Mastercam Wire Tutorial

May 2016

Mastercam® 2017

Mastercam® 2017 Mastercam Wire Tutorial

Date: May 2016

Software: Mastercam 2017

TERMS OF USE

Be sure you have the latest information!

Information might have been changed or added since this document was published. The latest version of this document is installed with Mastercam or can be obtained from your local Reseller. A ReadMe file (`ReadMe.pdf`)—installed with each release—includes the latest information about Mastercam features and enhancements.

Contents

Introduction

Mastercam Wire delivers comprehensive wire EDM software with powerful toolpaths and techniques. This tutorial provides an introduction to Mastercam Wire workflow and best practices.

Tutorial Goals

- Practice creating and selecting part geometry for your toolpaths.
- Enter appropriate toolpath parameters for different toolpath types.
- Make changes to your toolpath and update the tool motion.

Verify your toolpath motion using several different tools.

Estimated time to complete this tutorial: 2.5 hours

General Tutorial Requirements

All Mastercam tutorials have the following general requirements:

- You must be comfortable using the Windows® operating system.
- The tutorials cannot be used with Mastercam Demo/Home Learning Edition (HLE). The Demo/HLE file format (`emcam`) is different from the Mastercam file format (`mcam`), and basic Mastercam functions, such as file conversions and posting, are unavailable.
- Each lesson in the tutorial builds on the mastery of preceding lesson's skills. We recommend that you complete them in order.
- Additional files may accompany a tutorial. Unless the tutorial provides specific instructions on where to place these files, store them in a folder that can be accessed from the Mastercam workstation, either with the tutorial or in any location that you prefer.
- You will need an internet connection to view videos that are referenced in the tutorials. All videos can be found on our YouTube channel: www.youtube.com/user/MastercamTechDocs.
- All Mastercam tutorials require you to configure Mastercam to work in a default metric or English configuration. The tutorial provides instructions for loading the appropriate configuration file.

LESSON 1
Single Contour Toolpaths

A wire contour toolpath machines parts that have the same shape at the top and bottom. This lesson focuses on the basic workflow used to create a wire contour toolpath.

Lesson Goals

- Open a part file and assigning a machine definition.
- Create a contour toolpath with a single contour.
- Backplot to check the toolpath.
- Post the toolpath.

Exercise 1: Getting Started with Toolpath Creation

In this exercise, you open a part and select a machine definition that simulates a wire EDM machine.

1 Start Mastercam using your preferred method:

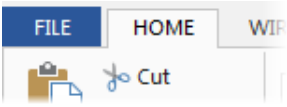
- Double-click Mastercam's desktop icon.

  Or

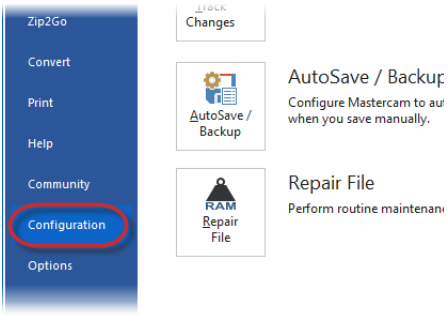
- Launch Mastercam from the Windows Start menu.

2 Select the default metric configuration file:

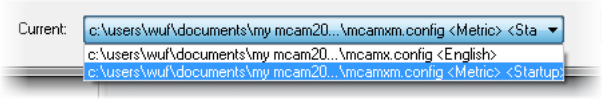
a Click the **File** tab.



b Choose **Configuration** from Mastercam's Backstage View to open the System Configuration dialog box.

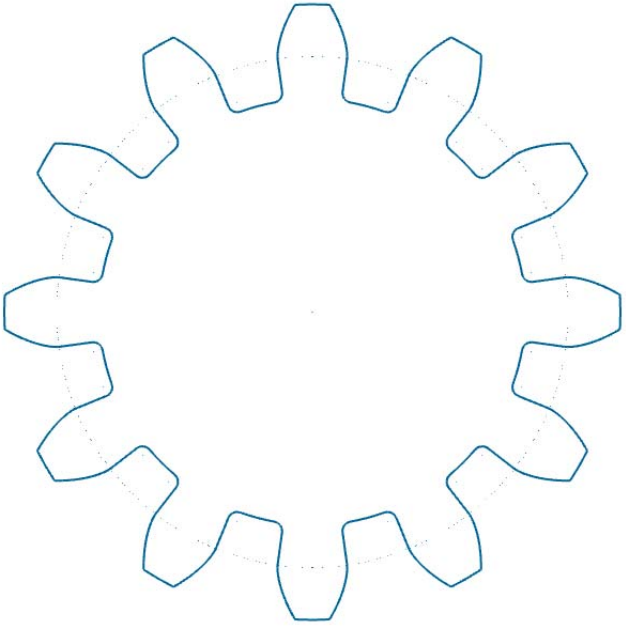


c Choose **...\mcamxm.config <Metric>** from the **Current** drop-down list.



d Click **OK**.

3 Choose **File, Open** and select the part file, `single_contour.mcam` provided with the tutorial.



The gear shape is the single contour for this toolpath. Mastercam Wire also needs a *thread point* for the toolpath, which is the point where the machine threads the wire, often a pre-drilled hole in the material. By creating a thread point in Mastercam and selecting it as part of the toolpath, it becomes *associative*, which means the toolpath updates if the point moves.

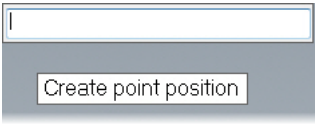
4 On the **Wireframe** tab, choose **Point Position**.

5 In the Point Position function panel, choose **Thread Point**.

The thread point symbol may be hard to see depending on your background color. Before you begin to identify the thread point, zoom into the area if you need to see it more clearly.

6 Drag and drop the graphics window prompt and press [**Spacebar**] to access FastPoint mode and type in the point coordinates.

Because the prompt window hides anything under it, you may not want to move it over information displayed in Toolpaths Manager, or any used sidebar areas.



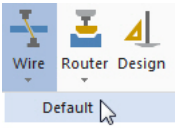
7 Type **`X-40,Y0.2,Z0`** and press [**Enter**]. A point with the thread symbol displays to the left of the part.

8 Click **OK** in the Point Position function panel to complete the function.

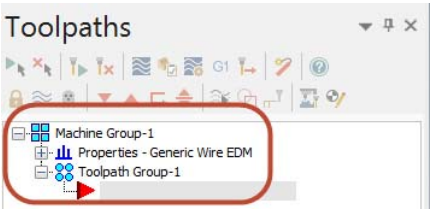
You now have the geometry you need for a basic contour toolpath.

9 On the Machine tab, choose **Wire, Default** to select the default wire machine definition.

The *machine definition* is a model of your machine tool's capabilities and features. It provides a template for setting up machining jobs.



Choosing the machine definition adds a machine group and a toolpath group to the Toolpaths Manager. Toolpaths you create are placed at the red arrow in this list.

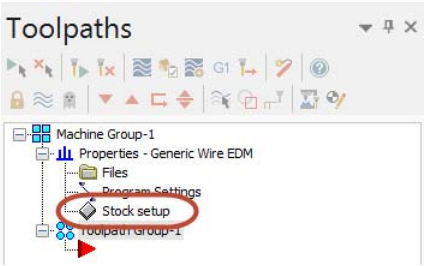


10 Choose **File, Save As** and save the part under a different file name. This protects the original tutorial file from being overwritten.

Exercise 2: Setting Up Stock

Defining a stock model helps you visualize your toolpaths more realistically. You can see your stock boundaries with the part geometry when you backplot or verify toolpath operations.

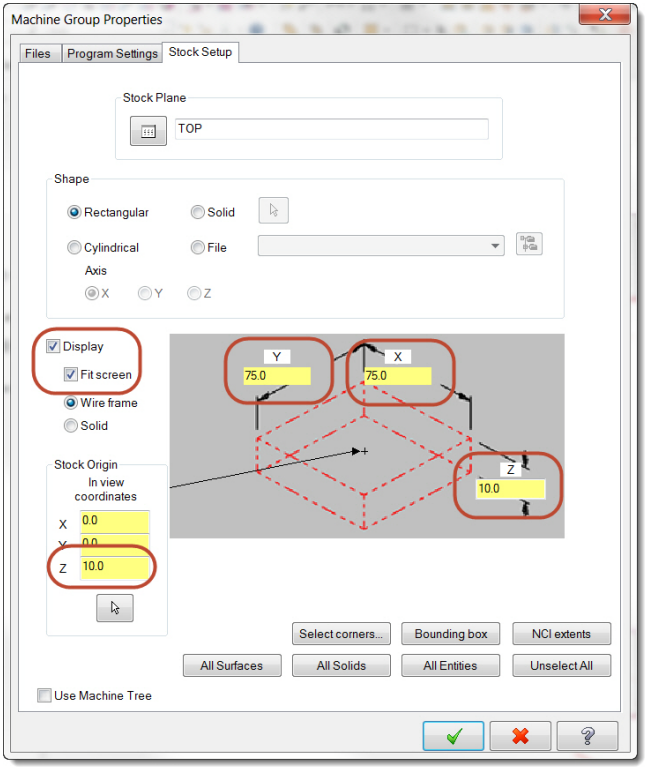
1 In the Toolpaths Manager, expand the Properties under the Machine Group, and click **Stock setup**.



2 Enter the following values to set the stock boundaries:

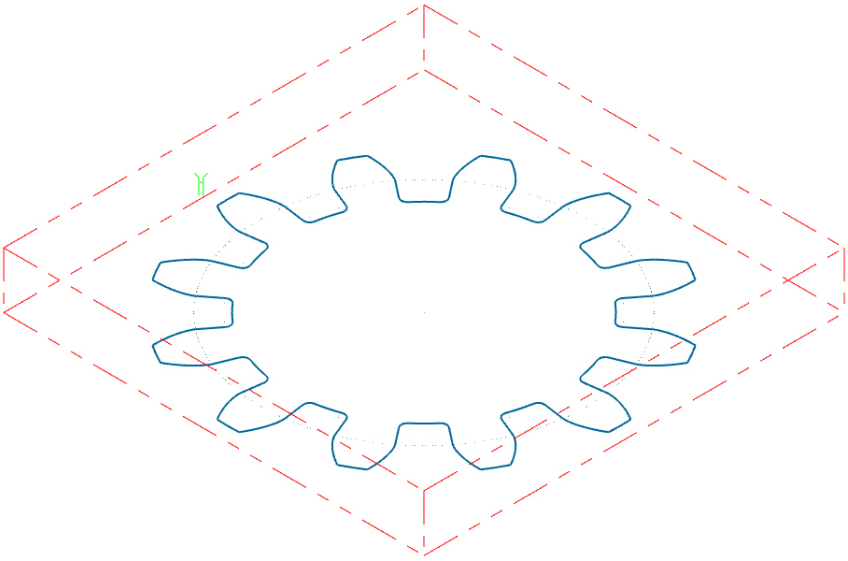
- Enter **Y75**, **X75**, and **Z10** for the stock dimensions.
- Enter a **Z** value of **10** for the stock origin.

3 Select **Display** and **Fit screen** to show the stock boundaries and include the boundaries when you use the **Fit to Screen** function.



4 Click **OK** to complete the stock setup.

5 Right-click in the graphics window and choose **Isometric (WCS)** to get a clearer view of the stock boundaries.

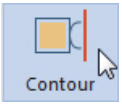


Exercise 3: Selecting Geometry

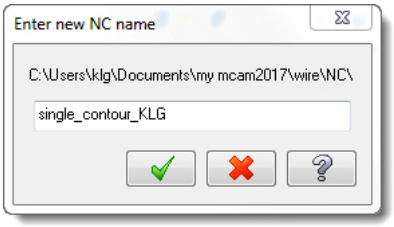
Selecting geometry for toolpaths in Mastercam Wire is called *chaining*. Chaining is the process of selecting and linking pieces of geometry to form the foundation of a toolpath, surface, or solid. When you chain geometry, you select one or more sets of curves (lines, arcs, and splines) that have adjoining endpoints. You can also chain points, which is important for setting thread and cut positions in your toolpath.

In Mastercam Wire, you can chain either wireframe or solid geometry for your toolpaths. This example involves chaining wireframe geometry.

1 On the Wirepaths tab, choose **Contour**.



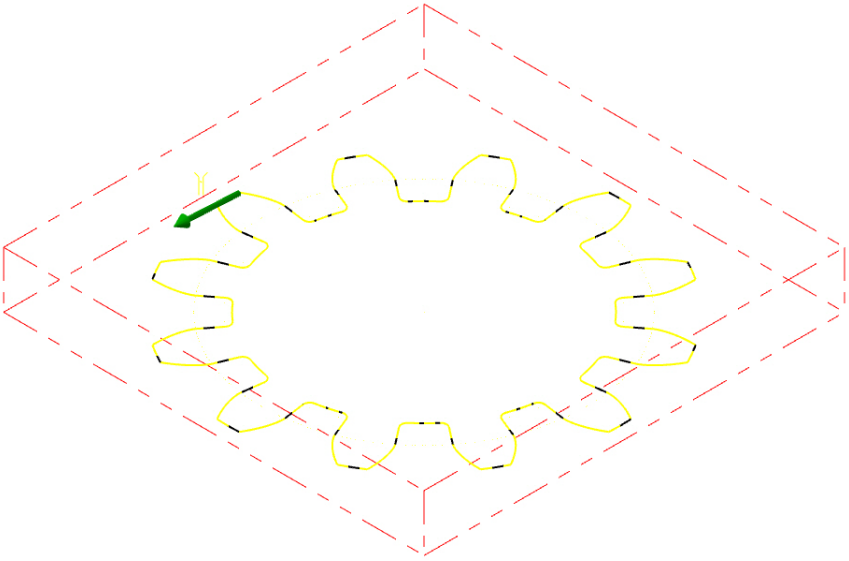
2 Click **OK** if prompted to enter a new NC file name.



3 To associate the point with the toolpath, select the thread point you created as the first chain.

NOTE: When the stock size is close to the part size, the thread point is often outside the stock.

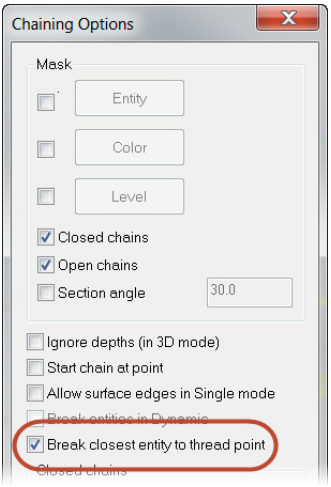
4 Select the gear shape as the second chain. Click on the area closest to the thread point. The green arrow showing chain direction should go in a counterclockwise direction.



5 Click **Options** on the Chaining dialog box to access additional chaining parameters.

6 Select **Break closest entity to thread point**.

This option breaks the entity closest to the thread point into two pieces so you begin the toolpath with a perpendicular move. This creates the shortest motion possible between the thread point and the chained geometry.



7 Click **OK** on the Chaining Options dialog box.

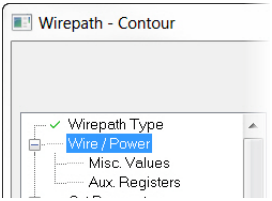
8 Click **OK** on the Chaining dialog box to continue to set the wirepath parameters in the Wirepath - Contour dialog box.

Exercise 4: Entering Wirepath Parameters

The next step is to set the values for each aspect of the toolpath. The Wirepath - Contour dialog box includes the options you need to program your part.

1 Choose **Wire/Power** from the Tree View pane on the left side of the dialog box.

A *power library* contains wire EDM machine-specific settings for the material you are cutting.



TIP: A power library can contain up to 24 passes. A *pass* is a single path made by the wire around a contour. Each pass includes unique power settings needed to cut a certain material type on a certain wire EDM machine. For example, Pass 1 in the library may correspond to a rough cut, Pass 2 a tab cut, and Passes 3 - 5 finish cuts (also known as *skim cuts*).

2 Enter **0.2** for the wire diameter.

This toolpath will only have one pass, so you do not need to make other changes.

3 Choose **Compensation** from the Tree View pane.

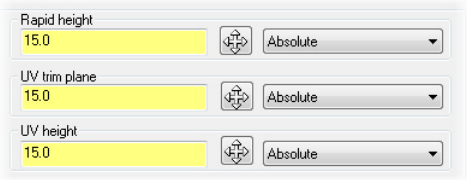
4 Set the compensation type to **Computer**.

This option computes the compensated wirepath and does not output control codes for compensation. This allows you to see a more accurate representation of the toolpath motion.

5 Choose **Taper** from the Tree View pane.

This page sets various toolpath heights, including the UV and XY heights that indicate the top and bottom of the stock.

6 Set the **Rapid height**, **UV trim plane**, and **UV height** to **15.0**. The *rapid height* is the Z height of the upper wire guide for rapid moves. The *UV trim plane* sets the position of the upper wire guide.

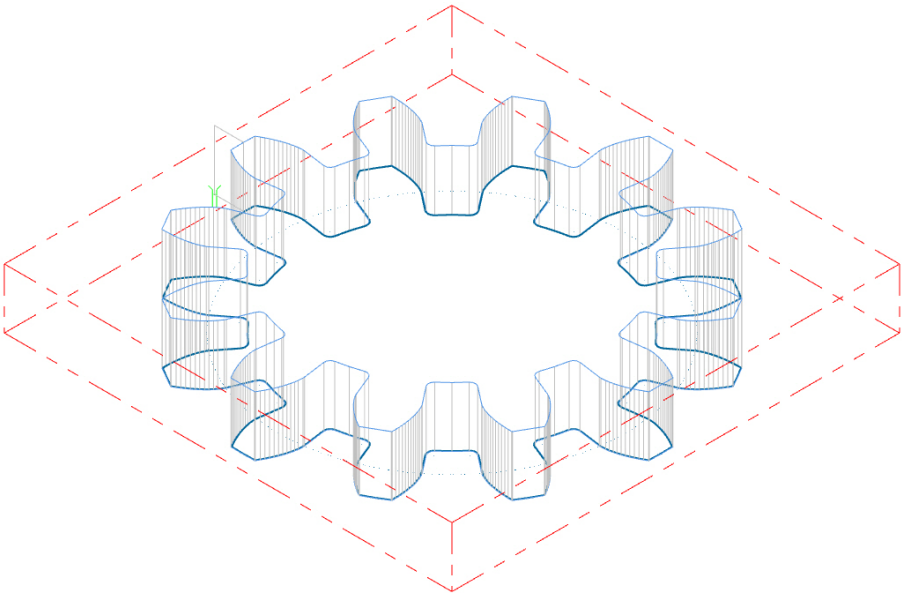


7 Click **OK** to complete the toolpath. 

8 Click **OK** to close the Chain Manager.

The Chain Manager displays for access to the Change at point or sorting options. These options are discussed in Lesson 2.

The following picture shows the completed toolpath.

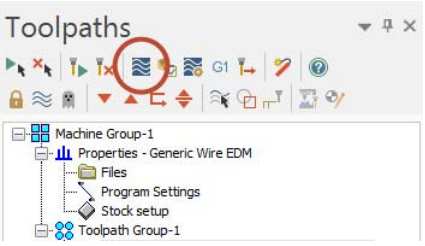


9 Choose **File, Save** from the menu to save your progress.

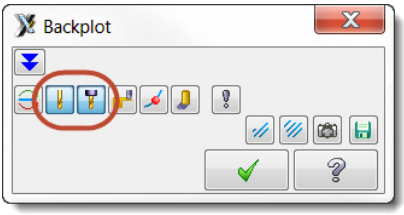
Exercise 5: Backplotting the Toolpath

Mastercam's Backplot shows the path the wire takes to cut your part. This display lets you spot errors in the program before you machine the part.

1 Click **Backplot selected operations** at the top of the Toolpaths Manager.

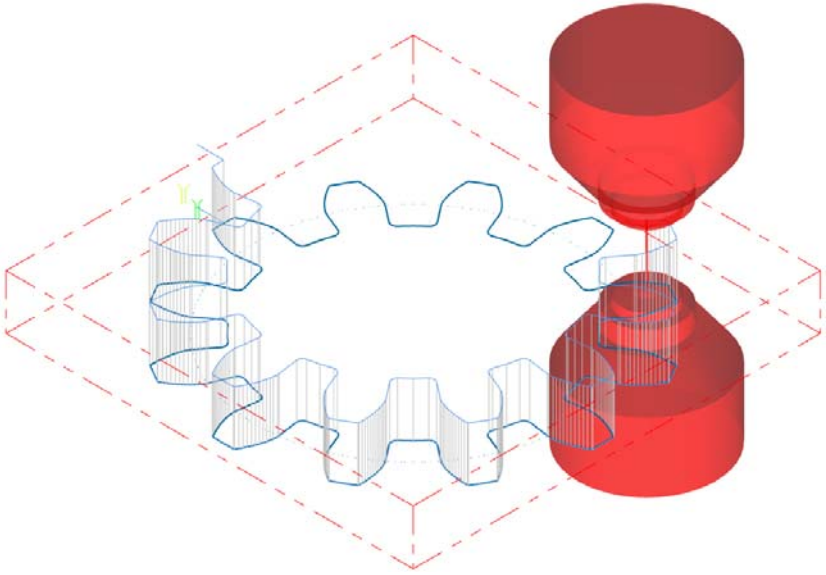


2 Click **Display tool** and **Display holder** on the Backplot dialog box to show the wire and guides during backplot.



3 Click **Play** on the Backplot VCR bar above the graphics window to run the backplot.

The wire moves around the part for a single rough pass.

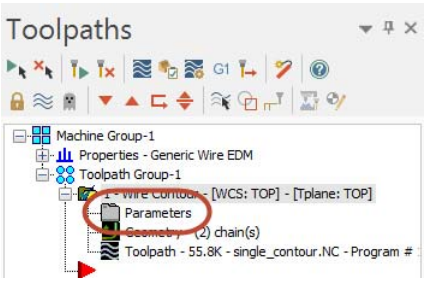


4 Click **OK** on the Backplot dialog box to exit the function.

Exercise 6: Changing Toolpath Parameters

Although the simple contour toolpath you just created will machine the part, additional options in Mastercam Wire can make your machining more effective.Change parameters and update the operation to include these new settings.

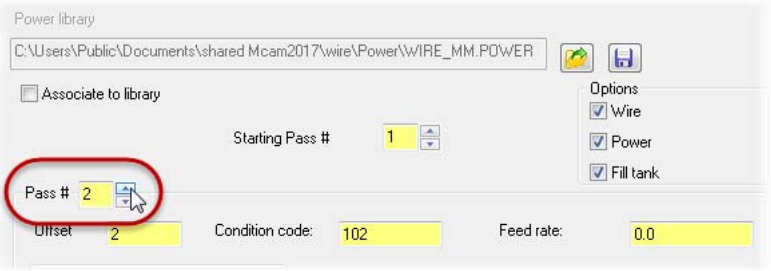
1 Click the **Parameters** icon in the Toolpaths Manager for the Wire Contour operation.



2 Choose **Wire/Power** from the Tree View pane.

3 For Pass 1, enter **0.035** for the **Wire Overburn**.

4 Click the up arrow next to Pass 1 to enter values for Pass 2.



5 Enter the following values for Passes 2, 3, and 4:

- Pass 2: Wire Diameter **0.2**, Wire Overburn **0.02**
- Pass 3: Wire Diameter **0.2**, Wire Overburn **0.01**
- Pass 4: Wire Diameter **0.2**, Wire Overburn **0**

TIP: As the passes get closer to the final part shape, the wire cuts less material and the overburn decreases.

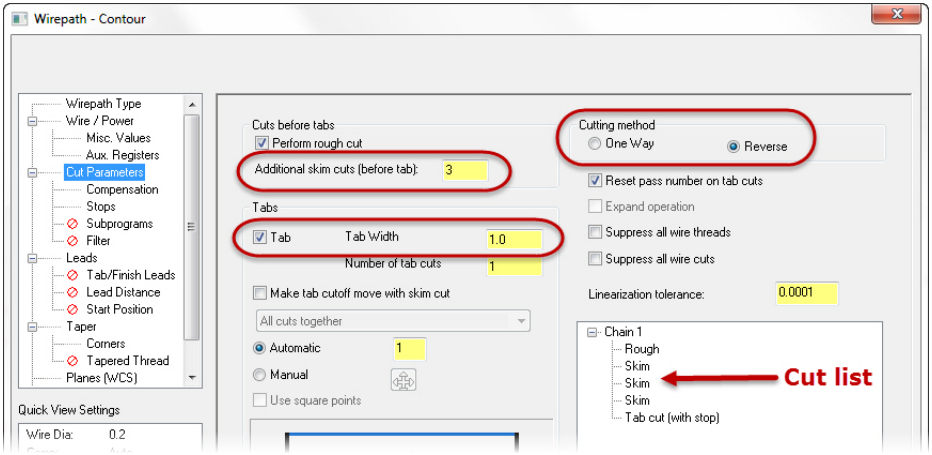
6 Choose **Cut Parameters** from the Tree View pane.

7 Set the following parameters on this page (see following graphic of new Cut Parameters values):

- Enter **3** for **Additional skim cuts (before tab)**. These additional passes create a better finish on the part.

- Select **Tab** and enter **1.0** for the **Tab Width**. Creating a tab on the part keeps it attached to the stock.
- Set the cutting method to **Reverse**. Instead of cutting in one direction and re-threading the wire, this option makes the wire go in the opposite direction at the end of each pass.

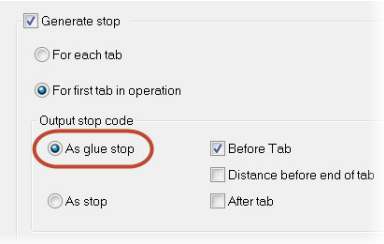
Notice the change in the cut list. The toolpath motion now includes four passes around the chain and a short move to cut the tab.



8 Choose **Stops** from the Tree View pane.

Because you added a tab cut with a stop on the Cut Parameters page by selecting the **Tab** option and setting the **Tab Width**, some of the options on the Stops page are already selected.

9 Select **As glue stop** for the output stop code.

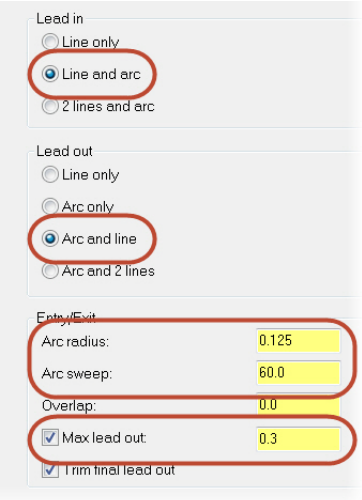


A *glue stop* is an optional stop (using an M01 code) that pauses the machine before the tab cut, and allows the operator to secure the part to prevent dropout after the tab cut.

10 Choose **Leads** from the Tree View pane.

11 Select **Line and arc** for the lead in and **Arc and line** for the lead out.

Starting and ending the toolpath away from the part reduces the possibility of leaving a burr on the part.



12 Enter **0.125** for the arc radius and **60** for the arc sweep on the entry and exit moves.

13 Select **Max lead out** and set the distance to **0.3**.

This option shortens the lead out move, instead of forcing the wire to travel from the end of the contour to the cut point.

14 Click **OK** to complete the parameter changes.

15 Click **Regenerate all selected operations** in the Toolpaths Manager to update the contour toolpath with your changes.

16 Choose **File, Save** to save your part.

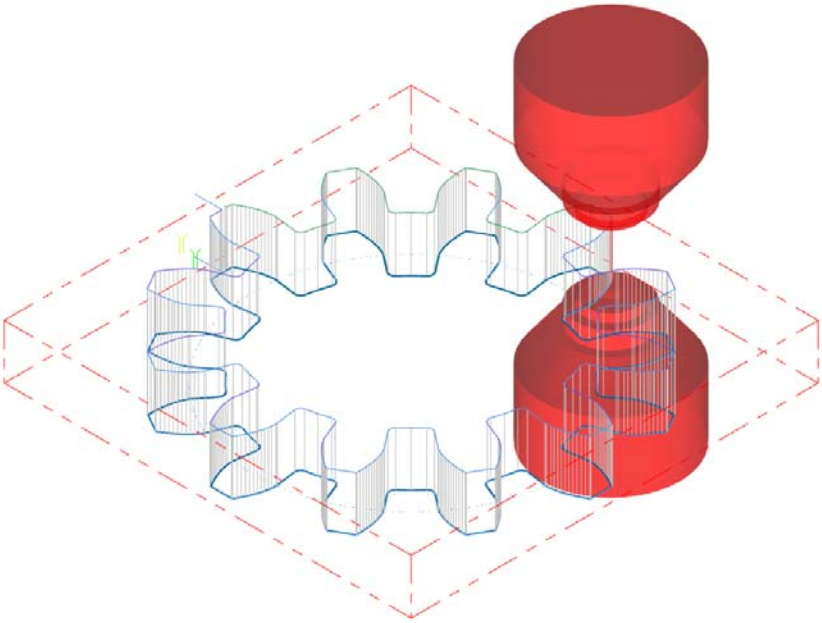
Exercise 7: Backplotting the Updated Toolpath

Use Backplot to verify the changes you made to the contour toolpath.

1 Click **Backplot selected operations**.

2 Click **Play** to run the backplot.

The wire moves to the end of the contour, leaves a tab, travels back and forth around the part during the skim cuts, then cuts the tab.



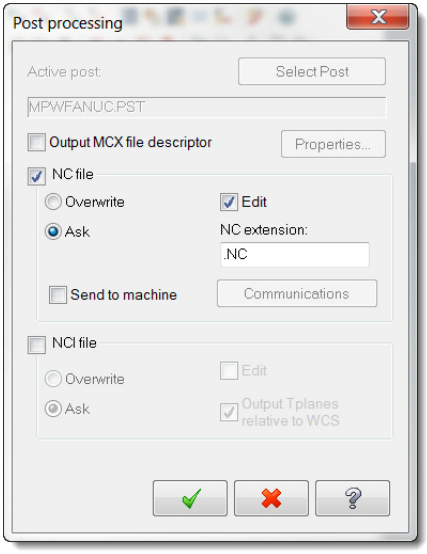
3 Click **OK** to complete the backplot.

Exercise 8: Posting the Toolpath

Post processing, or *posting*, refers to the process by which the toolpaths in your Mastercam part files are converted to a format that can be understood by your machine tool's control (for example, G-codes). A special program called a *post processor*, or *post*, reads your Mastercam file and writes the appropriate NC code. Generally, every machine tool or control will require its own post processor, customized to produce code formatted to meet its exact requirements.

1 Click **Post selected operations** at the top of the Toolpaths Manager.

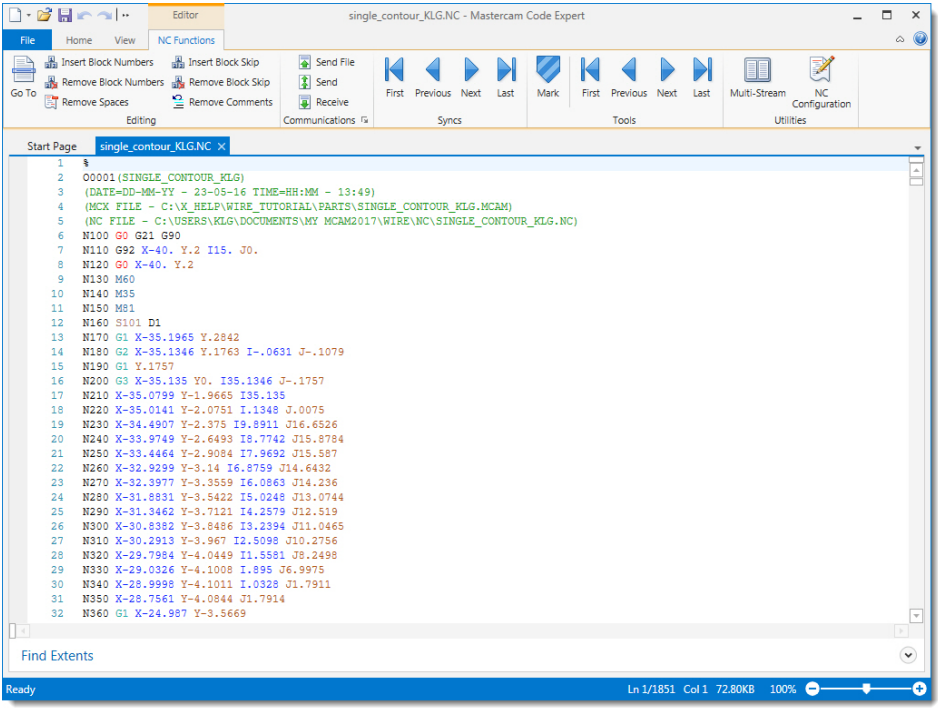
2 Set the post processing parameters as shown. These settings will ask if you want to save the NC file and will display the resulting file in your default text editor.



3 Click **OK**.

4 Select a location for the NC file and click **Save**.

5 Mastercam Code Expert opens in Editor mode and displays the posted NC code. Scroll through the code and see if it meets your expectations.



6 Edit the file and save if necessary.

7 Close the Code Expert window.

Congratulations on completing your first Wire toolpath! The next lesson involves multiple contours.

LESSON 2
Multiple Contour Wirepaths

In addition to single contour wirepaths, Mastercam Wire allows you to cut multiple contours from a single piece of stock in a single operation. You can select the parts as a group and then sort the chains to generate the toolpath motion you want.

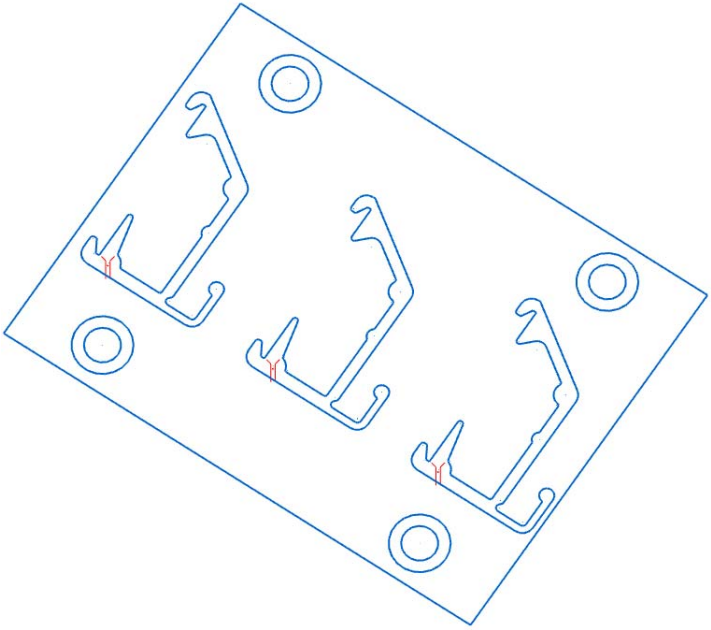
Lesson Goals

- Select. multiple contours.
- Create multiple tabs and stops.
- Use verification to check the toolpath.

Exercise 1: Preparing the Part

1 Choose **File, Open** and select the part file, `multiple_contours.mcam` provided with the tutorial.

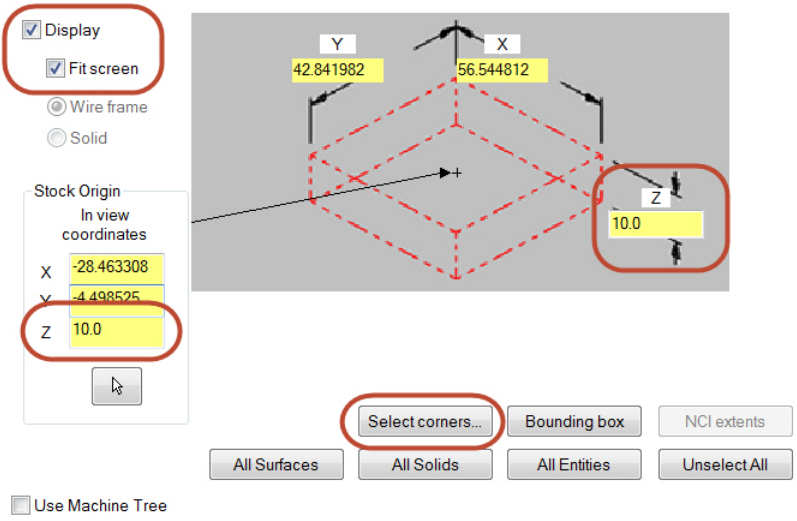
This part includes three contours, three thread points, and four mounting holes. The default Wire machine definition is already selected.



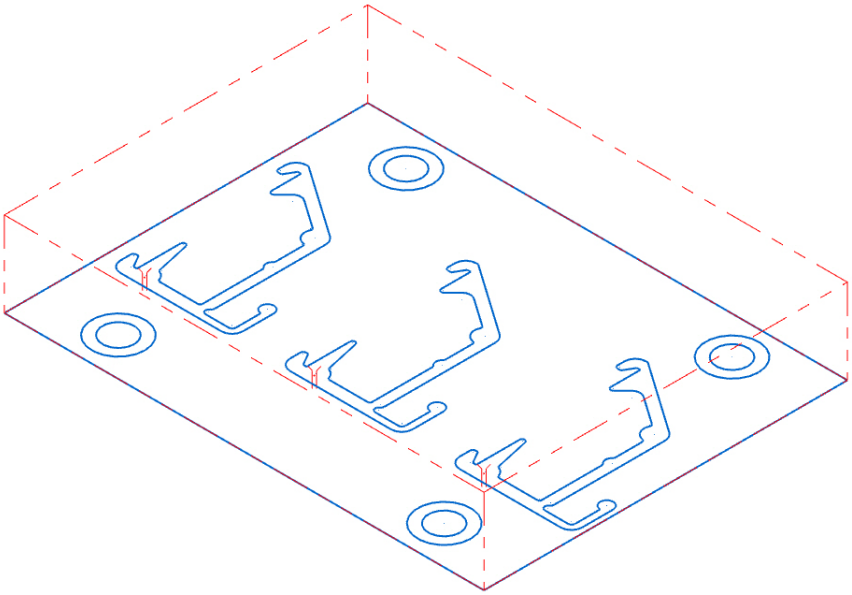
2 In the Toolpaths Manager, expand the Properties under the Machine Group and click **Stock setup**.

3 Enter the following values to set the stock boundaries and display them in the graphics window:

- Click **Select corners** and select opposite corners of the rectangle surrounding the contours.
- Enter **10** for the Z stock dimension.
- Change the Stock Origin Z coordinate to **10**.
- Select **Display** and **Fit screen** to show the stock boundaries and include the boundaries when you use the **Fit to Screen** function.



4 Click **OK** to complete the stock setup. If necessary, right-click in the graphics window and select **Isometric (WCS)**.



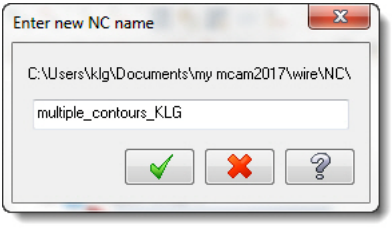
5 Choose **File, Save As** and save the part under a different file name. This protects the original tutorial file from being overwritten.

Exercise 2: Selecting Geometry

Instead of chaining each contour, you can quickly select multiple parts using window chaining.

1 On the Wirepaths tab, choose **Contour**.

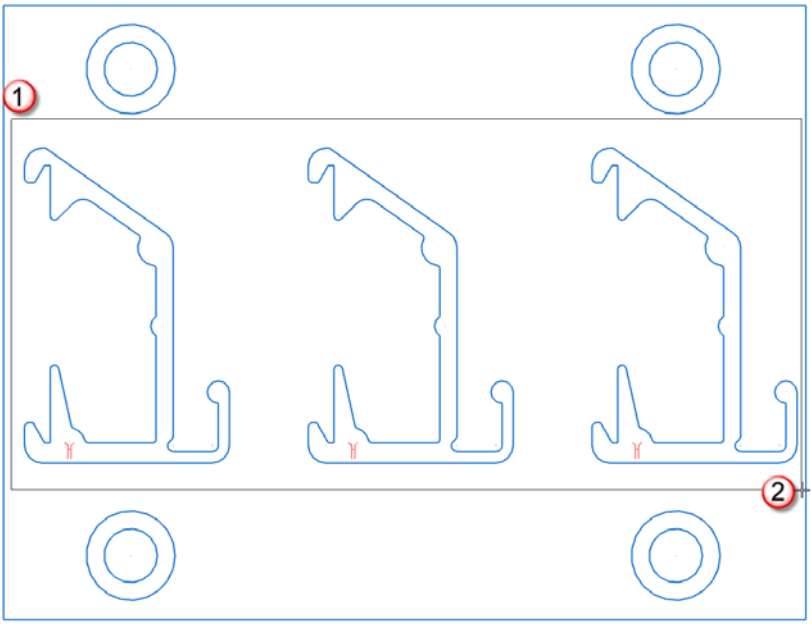
2 Click **OK** if prompted to enter a new NC file name.



3 Right-click in the graphics window and choose **Top (WCS)** to view the part from above.

4 Click **Window** on the Chaining dialog box. 

5 Click at the first point and then drag to the second point shown in the following picture and click again to draw the window around the parts.



6 Click the thread point on the left part as the approximate start point.

7 Click **OK** on the Chaining dialog box to continue to the toolpath parameters.

Exercise 3: Entering Toolpath Parameters

The parameters for this toolpath are similar to the ones you entered in Lesson 1.

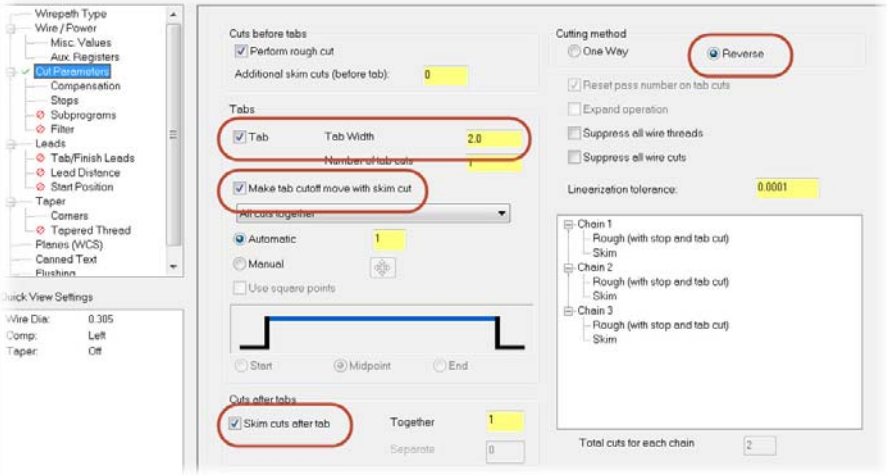
1 Select **Cut Parameters** in the Tree View pane.

2 Set the following parameters on this page:

- Select **Tab** and enter **2.0** for the tab width. Creating a tab on the part keeps it from dropping out of the stock.
- Select **Make tab cutoff move with skim cut** to make the toolpath motion more efficient.
- Select **Skim cuts after tab** to add a finish pass after cutting the tab.

- Set the cutting method to **Reverse**.

The toolpath motion now involves two cuts for each chain, with the rough cut including the stop and the tab cut.

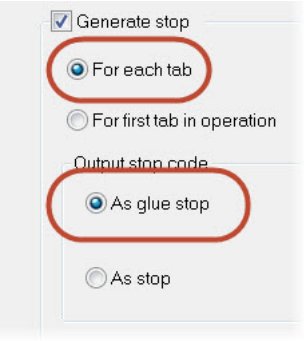


3 Choose **Compensation** from the Tree View pane.

4 Select **Computer** for the compensation type, and set the compensation direction to **Left**.

5 Choose **Stops** from the Tree View pane.

6 Select **For each tab** to add a stop code before the tab cut on each chain and **As glue stop** to add them as optional stops.

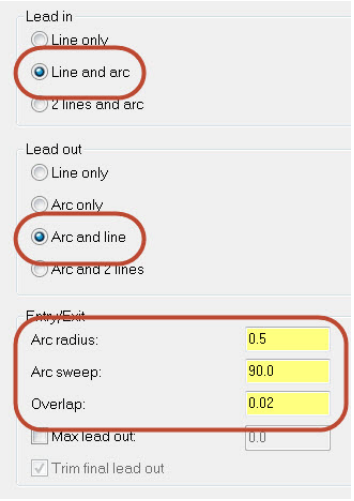


7 Choose **Leads** from the Tree View pane.

8 Set the following parameters on this page:

- Select **Line and arc** for the lead in and **Arc and line** for the lead out.
- Enter **0.5** for the arc radius and **90** for the arc sweep on the entry and exit moves.
- Enter **0.02** for the overlap.

  This option eliminates potential burrs by overlapping the start and end of the contour by the entered amount.



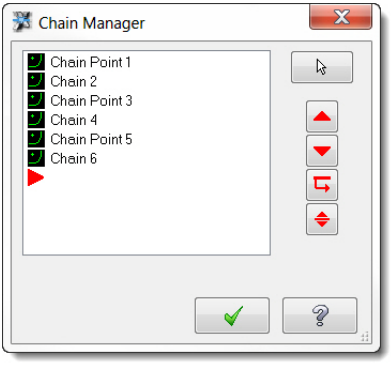
9 Choose **Taper** from the Tree View pane and set the **Rapid height**, **UV trim plane**, and **UV height** to **20.0**.

10 Click **OK** to finish entering the parameters. The Chain Manager opens.

Exercise 4: Adjusting the Chains

Because you selected the contours with window chaining, the chain order may not be what you would prefer for the toolpath. The Chain Manager displays after you enter the toolpath parameters so you can adjust the chains and get the toolpath motion you want.

1 Click each **Chain Point** and **Chain** in the Chain Manager to see how Mastercam chained the contours by default.



2 Right-click in the Chain Manager and choose **Sort options**. The Sorting dialog box displays.

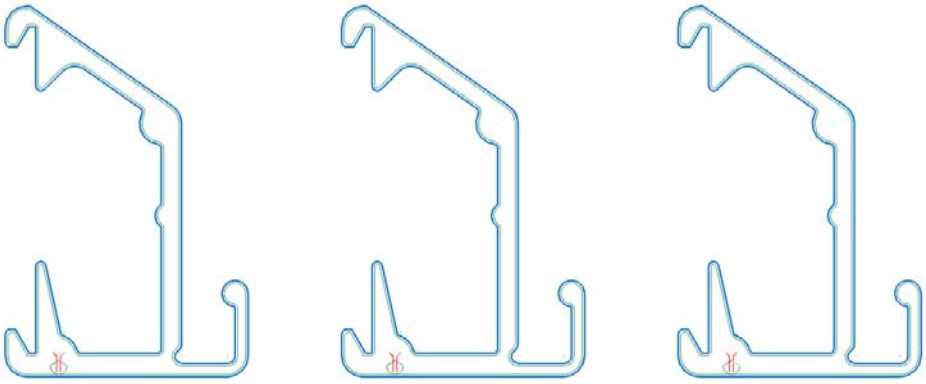
3 Select the **Y+ X-** option in the third row.

The red point indicates the start point and the arrow shows the sorting direction.

4 Click **OK** to close the Sorting dialog box.

5 Click through the points and chains again to see the new chaining order from right to left.

6 Click **OK** to close the Chain Manager and the toolpath displays on the three contours.



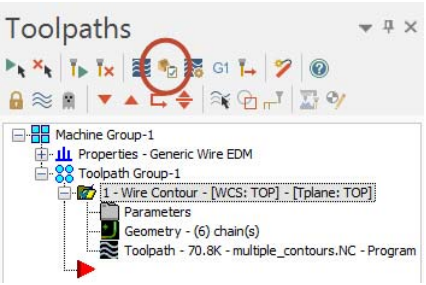
7 Save your part file.

Exercise 5: Verifying the Toolpath

Verification is different from backplot because it simulates material removal in addition to tool motion. The verification displays in a separate window called Mastercam Simulator.

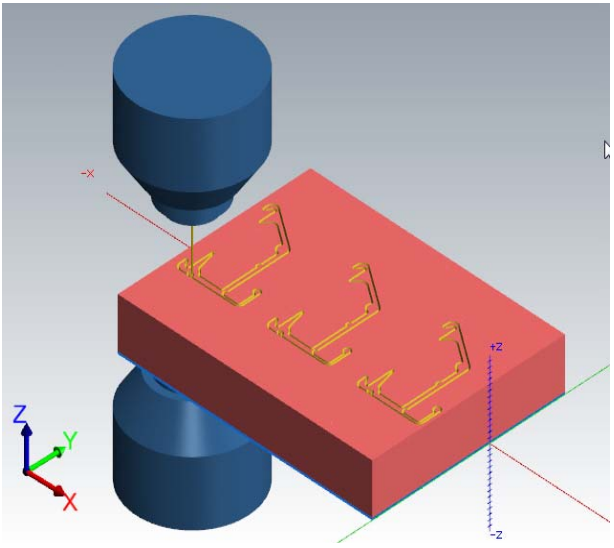
1 Click **Verify selected operations** at the top of the Toolpaths Manager.

The stock, wire, and guides display in the Mastercam Simulator window.



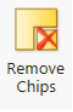
2 Right-click in the Simulator window and choose **Isometric**, then **Fit** to get a better view of the toolpath and wire.

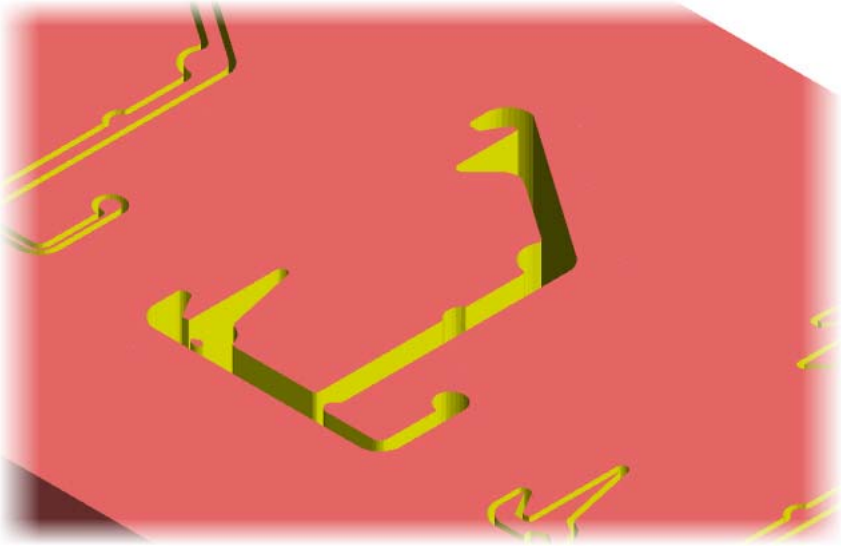
3 Click **Play** at the bottom of the Simulator window to move through the verification.



4 After the verification is complete, right-click and choose **Zoom Window**. Draw a window around the center part to look at the results more closely.

5 Click **Remove Chips** on the Verify tab and then click on the middle part to see the slug drop out of the stock.





You can click on the other slugs as well to see the final tool motion results.

6 Close the Simulator window to return to Mastercam.

Now that you have worked with single contours and multiple contours, the next lesson focuses on No Core toolpaths.

LESSON 3
No Core Toolpaths

No Core toolpaths in Mastercam Wire remove all material within a boundary without producing slivers or slugs. The wirepath typically starts at a pre-drilled hole in the material, and zigzags or spirals outward until all material within the chained geometry is removed.

Lesson Goals

- Use AutoCursor to select points.
- Create rough and finish passes.
- Use backplot to check the toolpath results.

Exercise 1: Preparing the Part

1 Choose **File, Open** and select the part file, `no_core.mcam`, provided with the tutorial.

This part is a solid model with a hole in the center. The No Core toolpath will machine the four additional slots. The default wire machine definition is already selected, but you need to define a thread point for each slot.

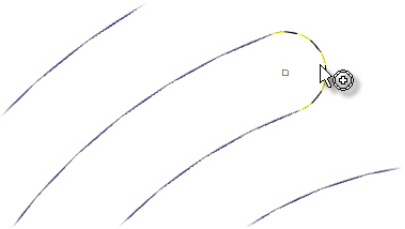
2 Press [**Alt+S**] to turn off shading, and view the wireframe geometry.

3 Right-click in the graphics window, and choose **Top (WCS)** to rotate the view.

4 On the **Wireframe** tab, choose **Point Position**.

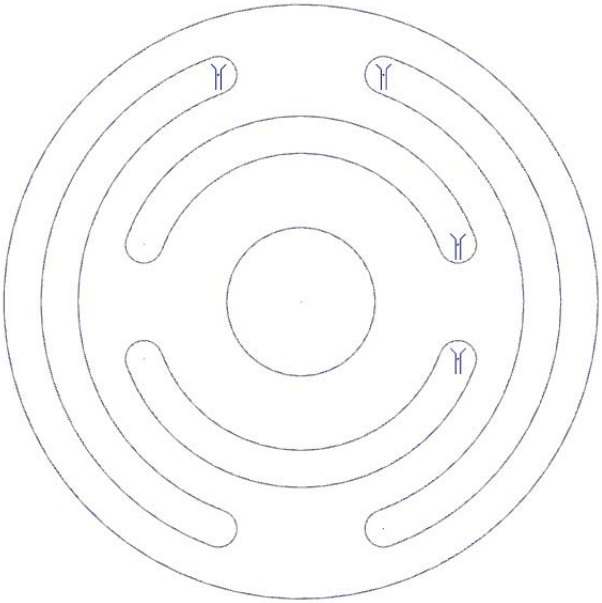
5 In the Point Position function panel, choose **Thread Point**.

6 Move your cursor to the end of a slot and notice the cursor change to indicate that you are selecting the center of an arc.



TIP: This is Mastercam's *AutoCursor*, which detects and snaps to points as you move the cursor over geometry. AutoCursor activates whenever Mastercam prompts you to select a position on the screen.

7 Select the four points shown in the following picture as thread points.



TIP: Placing the points close together reduces travel time between the slots.

8 Click **OK** in the Point Position function panel to complete the function.

9 Choose **File, Save As** and save the part under a different file name.

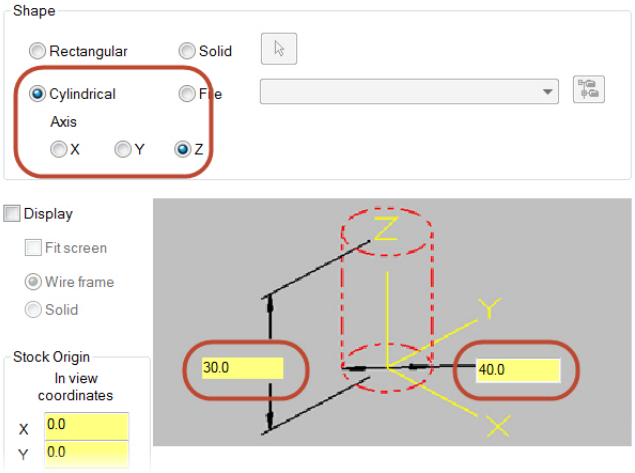
Exercise 2: Setting Up Stock

Mastercam Wire can model stock based on many different shapes, including solid models, castings, and cylinders.

1 In the Toolpaths Manager, expand the Properties under the Machine Group and click **Stock setup**.

2 Select **Cylindrical** in the Shape section and select **Z** as the cylinder axis.

3 Enter **30** for the cylinder height and **40** for the diameter.



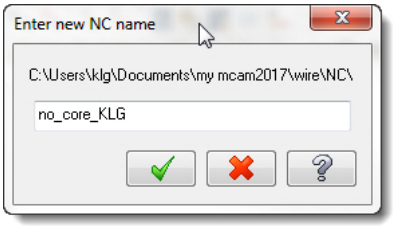
4 Click **OK** to complete the stock setup.

Exercise 3: Selecting Geometry

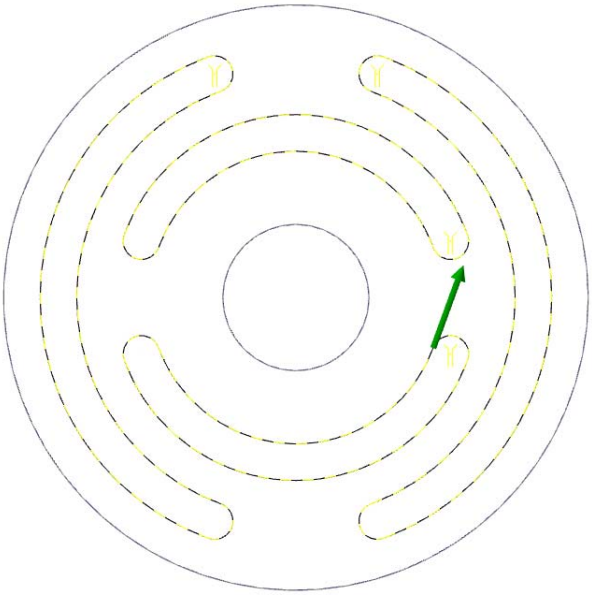
All chains (except thread points) must be a closed shape for No Core toolpaths.

1 On the Wirepaths tab, choose **No Core**.

2 Click **OK** if prompted to enter a new NC file name.



3 Select each thread point and its corresponding slot. Select the chain for each slot close to its thread point.



NOTES:

- If you were actually machining the part, each thread point would be the center of a pre-drilled hole for the wire.
- In the Chaining Options dialog box, make sure that **Break closest entity to thread point** is unchecked.

4 Click **OK** on the Chaining dialog box to complete chaining.

Exercise 4: Entering Toolpath Parameters

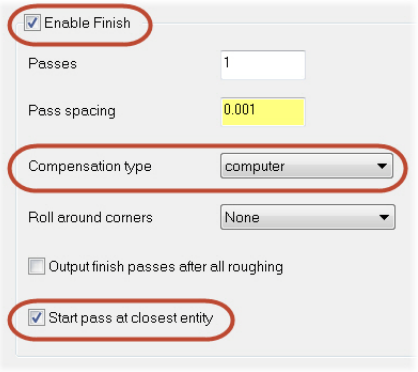
No Core toolpaths can include a rough pass and finish passes. The rough pass typically removes almost all of the material. The finish passes smooth out rough edges and can also take off additional material.

1. Select **No Core** in the Tree View pane.
2. Set the **Rapid height**, **UV trim plane**, and **UV height** to **40.0**.
3. Select **Rough** in the Tree View pane.

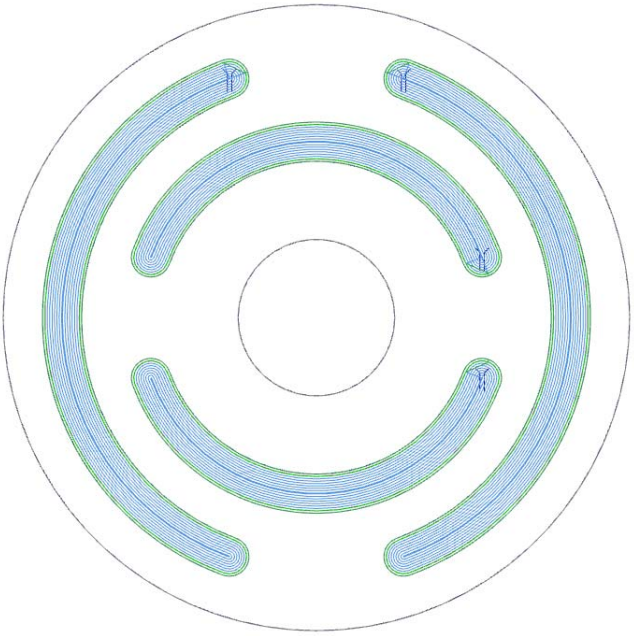
   Mastercam No Core wirepath offers six cutting methods for cleaning out the material. The best practice is to select a cutting method that follows the shape of your part. For this part, the default **Parallel Spiral** option works well.
4. Select **Finish** in the Tree View pane.
5. Select **Enable Finish** to activate the default finish pass parameters.
6. Set the Compensation type to **computer** for the finish pass.
7. Select **Start pass at closest entity**.

   When checked, begins the finish pass starting with the closest endpoint of the closest entity at the end of the roughing wirepath. When cleared, the finish pass begins with the first entity in the chain as it was originally selected.

   This option makes your toolpath motion more efficient.



8 Click **OK** to generate the toolpath.



9 Save your part file.

Exercise 5: Backplotting the Toolpath

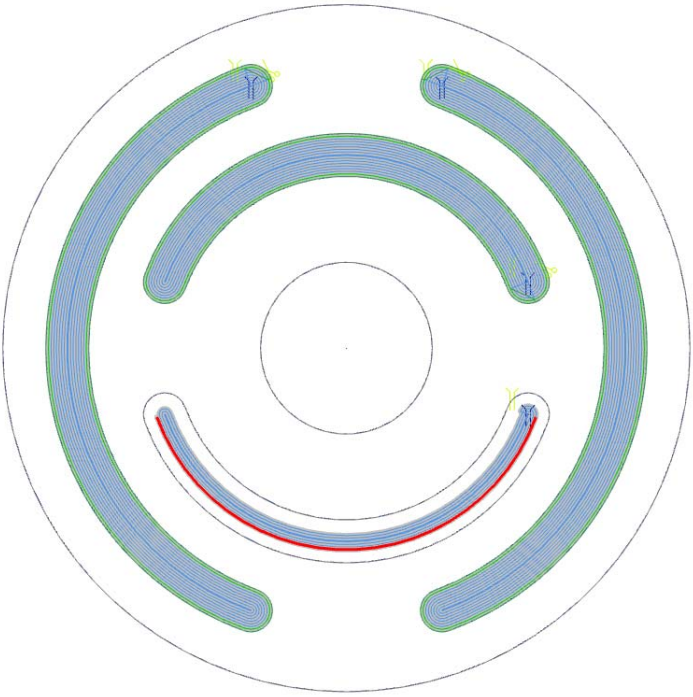
Backplot includes several tools for showing your wire motion.

1 Click **Backplot selected operations** at the top of the Toolpaths Manager.

2 Select **Quick verify** from the Backplot dialog box. 

This option displays a shaded path the width of the wire as the backplot progresses and gives you a quick way to check if all the material is removed from the slots.

3 Click **Play** to run the backplot. The wire cuts each slot from the inside out. The shaded area under the tool motion confirms that all the slots are machined completely.



4 Click **OK** to exit backplot.

The last lesson in this tutorial focuses on 4-axis toolpaths, which work well for parts with different contours in the top and bottom planes.

LESSON 4
4-axis Toolpaths

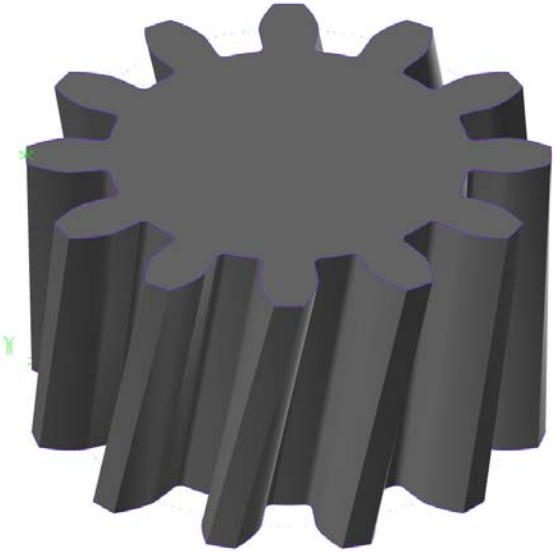
Use 4-axis toolpaths for parts that require the wire to be in a non-vertical orientation. These parts typically have different geometry in the XY and UV planes, or the same geometry in a different orientation, such as the gear used in this lesson.

Lesson Goals

- Select a different machine definition.
- Learn about synchronization methods.
- Use TECH libraries.

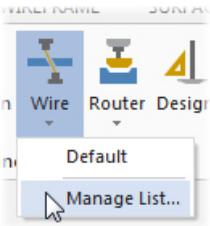
Exercise 1: Preparing the Part

1 Choose **File, Open** and select the part file, `4_axis.mcam`, provided with the tutorial. If the part is not shaded, press [**Alt +S**] to turn on shading.



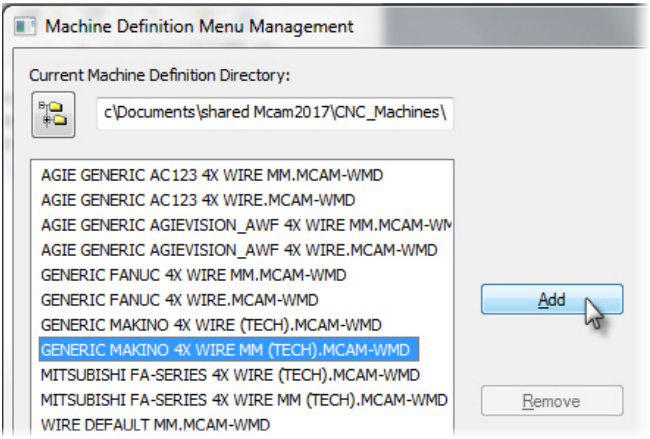
This part is a solid model with the thread point and chaining start points already defined.

2 On the Machine tab, choose **Wire, Manage List** to select a wire machine definition.



This option lets you select a different machine definition than the default Wire machine.

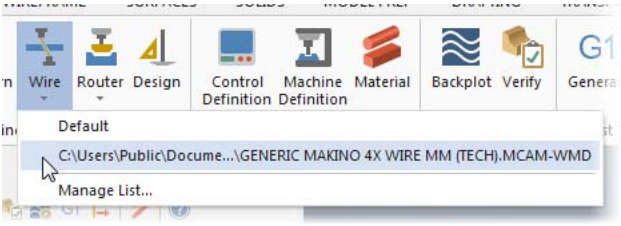
3 Select `GENERIC MAKINO 4X WIRE MM (Tech).MCAM-WMD` from the machine definition list, and click **Add**.



These wire machine definitions (`.MCAM-WMD` files) are installed with Mastercam Wire. If your Mastercam Wire license included another machine definition, it would display in this list. This Makino machine uses a `.TECH` library for the wire power settings, which you will work with in Exercise 4.

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

5 Choose **Wire** and select the machine you just added to the list.



The Makino machine definition is added to the Toolpaths Manager.

6 Choose **File, Save As** and save the part under a different file name.

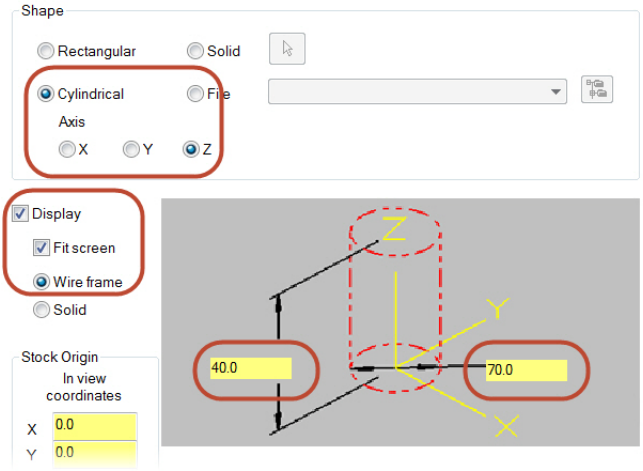
Exercise 2: Setting Up Stock

Similar to Lesson 3, a simple cylinder is the best stock model for this part.

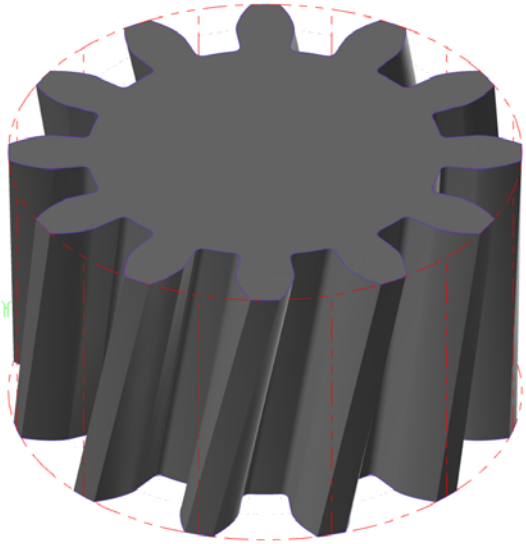
1 In the Toolpaths Manager, expand the Properties under the Makino Machine Group and click **Stock setup**.

2 Enter the following values in the dialog box:

- Select **Cylindrical** in the Shape section, and select **Z** as the cylinder axis.
- Enter **40** for the cylinder height and **70** for the diameter.
- Select **Display** and **Fit screen** to show the stock boundaries.



3 Click **OK** to complete the stock setup.



Exercise 3: Selecting Geometry

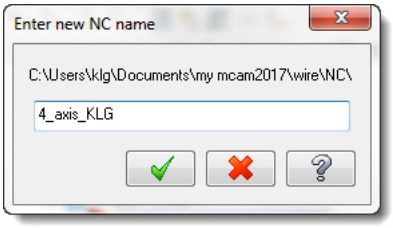
For a 4-axis toolpath, you chain geometry in two planes: the XY plane (lower contour), and UV plane (upper contour). Mastercam Wire uses synchronization to determine the wire motion between the two chains. The sync mode is typically determined by the part geometry, but Mastercam Wire offers several options:

- **By entity** - Matches the endpoint of each entity. Requires both chains to have the same number of entities.
- **By point** - Matches previously created point entities on each chain.
- **By node** - Matches two or more parametric splines by their node points. Requires each spline to have the same number of node points.
- **By branch** - Matches the contours by branch points. Requires 3D geometry connecting the upper and lower contours.
- **Manual** - Matches the chains of user-defined areas.
- **Manual/density** - Matches the chains and assigns a density for each chain. If an area has small radii, use a higher density for a better finish.

NOTE: A *branch point* is a position where the endpoints of three or more entities meet and where the chain direction can change.

1 In the Wirepaths tab, choose **4 Axis**.

2 Click **OK** if prompted to enter a new NC file name.

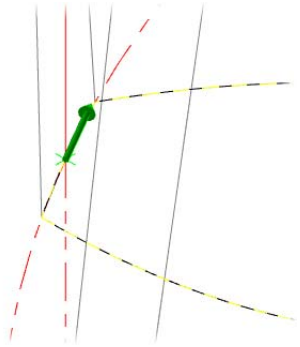


3 Press [**Alt+S**] to turn off shading and make it easier to chain the part.

4 Select the thread point to the left of the part as the first toolpath chain.

5 Select the lower purple contour to the right of the green point. The chain start arrow should be at the point.

This is the XY plane for the toolpath.



TIP: Press [**F1**] and draw a zoom window to focus on a certain area of the part.

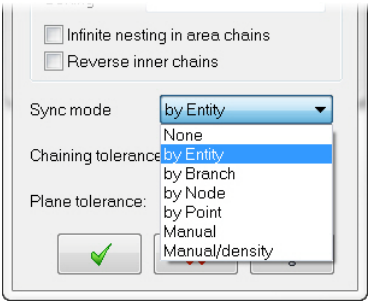
6 Chain the upper contour starting at the green point. Both chains must go in the same direction and should be clockwise around the part.

TIP: Click **Reverse** on the Chaining dialog box to switch the chain direction.

7 Click **Options** on the Chaining dialog box to access the synchronization methods.

8 Select **by Entity** from the Sync mode drop-down list.

This sync mode works for this part because the top and bottom contours have the same number of entities.



9 Click **OK** to close the Chaining Options dialog box.

10 Click **OK** again to close the Chaining dialog box and to open the Wirepath - 4 Axis dialog box.

Exercise 4: Entering Toolpath Parameters

The machine definition you selected is connected to a `.TECH` library that contains all the power information for the wire passes, including register settings, offsets, and feeds. These settings are designed for this wire machine and control.

1 Choose **Wire/Power** from the Tree View pane.

2 Click the **Tech** button to change the power settings for this toolpath.

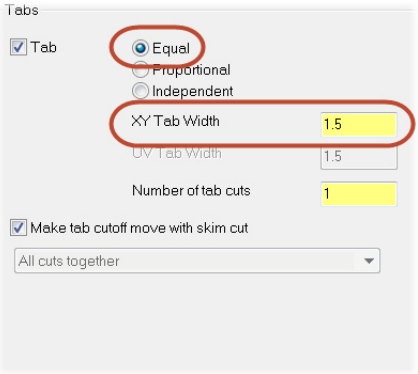
The Technology Database dialog box displays.

3 Select **Rough & 2 skim(s)** from the Sequence drop-down list at the bottom of the dialog box. This matches the number of passes you will program for this part.

4 Click **OK** to return to the toolpath parameters.

5 Choose **Cut Parameters** from the Tree View pane.

6 In the Tabs section, select **Equal** to make the UV and XY tabs the same size and enter **1.5** for the **XY Tab Width**.

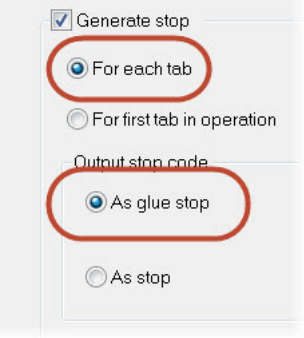


7 Choose **Compensation** from the Tree View pane.

8 Set the compensation type to **Computer** and set the compensation direction to **Left**.

9 Choose **Stops** from the Tree View pane.

10 Select **For each tab** to add a stop code before the first tab cut on each chain and **As glue stop** to add them as optional stops.

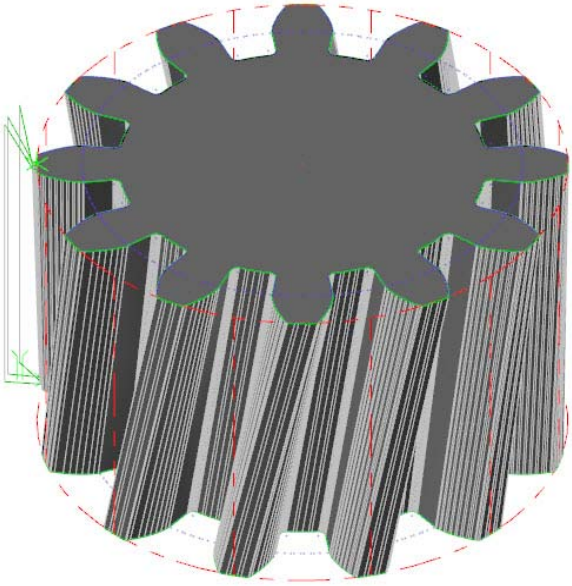


11 Choose **Leads** from the Tree View pane.

12 Select **Line and arc** for the lead in and **Arc and line** for the lead out.

13 Click **OK** to generate the toolpath.

14 Press [**Alt+S**] to view the shaded part with the toolpath.



15 Save your part file.

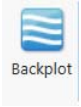
Exercise 5: Backplotting the Toolpath

Besides the backplot tools you've already used, you can backplot your tool motion in the same Mastercam Simulator window as verification.

1 Click **Verify selected operations** at the top of the Toolpaths Manager.

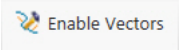
2 On the Home tab, check **Workpiece** to shade the solid.

3 Click **Backplot** to change to Backplot mode in Mastercam Simulator.



The stock no longer displays in the window and the Backplot tab displays at the top of the screen.

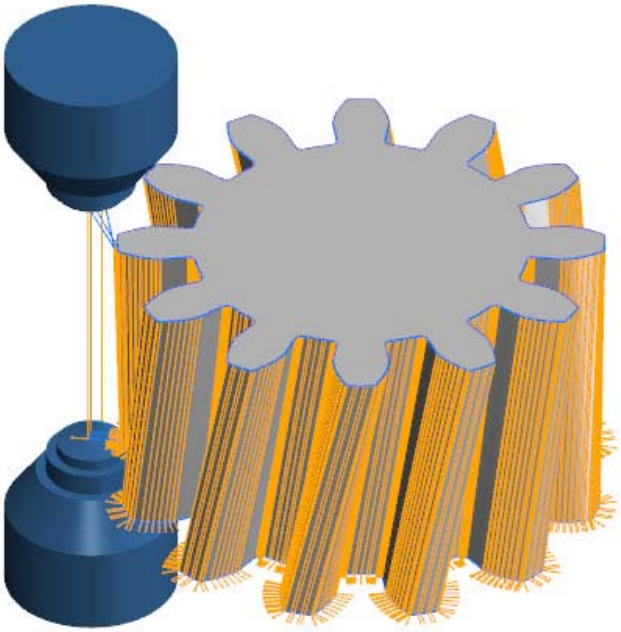
4 Click the Backplot tab and select **Enable Vectors** to view the 4-axis



wire positions as it moves around the part.

5 Click **Play** at the bottom of the Simulator window to run the backplot. The wire moves around the part three times for the one rough and two skim passes.

6 Click the Home tab, and select **Workpiece** to see the final part shape so you can compare it to the tool motion. The **Workpiece** checkbox toggles to view the part in different shades of translucency.



7 Close the Simulator window to return to Mastercam.

8 Save your part.

Conclusion

Congratulations! You have completed the *Mastercam Wire* tutorial. Now that you have mastered the skills in this tutorial, explore Mastercam's other features and functions.

You may be interested in other tutorials that we offer. The Mastercam tutorial series is in continual development, and we will add modules as we complete them. Visit our website, or select **Help, Tutorials** from the **File** tab to see the latest publications.

Mastercam Resources

Enhance your Mastercam experience by using the following resources:

- *Mastercam Help*—Access Mastercam Help by selecting **Help, Contents** from Mastercam's **File** tab or by pressing [**Alt+H**] on your keyboard. Also, most dialog boxes, function panels, and ribbon bars feature a Help button that opens Mastercam Help directly to related information.
- *Mastercam Reseller*—Your local Mastercam Reseller can help with most questions about Mastercam.
- *Technical Support*—CNC Software's Technical Support department (860-875-5006 or support@mastercam.com) is open Monday through Friday from 8:00 a.m. to 5:30 p.m. USA Eastern Standard Time.
- *Mastercam Tutorials*—CNC offers a series of tutorials to help registered users become familiar with basic Mastercam features and functions. The Mastercam tutorial series is in continual development, with new modules added as we complete them. Visit our website, or select **Tutorials** from the **Help** menu to see the latest publications.
- *Mastercam University*—CNC Software sponsors Mastercam University, an affordable online learning platform that gives you 24/7 access to Mastercam training materials. Take advantage of more than 180 videos to master your skills at your own pace and help prepare yourself for Mastercam Certification. For more information on Mastercam University, please contact your Authorized Mastercam Reseller, visit www.mastercamu.com, or email training@mastercam.com.
- *Online communities*— You can find a wealth of information, including many videos, at www.mastercam.com. For tech tips and the latest Mastercam news, follow us on Facebook (www.facebook.com/mastercam), Twitter (www.twitter.com/mastercam), or Google+ (plus.google.com/+mastercam).

Visit our YouTube channel to see Mastercam in action (www.youtube.com/user/MastercamCadCam)!

Registered users can search for information or ask questions on the Mastercam Web forum, forum.mastercam.com, or use the knowledge base at kb.mastercam.com.

To register, select **Community**, **Link Account** from the **File** tab, and follow the instructions.

Mastercam Documentation

Mastercam installs the following documents in the `\Documentation` folder of your Mastercam installation:

- *What's New in Mastercam 2017*
- *Mastercam 2017 Installation Guide*
- *Mastercam 2017 Administrator Guide*
- *Mastercam 2017 Transition Guide*
- *Mastercam 2017 Quick Reference Card*
- *Mastercam 2017 ReadMe*

Contact Us

For questions about this or other Mastercam documentation, contact the Technical Documentation department by email at techdocs@mastercam.com.

Attention! Updates may be available.
Go to Mastercam.com/Support for the latest downloads.

CNC software, inc.

671 Old Post Road
Tolland, CT 06084 USA
www.mastercam.com