



Advanced
Manufacturing
Solutions

FeatureCAM 2012

Training Course

FeatureCAM 2012

Training Course

FeatureWIRE



FeatureCAM

Copyright © 1995-2011 Delcam plc. All rights reserved.

Delcam plc has no control over the use made of the software described in this manual and cannot accept responsibility for any loss or damage howsoever caused as a result of using the software. Users are advised that all the results from the software should be checked by a competent person, in accordance with good quality control procedures.

The functionality and user interface in this manual is subject to change without notice in future revisions of software.

The software described in this manual is furnished under licence agreement and may be used or copied solely in accordance with the terms of such licence.

Delcam plc grants permission for licensed users to print copies of this manual or portions of this manual for personal use only. Schools, colleges and universities that are licensed to use the software may make copies of this manual or portions of this manual for students currently registered for classes where the software is used.

Acknowledgements

This documentation references a number of registered trademarks and these are the property of their respective owners. For example, Microsoft and Windows are either registered trademarks or trademarks of Microsoft Corporation in the United States.

FeatureCAM overview

FeatureCAM is a CAD/CAM software suite that automates machining and minimizes programming times for parts on mills, lathes, and wire EDM. Unlike operations-based CAM systems, FeatureCAM generates toolpaths based on the features of the part, and automatically selects appropriate tools, determines roughing and finishing passes, and calculates feeds and speeds. The selections made can be based on the built-in machining knowledge that Delcam supplies 'out-of-the-box' with FeatureCAM, or from experience captured from your company, project or individual users' preferences.

FeatureCAM includes five stand-alone modules:

- **2.5D Milling** - 2.5D design and toolpath generation for 2- and 3-axis mills.
- **3D Milling** - 3D surface modeling and 3-axis toolpath generation.
- **3D Lite** - a limited version of **3D milling**.



***3D Lite** lets you mill only one surface per feature, but you can create multiple features. The strategies available in **3D Lite** are Z-level rough, Parallel rough, Parallel finish, Isoline, and 2D spiral.*

- **Turning** - 2-axis design and toolpath generation for 2-axis lathes.
- **Turn/Mill** - Supports lathes with C and Y-axis milling capabilities.
- **Wire EDM** - 2- and 4-axis wire EDM toolpath creation.

The following add-on modules are also available:

- **RECOGNITION** - 3D surface and solid import and the recognition of 2.5D features from solid models. Accelerates making 2.5D and turned parts from solid models.
- **Tombstone** - Multiple part manufacturing for horizontal or vertical milling machines with indexers.

- **Solid Modeling** - Solid modeling and tools for creating molds from solid models.
- **5-Axis Positioning** - Manufacture 2.5D features from 5-axis orientations.
- **Native Import Modules** - Native data can be read directly from SolidWorks, SolidWorks Assemblies, Autodesk Inventor, SolidEdge, Catia, NX, Pro-Engineer, and Step files.
- **Machine Simulation** - Modeling and simulation of a CNC machine.
- **Advanced Turn/Mill (MTT)** - Includes support for Turn/Mill in addition to support for B-axis (5-axis positioning) and multiple turret synchronization.
- **Network Database and Licensing** - Flexible product licensing allows sharing FeatureCAM licenses across a network.
- **5-Axis Simultaneous** - Manufacture 3D features while changing the tool axis.

Why creating toolpaths is so fast

FeatureCAM has the unique ability to generate toolpaths and create NC code to run the machines with a minimum amount of user input.

Traditional CAM systems are *operations-based* and require you to program every operation, one at a time, to create your part.

FeatureCAM is *feature-based*; this means the part is created using features that describe that part, from simple holes, to complex pockets, to turned grooves. Machinable features contain information and rules describing how and where material removal should occur, cutting depths, whether to use climb cutting, whether to spot drill or center drill, and preferred machining strategies for roughing and finishing. This means that after you import or draw the part and identify its features, FeatureCAM automatically:

- Selects the most appropriate tools and operations;
- Recommends machining strategies;
- Calculates speeds and feeds;
- Generates toolpaths and creates the NC code.



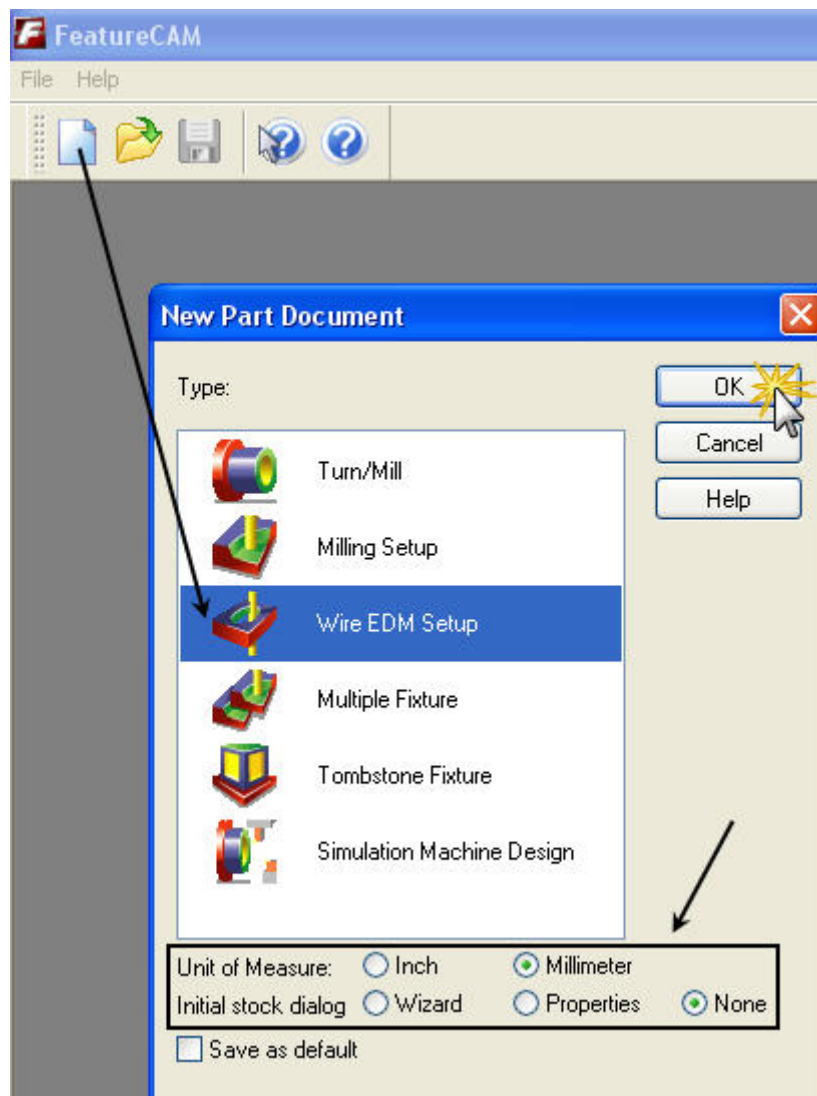
You can customize this built-in 'intelligence' to your own style of cutting.

Introduction

FeatureWIRE is structured similarly to all other **FeatureCAM** products and the steps are used to walk you through the process.

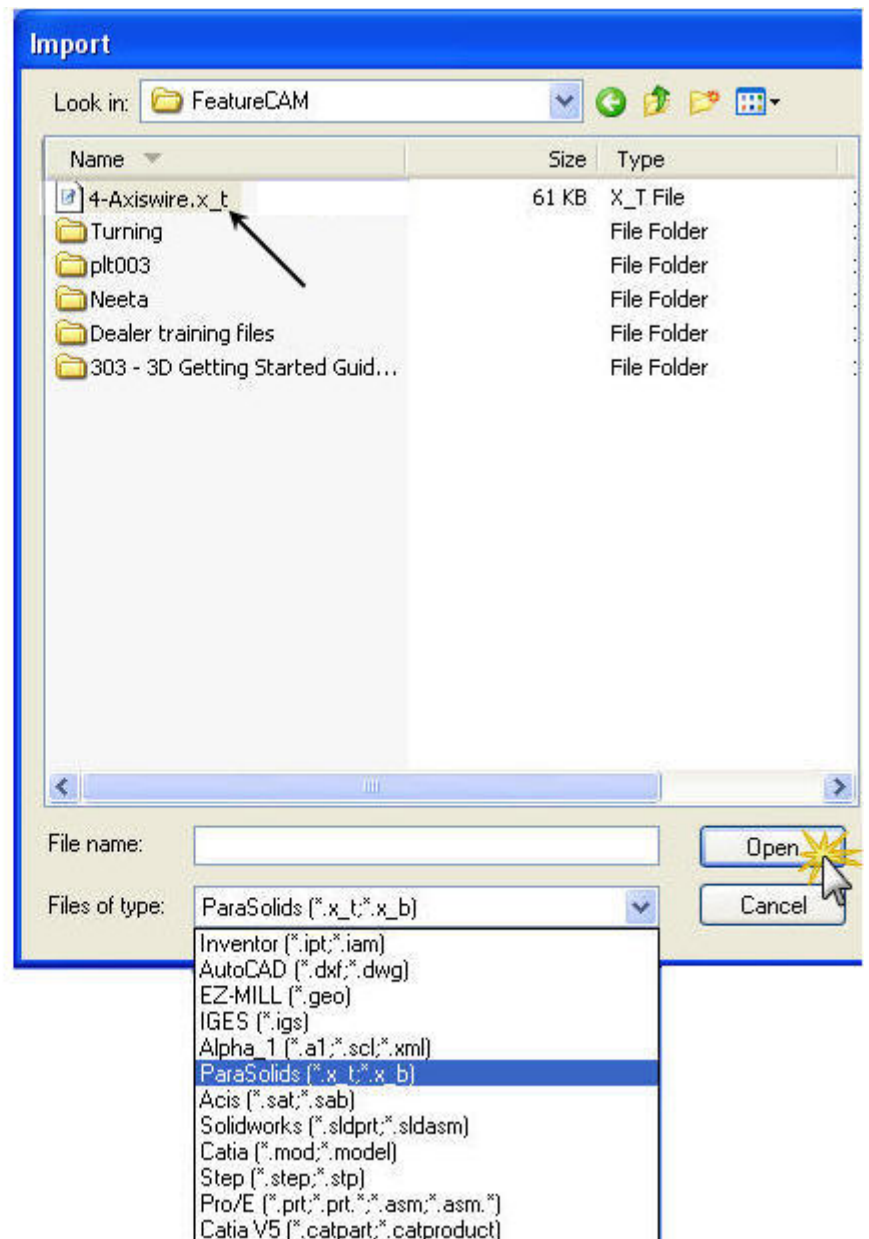
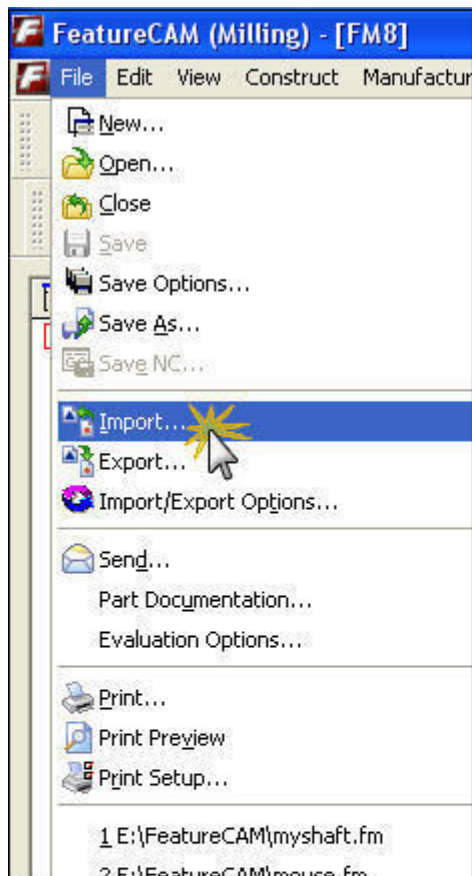
4-Axis Wire EDM feature recognition from Solid

- Open **New Part Document**. Select **Milling setup**.
- Select the units you will use to model your part from among: **Inch / Millimetre**.
- Select **Millimetre**.
- Select how you would like to setup your **stock**. Select from:
Wizard / Properties / None.
- Select **None** and Click **OK**.



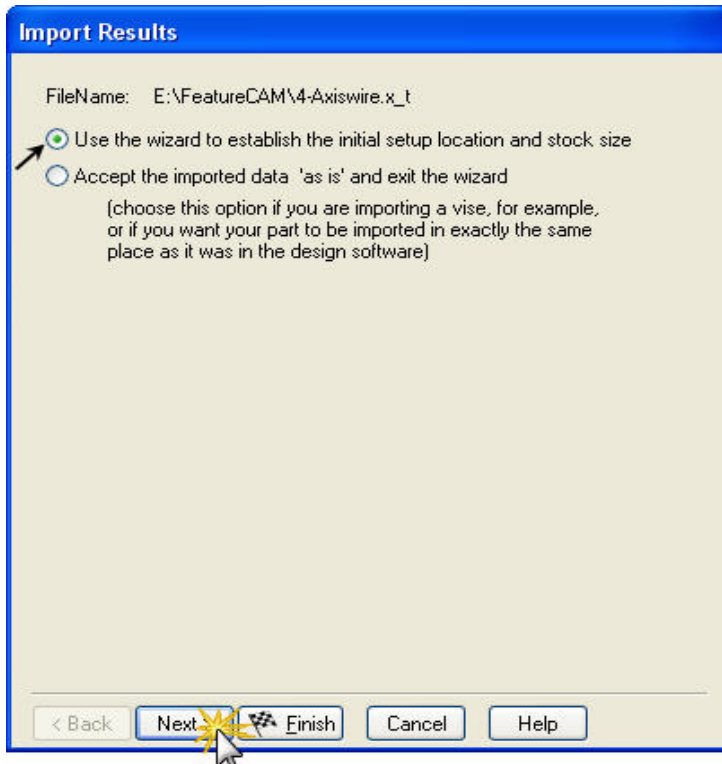
Importing files from CAD systems

- Open a new part file. You must have a **part open to import geometry**.
- Select **Import** from the File menu. Select “**4-axiswire.x_t**” file as shown below. Click **Open**.



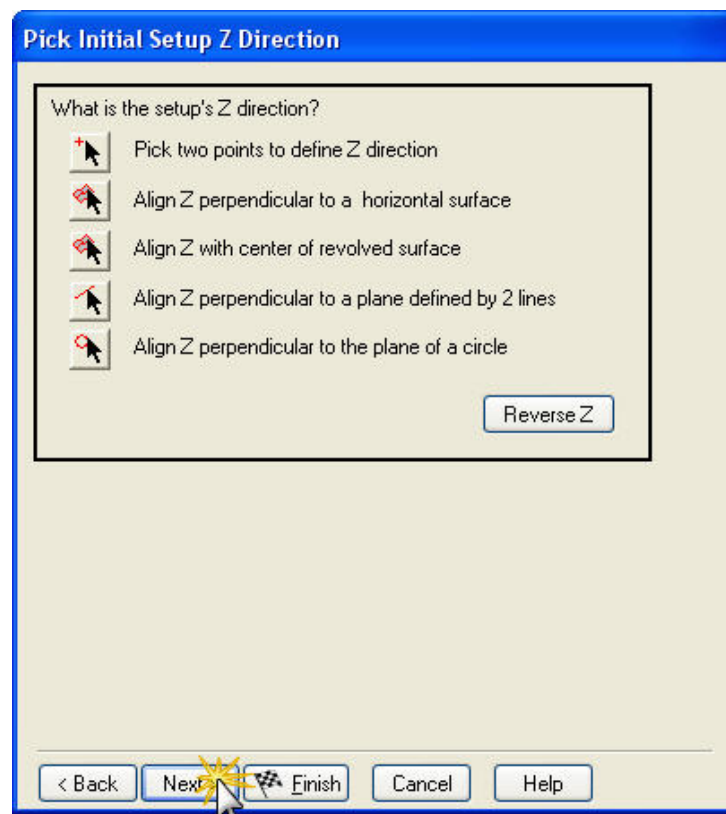
Import Results

The import of the **CAD data** file is just the beginning of the import process in **FeatureCAM**. After the file is imported, the Import wizard can be run to align the part to stock, size the stock and for some file formats, even perform some initial feature recognition.



- Select “**Use the wizard to establish the initial setup location and stock size**” as shown below and **Click Next**.

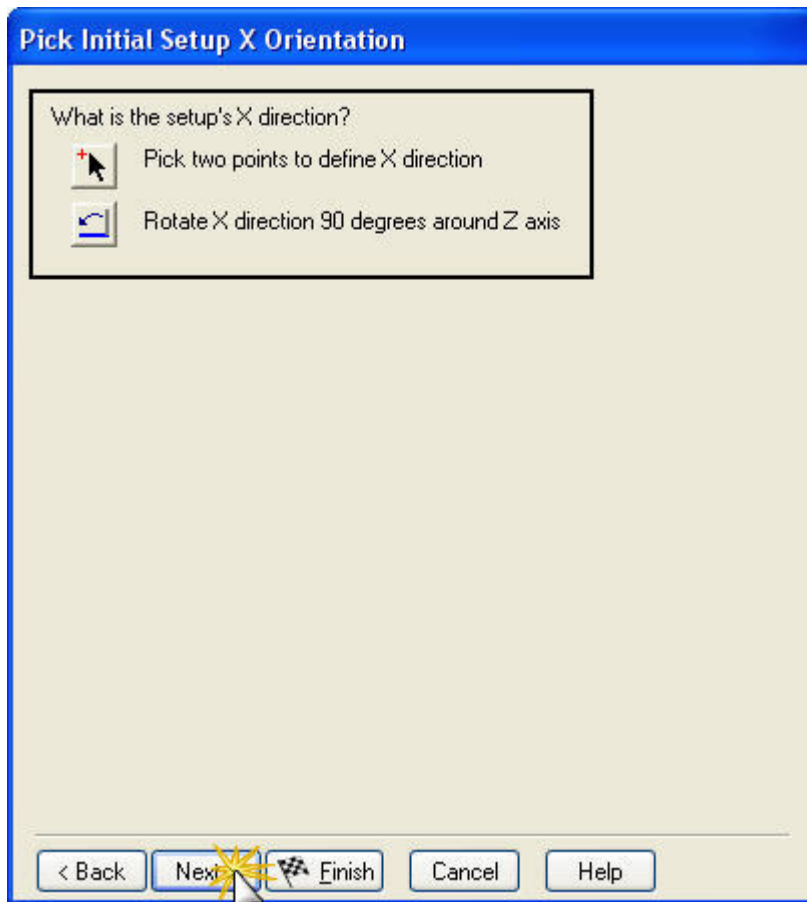
Pick Initial Setup Z Direction



In this dialog box, you set the **Z direction** for the setup. This will correspond to the **Z axis** of your machining coordinates. If you are happy with the current **Z direction**, just **Click Next**.

Options to change the **direction of the Z axis**:

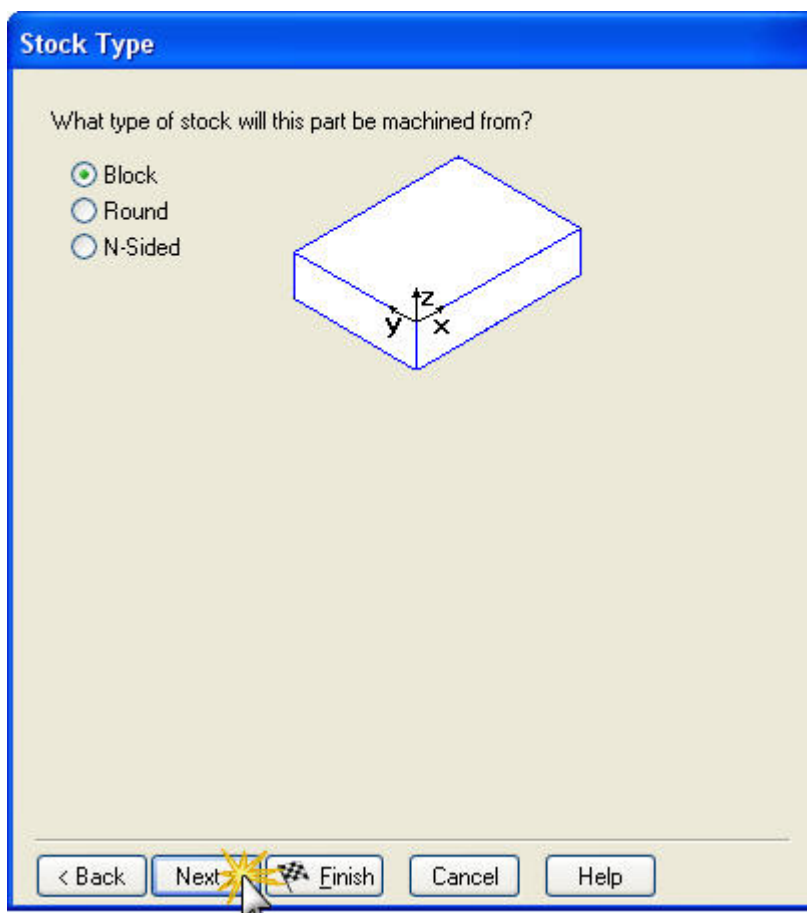
Keep as it is and **Click the Next** button.



Pick Initial Setup X Orientation

In this dialog box you define the **orientation** of the setup **X axis**.

Options to **orient X direction**, Keep as it is and **Click Next**.



Stock Type

In this dialog box, you set the shape of your stock.

To complete this dialog box: Select **Block** for rectangular stock.

Select **Round for cylindrical stock**. Select the axis that runs down the center of the stock. Select N-sided for the stock, and specify the number of sides for the stock. Symmetric shapes like hexagonal or triangular stock.

Select **Block** and **Click Next**

Stock dimensions

On this page you specify the size of your stock. The page is basically a three-column table showing you the size of the data and the size of the stock.

- Block Stock.
- Select **Compute stock size from the size of the part**, the stock is sized to fit the part. In the second column, you can enter additional amounts to add to any stock dimension. **Enter the dimensions** as shown below.
- Click Next.

Stock Dimensions

☐ Enter specific stock dimensions
☒ Compute stock size from the size of the part

Width: 147.619
 Length: 112.619
 Thickness: 50.000

Imported Data:	Extra stock size	Stock size
Length: 147.619	+ Extra stock size (-X): 5.000 mm.	= 157.619
	+ Extra stock size (+X): 5.000 mm.	
Width: 112.619	+ Extra stock size (-Y): 5.000 mm.	= 122.619
	+ Extra stock size (+Y): 5.000 mm.	
Thickness: 50.000	+ Extra stock size (-Z): 0.000 mm.	= 50.000
	+ Extra stock size (+Z): 0.000 mm.	

Preview

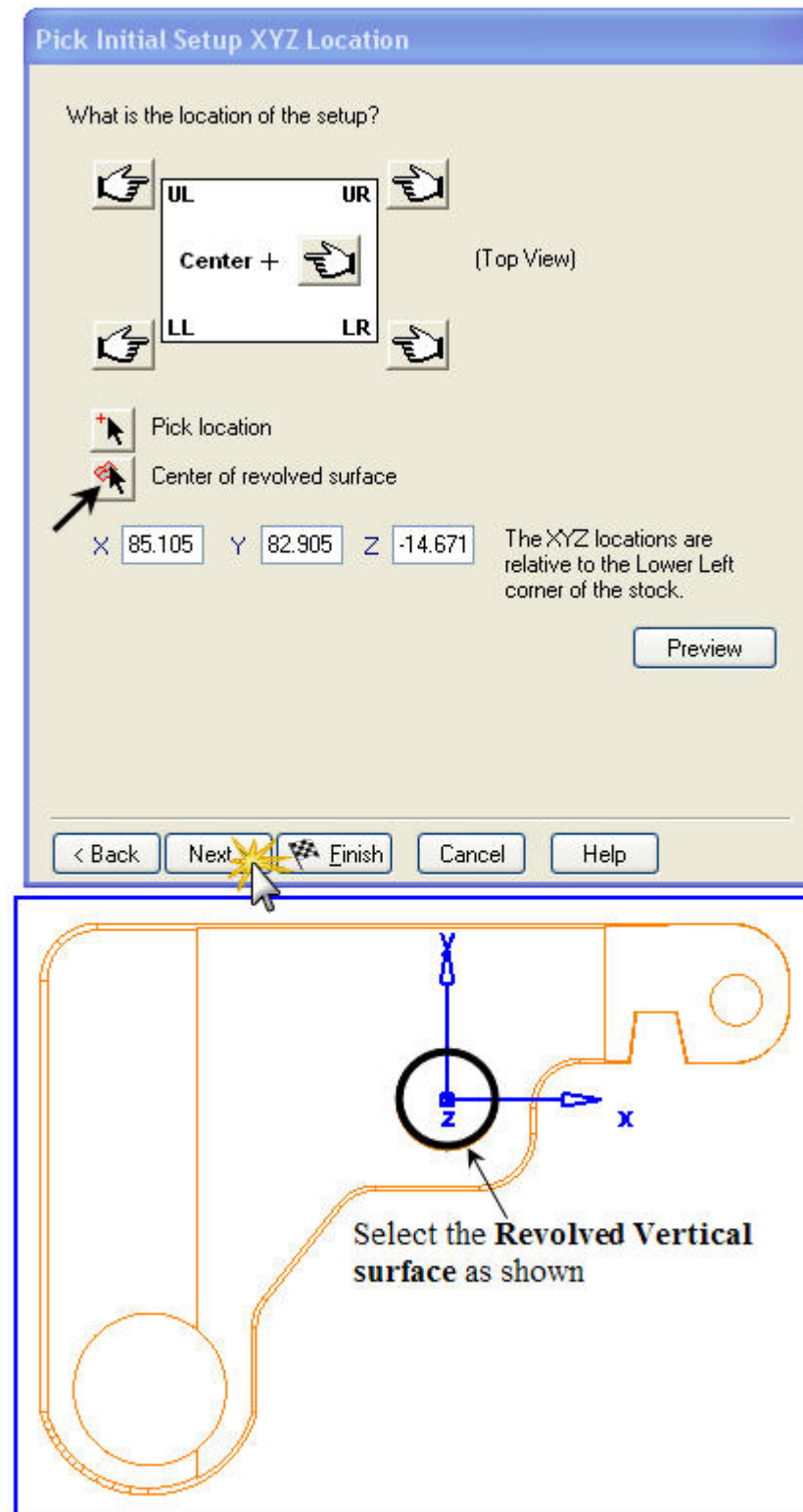
< Back Next > Finish Cancel Help

Pick initial setup XYZ location

This dialog box helps you set the part program zero. This will be the origin of the coordinate system for the NC program for this setup.

- If you want to locate the origin at one of the points indicated by a pointing finger button, click the button.

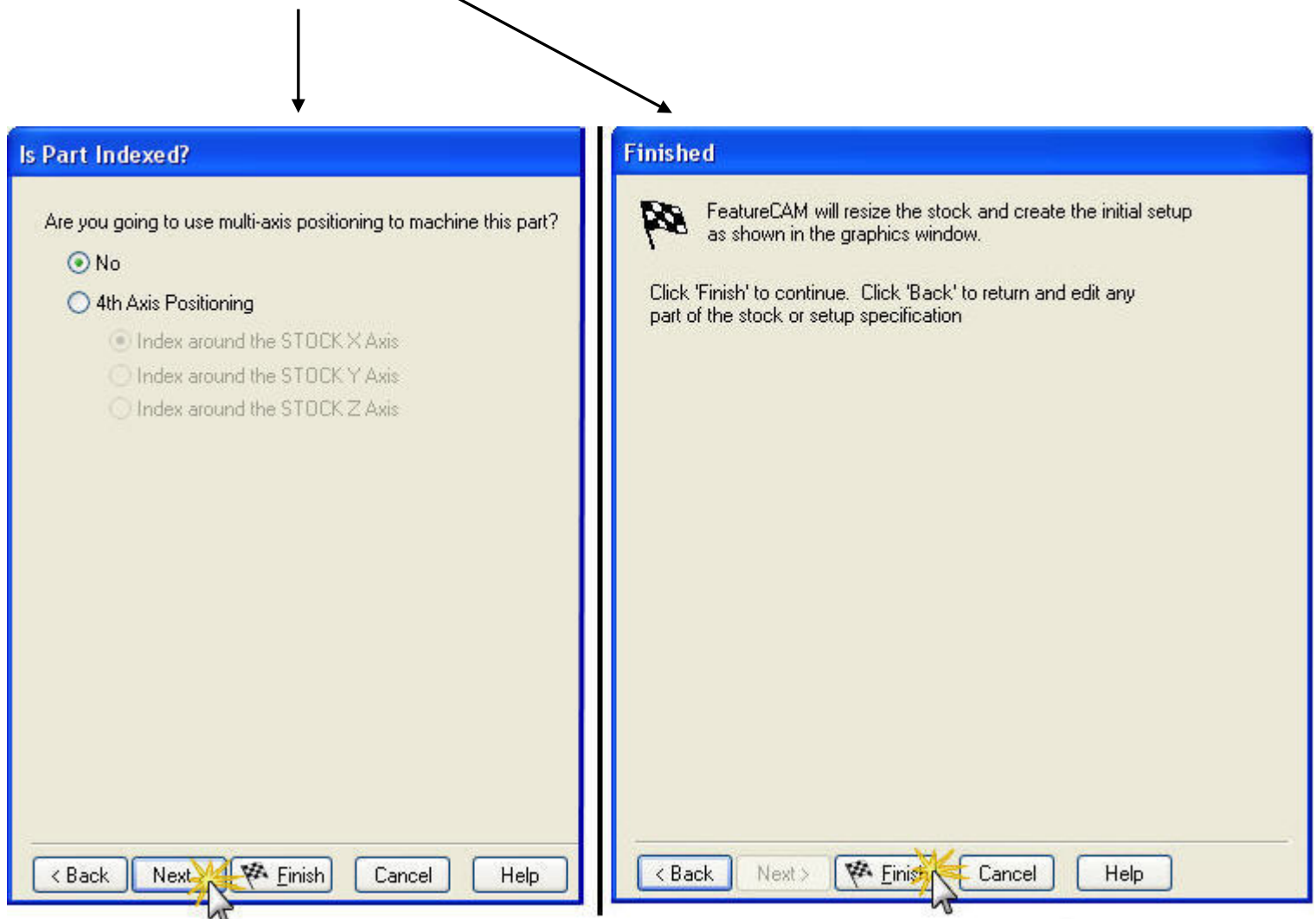
- **Pick Location:** If you want to type the coordinates, enter the X, Y and Z coordinates.
- **Center of revolved surface:** Select the **revolved surface** as shown below. Click **Next**.

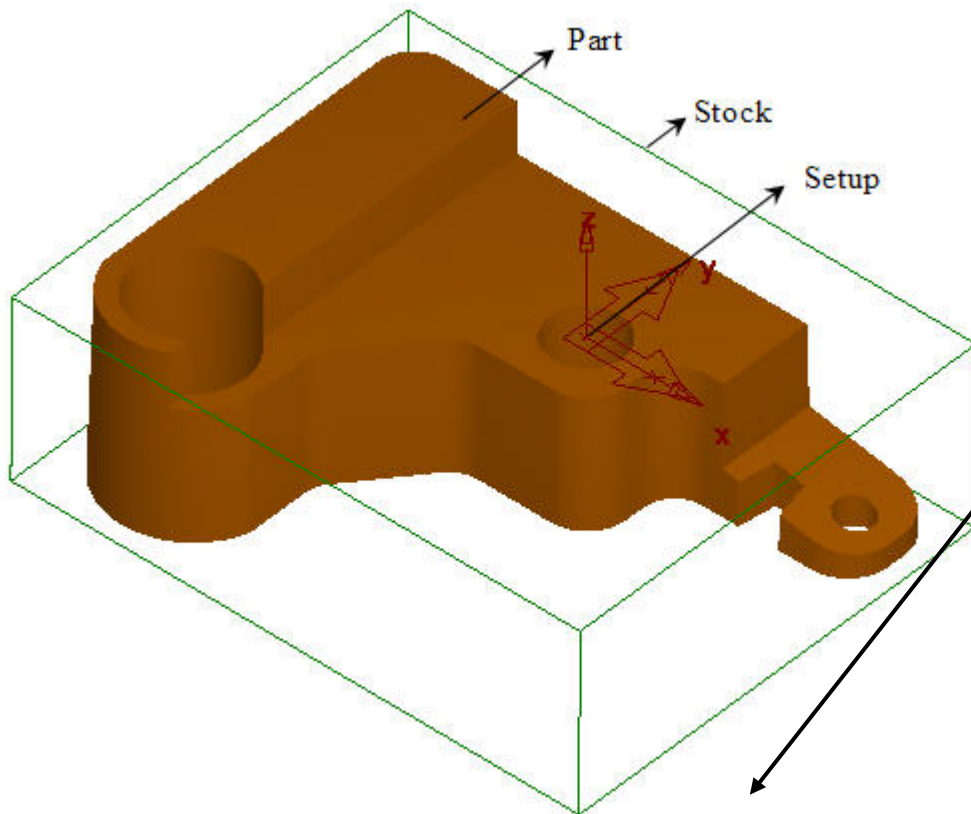


Is part indexed?

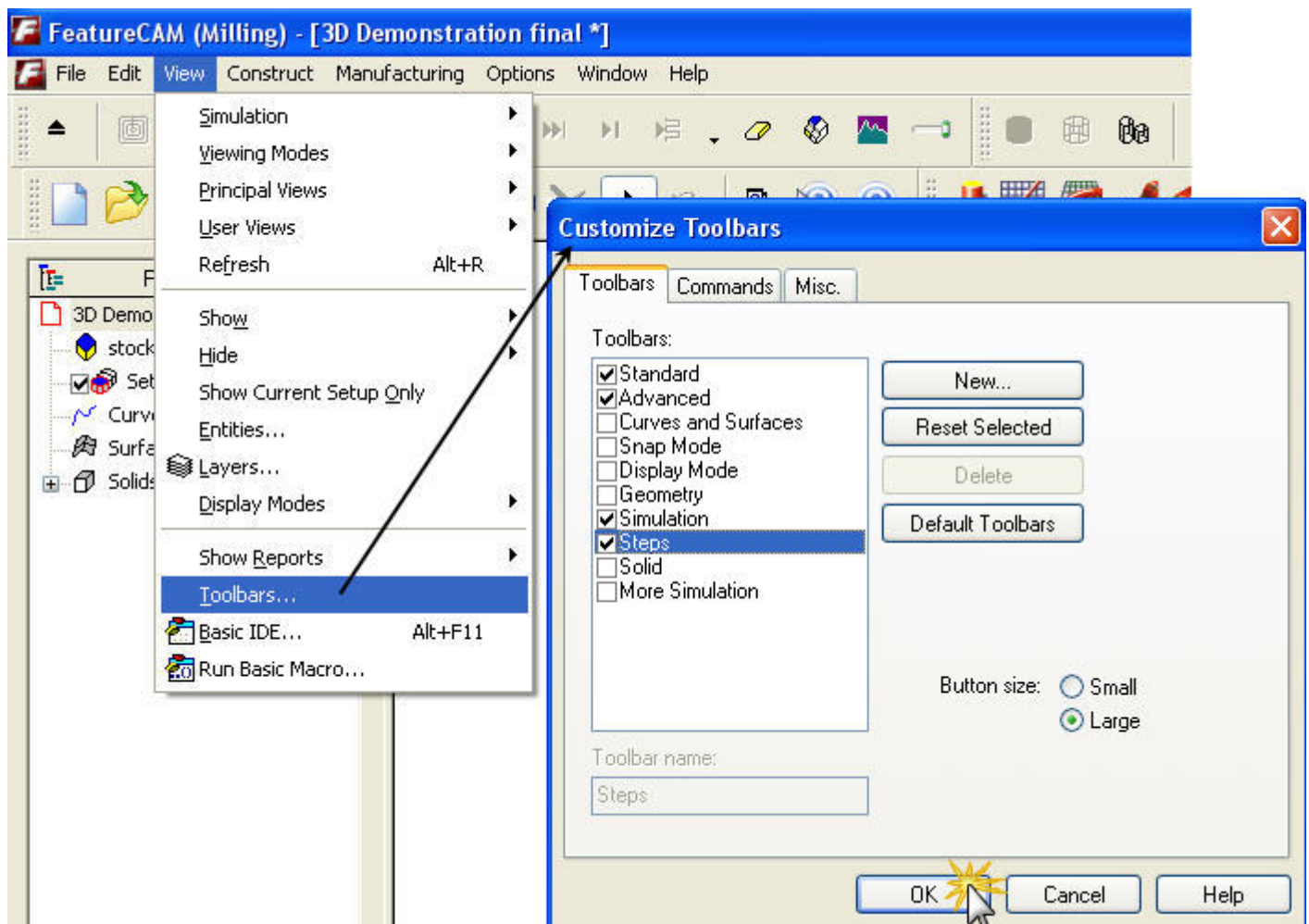
In this dialog box you specify if you will be creating your part with 4th axis indexing or 5th axis positioning.

- Multi axis positioning is not required to this part. **Click No.**
- **Stock and Setup are created according to the requirement.**
- **Click Next and Finish** as shown.



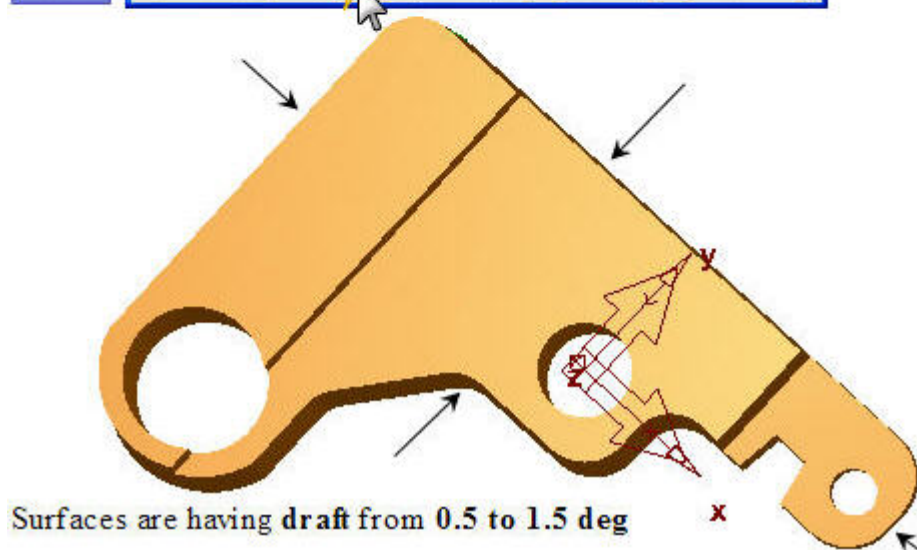
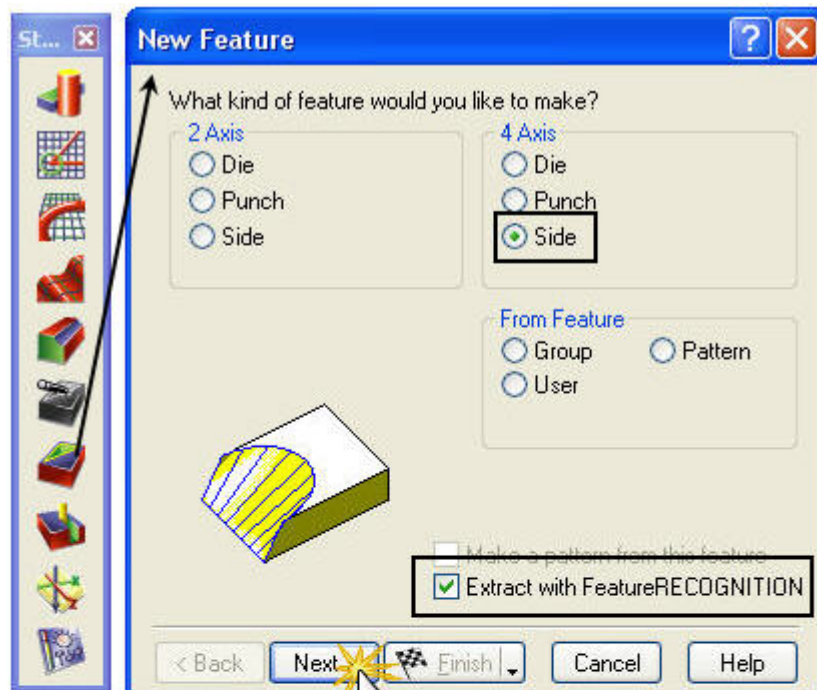


Stock creation and Setup creation are completed, as shown in the graphic window. **Turn on the Toolbars required from the View menu**, as shown. **Click OK.**



New Feature

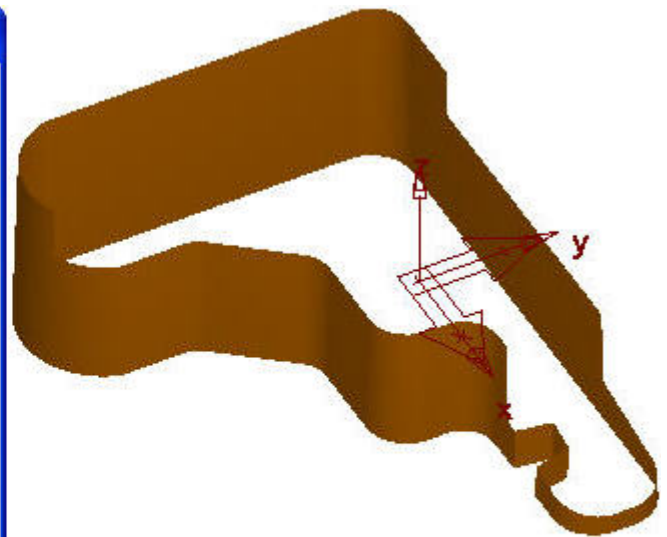
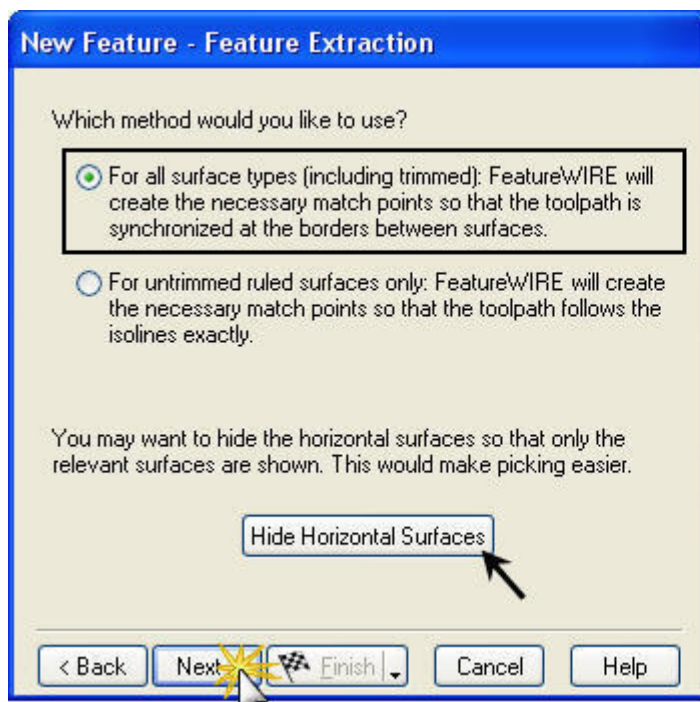
- Create **New Feature**. Select **Side** from **4-axis** and Turn on **Extract with Feature recognition**
- Click **Next**.




New Feature – Feature Extraction

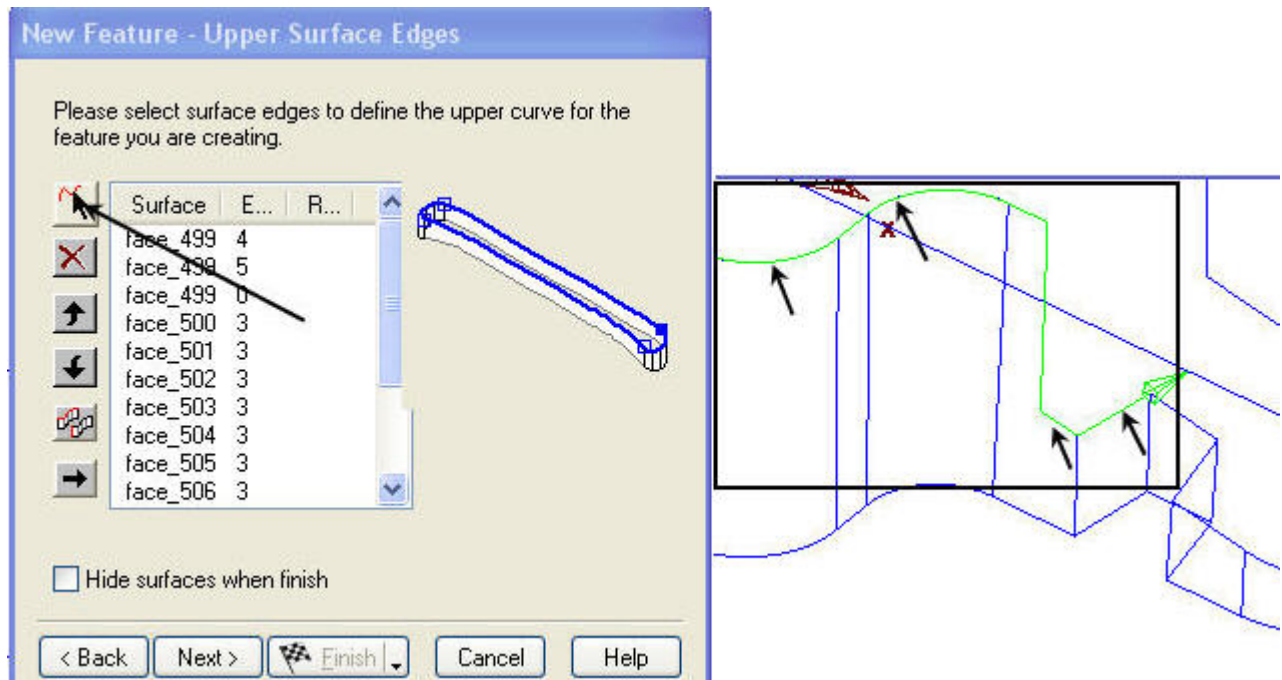
FeatureREGCONITION saves you time in creating a feature. It will extract the upper and lower curves from the surfaces for you. It will also create the necessary match points so that the toolpath is synchronized at the borders between surfaces. Without FeatureREGCONITION, you would have to extract the curves using the Surface Edges constructor and then tediously create the match points.

- Select the first method and use **Hide Horizontal Surfaces** to hide Flat surfaces automatically.
- Remaining has to select manually and hide using Right click of mouse.
- Part will be **displayed as shown below**. **Click Next**.




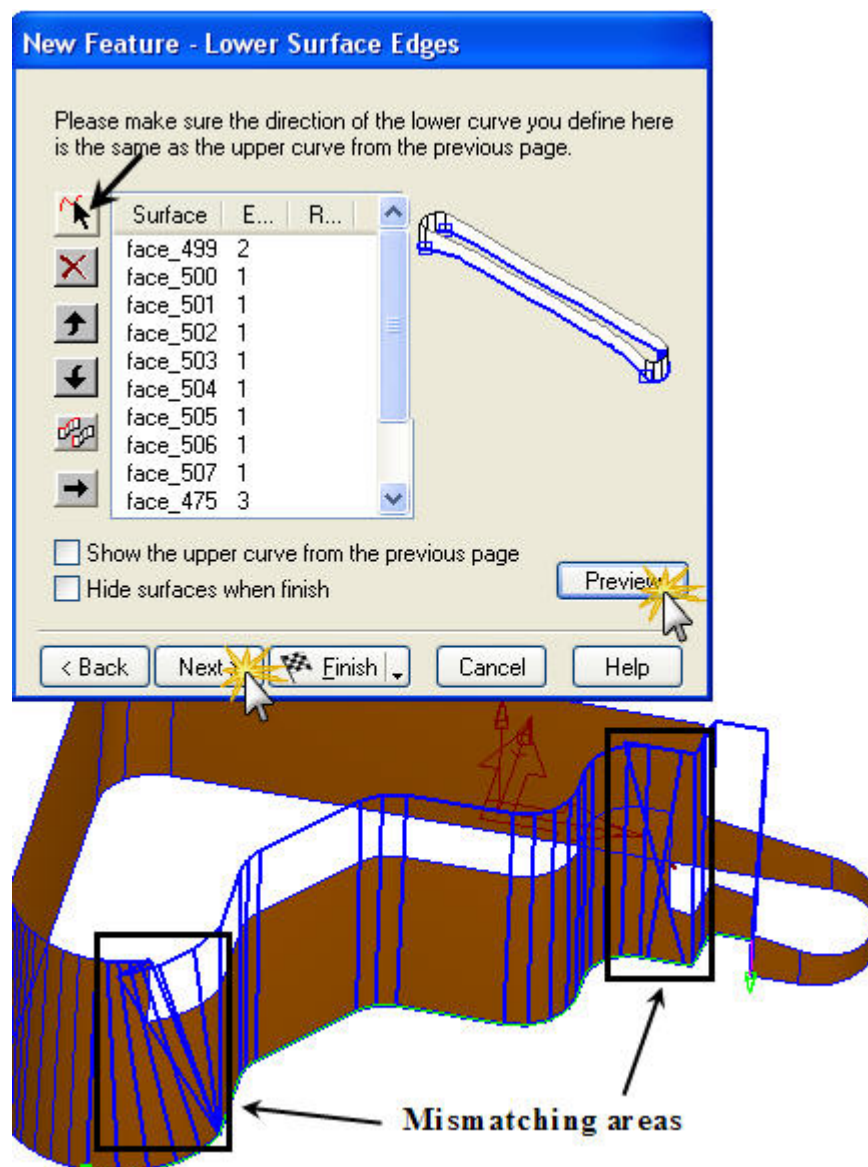
New Feature Upper Surface Edges

Select Upper Surface edges using pick curve on surface icon  as shown. While picking the edges, take a close look (Zoom) then select the edges. Select those required from **Upper Surface Edges** and **Click Next**.




New Feature Lower Surface Edges

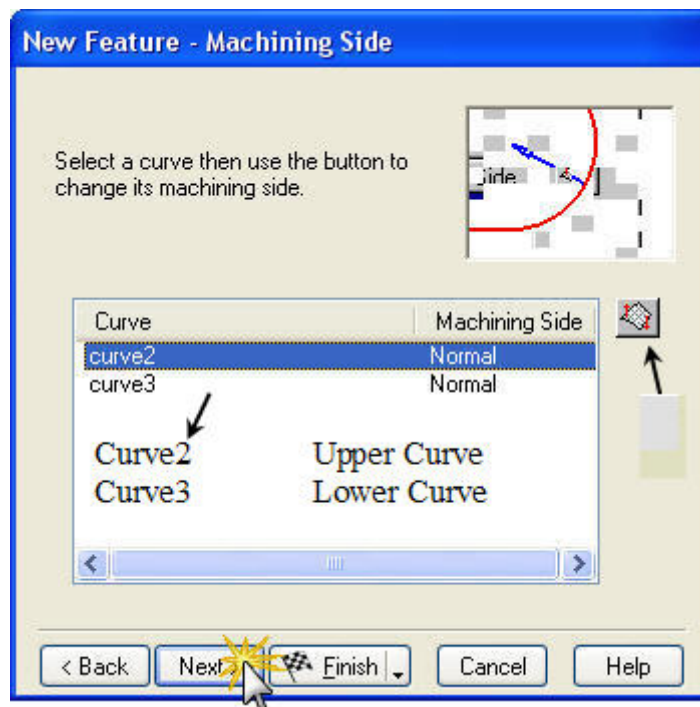
- Select **Lower Surface edges** using **pick curve on surface** icon  as shown.
- While picking the edges, take a close look (Zoom) then select the edges.
- Select those required from **Lower Surface Edges** and click **Preview**.
- There are some **mismatching areas** (highlighted). They are caused by the vertical drops in the upper curve and we will fix this problem in a later step. Note that FeatureREGCONITION can fix the vertical drops automatically if they are aligned with the Z-axis (the vertical drops here are slightly tilted).
- Click **Next**.



New Feature - which side

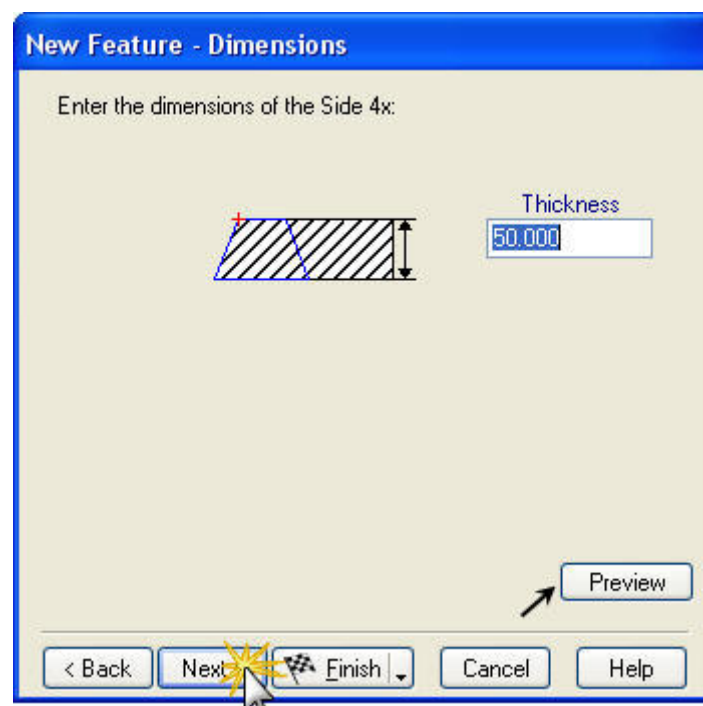
A large blue arrow is displayed pointing to the **cutting side** for each feature. You can modify the cutting side by selecting the curve in the table.

- Clicking the reverse **side**  button.
- **Click Next** when all the arrows are pointing in the **out side** direction.



New Feature – Dimensions

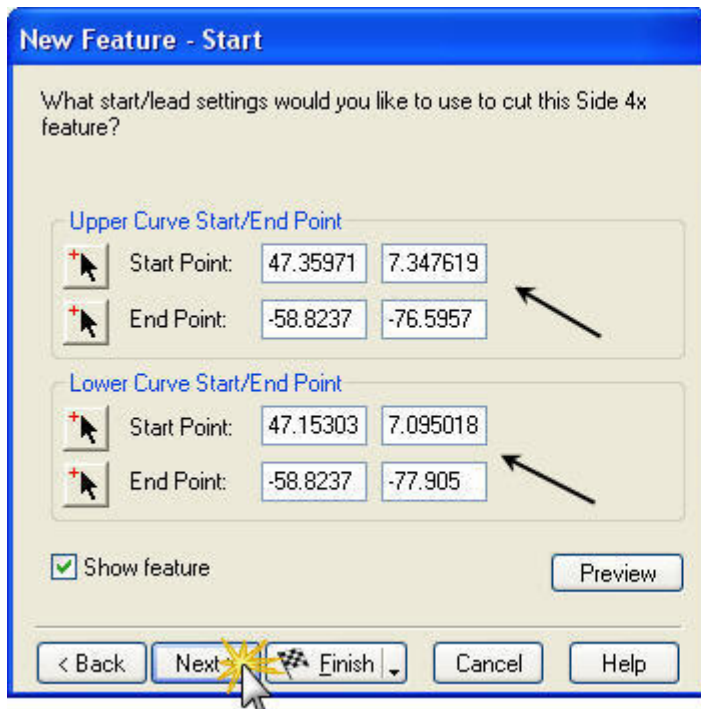
- Enter the **thickness** of stock and **Click Next**.



New feature - Start

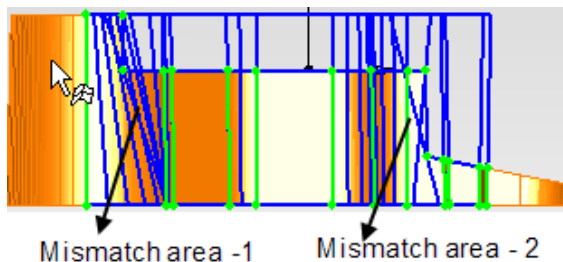
For 2 and 4-axis side features, the default is to start at the first point of the curve. To add new linear moves at the start or end of a curve.

- If the feature has multiple curves, select the appropriate curve from the Curve drop down list.



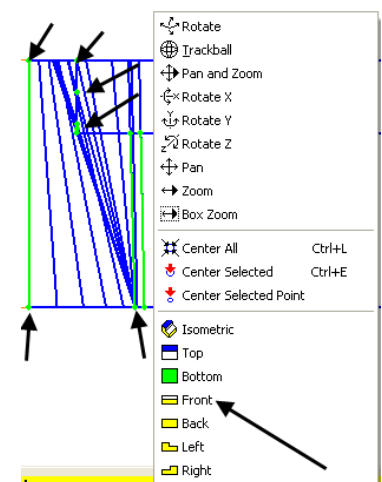
- Click the Pick point button for either the start or end point.
- Click a point at the new location.
- Alternatively, you can simple enter the new end point coordinates.
- Click Next.

New feature - Match curve



FeatureCAM attempts to match up the points of the upper and lower curves of a 4-axis feature. If the curves are not matched automatically, you can manually match the two curves (Upper and Lower).

Keep View as Front. Take a close look into Mismatch area-1; you can see there are four points on Upper edge and two points on Lower edge.



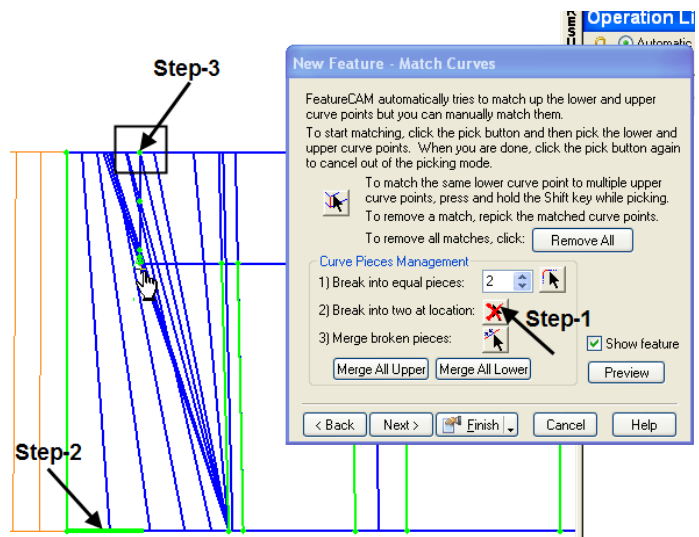
Use following Steps to match the curves properly:

Keep View as Front.

Step1. Select Break into two at location.

Step2. Select the Lower edge.

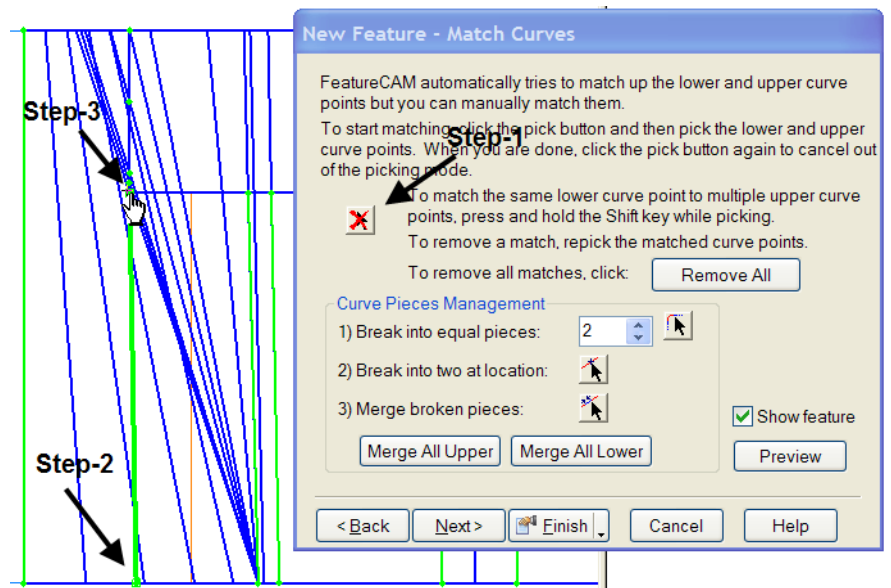
Step3. Click at upper edge point, as shown.



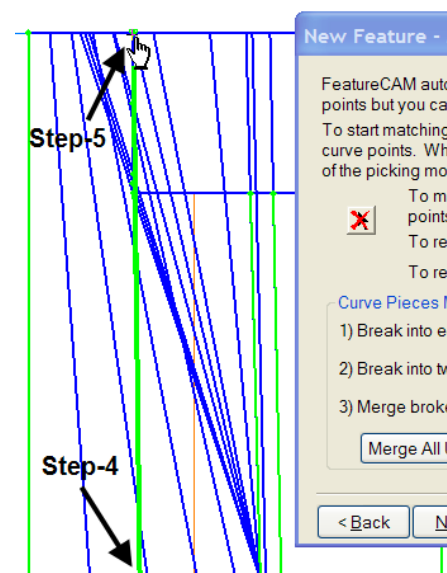
Step1. Select .

Step2. Select the Lower point.

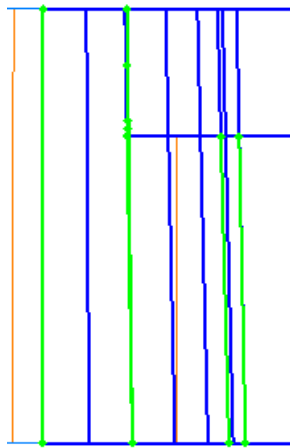
Step3. Select the Point **exactly** as shown in the Picture.



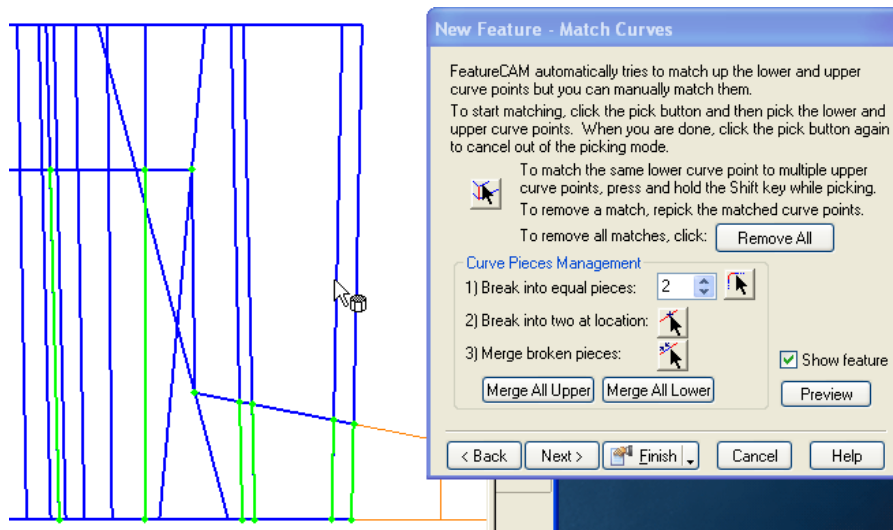
Step4 & 5. To select the same lower point to another upper point, hold the Shift key and select the points **exactly** as shown in the picture.



Step6. Click Preview. Results will be OK as shown in below picture.



Now take a look into Mismatch area-2.

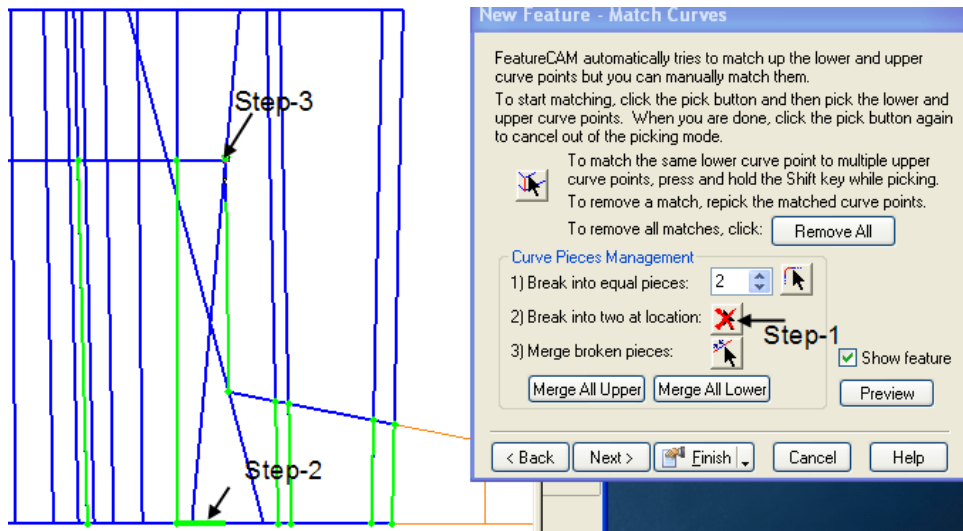


Use following Steps to match the curves properly:

Step1. Select Break into two at location.

Step2. Select the Lower edge.

Step3. Click at upper edge point, as shown. Now the curve as broken into two pieces.

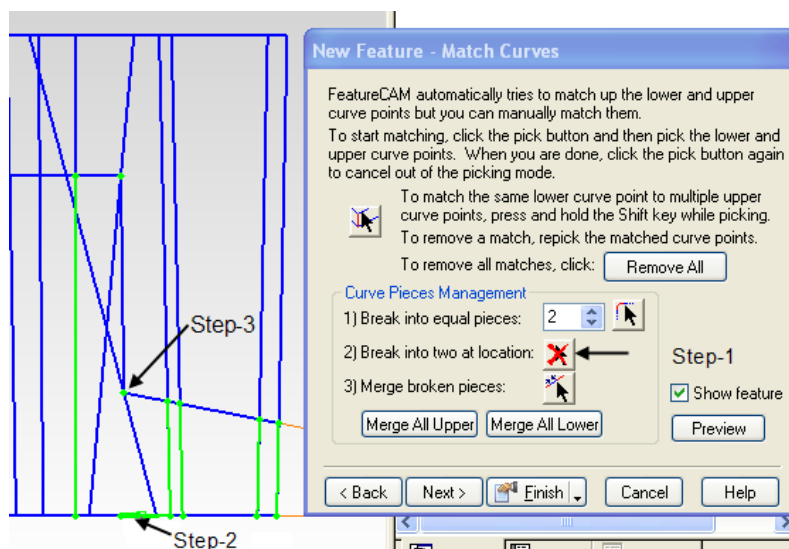


Step1. Select Break into two at location.

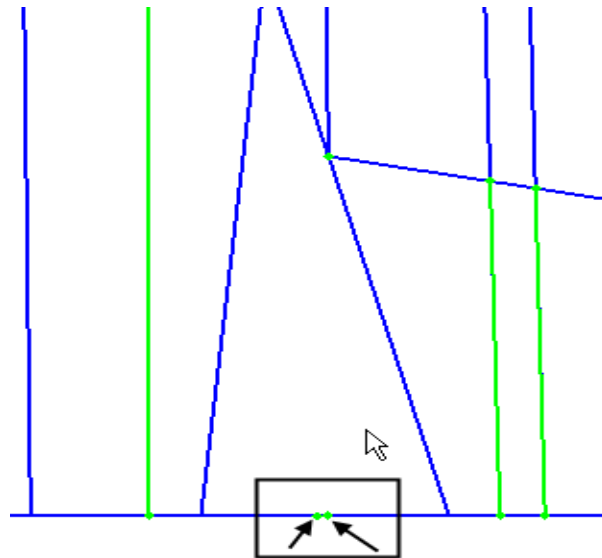
Step2. Select the Lower edge as shown.

Step3. Click at upper edge point **exactly** as shown. Now the curve as broken into three pieces.

All these you need to do in a Closer view as the distance is very less between points.



Use **Box Zoom** and you can see there are two points showing in the lower edge.



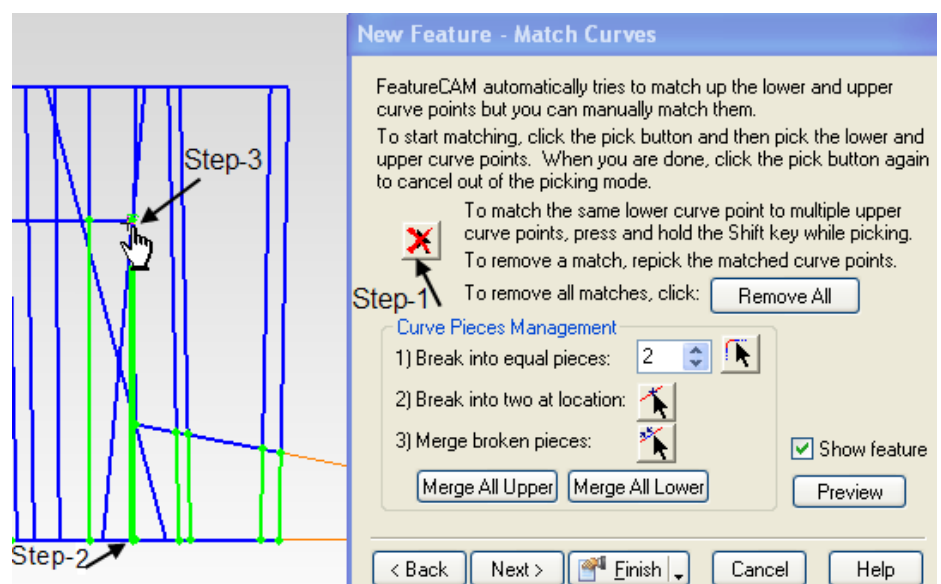
Use **Box Zoom** for selecting each point.

Step1. Select .

Step2. Select the Lower point.

Step3. Select the Upper point as shown in the Picture.

Click on preview.



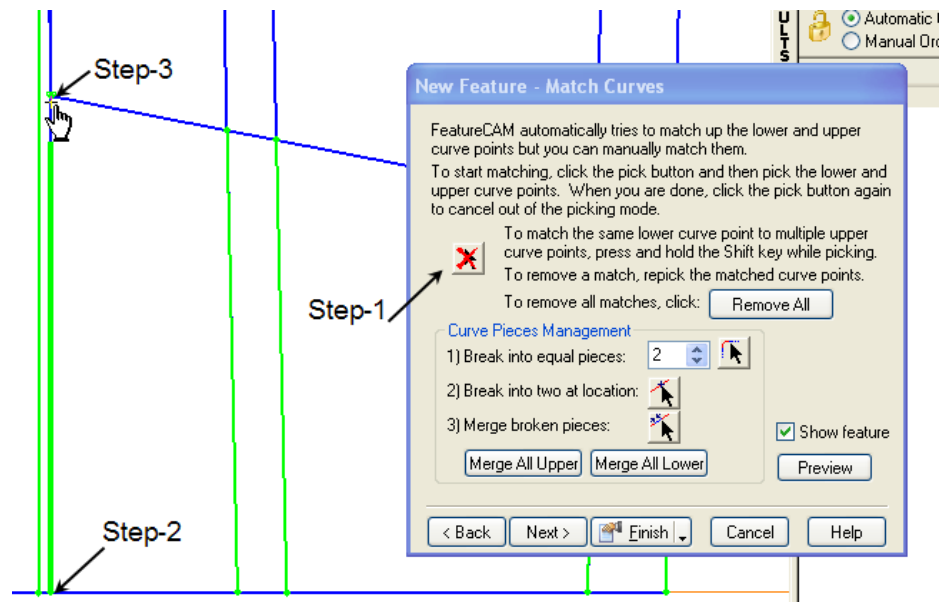
Steps for matching the second point.

Step1. Select .

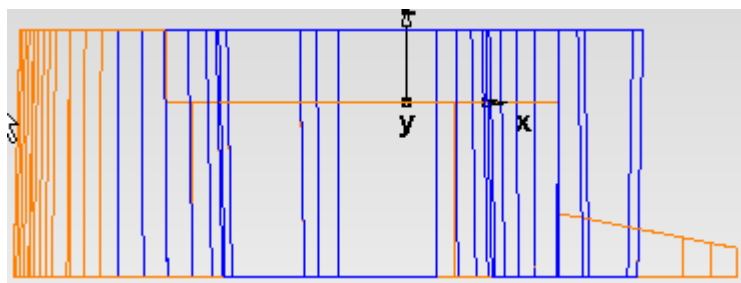
Step2. Select the Lower point.

Step3. Select the Upper point as shown in the Picture.

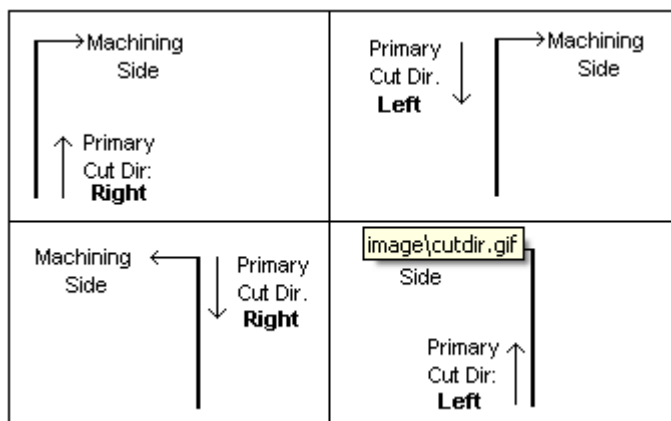
Click on preview



Final Matching Curve Results will be as shown:



New Feature – Strategies

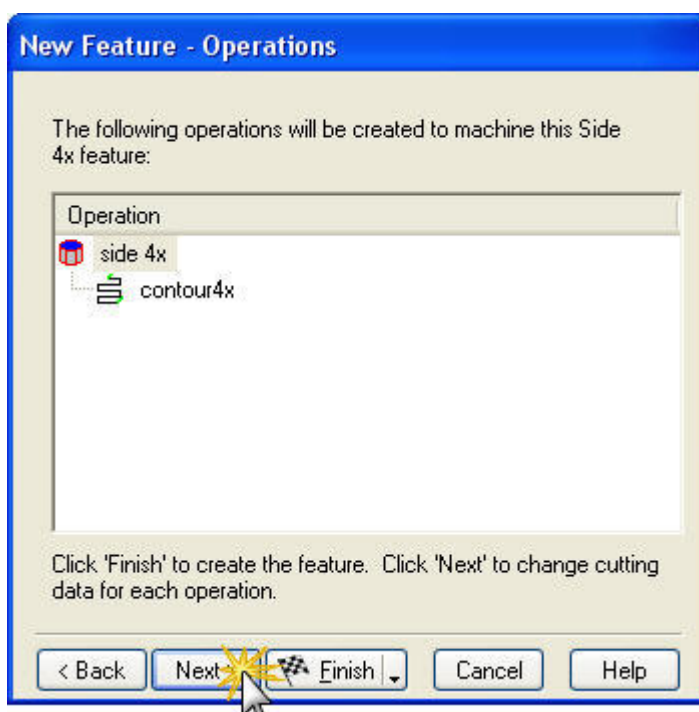
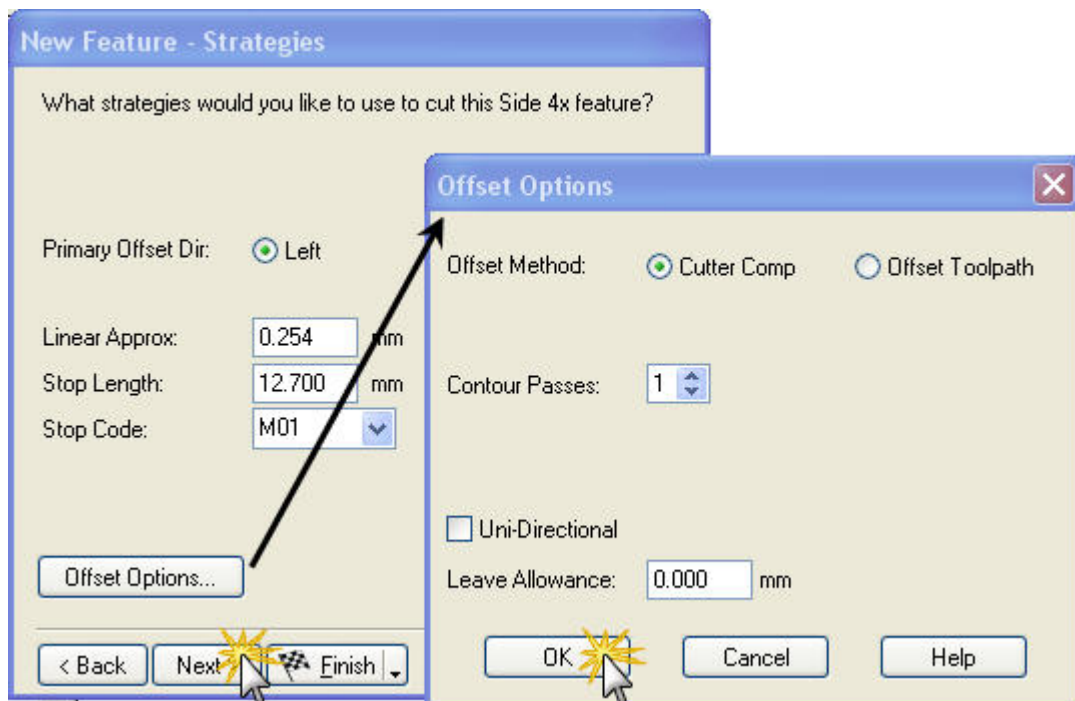


Primary cut direction

- This attribute **controls the direction of a cut**. For closed curves the options are **clockwise (CW)** or **counter-clockwise (CCW)**. For open curves the choices are **right or left**. These settings are relative to the machining side setting.

Offset method

- This radio box controls whether the offsetting of the wire path is performed on the **machine using cutter compensation** or by **FeatureCAM**. Select **Cutter comp** to perform the **offsetting** on the machine, or **Offset toolpath** to have **FeatureCAM** perform the **offsetting**. **Contour passes**
- **The number of passes to take for the contour operation.**
- **Click Next.**

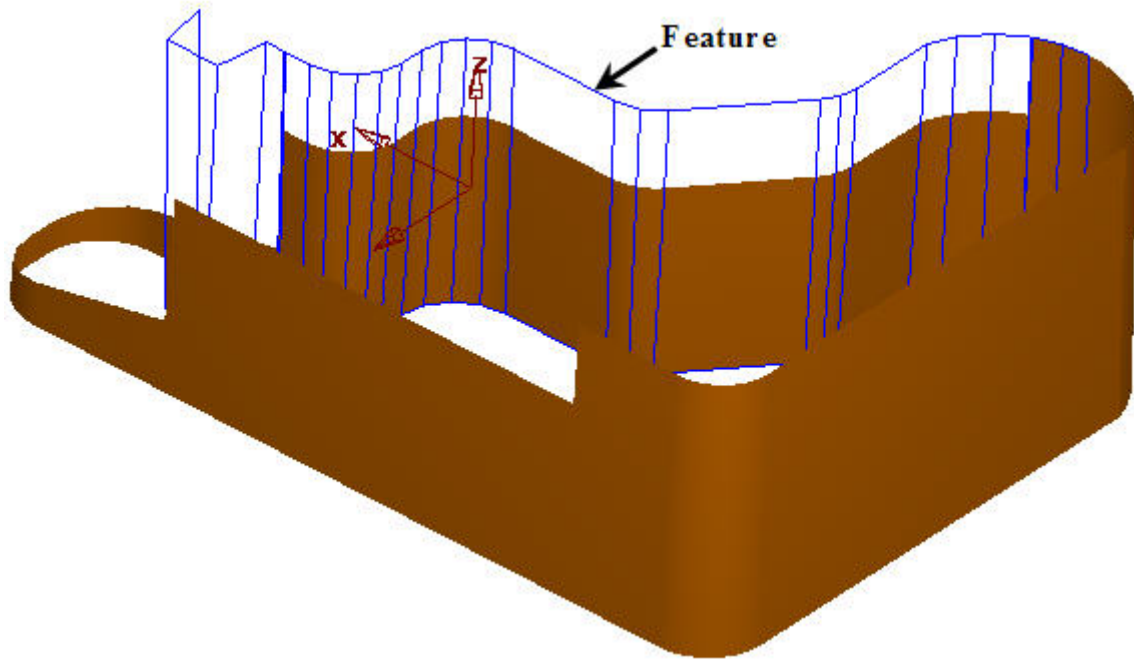


New feature operations

This page contains a summary of the operations that will be created to manufacture the feature, the names of the tools that have been automatically selected, and the feed and speed values that were calculated automatically. Each row of the table represents an operation.

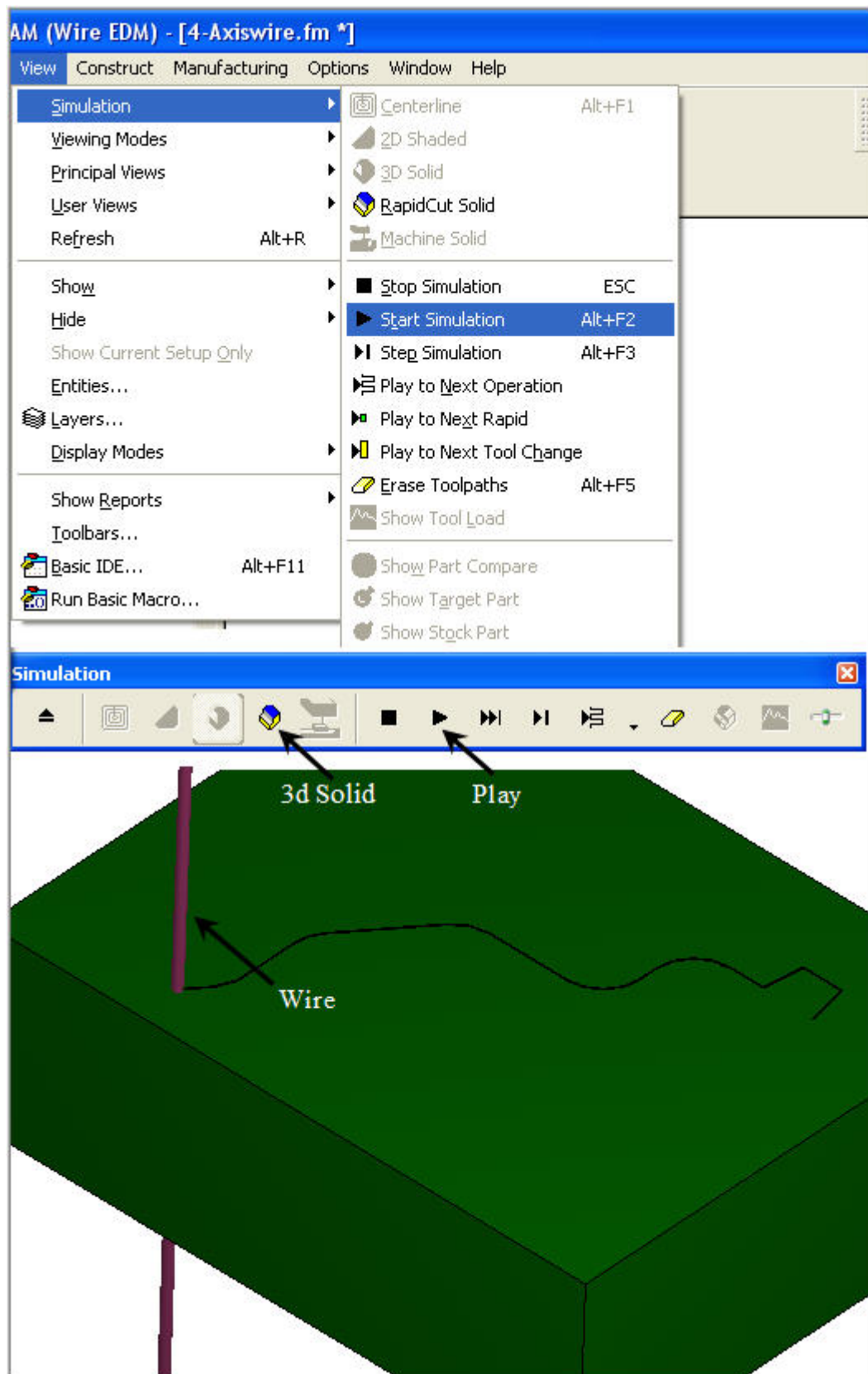
- If you are familiar with the tooling names and satisfied with the selected tooling and feeds and speeds, **Click Finish.**

New Feature - Created



3D Simulation

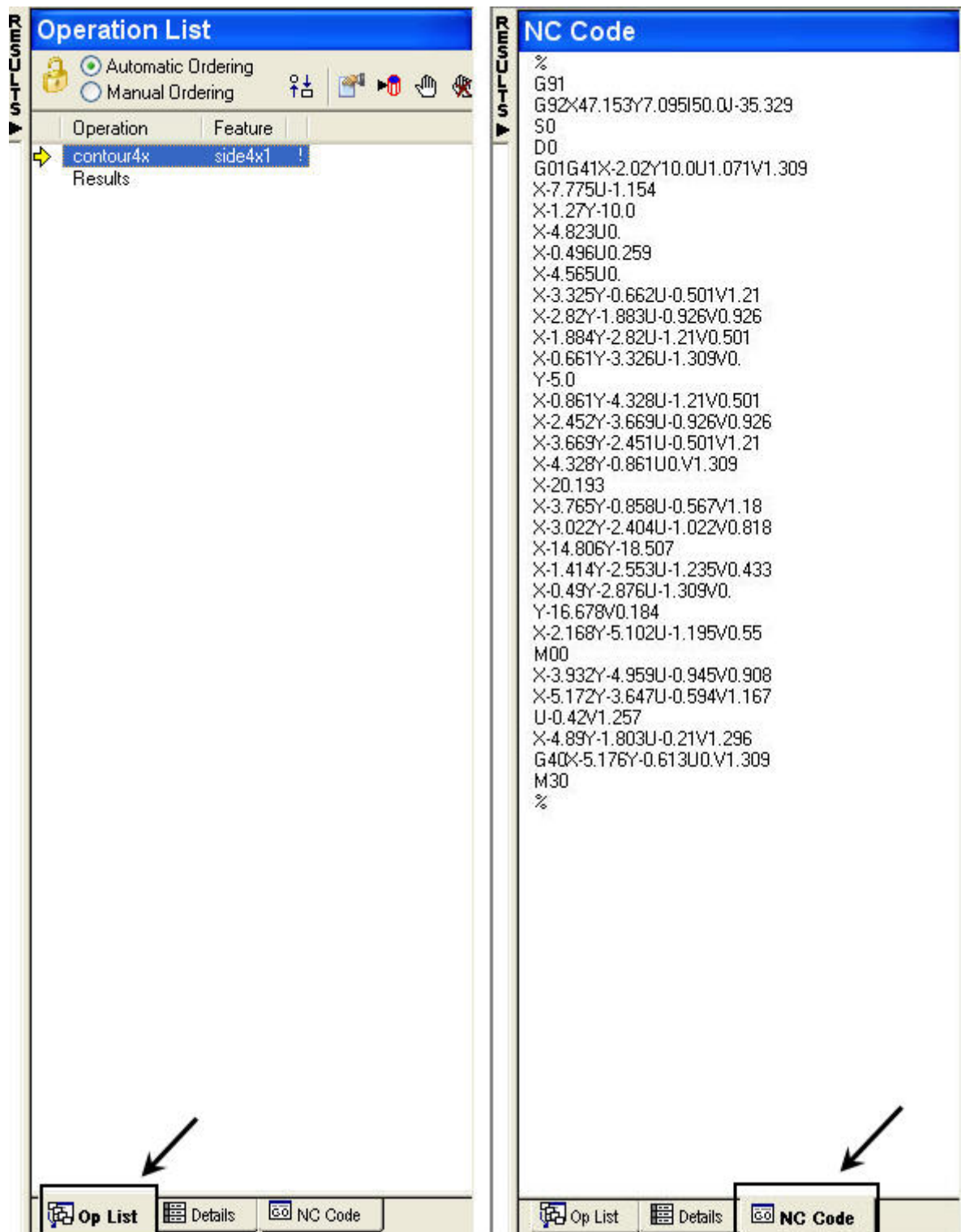
- Go to **Simulation** from the **View** menu and Click **Start Simulation**.
- **Play** Simulation from the **Simulation Menu** to know the results.



NC Code

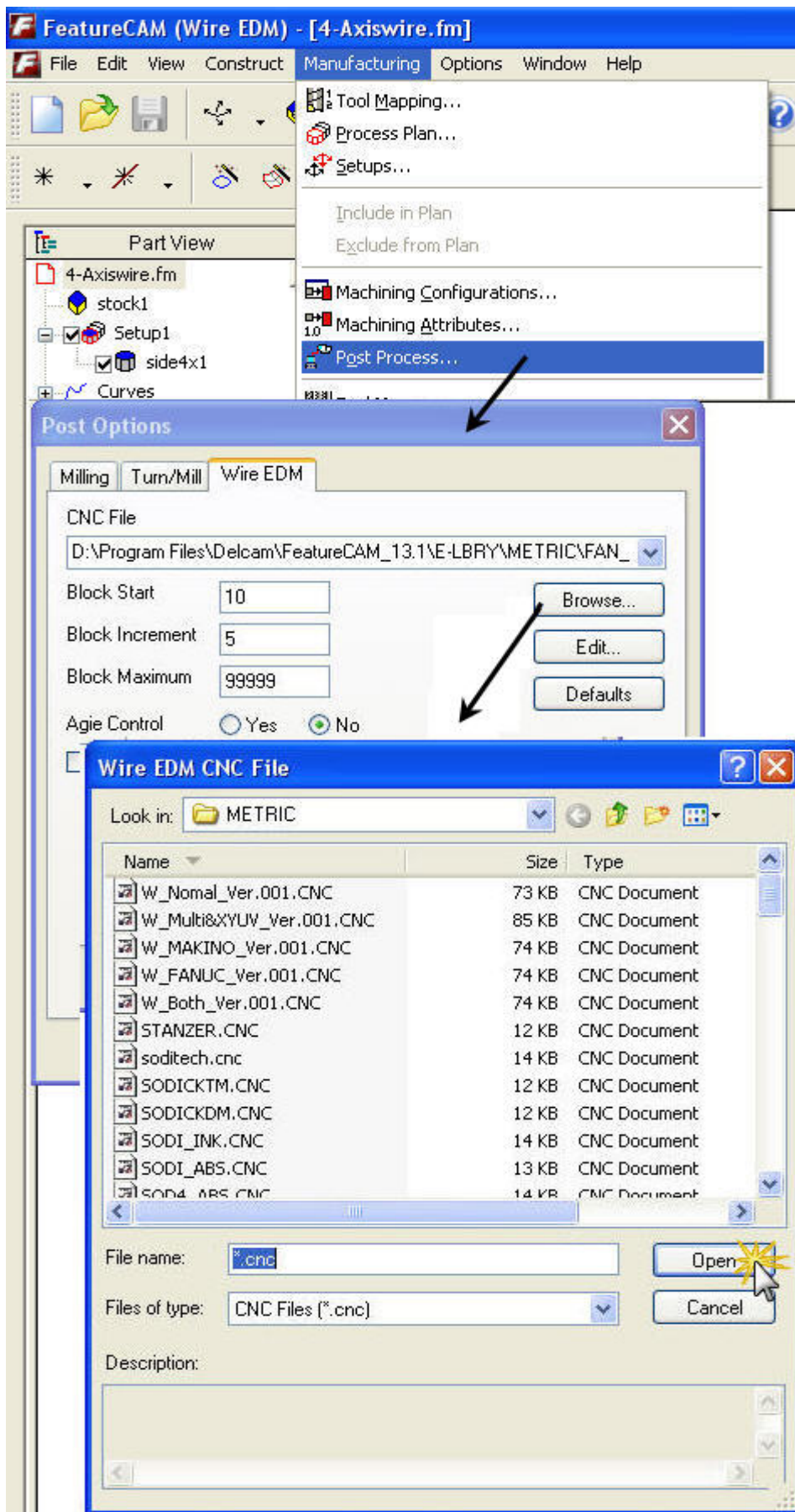
Once you run the **simulation** Successfully, **NC-Codes** will be generated **automatically**.

Click on the **NC Code** Tab as shown below.




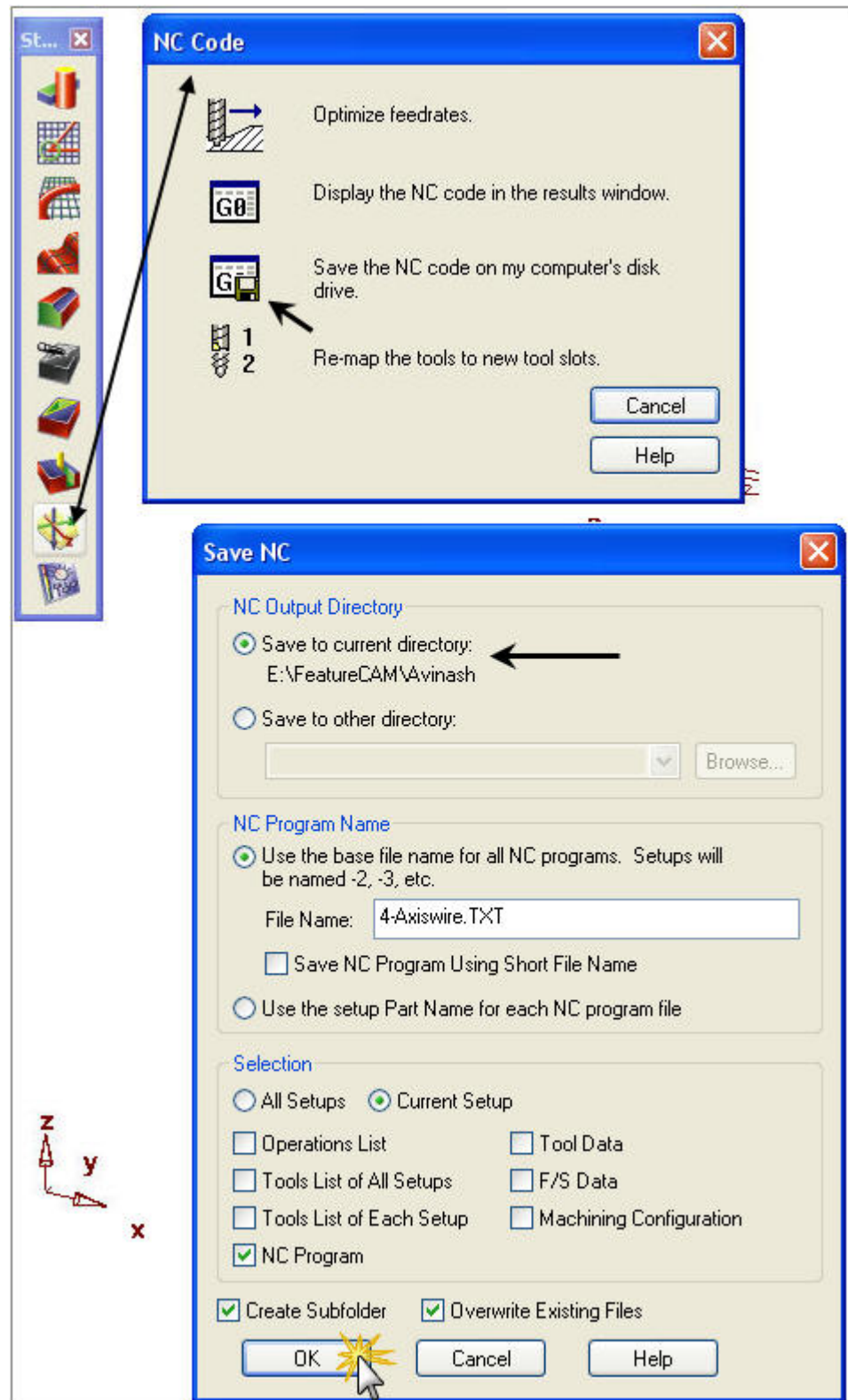
Post Processor Selection

- Open Manufacturing from the main menu and Select the **Required Post processor** as shown.



Save NC

- **Open NC Code** window from **Nc code Icon**  and select Save the NC code on my computer's disk drive.
- **Click OK.**



Demo: Quick 2-axis

- Open demo2x.fm.
- Bring up the New Feature Wizard.
- "We have die, punch and side features which corresponds to the types of parts people make with wire EDM."
- Switch between die, punch and side.
- Pick die, click Next, and then click Curve Chaining.
- Chain the geometry by clicking on the bottom left horizontal line.
- Click Next and Finish.
- Rotate the part around.
- Go to isometric view, run 3D sim, show that you can remove the core, and then rotate the part around.
- Post NC code.



Demo: Set Start Hole

- Go to the top view.
- Bring up the Feature Properties dialog.
- Go to the Start page.
- Pick (5.75, 3.25) and then the bottom right horizontal line.



Demo: Constant Taper

- Go to the Dimensions page.
- Set taper to constant left.
- Go to isometric view.
- Set taper to constant right.
- "We also support ISO or cylindrical taper vs. the regular or conical taper."
- Go to top view.
- Set taper to constant ISO right.
- Zoom in to the bottom arc to show that the taper is cylindrical (same radius).
- Switch back to constant right to show the difference.



Demo: Variable Taper

- Zoom out to show the part again.
- Set taper to variable.
- In the Variable Taper dialog, set everything to (right, 10, 10).
- "Let me first set everything to have the same setting as constant taper right 10 degrees. As you can see the shape didn't change."
- Uncheck Auto Fill Down.
- Pick the arcs and change them to ISO.
- "You can also vary the taper angle."
- Change Angle to 20 on the next line.
- "If you don't want a in-place taper angle change."
- Change First Angle to 20 on the arc after the line.



Demo Round/ISO

- Open RoundedISO.fm
- Create a new Die feature
- Make a constant 10 degree right taper
- Try the different rounded and taper strategies



Demo: Skim Cuts

- Set taper to none.
- Go to the Strategy page.
- Point at Contour Passes.
- "The number of passes is currently set to 1 so you will see only 1 pass in the simulation."
- Isometric view.
- Lower sim speed.
- 3D sim.
- Change Contour Passes to 3.
- 3D sim.
- "Right now we are using cutter comp as the offset method so the toolpath for the 3 passes are not offset. The NC code will have different cutter comps for each pass for the wire EDM machine to do the offsetting. I will talk more about this later. Another way of making skim cuts is to actually generate offset toolpath..."
- Change to Offset Toolpath.
- Top view.
- Centerline sim.
- Zoom to show offset toolpath.
- Center all.



Demo: Power Settings

- Change back to Cutter Comp.
- Click on the operation to show the Cutting Data page.
- "As you can see the feed, water and comp reg are different for each pass. The comp reg is for the offsetting that we just talked about. These numbers are actually register numbers that refer to specific settings on the wire EDM machine. For example, the 415 feed register number refers to a specific electrical power setting on the machine."
- Show the feeds and comp regs in the NC code.
- "If you want different numbers, you can change them in..."
- Show changing a number in the Cutting Data page.
- "The defaults comes from a table in the Cutting Data Tables dialog. To find the table, we use your stock material, wire and machine combination. You can see your combination on..."
- Open stock dialog.
- Point to the material, wire and machine area.
- "You can change your combination by..."
- Click Condition.
- "Here is the Cutting Data Tables dialog. Here is a table for 2 passes, 3 passes, and a table for Carbide."
- Click Cutting Data.
- Change passes to 2 and 3.
- Change material to carbide.



Demo: Retract and Cutoff Ops

- Set operations to Retract+Cutoff.
- Isometric view.
- "Let me run the simulation to the end of the Retract operation."
- 3D sim to Next Operation.
- "This operation could be run overnight. When it is done, the machine would stop. The uncut region keeps the core in place. If the core were cut loose, it would have fell and interfered with the wire movement. Also, the retract move makes sure the burn mark from the wire changing cut direction is away from the part. So in the morning, the operator would put magnets or glues to hold the core in place, and then he would continue the cutting with the Cutoff operation."
- 3D sim and remove core.
- "Pocketing and zigzag are two other useful operations. They are no-core operations meaning that the entire core is burned away."
- Top view.
- Create a die with a small circle.
- Set operations to pocketing.
- Centerline sim.
- Change operations to zigzag.
- Centerline sim.



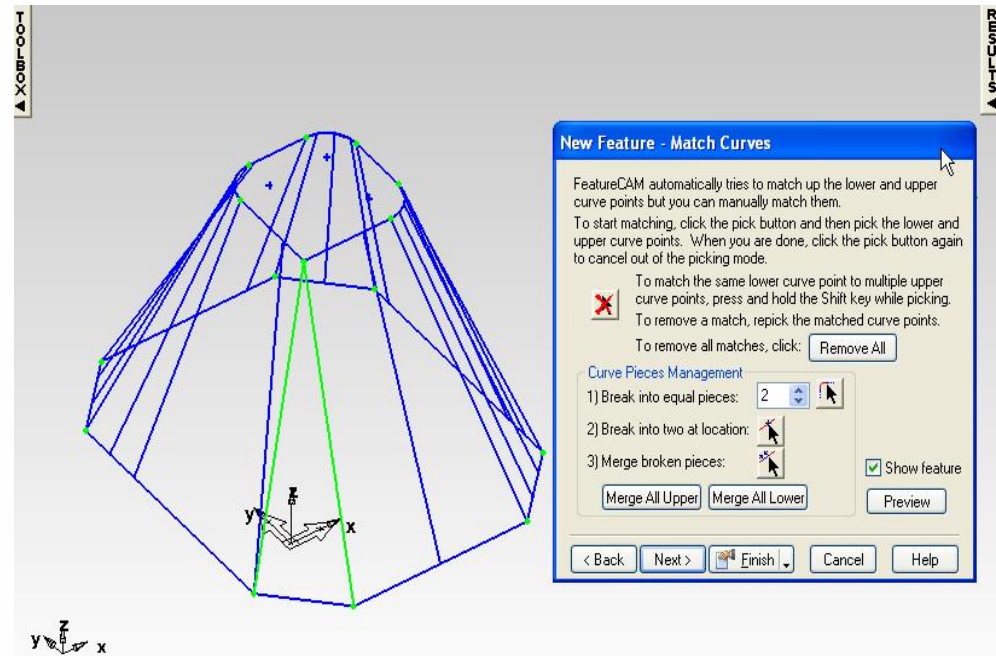
Demo: 4-axis Wire

- Open 4x1.fm.
- Bring up the New Feature Wizard.
- "Again, we have die, punch and side features just like in 2-axis."
- Pick die.
- Use the inner geometry for the upper curve and the outer for the lower.
- Set thickness to 2.
- "The feature is all twisted right now because we have not set the start points."
- Set start points to the middle.
- Finish.
- Rotate the part around.
- "Since there are the same number of geometry pieces on the upper and lower curves, the part's shape came out nicely. "



Curve Matching

- FeatureCAM attempts to match up the points of the upper and lower curves of a 4-axis feature.
- If the curves are not matched as you would like, then you can manually match the two curves.



Demo: 4-axis Matching

- Open 4x2.fm.
- Bring up the New Feature Wizard.
- Pick die.
- Use the inner geometry for the upper curve and the outer for the lower.
- Set thickness to 2.
- Set start points to the middle.
- "Since there are different number of geometry pieces on the upper and lower curves, FeatureWIRE does a length-based synchronization of the two curves, which is not what I want for this part. To control the synchronization, I will go to the next page..."
- Next.
- Match the two lower left points to the upper left point.
- Finish.
- Rotate the part around.



Advanced Demos

- 3D curves in 4-axis
- Posting for older Agie machines
- Posting for newer Agie machines
- UDF



Demo: 3D Curve in 4-axis

- This is useful when you get a model from someone and the surfaces that you want to cut do not give you planar curves.
- Open 3d.fm.
- Shade the surface and rotate it around.
- Unshade the surface
- Bring up the New Feature Wizard.
- Make a 4-axis side with curve1 as upper and curve2 as lower.
- Set the thickness to the distance between the curves.
- Finish.
- Shade the surface.
- Rotate the part around to show that it matches with the surface.



Demo: Posting for Older Agie Machines

- For older Agie machines that take NC code with full toolpath.
- Those machines have special requirements for cutter comp.
- Open agie.fm.
- Rotate it around.
- Change post to "sodickdi.cnc". Not using Agie posts since the the XY coordinates in those posts are incremental.
- Generate NC code.
- Show that the XY match with the part.
- Open the Post Options dialog.
- Set Agie Control to Yes and click OK to repost.
- The first two moves are swapped.
- G40 in the second move.
- A dummy perpendicular move is added before the last move.
- Change Agie Control back to No.



Demo: Posting for Newer Agie Machines

- For newer Agie machines that take ISO, SBR, SBL files.
- ISO file for the shape of the part.
- SBR and SBL files for the script that controls the cutting.
- Open agie2.fm.
- The operation has to be 'Contour' with 1 pass.
- Change the post to "agie-die.cnc".
- Open the Add-ins dialog and add PostToAgie.bas.
- Open the Run Basic Macro dialog and run PostToAgie.
- Show that the XY match with the part.
- A dialog pops up showing what files are generated.
- You can customize the agie-die.cnc.



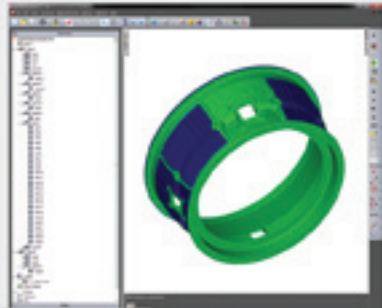
Demo: UDF

- UDF stands for User Defined Feature.
- The idea is to be able to insert custom NC code.
- Open a new wire part.
- Change the post to "sodickdi.cnc".
- Open the post in Xbuild.
- The UDF Text format has to be filled in.
- Open the Add-ins dialog and add "Code-Comment operation.bas".
- Bring up the New Feature Wizard.
- Create the UDF.
- Add some code and comment.
- Generate NC code.
- If you have other features, you can move the UDF around.
- You can create your own UDF (e.g. probe.bas)



FeatureCAM 2012

FeatureCAM 2012



new features

Delcam TV



www.delcam.tv

FeatureCAM Learning Zone



www.delcam.tv/lz

FeatureCAM Website



www.featurecam.com



THE QUEEN'S AWARDS
FOR ENTERPRISE:
INNOVATION
2010



THE QUEEN'S AWARDS
FOR ENTERPRISE:
INTERNATIONAL TRADE
2011

Powering your productivity

Delcam Headquarters | Small Heath Business Park | Birmingham | B10 0HJ | UK
+44 (0)121 766 5544 | marketing@delcam.com | www.delcam.com

To contact your local reseller, visit www.delcam.com/resellers