting Conditions file, the folder location where the files are to be copied is:

  Drive:\CAMWorksData\CAMWorks2022x64\EDM\Legacy-MSAccessDatabases\ EDMCuttingConditions

Step 3: Exit the SOLIDWORKS/CAMWorks Solids application, if open

If SOLIDWORKS or CAMWorks Solids application is open, then exit the application.

It is vital that SOLIDWORKS/CAMWorks Solids not be open when you add a new EDM Machine Make/Model. The reason is that any changes/additions made to the EDM Machines are saved to the MachineSetup.bin file. If the CAMWorks Add-In is open with the SOLIDWORKS/CAMWorks Solids application, then the MachineSetup.bin will be overwritten when you later close the application, thus undoing the EDM machine addition you made.

Step 4: Open the EDM Machine Setup utility

To open the *EDM Machine Setup* utility, perform any one of the following steps:

- Browse to the \EDM\Posts folder within the CAMWorks data folder and double-click ***CWEdmMachinesSetupx64.exe*** file. The folder path to this file would be:

  Drive:\CAMWorksData\CAMWorks2022x64\EDM\Posts

  OR

- Alternatively, you can click on the ***Windows Start menu*** and select

All Programs>>CAMWorks2022x64>>EDM Post Utilities>>EDM Machine Setup.

The above action will display the *EDM Machine Setup* dialog box.

Step 5: Assign customized Wire EDM Processor to desired Wire EDM Machine using EDM Machine Setup Utility

To assign a customized Wire EDM post processor file to an existing Machine Make and Model, following are the steps:

i. Click on the *Machines* tab in the *EDM Machine Setup* dialog box.

ii. Within the *Machines* group box, select the Machine Make in the *Make* dropdown list.

iii. Select the Machine Model from the *Model* dropdown list.

iv. Switch to the *Settings* tab.

v. In the *Post* group box, from the *Type* dropdown list, select the type of code you need to output (*G code* or *Command/ISO*) and then click on the *Select* button. Use the displayed *Open* dialog box to browse and select the customized Post processor that is to be assigned to the machine. (The Post Processor copied to the *Posts* folder in Step 1.)

vi. In the *Parameters* group box, adjust the settings to correspond to the post processor. Place a check in the applicable checkboxes.

Step 6: (Optional) Assign the customized CTC Database using EDM Machine Setup Utility

To assign a customized Cutting Conditions (CTC) database to an existing Machine Make and Model, following are the steps:

i. Click on the *Machines* tab in the *EDM Machine Setup* dialog box.

ii. In the *Cutting conditions database* group box, ensure that a check is placed in the *Cutting Cond Allowed* and *Database Allowed* checkboxes.

iii. Click the *Select* button. Use the displayed *Open* dialog box to browse to the folder containing the applicable Cutting Conditions (CTC) database.

iv. Select the desired database file and click the *Open* button.

Step 7: Save the Changes and exit the EDM Machine Setup utility

i. Click the *Save* button. This action updates the information associated with the edited EDM Machine Make and Model in the MachineSetup.bin file.

ii. Click the *Close* button to exit the *EDM Machine Setup* utility.

The customized Wire EDM Post Processor files (*.pst) will now be automatically assigned to the specific Machine Make and Model. If a customized CTC database has been assigned, then it will now be assigned as the default CTC database for all parts machined using the specific Machine Make and model.

Verifying the successful reassigning of EDM Post Processor/CTC Database

Once the Post processor and/or CTC Database assigned to an EDM Machine Make/Model has been reassigned, you can verify whether the Machine Makes/Models have been successfully listed and whether the paths of associated post files are correct.

1. Launch *CAMWorks* as an Add-In within *SOLIDWORKS/CAMWorks Solids*.

2. Double click the *Machine* item in the CAMWorks Feature tree.

The *Machine* dialog box will be displayed.

3. In the *Machine* tab of this dialog box, select the *Wire EDM* machine and click *OK*.

4. Right-click the *Wire EDM* Machine node in the CAMWorks Feature tree and click *Select Machine* command from the context menu.

5. The **Machine Selection** dialog box will be displayed. Select the EDM Machine Make and model for which the Post Processor/ CTC Database has been reassigned and click *OK*.

6. Right-click the *Wire EDM* Machine in the CAMWorks Feature tree and select *Edit Settings* from the context menu.

   The **EDM Settings** dialog box will be displayed.

7. On the **Machine Setup** tab, click on the **CTC Database** button. This action displays the **File Open** dialog box. Verify within this dialog box that the correct *Cutting Conditions Database* has been selected and click the **Cancel** button to close this dialog box.

8. On the **Posting** tab of the **EDM Settings** dialog box, click on the **Post** button. This action displays the **File Open** dialog box. Verify within this dialog box that the correct Post processor file has been assigned and click the **Cancel** button to close this dialog box.

9. Click *OK* button in the **EDM Settings** dialog box after verifying the changes to close the dialog box.

APPENDIX C: ADDING EDM MACHINE MAKES & MODELS

CAMWorks comes shipped with a set of EDM Machine Makes and Models and associated post information. These EDM Machine Makes and Models are listed in the **Machine Selection dialog box** within the CAMWorks User interface. (This dialog box is displayed when you right-click on the Wire-EDM Machine node in the CAMWorks Feature tree and select the *Select Machine* command on its context menu.)

If the EDM Machine Make and/or model that you wish to use is not listed in the **Machine Selection dialog box**, you can add the machine and model as explained below.

To reassign a custom EDM Post processor and/or CTC Database to an existing Machine Make and/or Model, refer the topic: Appendix B: Assigning Post Processors to existing EDM Machine Makes and Models.

Steps to add a new Machine Make and/or Model

1. Copy the customized Wire EDM Post Processor files (*.pst) and any additional post information files that you wish to assign to the new EDM machine in to the applicable *Posts* folder within the CAMWorks Data folder.

   - For SQLite based customized Wire EDM Post Processor files (*.pst), the folder location where the files are to be copied is:

     Drive:\CAMWorksData\CAMWorks2022x64\EDM\Posts

   - For MS Access based customized Wire EDM Post Processor files (*.pst), the folder location where the files are to be copied is:

     Drive:\CAMWorksData\CAMWorks2022x64\EDM\Legacy-MSAccessDatabases\Posts

2. If the EDM Machine Make/Model you intend to add supports a Cutting Conditions Database, then copy the custom Wire EDM Cutting conditions file (*.brcc, *.ecc, *.sbcc, *.sdccc, etc.) to the applicable Cutting conditions folder.

   - For SQLite based customized Wire EDM Cutting conditions file, the folder location where the file is to be copied is:

     Drive:\CAMWorksData\CAMWorks2022x64\EDM\EDMCuttingConditions

   - For MS Access based customized Wire EDM Cutting conditions file, the folder location where the files are to be copied is:
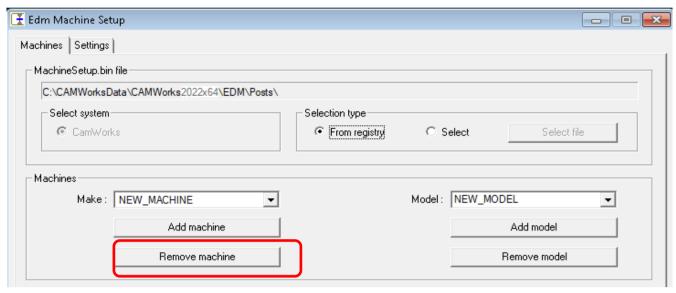
     Drive:\CAMWorksData\CAMWorks2022x64\EDM\Legacy-MSAccessDatabases\ EDMCuttingConditions

3. If SOLIDWORKS or CAMWorks Solids application is open, then exit the application.

   It is vital that SOLIDWORKS/CAMWorks Solids not be open when you add a new EDM Machine Make/Model. The reason is that any changes/additions made to the EDM Machines are saved to the MachineSetup.bin file. If the CAMWorks Add-In is open with the SOLIDWORKS/CAMWorks Solids application, then the MachineSetup.bin will be overwritten when you later close the application, thus undoing the EDM machine addition you made.

4. Open the **EDM Machine Setup** utility. The **EDM Machine Setup** dialog box will be displayed.

5. Click on the *Machines* tab in this dialog box. Under this tab, in the *Machines* group box, click the *Add Machine* button. In the *Make* and *Model* dropdown lists within the *Machines* group box, *NEW_MACHINE* and *NEW_MODEL* will be displayed respectively.
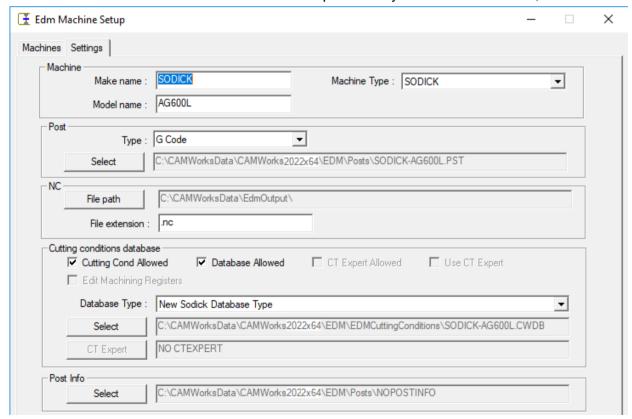
Click on the 'Add Machine' button in the Machines tab of EDM Machine Setup dialog box

6. Switch to the *Settings* tab.

   IMPORTANT: The settings on this tab are based on how the post processor was written and must match the information provided by *Geometric Americas, Inc*.

Settings tab of EDM Machine Setup dialog box

7. In the ***Machine*** group box:
   i. Edit the *Make name* and *Model name* fields to assign desired names to the Make and Model of the Machine to be added.
   ii. Use the *Machine Type* dropdown list to select the Machine type.

---

8. In the *Post* group box:
    i.   From the *Type* dropdown list, select the type of code you need to output (*G code* or *Command/Iso*).
    ii. Click the *Select* button. Use the displayed *Open* dialog box to browse and select the Post processor that is to be assigned to the machine.

9. In the *NC* group box:
    i.   Click on the *File path* button. Use the displayed **Browse for Folder** dialog box to set the path where the output file generated is to be saved.
    ii. In the **File extension** field, set the file extension. (For example: *.txt.)

10. If a Cutting Conditions (CTC) database is to be supported by the Machine Make/Model you intend to add, then in the *Cutting conditions database* group box:
    i.   Place a check in the **Cutting Cond Allowed** and **Database Allowed** checkboxes.
    ii. Click the *Select* button. Use the displayed **Open** dialog box to browse to the folder containing the applicable *Cutting Conditions (CTC) database*. Select the desired database file and click the *Open* button.

11. In the **Parameters** group box:
    i.   Adjust the settings to correspond to the post processor.
    ii. Place a check in the applicable checkboxes.

12. Click the **Save** button. This action saves the newly added Machine Make/Model to the MachineSetup.bin file.

13. Click the **Close** button to exit the *EDM Machine Setup* utility.

Verifying the successful addition of an EDM Machine Make/Model

Once an EDM Machine Make/Model has been added, you can verify whether the Machine Makes/Models have been successfully listed and whether the paths of associated post files are correct.

1. Launch *CAMWorks* as an Add-In within SOLIDWORKS/CAMWorks Solids.

2. Double click the *Machine* item in the CAMWorks Feature tree.

    The **Machine** dialog box will be displayed.

3. In the **Machine** tab of this dialog box, select the *Wire EDM* machine and click *OK*.

4. Right-click the *Wire EDM* Machine node in the CAMWorks Feature tree and click *Select Machine* command from the context menu.

5. The **Machine Selection** dialog box will be displayed. If you have added a new Machine Make/Model, then verify that the machine is listed in this dialog box. Select the newly added *Machine and Model* and click *OK*.

6. Right-click the *Wire EDM* Machine in the CAMWorks Feature tree and select *Edit Settings* from the context menu.

    The **EDM Settings** dialog box will be displayed.

7. On the **Machine Setup** tab, click on the **CTC Database** button. This action displays the **File Open** dialog box. Verify within this dialog box that the correct *Cutting Conditions Database* has been selected and click the *Cancel* button to close this dialog box.

8. On the *Posting* tab of the *EDM Settings* dialog box, click on the *Post* button. This action displays the *File Open* dialog box. Verify within this dialog box that the correct Post processor file has been assigned and click the *Cancel* button to close this dialog box.

9. Click *OK* button in the *EDM Settings* dialog box after verifying the changes to close the dialog box.

APPENDIX D: RESTORING CUSTOMIZED EDM FILES AFTER UPGRADING CAMWORKS

If you have customized EDM post files data (optionally inclusive of CTC Databases), then you need to copy those post files to their designated folder locations within the CAMWorks Installation folder in the following cases:

- When you upgrade CAMWorks with a fresh Service Pack (E.g.: From *CAMWorks-2020 SP0* to *CAMWorks-2020 SP1*).

- When you upgrade CAMWorks from a previous release to a new release (E.g.: *CAMWorks-2019* to *CAMWorks-2020*).

In both the above cases, unless you copy these customized EDM post files to the designated folder location, the customized components for EDM Post processing won't work.

Restoring customized EDM Post Files after upgrading an existing CAMWorks installation with a new Service Pack

A Service Pack updates specific files for an existing CAMWorks installation. When a Service Pack is installed, existing CAMWorks files will be overwritten. However, the Service Pack creates backups of customized files Technology Database, EDM Post Files, CTC Databases, etc. before upgrading the existing installation. These backups are created and saved within a backup folder located in the same folder where the customized files were originally present. To reuse the customized data after installing the service pack, you need to copy the customized EDM Post files from the backup folder and restore them to their original folder location.

Steps to retrieve and use backed up EDM Post Files in a CAMWorks Installation after upgrading with a Service Pack

When you upgrade your CAMWorks installation with a Service Pack, the CAMWorks Installation Wizard will automatically create backups of all customized data files. Following are the steps to retrieve and use backed up EDM Post Files after your existing CAMWorks installation has been upgraded with a Service Pack:

1. The SQLite based customized EDM Post files need to be copied from their Backup location and pasted back to their original location:

    Copy from Backup folder location:
    Drive:\CAMWorksData\CAMWorks202Xx64\EDM\Posts\backup\backup_xx
    Paste to Original folder location:
    Drive:\CAMWorksData\CAMWorks202Xx64\EDM\Posts

2. If you use MS Access based customized EDM Post files, then those files need to be copied from their Backup location and pasted back to their original location:

    Copy from Backup folder location:
    Drive:\CAMWorksData\CAMWorks202Xx64\EDM\Legacy-MSAccessDatabases\Posts\backup\backup_xx
    Paste to Original folder location:
    Drive:\CAMWorksData\CAMWorks202Xx64\EDM\Legacy-MSAccessDatabases\Posts

> *The "xx" of the **backup_xx** folder indicates a two digit numerical indicating the number of times the CAMWorks installation has been upgraded. For example, the folder **backup_02** indicates the second time the CAMWorks installation was upgraded with a Service Pack. In case of multiple backups, the backup files must always be retrieved from the latest backup folder. Such a folder is easily identifiable as the affixed number in its folder name is higher than those of other backup folders.*

3. While copying and pasting, in the case of MachineSetup.bin and MACHINESETUP.TXT, you have to overwrite the files installed by the Service pack.

4. Similarly, the customized SQLite based CTC Database files need to be copied from their Backup location and pasted back to their original location:

    Copy from Backup folder location:

    Drive:\CAMWorksData\CAMWorks2022x64\EDM\EDMCuttingConditions\backup\backup_xx

    Paste to Original folder location:

    Drive:\CAMWorksData\CAMWorks2022x64\EDM\EDMCuttingConditions

5. If you use customized MS Access based CTC Database files, then the customized files needs to be copied from their Backup location and pasted back to their original location:

    Copy from Backup folder location:

    Drive:\CAMWorksData\CAMWorks2022x64\EDM\Legacy-MSAccessDatabases\ EDMCuttingConditions\backup\backup_xx

    Paste to Original folder location:

    Drive:\CAMWorksData\CAMWorks2022x64\EDM\EDMCuttingConditions

6. Once all the files are copied, verify the successful listing of Machines/Models and their associated file paths. A successful listing indicates that EDM post files were successfully copied and activated and ready for use.

Restoring customized EDM Post Files after installing a new CAMWorks Release

Each annual CAMWorks release is installed as a separate installation without affecting the previous CAMWorks release already installed. To link the customized EDM Post Files with the newly installed CAMWorks release, you need to import them by copying them to the CAMWorks Data folder location within the new CAMWorks installation.

1. Barring the Cutting Conditions files (which is located in another folder), all these EDM Post files need to be copied from data folder location of previous CAMWorks release to the corresponding folder location in the new CAMWorks release.

    i.   For customized SQLite based EDM Post files, the files need to be copied from their original location at

        Drive:\CAMWorksData\CAMWorks2021x64\EDM\Posts

        to

        Drive:\CAMWorksData\CAMWorks2022x64\EDM\Posts

    ii. For customized MS Access based Wire EDM Post files, the files need to be copied from

        Drive:\CAMWorksData\CAMWorks2021x64\EDM\Posts

        to

        Drive:\CAMWorksData\CAMWorks2022x64\EDM\Legacy-MSAccessDatabases\Posts

2. Overwrite the MachineSetup.bin and MachineSetup.text files present in the new CAMWorks release's data folder with the files from the previous release.

3. Customized Wire EDM Cutting Conditions files need to be copied from the data folder location of previous release to the corresponding folder location in the new CAMWorks release.

   i. For customized SQLite based Wire EDM Cutting Conditions files, the files need to be copied from

   Drive:\CAMWorksData\CAMWorks2021x64\EDM\EDMCuttingConditions

   to

   Drive:\CAMWorksData\CAMWorks2022x64\EDM\EDMCuttingConditions

   ii. For customized MS Access based Wire EDM Post files, the files need to be copied from

   Drive:\CAMWorksData\CAMWorks2021x64\EDM\EDMCuttingConditions

   to

   Drive:\CAMWorksData\CAMWorks2022x64\EDM\Legacy-MSAccessDatabases\ EDMCuttingConditions

4. Once the files are copied, verify the successful listing of Machines/Models and their associated file paths. A successful listing indicates that the EDM post files were successfully copied and activated. How to verify the successful listing is explained in the next paragraph.

Verifying successful activation of EDM Post files after upgrading your CAMWorks Installation

After copying customized EDM post files to their designated locations within the CAMWorks Data folder, you can verify their successful activation as follows:

1. Launch CAMWorks as an Add-In within SOLIDWORKS/CAMWorks Solids.

2. Double click the *Machine* node in the Feature tree.

3. The *Machine dialog box* will be displayed. Select the *Wire EDM machine* on the Machine tab and click *OK*.

4. Right click the *Wire EDM Machine* item in the CAMWorks Feature tree and click *Select Machine* from the context menu.

5. The *Machine Selection dialog box* will be displayed. Within this dialog box, select the applicable Machine and Model and click OK.

6. Once again right click the *Wire EDM Machine* item in the CAMWorks Feature tree and select *Edit Settings* from the context menu.

7. The *EDM Settings dialog box* will be displayed. Verify the folder paths for Cutting Conditions database files and Post Processor files in the Machine Setup tab and Posting tab respectively to make sure they are correct. If the EDM post files were copied correctly, the correct paths will be displayed.

8. Click *OK* to close this dialog box.