Getting Started with Automatic Toolpathing

May 2016

Mastercam® 2017

Mastercam® 2017 Mastercam ATP

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

Welcome to *Getting Started with Automatic Toolpathing*. Once Automatic Toolpathing (ATP) is setup, it automates Mastercam to create part files, NC files, nesting files, and report images with the click of a button. More specifically, ATP automates the process of assigning toolpaths to geometry for recurrent parts, such as in cabinet and closet making. It does this by linking operations to specific levels (layers) containing geometry, which creates a machining strategy. In this tutorial, you discover how to setup ATP and use it to process part files exported from your cabinet software.

Tutorial Goals

- Set up Mastercam 2017 for ATP.
- Create operation libraries for ATP.
- Configure ATP.
- Create ATP strategies.
- Run a cutlist.

This tutorial assumes that you are familiar with the basics of using Mastercam.

Estimated time to complete this tutorial: 4 hours

General Tutorial Requirements

All Mastercam tutorials have the following general requirements:

- You must be comfortable using the Windows® operating system.
- 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.

Files Included with this Tutorial

This tutorial is accompanied by part files that you can use to complete each lesson's exercises. Place these files (extracted from `Getting Started with Mastercam ATP.zip`) anywhere convenient on your system, but be sure to also keep unmodified copies. After extraction, you will have the folders described in the following table.

Table 1: Tutorial Folders and Files

Folder Name	Description
`AcmeSmall`	Cabinet `DXF` files and cutlist for Lessons 1 through 3.
`AcmeSmallFinished`	Completed `MCAM`, `mcam-operations`, level list, and strategy files.
`AcmeSmallOutput`	ATP output for the cabinet lessons.
`CustomColumns`	Cabinet `DXF` files and cutlist for Appendix A.
`CustomColumnsFinished`	Completed operation, level list, and strategy files.
`CustomColumnsOutput`	ATP output for the Appendix A lessons.
`MoldBases`	Mold base `DXF` files for Lessons 4 and 5.
`MoldBasesFinished`	Completed `MCAM`, `mcam-operations`, level list, and strategy files.
`MoldBasesOutput`	ATP output for the mold base lessons.

LESSON 1
ATP Cabinet Overview

ATP automates the assignment of toolpaths to geometry for recurrent parts by linking operations to specific levels, each level containing geometry for a single type of operation. ATP automatically scans all levels and programs for the geometry on each level with the toolpath types you have chosen.

Once you setup ATP (create a strategy), you choose the cutlist to be processed. ATP automatically batch processes each part file using the chosen strategy. It then nests the toolpaths and posts the results to one or more NC files (depending on the number of sheets created). ATP sorts these files into folders by material name.

ATP is especially useful in large projects with many pieces of geometry. With a modest amount of setup, you can save valuable time by letting the software automatically find, chain, and toolpath all the elements of a job.

The ATP Workflow

Most of the ATP workflow involves preparing your part's operations and mapping them to the levels in the part files. After this one-time setup, running a cutlist to generate and nest your part files is a quick process. The following list is a summary of the complete workflow.

- Assign toolpaths to geometry in the part files.
- Export the toolpaths as an operation library.
- Generate a level list from the part files.
- Map operations to levels.
- Run the cutlist.

Assigning Toolpaths to Geometry

In most cases, you start with the DXF file with the largest number of operations. This file usually contains all of the operations you need for all the other files in the job. You open this file in Mastercam and create operations just as you would for any other Mastercam part. However, you need only create one of each operation type. For example, if a job has several dado cuts that use the same operation type and tool, you only need to toolpath one of them.

NOTE: ATP supports the following toolpaths: Contour, Pocket, Drill, Block Drill, Engrave, Circle Mill, and Helix Bore.

Exporting the Toolpaths

Once you have created the operations needed to complete the job, you export the operations to a library. When you work with ATP, you load this library so that the operations you created are available for mapping to levels. A single operation can be mapped to multiple levels in multiple part files.

Generating a Level List

Just as ATP needs to know about the operations it has to work with (stored in the exported operations library), it needs to know about the levels included in your job's part files. ATP creates a level list for you and displays the results.

Mapping Operations to Levels

After creating your level list, you load the operations library, which makes the operations available for mapping. Each level in the list has a drop-down menu from which you can choose the appropriate operation. You must map an operation to every level in the level list.

Run the Cutlist

You complete the previous steps only once for each job type. After that, you can simply run the job's cutlist to generate the files needed to cut the parts for the job, including nesting the parts on the sheet stock you define.

Levels and Operations

As you can tell from the ATP workflow, levels and operations play a central role in automating Mastercam to create your final part files. Typically, you define the levels when creating your job's files in your cabinet software. Each level must contain the

geometry for a single type of operation. For example, the part files you use in this tutorial's lessons, include the following levels:

Level
DrillDrawer
DrillPilot
DrillShelf
Panel_0p2500
Panel_0p7500
Rout
RoutTab

In the exercises, you create a library containing the following operations:

Operation Type	Tool Comment	Tool #	Operation Comment
Contour	0.5 Flat Endmill	2	0.5" outside route
Contour	1 Chamfer Mill	1	Chamfer route
Contour	0.1 Flat Endmill	3	Dado route
BlockDrill	Drill Block	100	Block drill

You bring these two lists together by assigning an operation to each level, as shown below.

Level	Operation	Edit
DrillDrawer	Block drill	
DrillPilot	Block drill	
DrillShelf	Block drill	
Rout	Dado route	
RoutTab	Dado route	
Panel_0p2500	0.5 outside route	
Panel_0p7500	0.5 outside route	
Panel_0p7500	Chamfer route	

How ATP Processes Your Files

A lot happens when ATP automates Mastercam. The following is a general overview of the process.

1. Imports a part file into Mastercam.
2. Applies operations to the part file using the active strategy.
3. Saves results to the applicable material folder.
4. Repeats Steps 1 through 3 for all part files.

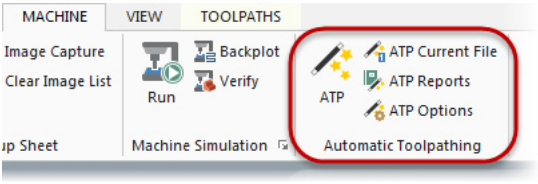
After processing the part files, ATP performs nesting and then posts the results, as follows.

1. If nesting is on, runs a material nest list.

2. Saves the nested sheet as a drawing to the applicable material folder.
3. Posts NC code to the applicable material folder.
4. Repeats Steps 1 through 3 for all nest lists.

LESSON 2
Preparing ATP - Cabinet Example

Before you can use ATP effectively, you must create the operations required by your job and then export those operations into a library that ATP can use. Access Mastercam ATP through the Machine tab on the ribbon bar.



You might also want to set up Mastercam so that the Nesting McLink function is more easily accessible, by adding its icon to your Mastercam ribbon. The following procedure shows how to accomplish this task.

Exercise 1: Preparing the Interface

The ATP components are on the Machine tab by default, making it easy to run ATP when needed. If you use ATP often, you will want to complete the following procedure to add the Nesting McLink icon to the ribbon in Mastercam.

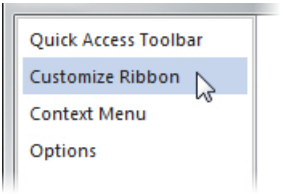
NOTE: Use the Options dialog box (File, Options) to add frequently used C-Hooks or NET-Hooks to a tab (Customize Ribbon) or the graphics window context menu (Context Menu).

▶ Adding the McLink Icon

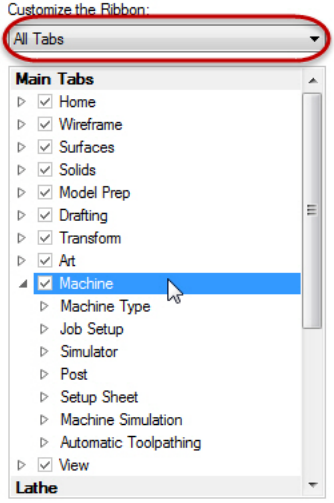
The McLink Nesting utility provides a handy way to edit nesting files generated by ATP. You can add McLink Nesting to Mastercam's ribbon. For additional information about McLink, open the Help in Mastercam and search for *Nesting with MCLink*.

1 Choose **Options** from the **File** tab to open the Options dialog box.

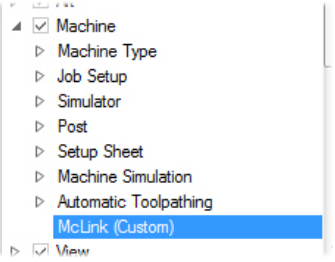
2 Select the **Customize Ribbon** page in the left pane.



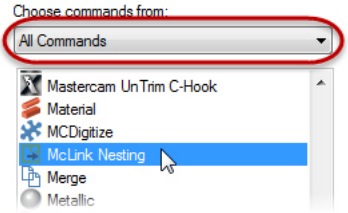
3 In the **Customize the Ribbon** list, choose All Tabs in the drop-down list, and then **Machine**. Expand the Machine choice.



4 Click the **New Group** button, and use the Rename button to name it **McLink**. (When you name it McLink, it adds the "(Custom)" to the name.)



5 Set the **Choose commands from** drop-down to display **All Commands** and highlight **McLink Nesting**.



6 Click the **Add** button to add the McLink Nesting icon to the McLink group in the Machine tab. Click the **OK** button to add the new option to the ribbon.

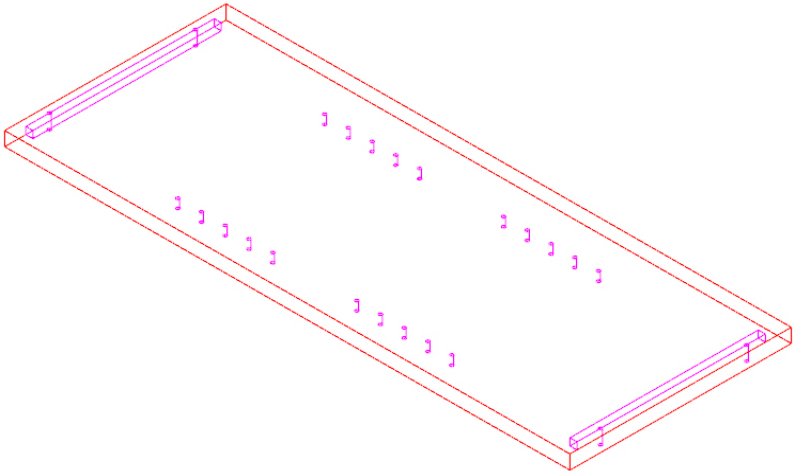
Exercise 2: Opening the Part

In the `AcmeSmall` folder included with this tutorial, you can find the `DXF` files for a complete cabinet job. In this procedure, you open the largest of these files. Typically, the largest file in a job contains the geometry for all of the operations you need to create for ATP.

NOTE: Be sure that Mastercam is set to the inch configuration, rather than metric.

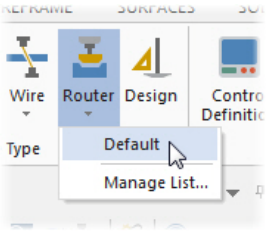
1 In Mastercam, open the `ACME_1_Side_Right_001F.dxf` file.

2 Adjust the display in the graphics window so that you can see the entire part in the isometric view.



3 Choose **Machine, Router, Default**.

Mastercam creates a router machine group in the Toolpaths Manager.



Exercise 3: Creating the Operations

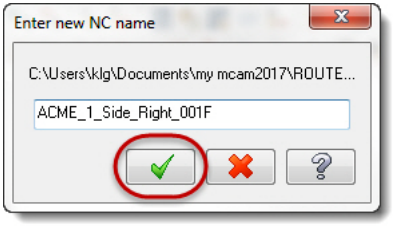
Before ATP can do its job, it must have a library of operations that you can map to the levels in your part files. In this exercise, you create these operations.

▶ Creating the Outside Contour Operation

1 From the Toolpaths tab, choose **Contour** from the 2D gallery. The **Enter new NC name** dialog box opens.

2 Click **OK** to accept the default NC file name.

The Chaining dialog box opens.

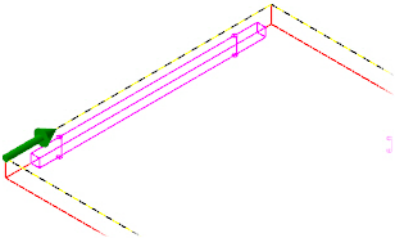


3 Activate the **C-plane** option.

Mastercam will now only chain entities that are parallel to the construction plane and at the same Z depth as the first entity you chain.

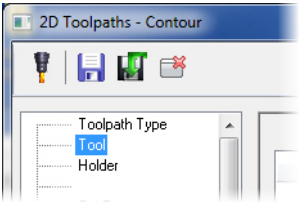
4 Chain the part's outside edge, as shown, and click **OK**.

The **2D Toolpaths - Contour** dialog box opens.



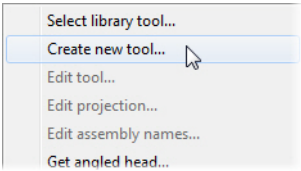
IMPORTANT: For ATP to work correctly, you must do all chaining in the clockwise direction.

5 In the dialog box, select the **Tool** page.

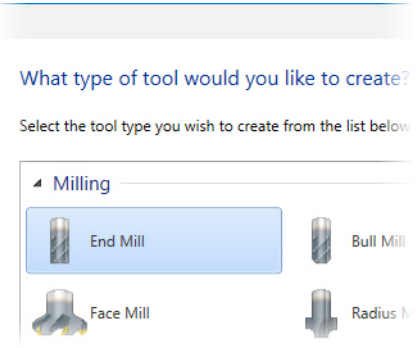


6 Right-click in the dialog box, and choose **Create new tool** from the pop-up menu.

The Tool Wizard opens.

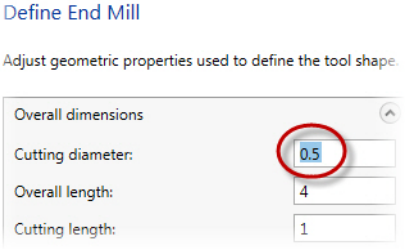


7 Select **End Mill**, and click **Next**.



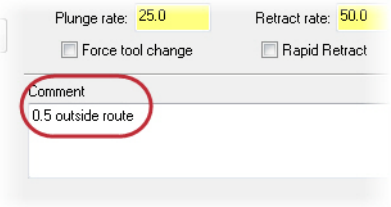
8 Ensure that **Cutting diameter** is set to **0.5**, and then click **Finish**.

Mastercam creates a new 0.5" flat endmill for the contour operation.

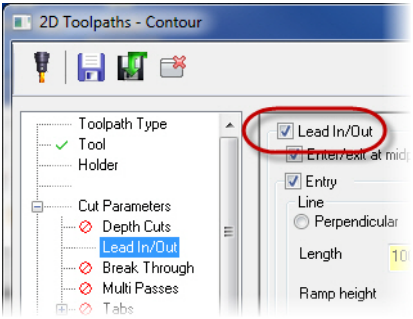


9 In the Comment box, enter **0.5 outside route**.

Enter comments for all operations that you will export for ATP. Later in the process, ATP uses these comments as operation descriptions.



10 Select the **Lead In/Out** page, and ensure that Lead In/Out is turned on.



11 Click **OK** to create the toolpath.

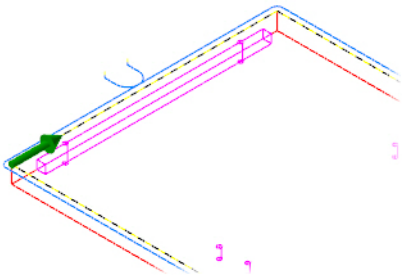
Creating the Outside Chamfer Operation

1 From the Toolpaths tab, choose **Contour**. The Chaining dialog box opens.

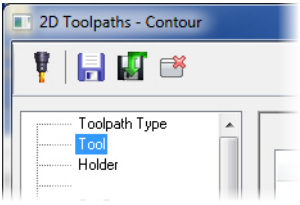
2 Activate the **C-plane** option.

3 Chain the part's outside edge, as shown, and click **OK**.

The **2D Toolpaths - Contour** dialog box opens.

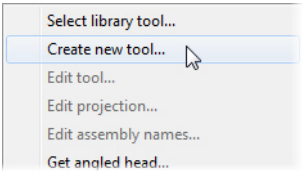


4 In the dialog box, select the **Tool** page.



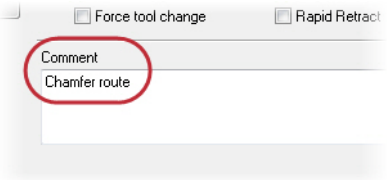
5 Right-click in the dialog box, and choose **Create new tool** from the pop-up menu.

The Tool Wizard opens.



6 Create a 1.0" diameter chamfer mill, and then click **Finish**.

7 On the Tool page, enter **Chamfer route** in the Comment box.



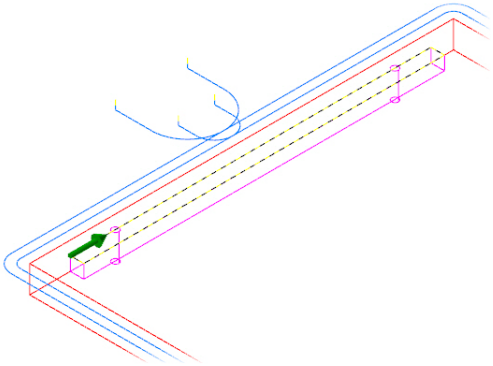
8 Click **OK** to create the toolpath.

Creating the Dado Operation

1 From the Toolpaths tab, choose **Contour**. The Chaining dialog box opens.

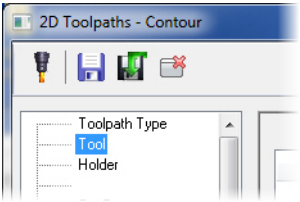
2 Activate the **C-plane** option.

3 Chain the dado geometry, as shown below, and click **OK**.

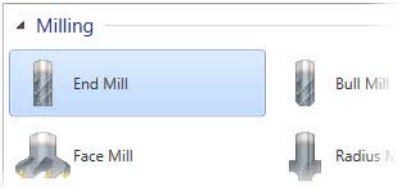


The **2D Toolpaths - Contour** dialog box opens.

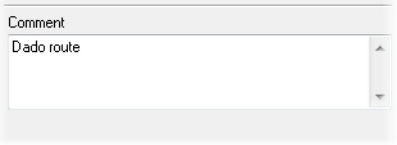
4 In the dialog box, select the **Tool** page.



5 Create a 2" long, 0.1" diameter endmill for the dado cut, naming the tool **0.1 Flat Endmill**. (After entering the Cutting diameter and Overall length, click **Next** to find the Name field.)



6 Set **Comment** to **Dado route**.



7 On the Cut Parameters page, set **Compensation direction** to **Right**, and on the Lead In/Out page, turn off **Lead In/Out**.

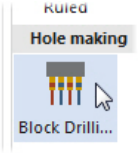
8 Click **OK** to create the toolpath.

NOTE: Although the part file contains two dados, you only need to toolpath one. When run, ATP will automatically find other matching dado operations.

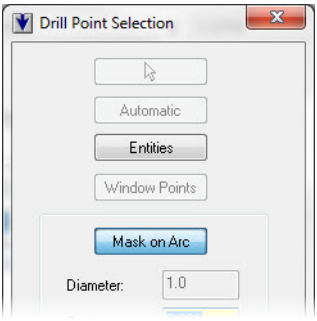
Creating the Block Drilling Operation

1 Choose **Toolpaths**, expand the **2D** gallery, and choose **Block Drilling**.

The Drill Point Selection dialog box opens.

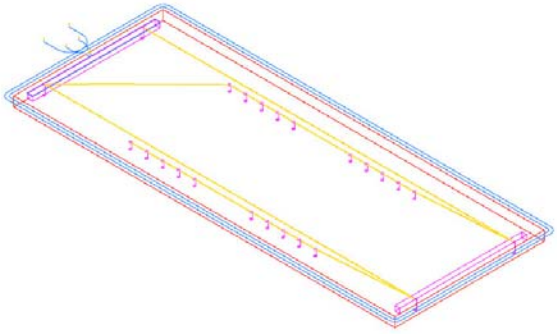


2 Choose **Mask on Arc**, and in the graphics window, click one of the drill holes. (You might need to zoom in on the geometry.)



3 Use window select to select all of the part's geometry, and press [**Enter**].

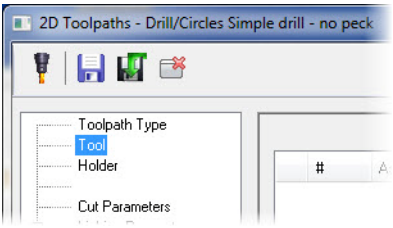
Mastercam selects all of the holes that match your arc selection.



4 In the dialog box, click **OK**.

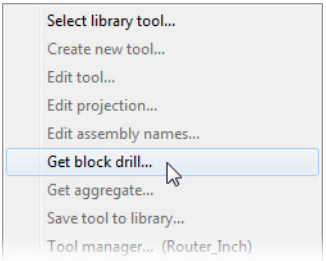
The **2D Toolpaths - Drill/Circles Simple drill - no peck** dialog box opens.

5 In the dialog box, select the **Tool** page.



6 Right-click in the dialog box, and choose **Get block drill** from the pop-up menu.

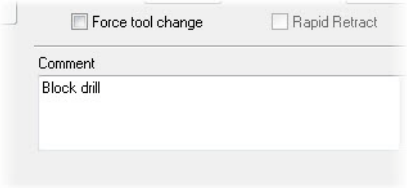
The Drill Block Selection dialog box opens.



7 Click **OK** to accept the default drill block settings.

8 Enter **Block drill** into the Comment box, and click **OK**.

Mastercam adds the operation to the Toolpaths Manager.

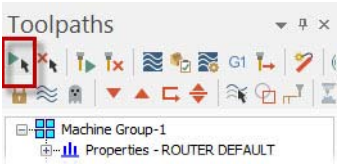


Exercise 4: Exporting the Operations

Now that you have created the operations needed for the parts, you must export them to a library that you can use when mapping level names to operations.

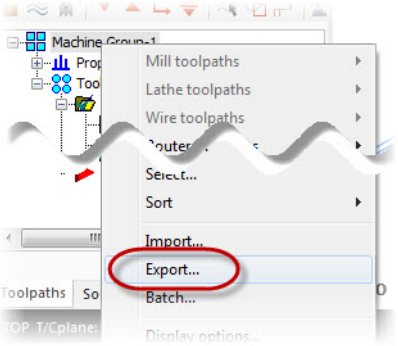
1 In Toolpaths Manager, click **Select all operations**.

Mastercam selects all operations in the machine group.



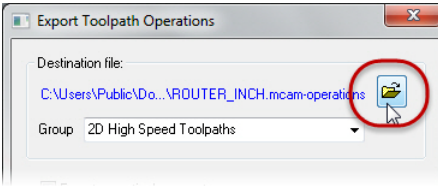
2 Right-click **Machine Group-1**, and choose **Export** from the pop-up menu.

The Export Toolpath Operations dialog box opens.

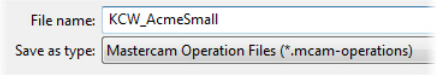


3 Click the **Select Operations File** button.

The **Export toolpath operations** dialog box opens.



4 In the **File name** box, type **KCW_AcmeSmall**, and click **Save**.



IMPORTANT: When saving your operations, never choose a library that was installed with Mastercam. Doing so will add invalid operations to the chosen library. Always give your operations library a custom name.

5 In the Export Toolpath Operations dialog box, click the **Apply** button.

Mastercam saves the operations to the selected library.

6 Dismiss the message box that appears, and then click the **Done** button.

You are now ready to run ATP and map the operations to the levels in your part files.

LESSON 3
Using ATP - Cabinet Example

Once you have your operation library created, it is time to create a strategy for the job. This requires generating a level list (a list of all levels in the job's part files) and assigning an operation to each level. When that is done, you can run your cutlist to create the job's final files.

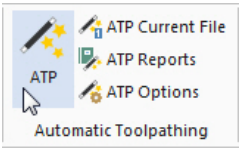
Exercise 1: Creating a Strategy

To create a strategy, you must start a new strategy file and generate a level list from the parts.

▶ Starting a New Strategy

1 In the Machine tab, click the **ATP** button.

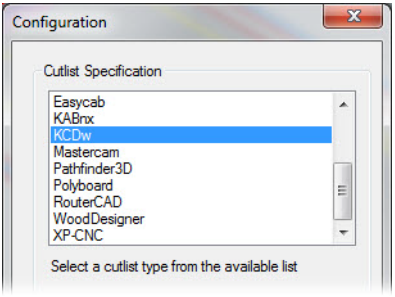
The Configuration dialog box opens.



NOTE: If the Configuration dialog box does not open, the ATP configuration was previously set. In this case, you need to be sure that, when the Mastercam ATP dialog box opens, **Configuration** is set to **KCDw**.

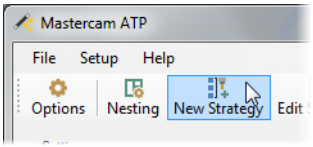
2 Select **KCDw** from the Cutlist Specification list, and click **OK**.

ATP is now configured to read cutlists generated by the KCDw software package.



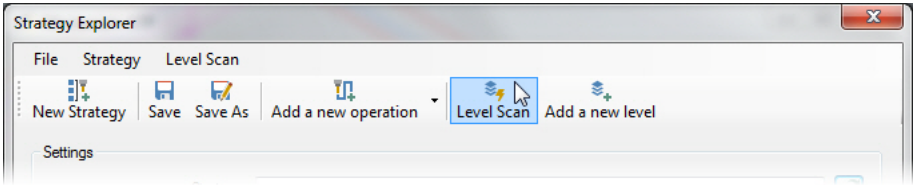
3 In the Mastercam ATP dialog box, click the **New Strategy** button.

The Strategy Explorer window opens.



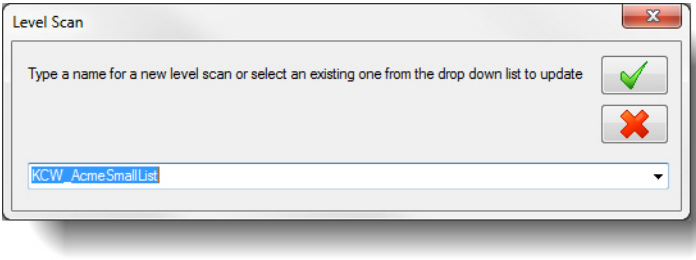
4 Click the **Level Scan** button.

The Level Scan dialog box opens, in which you can type a file name for the level list you are about to create.



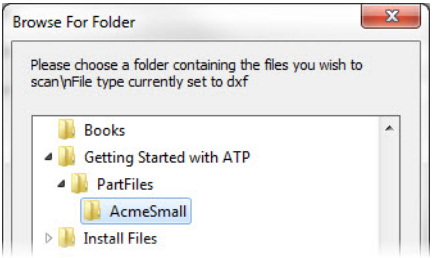
5 Enter **KCW_AcmeSmallList** into the text box, and click **OK**.

The Browse For Folder dialog box opens.

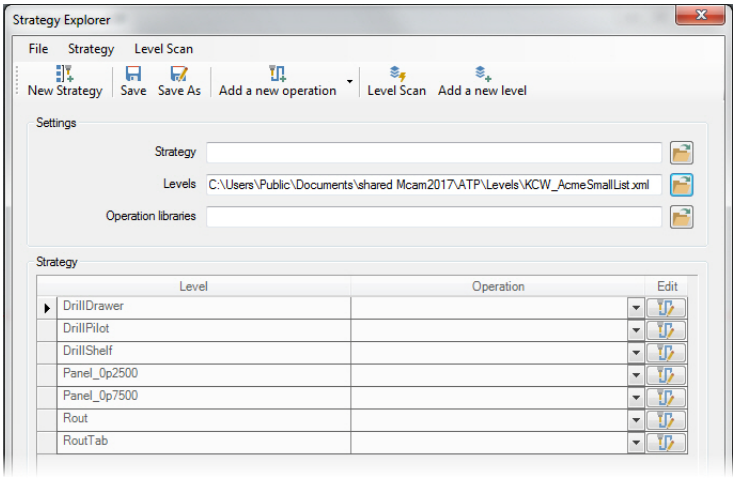


TIP: If you choose an existing file from the drop-down in the Level Scan dialog, you can merge the chosen level list file with levels ATP finds in the level scan.

6 Navigate to the `PartFiles\AcmeSmall` folder, which is supplied with this tutorial, and click **OK**.



ATP scans the `DXF` files in the selected folder, and builds a list from the levels found in those files, as shown below.

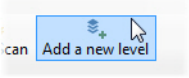


Exercise 2: Modifying the Level List

Once you have the level list, you must add levels and place operations in the correct order for cutting the parts.

▶ Adding Levels

1 Click the **Add a new level** button.

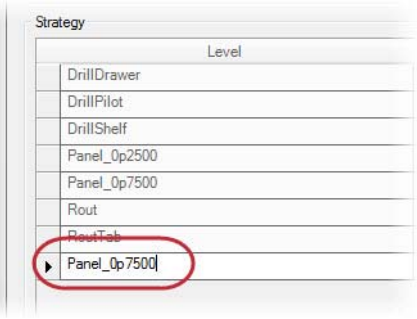


ATP creates a blank row in the Level list.

You must create this new level in order to include the chamfer operation in the list, due to the part's using the same chain for the outside contour cut and the chamfer.

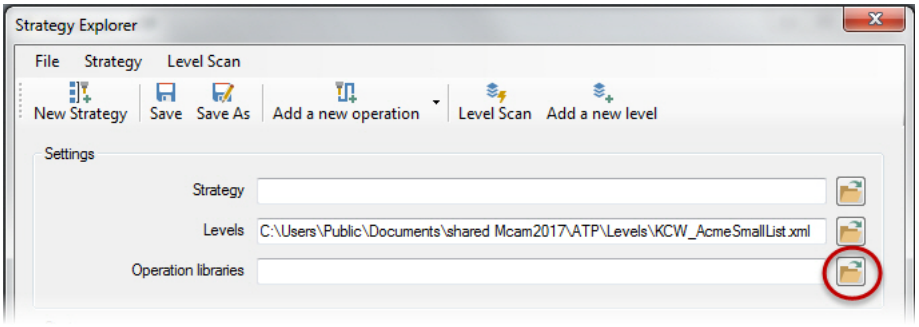
2 Click in the blank row, and type **Panel_0p7500** as the level name.

This gives you two references to the Panel_0p7500 level, one for the contour operation and one for the chamfer operation.

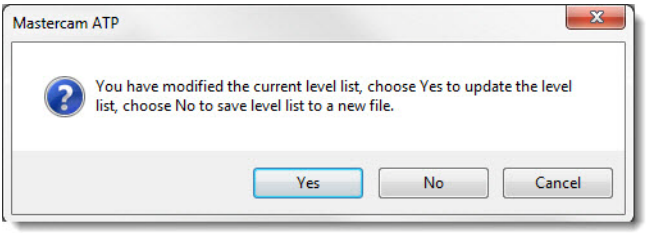


Assigning Operations to the Levels

1 Click the **Select operations file** button, located to the right of the **Operation libraries** text box.



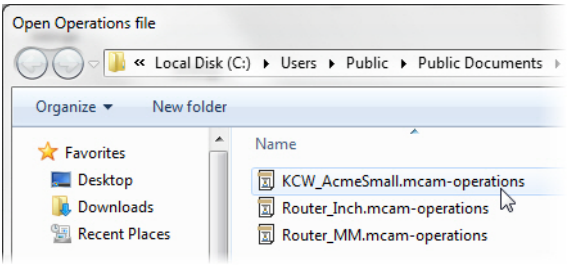
2 When asked if you want to save your changes to the level list, click **Yes**.



The **Open Operations file** dialog box opens.

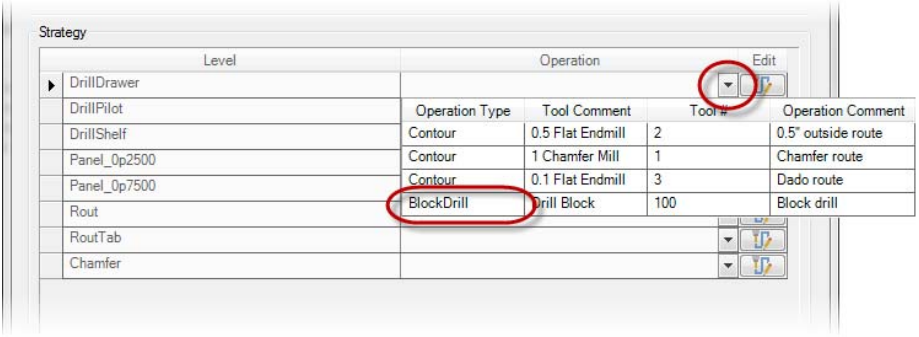
3 Select the `KCW_AcmeSmall.mcam-operations` file, and click **Open**.

ATP loads the operation library.



4 In the Operation column for the DrillDrawer level in the strategy, click the black drop-down arrow, and select the operation with the **Block drill** comment.

ATP maps the Block drill operation to the DrillDrawer level. Notice that the Operation Comment column contains the comments you entered when creating the operations.



5 Assign the **Block drill** operation to the DrillPilot and DrillShelf levels.

6 Assign the **0.5 outside route** operation to the Panel_Op2500 level and the first Panel_Op7500 level.

7 Assign the **Dado route** operation to the Rout and RoutTab levels.

8 Assign the **Chamfer route** operation to the second reference to the Panel_Op7500 level.

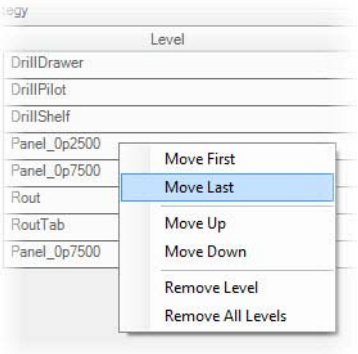
You should now have the operation mappings shown in the following picture.

Level	Operation	Edit
DrillDrawer	Block drill	
DrillPilot	Block drill	
DrillShelf	Block drill	
Panel_0p2500	0.5 outside route	
Panel_0p7500	0.5 outside route	
Rout	Dado route	
RoutTab	Dado route	
Panel_0p7500	Chamfer route	

Reordering the Operations

1 Select the **Panel_Op2500** level, right-click, and choose **Move Last** from the pop-up menu.

The level moves to the end of the list. It's important to arrange the operations in the correct order before generating files with ATP.



2 Move the first Panel_Op7500 and the chamfer route operations to the bottom.

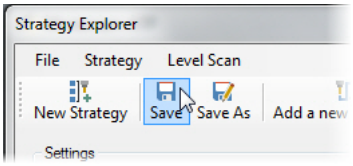
Your operations should be in the order shown below.

Strategy

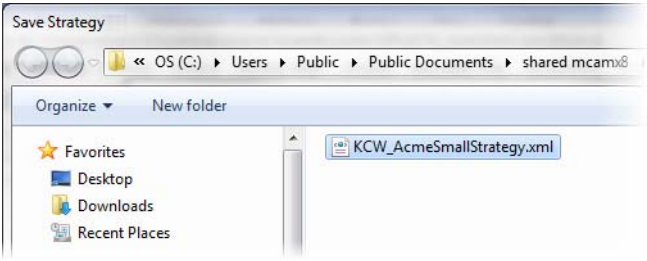
Level	Operation	Edit
DrillDrawer	Block drill	
DrillPilot	Block drill	
DrillShelf	Block drill	
Rout	Dado route	
RoutTab	Dado route	
Panel_0p2500	0.5 outside route	
Panel_0p7500	0.5 outside route	
Panel_0p7500	Chamfer route	

3 In Strategy Explorer's toolbar, click **Save**, and then click **Yes** if asked to save the level list.

The Save Strategy window opens.



4 Save the strategy under the name `KCW_AcmeSmallStrategy`, and then click **OK** in the Strategy Explorer window.



TIP: You can create multiple strategies for the same level list, allowing you to apply different operations in different situations.

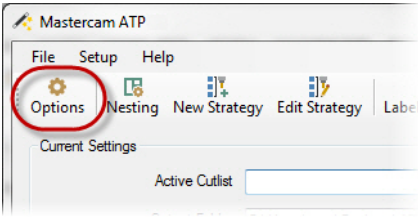
Exercise 3: Creating and Nesting Your Part Files

Now you are ready to set up and run your cutlist with ATP, which creates the files for your parts. After generating the parts, you can refine your nesting parameters as needed.

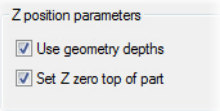
▶ Setting Up and Running the Cutlist

1 In the Mastercam ATP dialog box, click the **Options** button.

The Options dialog box opens.



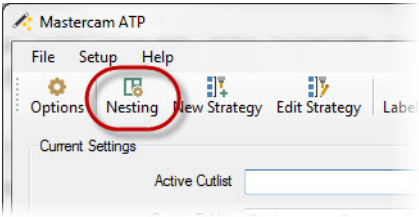
2 Make sure that the **Use geometry depths** and **Set Z zero top of part** options are selected, and click **OK**.



NOTE: When **Use geometry depths** is selected, ATP determines the operation depths from the 3D geometry. If you are processing 2D geometry, **Use geometry depths** must be off and the depths defined in each operation's Linking Parameters.

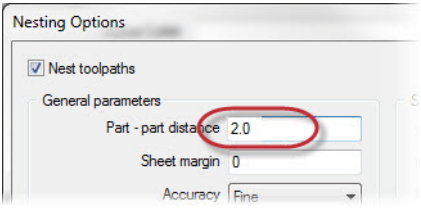
3 Click the **Nesting** button.

The Nesting Options dialog box opens. Use this dialog box to define nesting parameters for the current ATP job.

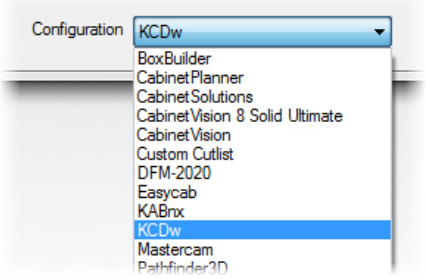


4 Change the **Part - part distance** setting to **2.0**, and then click **OK** to accept the change.

This change will force more space between the parts when they are nested onto the sheet material. (You nest them closer later in the tutorial.)

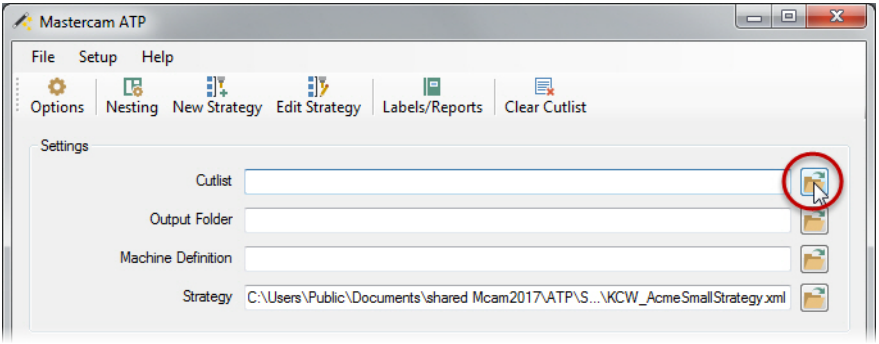


5 At the bottom of the Mastercam ATP dialog box, ensure that **Configuration** is set to **KCDw**.

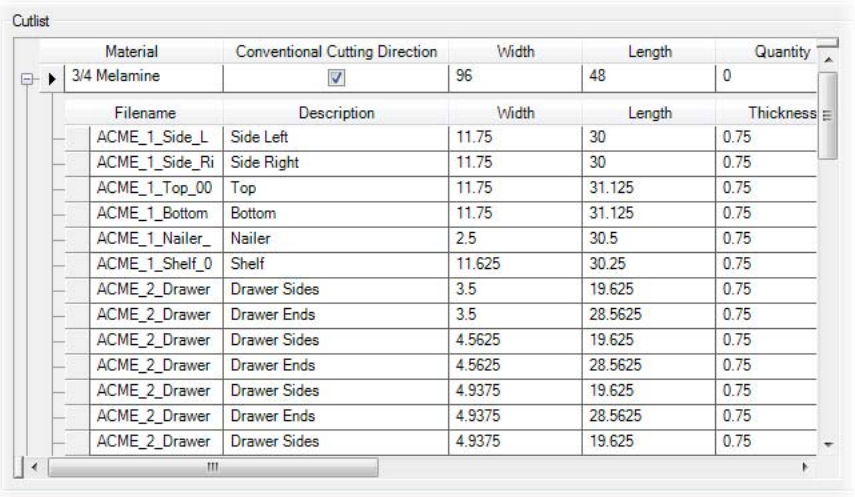


This setting specifies the software that created the DXF files and ensures that ATP reads the files correctly.

6 Click the **Open Cutlist** button, browse to `PartFiles\AcmeSmall`, and open the file `ACME_CNCList.txt`.



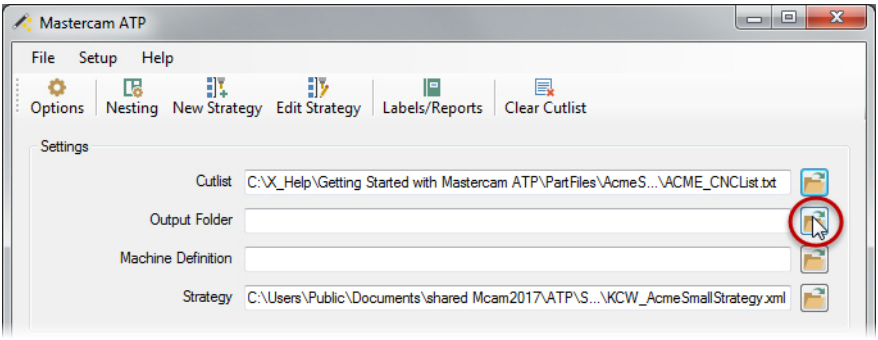
ATP reads the cutlist and displays the data in the Cutlist pane, as shown below.



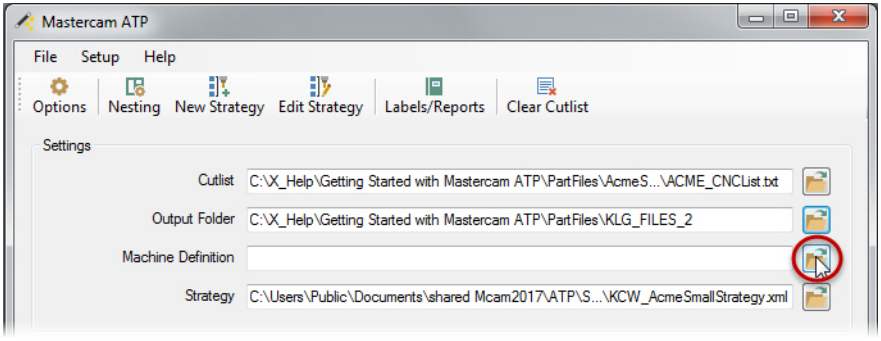
Material	Conventional Cutting Direction	Width	Length	Quantity
3/4 Melamine	☑	96	48	0

Filename	Description	Width	Length	Thickness
ACME_1_Side_L	Side Left	11.75	30	0.75
ACME_1_Side_Ri	Side Right	11.75	30	0.75
ACME_1_Top_00	Top	11.75	31.125	0.75
ACME_1_Bottom	Bottom	11.75	31.125	0.75
ACME_1_Nailer_	Nailer	2.5	30.5	0.75
ACME_1_Shelf_0	Shelf	11.625	30.25	0.75
ACME_2_Drawer	Drawer Sides	3.5	19.625	0.75
ACME_2_Drawer	Drawer Ends	3.5	28.5625	0.75
ACME_2_Drawer	Drawer Sides	4.5625	19.625	0.75
ACME_2_Drawer	Drawer Ends	4.5625	28.5625	0.75
ACME_2_Drawer	Drawer Sides	4.9375	19.625	0.75
ACME_2_Drawer	Drawer Ends	4.9375	28.5625	0.75
ACME_2_Drawer	Drawer Sides	4.9375	19.625	0.75

7 Click the **Output Folder** button, and browse to where you want ATP to place the files it generates. (Do not choose the folder that contains your source DXF files.)



8 Click the **Select Machine Definition** button, choose ROUTER DEFAULT.MCAM-RMD in the dialog box that opens, and click **Open**.



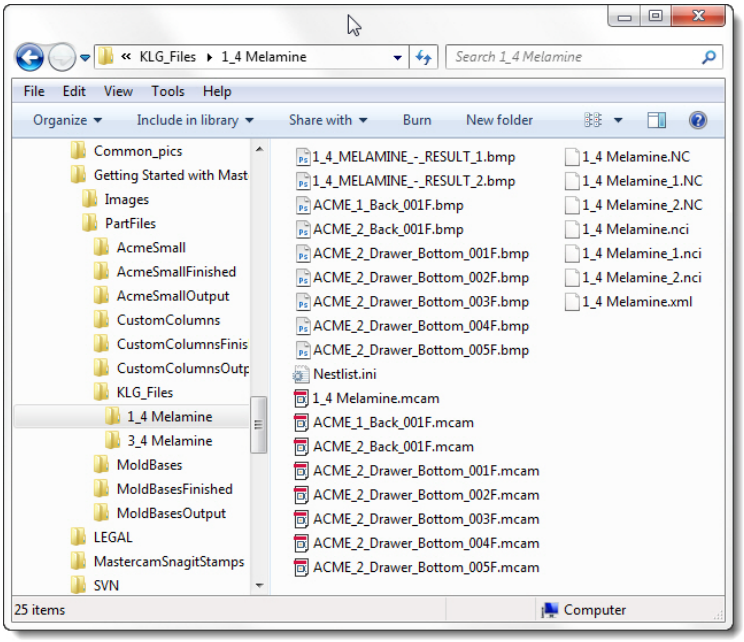
9 In the Mastercam ATP dialog box, click **OK** to run the cutlist with the settings you selected.

ATP automates Mastercam to create and nest the parts from the DXF files.

IMPORTANT: Do not do anything with Mastercam until you see the message **Process Complete**.

10 Dismiss the Process Complete message, and open the output folder you created in Step 7.

In the output folder you selected, you now have two subfolders, one each for the 1/4" Melamine and 3/4" Melamine materials. Each folder contains Mastercam part files, `NC` and `NCI` files, bitmaps for use with reports, and the nesting file (`Nesting.ini`), as shown below.



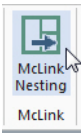
11 Open the file `3_4 Melamine.mcam` with Mastercam (found in the `3_4 Melamine` subfolder you just created). This file shows the nesting for the 3/4" parts.

12 Right-click in the graphics window and choose [**Fit**] to view how the parts are nested.

Refining the Nesting

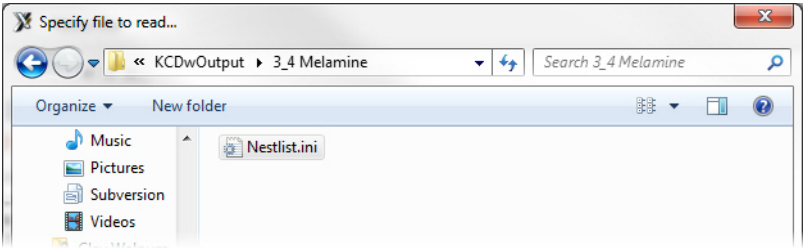
At this point, you might be ready to cut the parts, or you might want to adjust the nesting parameters to fit the parts better on the sheets. The following procedure shows how to make such adjustments.

1 In the Machine tab, click the **McLink Nesting** button.



The **Specify file to read** dialog box opens.

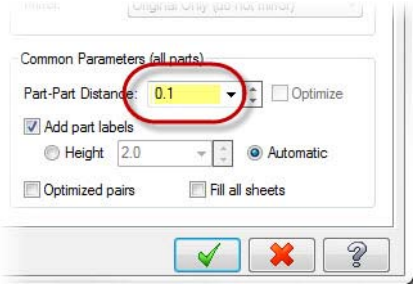
2 Navigate to the `3_4 Melamine` folder where you had ATP generate the job's output files, and open the `Nestlist.ini` file.



After some processing, the Nesting 2017 V2013R2 dialog box opens. This dialog box provides access to your nesting parameters.

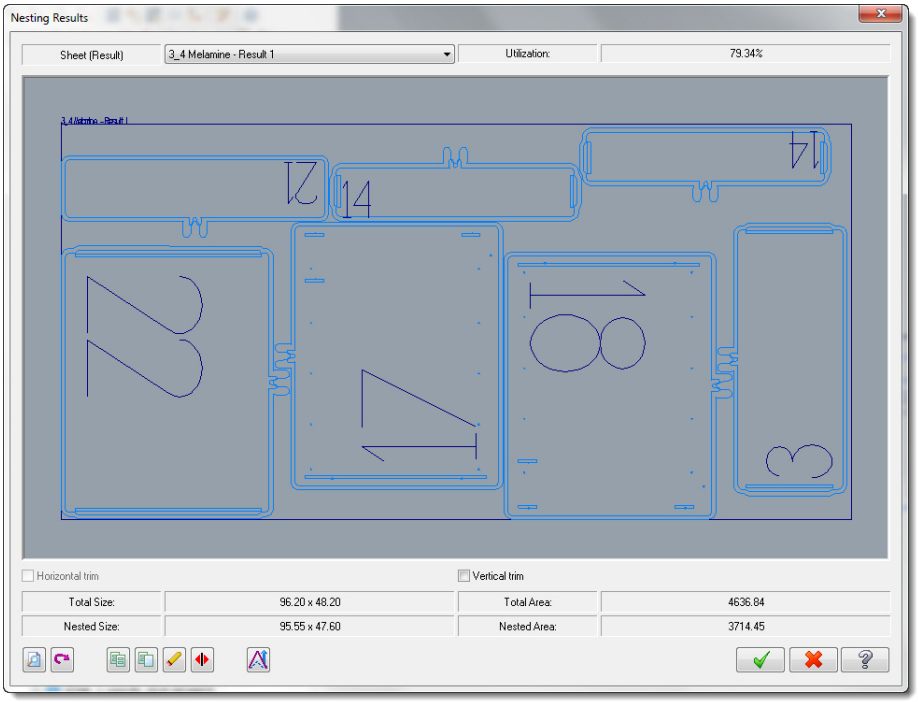
3 Click the **Parts** tab, change the **Part-Part Distance** setting to **0.1**, and click **OK**.

After some processing, the Nesting Results dialog box opens.



TIP: You can save the changed nesting parameters as defaults by clicking the **Save** button in the Nesting 2017 V2013R2 dialog box.

Notice that the parts are more tightly nested in the sheet, as shown below.



TIP: If you want to further refine your nesting parameters, click the **Redo** button to return to the Nesting 2017 V2013R2 dialog box. You may notice an increase or decrease in the number of operations depending on your default nesting parameters (options) set in this dialog box.

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

TIP: For more information about nesting, please refer to the Mastercam Help.

LESSON 4
Preparing ATP - Mastercam Example

When using ATP in the Mastercam configuration, you use a file list instead of a cutlist. As with the cabinet example, you must still create an operations library and map the operations to the levels in the files.

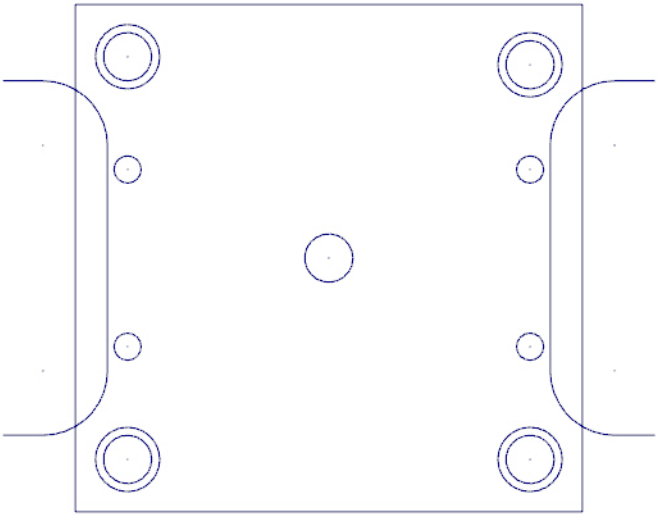
NOTE: This lesson assumes that you have set up ATP as described in the section **Preparing the Interface** on page 13.

Exercise 1: Opening the Part

In the `MoldBases` folder included with this tutorial, you can find the `DXF` files for this lesson. In this procedure, you open the largest of these files, because the largest file in a job typically contains the geometry for all of the operations you need to create for ATP.

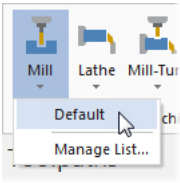
1. Start a new Mastercam session. Make sure you are using the inch configuration.
2. In Mastercam, open the `1.dxf` file. (You may want to change the geometry's color to make it easier to see.)

The part should look like the following image.



3 From the Machine tab, choose **Mill, Default**.

Mastercam creates a mill machine group in the Toolpaths Manager.



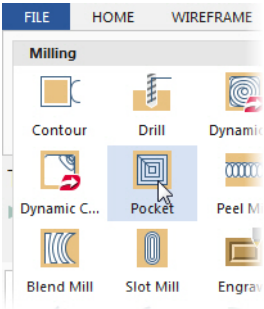
Exercise 2: Creating the Operations

The mold bases project contains much more geometry than you will create operations for. To keep the example short, in this exercise, you create only the seven operations represented on the levels in the `1.dxf` file.

Creating the Notch Contour Operation

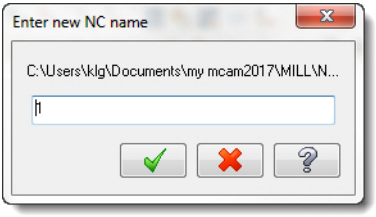
1 From the Toolpaths tab, choose **Pocket** from the **2D** gallery.

The **Enter new NC name** dialog box opens.



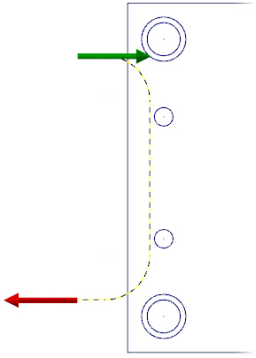
2 Click **OK** to accept the default NC file name.

The Chaining dialog box opens.



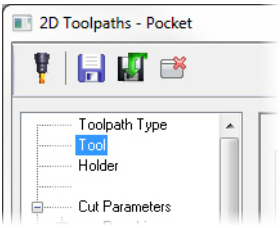
3 Chain the left notch, as shown, and click **OK**.

The **2D Toolpaths - Pocket** dialog box opens.



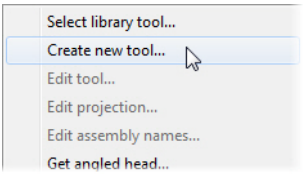
IMPORTANT: Remember, for ATP to work correctly, you must do all chaining in the clockwise direction.

4 In the dialog box, select the **Tool** page.

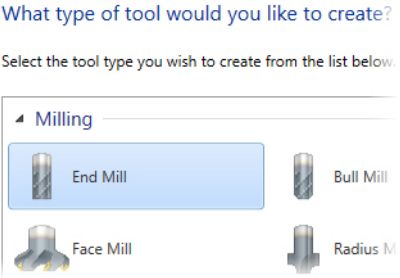


5 Right-click in the dialog box, and choose **Create new tool** from the pop-up menu.

The Tool Wizard opens.

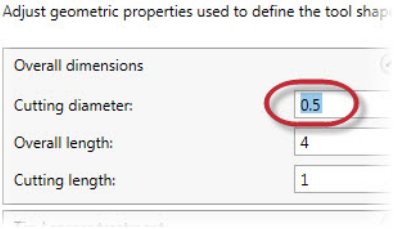


6 Select **End Mill**, and click **Next**.



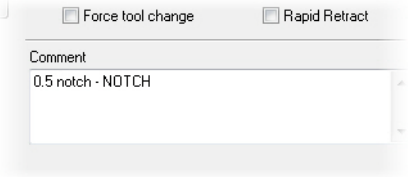
7 Ensure that **Cutting diameter** is set to **0.5**, and then click **Finish**.

Mastercam creates a new 0.5" flat endmill for the operation.



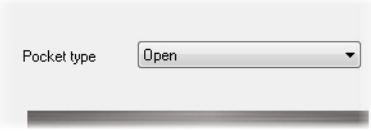
8 In the Comment box, enter **0.5 notch - NOTCH**.

Be sure to enter comments for all operations you will export for ATP. Later in the process, ATP uses these

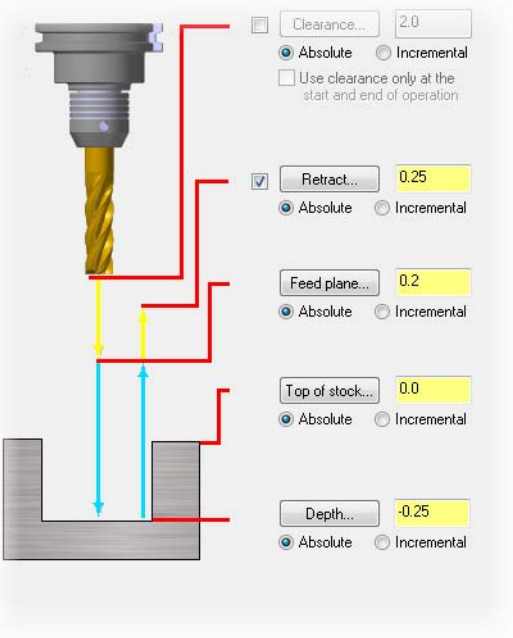


comments as operation descriptions.

9 In the Cut Parameters page, set Pocket type to **Open**.



10 In the Linking Parameters page, set Depth to **-0.25**, and change all settings to **Absolute**.

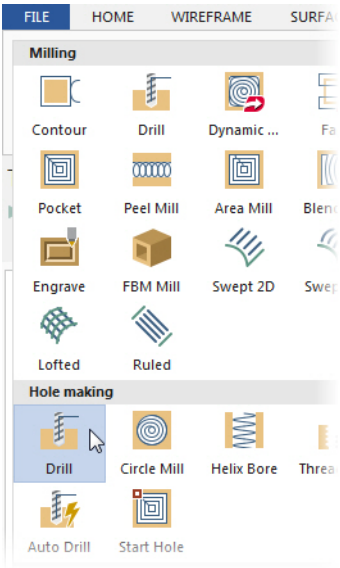


11 Click **OK** to create the toolpath.

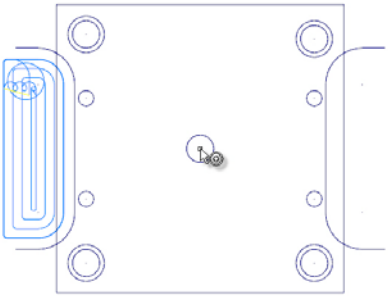
Creating the Rough Drill Operation

1 From the Toolpaths tab, choose **Drill** in the **Hole making** section of the **2D** gallery.

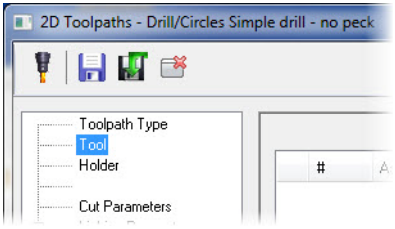
The Drill Point Selection dialog box opens.



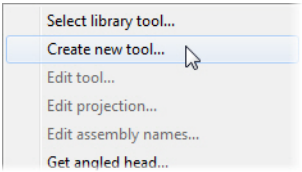
2 In Mastercam's graphics area, choose the hole's center, as shown, and click **OK** in the dialog box.



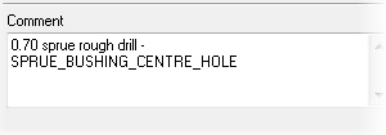
3 In the dialog box, select the **Tool** page.



4 Create a 4" long, 0.7" diameter drill named **0.70 drill**, and click **Finish**.

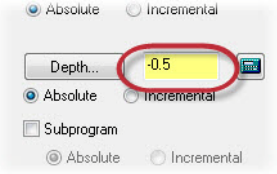


5 In the Comment box, type **0.70 sprue rough drill - SPRUE_BUSHING_CENTRE_HOLE**.



6 In Linking Parameters, set Depth to **-0.5**, all parameters to **Absolute**, and click **OK**.

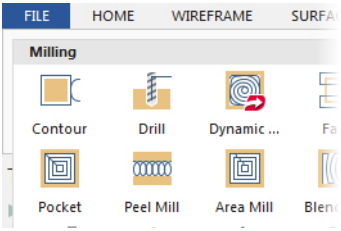
Mastercam creates the toolpath.



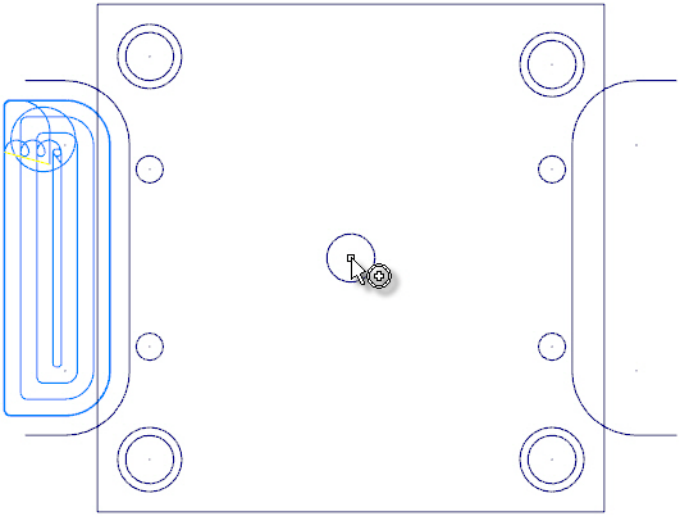
Creating the Finish Bore Operation

1 From the Toolpaths tab, choose **Drill**.

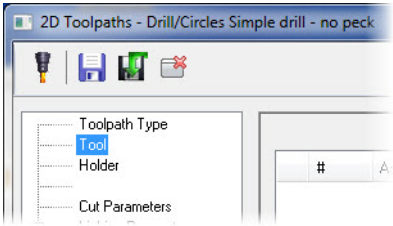
The Drill Point Selection dialog box opens.



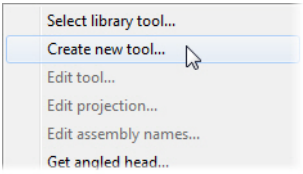
2 In Mastercam's graphics area, choose the hole's center, and click **OK** in the dialog box.



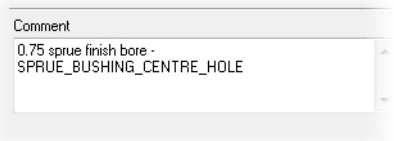
3 In the 2D Toolpaths dialog box, select the **Tool** page.



4 Create a 0.75" boring bar tool named **0.75 bore bar**, and click **Finish**.

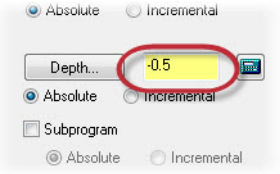


5 In the Comment box, type **0.75 sprue finish bore - SPRUE_BUSHING_CENTRE_HOLE**.



6 In Linking Parameters, set Depth to **-0.5**, all settings to **Absolute**, and click **OK**.

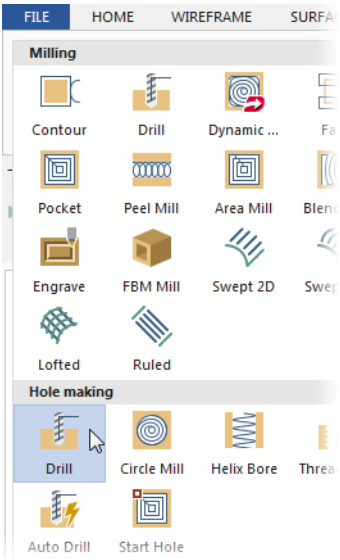
Mastercam creates the toolpath.



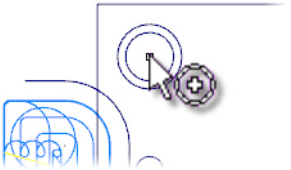
Creating the Leader Pin Drill Operation

1 From the Toolpaths tab, choose **Drill** in the **Hole making** section of the **2D** gallery.

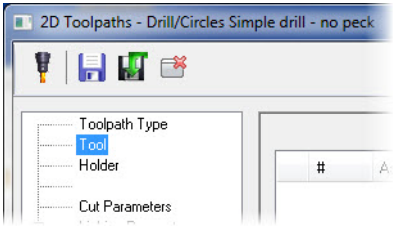
The Drill Point Selection dialog box opens.



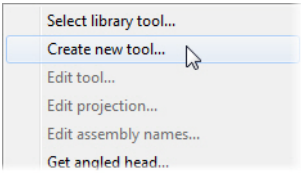
2 In Mastercam's graphics area, choose the hole's center shown, and click **OK** in the dialog box.



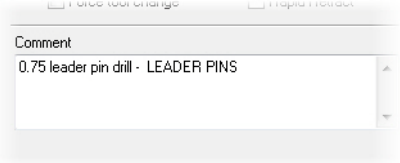
3 In the dialog box, select the **Tool** page.



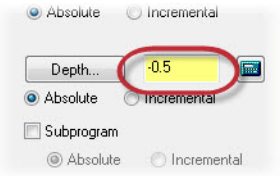
4 Create a 4" long, 0.75" drill named **0.75 drill**, and click **Finish**.



5 In the Comment box, type **0.75 leader pin drill - LEADER PINS**.

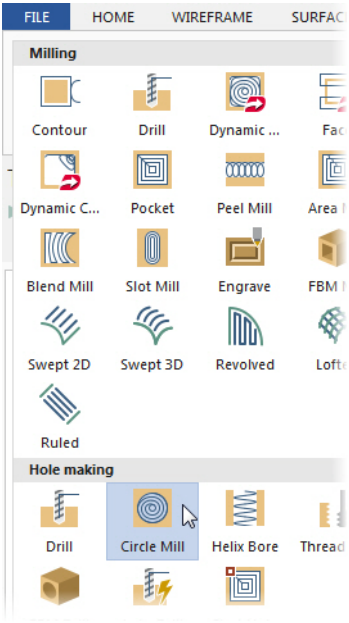


6 In Linking Parameters, set Depth to **-0.5**, all settings to **Absolute**, and click **OK**.



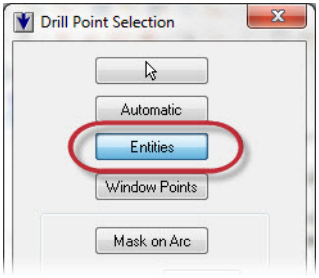
Creating the Leader Pin Counter Bore Rough Operation

1 From the Toolpaths tab, choose **Circle Mill** in the **Hole making** section of the **2D** gallery.



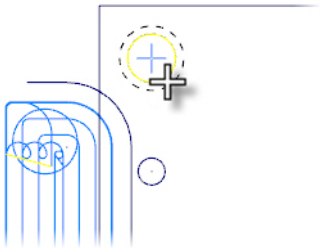
The Drill Point Selection dialog box opens.

2 Select **Entities** in the dialog box.

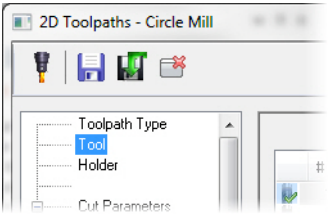


3 In Mastercam's graphics area, choose the hole shown, and click **OK**.

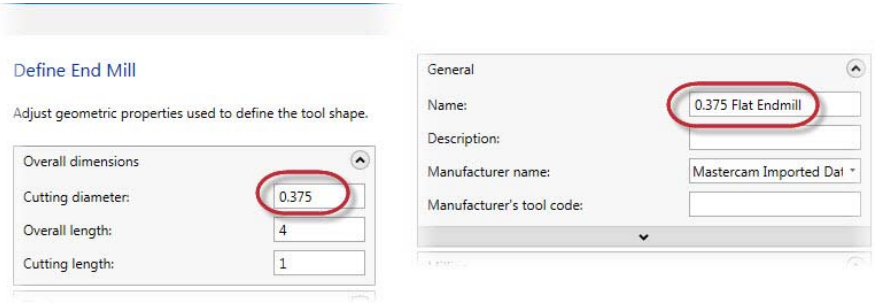
The 2D Toolpaths - Circle Mill dialog box opens.



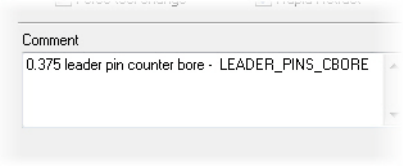
4 In the dialog box, select the **Tool** page.



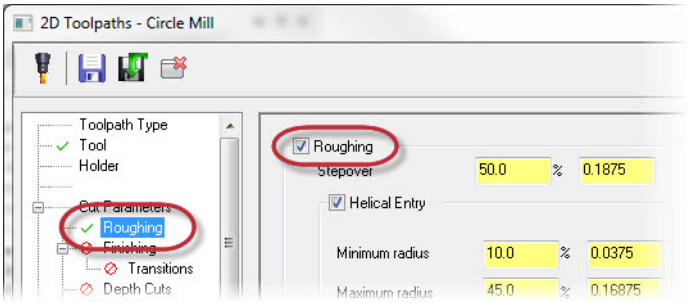
5 Create a **0.375 Flat Endmill** tool, and click **Finish**.



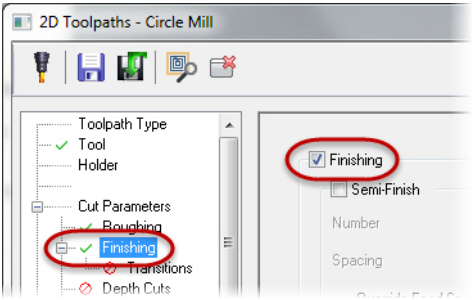
6 In the Comment box, type **0.375 leader pin counter bore - LEADER_PINS_CBORE**.



7 On the **Roughing** page, turn on **Roughing**.

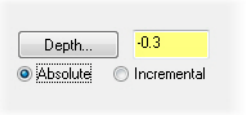


8 On the **Finishing** page, turn on **Finishing**.



9 Turn on **Finish**, and set Number to **1**.

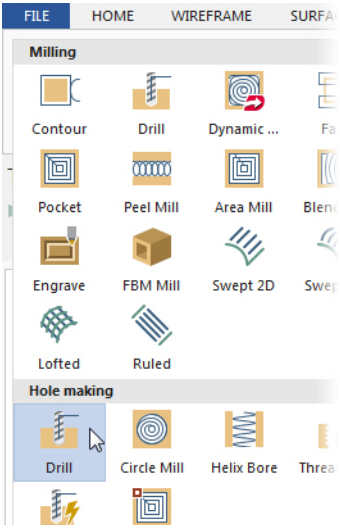
10 In Linking Parameters, set Depth to **-0.3**, all settings to **Absolute**, and click **OK**.



Creating the Assembly Screw Drill Operation

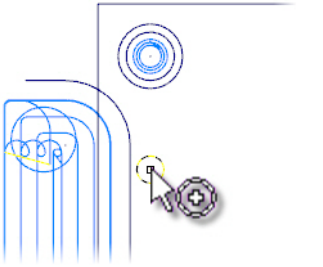
1 From the Toolpaths tab, choose **Drill** in the **Hole making** section of the **2D** gallery.

The Drill Point Selection dialog box opens.

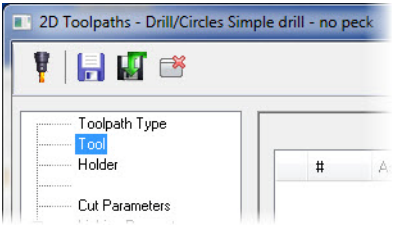


2 In Mastercam's graphics area, choose the hole's center, as shown, and click **OK** in the dialog box.

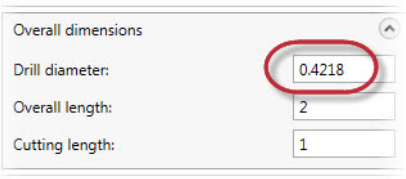
The **2D Toolpaths - Drill/Circles Simple Drill - no peck** dialog box opens.



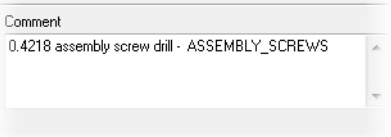
3 In the dialog box, select the **Tool** page.



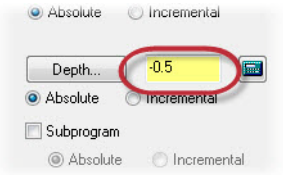
4 Create a 0.4218 drill, name it **0.4218 drill**, and click **Finish**.



5 In the Comment box, type **0.4218 assembly screw drill - ASSEMBLY_SCREWS**.



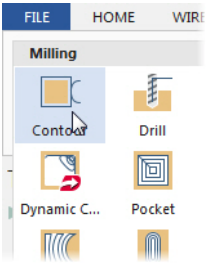
6 In Linking Parameters, set Depth to **-0.5**, set all to **Absolute**, and click **OK**.



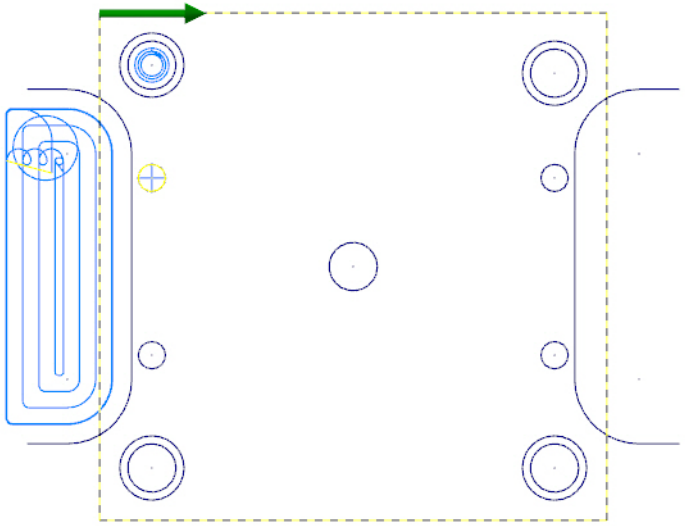
Creating the Outer Contour Operation

1 From the Toolpaths tab, choose **Contour** in the **2D** gallery.

The Chaining dialog box opens.

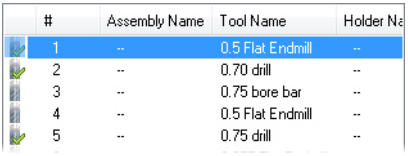


2 Chain the outside of the part, as shown, and click **OK**.

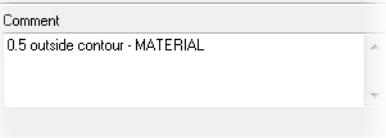


3 In the Contour dialog box, select the **Tool** page.

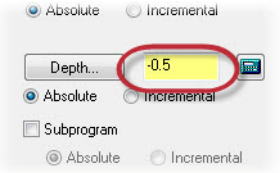
4 Select the **0.5 Flat Endmill**.



5 In the Comment box, type **0.5 outside contour - MATERIAL**.



6 In Linking Parameters, set Depth to **-0.5**, set all values to **Absolute**, and click **OK**.

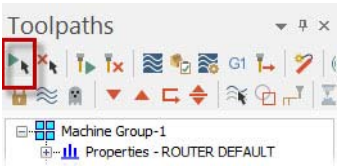


Exercise 3: Exporting the Operations

Now that you have created the operations needed for the parts, you must export them to a library that you can use when mapping level names to operations.

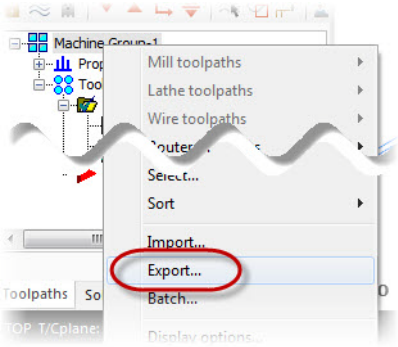
1 In Toolpaths Manager, click **Select all operations**.

Mastercam selects all operations in the machine group.



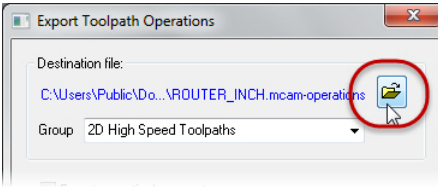
2 Right-click **Machine Group-1**, and choose **Export** from the pop-up menu.

The Export Toolpath Operations dialog box opens.

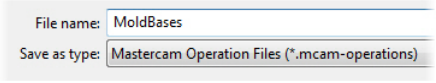


3 Click the **Select Operations File** button.

The **Export toolpath operations** dialog box opens.



4 In the **File name** box, type **MoldBases**, and click **Save**.



IMPORTANT: When saving your operations, never choose one of the libraries that was installed with Mastercam. Doing so will add invalid operations to the chosen library. Always give your operations library a custom name.

5 In the Export Toolpath Operations dialog box, click the **Apply** button.

Mastercam saves the operations to the selected library.

6 Dismiss the message box that appears, and then click the **Cancel** button.

You are now ready to run ATP and map the operations to the levels in your part files.

LESSON 5
Using ATP - Mastercam Example

In this lesson, you create an ATP strategy for generating the mold base project's final files. This process is similar to what you did to create a cabinet strategy. In the Mastercam case, however, you set up a file list in place of a cutlist.

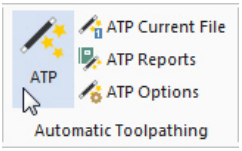
NOTE: This section assumes that you are continuing the tutorial from Lesson 4. If you are starting this section without completing Lesson 4, open the file `1.mcam` included with this tutorial.

Exercise 1: Creating a Strategy

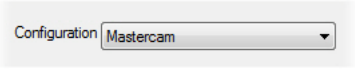
To create a strategy, you must tell ATP about the files to use, as well as set the appropriate options, and map operations to levels.

▶ Creating the File List and Setting Options

1. From the Machine tab, choose **ATP**. The Mastercam ATP window opens.

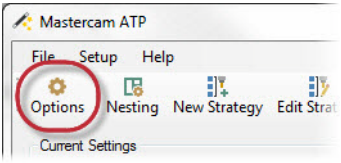


2. If necessary, change Configuration to **Mastercam**, and click **Yes**. (You must restart ATP if you make this change.)

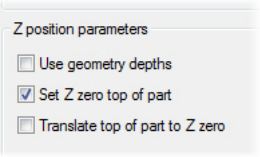


3. Click the **Options** button.

   The Options dialog box opens.

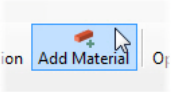


4 Turn off the **Use geometry depths** option (the **Set Z zero top of part** option should be on), and click **OK.**



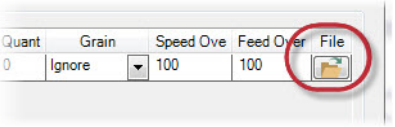
You are not using the **Use geometry depths** option because the 2D geometry has no defined depths. In this case, ATP will get depths from each operation's Linking Parameters.

5 Click the **Add Material** button.

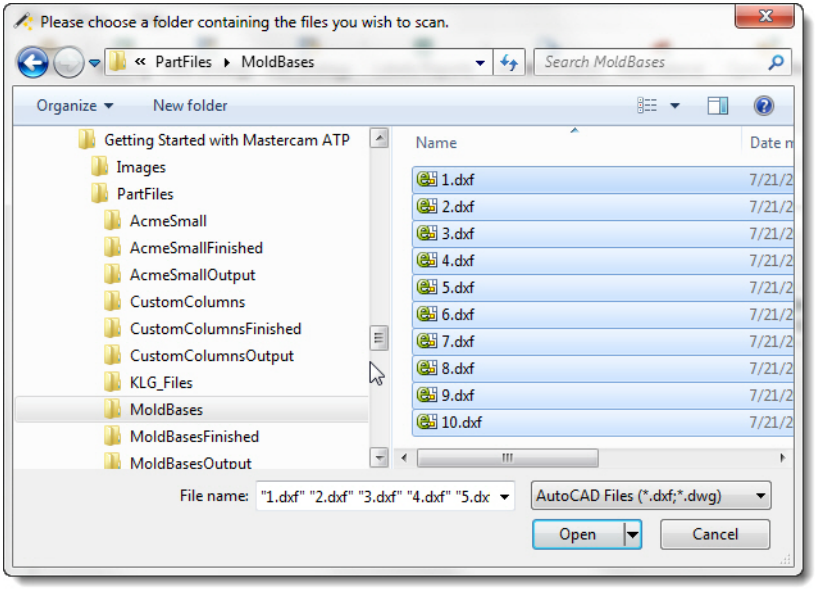


ATP adds the default material in the Options dialog box.

6 Click the **File** button, located to the right of the material row.



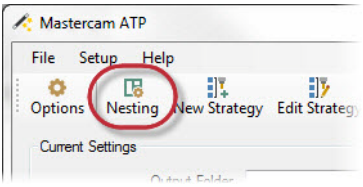
7 Select all `DXF` files in the `MoldBases` folder, and click **Open**.



ATP adds the files to its Files list.

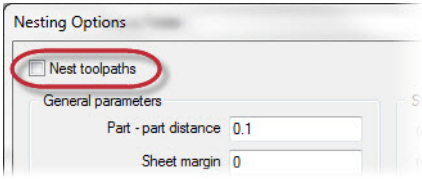
8 Click the **Nesting** button.

The Nesting Options dialog box opens.



9 Turn off the **Nest toolpaths** option, and click **OK**.

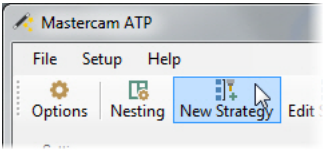
The mold bases do not need nesting.



Scanning for Levels

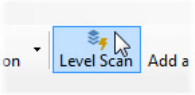
1 Click the **New Strategy** button.

The Strategy Explorer window opens.



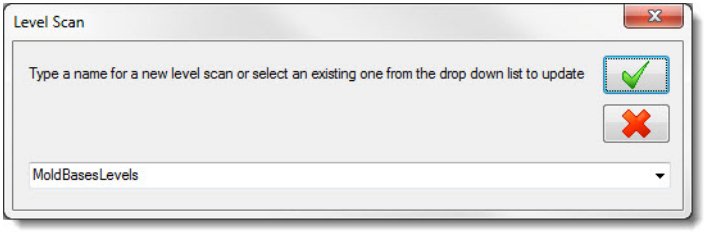
2 Click the **Level Scan** button.

The Level Scan window opens.



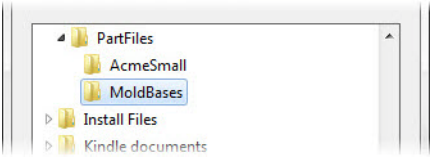
3 Enter **MoldBasesLevels** into the text box, and click **OK**.

The Browse For Folder dialog box opens.

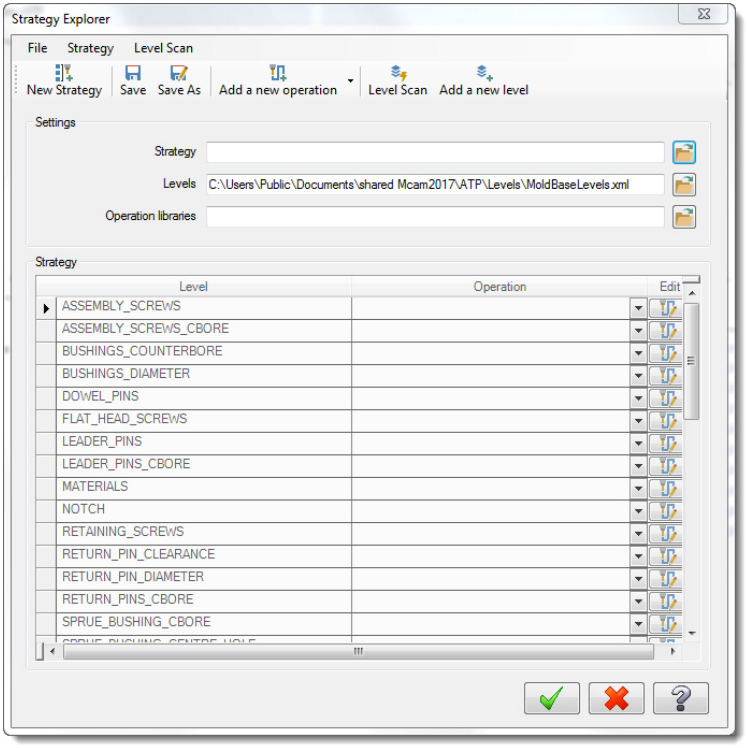


TIP: If you choose an existing file from the drop-down in the Level Scan dialog, you can merge the chosen file with levels ATP finds in the level scan.

4 Navigate to the `PartFiles\MoldBases` folder, which is supplied with this tutorial, and click **OK**.



ATP scans the `DXF` files in the selected folder, and builds a list of levels found in those files, as shown below.



Removing Unused Levels

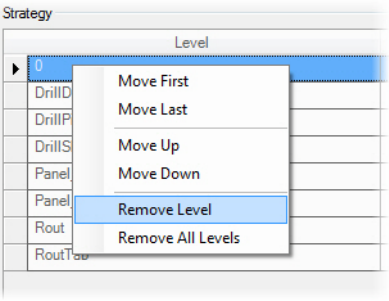
To keep this tutorial shorter, in Lesson 4, you created operations for only a subset of the many levels included in the files. To run ATP successfully, you must remove the unused levels from the level list.

1 If necessary, click the black arrow to the left of any unused levels.

ATP selects the row.

2 Right-click the selected row, and choose **Remove Level** from the pop-up menu.

ATP removes the level from the list.



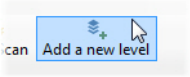
3 Delete all levels *except* ASSEMBLY_SCREWS, LEADER_PINS, LEADER_PINS_CBORE, MATERIALS, NOTCH, and SPRUE_BUSHING_CENTRE_HOLE.

TIP: Hold down the [**Ctrl**] key to select multiple levels.

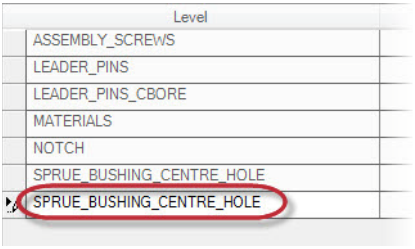
The Strategy list should then look as shown.

4 Click the **Add a new level** button.

ATP creates a blank row in the level list.



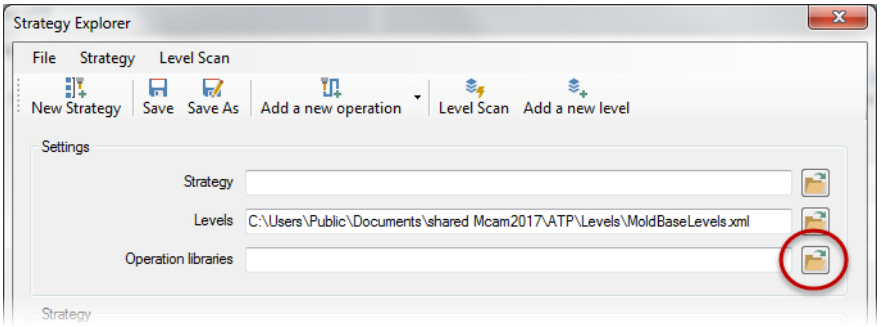
5 Click in the blank row, and type **SPRUE_BUSHING_CENTRE_HOLE** as the level name.



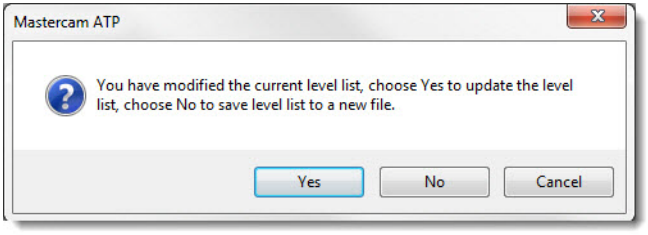
TIP: To avoid errors, you can copy the level name from the existing one, and then paste it into the new level.

Assigning Operations to the Levels

1 Click the **Select operations file** button, located to the right of the **Operation libraries** text box.



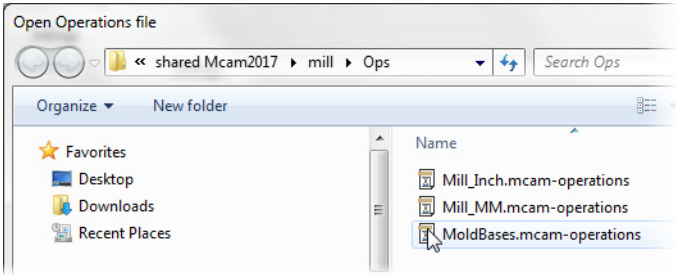
2 When asked if you want to save your changes to the level list, click **Yes**.



The Open Operations file dialog box opens.

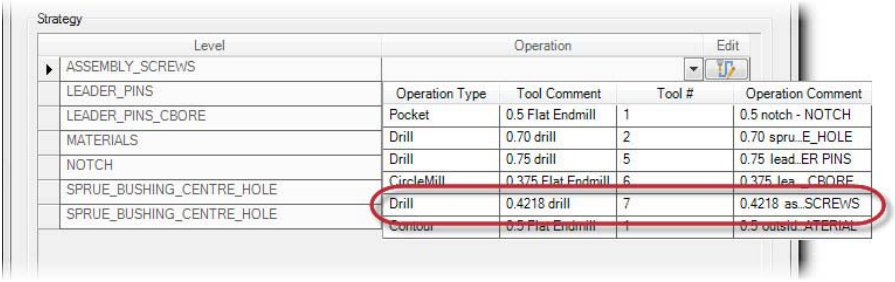
3 Select the `MoldBases.mcam-operations` file in your `shared Mcam2017\mill\Ops` folder, and click **Open**.

ATP loads the operation library.



4 In the Operation column for the ASSEMBLY_SCREWS level in the strategy, click the black drop-down arrow, and select the operation with the **0.4218 drill** tool comment.

ATP maps the drill operation to the ASSEMBLY_SCREWS level.



5 Assign the **0.75 drill** operation to the LEADER_PINS level.

6 Assign the **0.375 Flat Endmill** operation to the LEADER_PINS_CBORE level.

7 Assign the **0.5 Flat Endmill** (for the Contour Operation Type) operation to the MATERIALS level.

8 Assign the **0.5 Flat Endmill** (for the Pocket Operation Type) operation to the NOTCH level.

9 Assign the **0.70 drill** operation to the first occurrence of the SPRUE_BUSHING_CENTRE_HOLE level.

10 Assign the **0.75 bore bar** operation to the second occurrence of the SPRUE_BUSHING_CENTRE_HOLE level.

You should now have the operation mappings shown in the following picture.

Level	Operation	Edit
ASSEMBLY_SCREWS	0.4218 assembly screw ... ASSEMBLY_SCREWS	
LEADER_PINS	0.75 leader pin drill - LEADER PINS	
LEADER_PINS_CBORE	0.375 leader pin counter...EADER_PINS_CBORE	
MATERIALS	0.5 outside contour - MATERIAL	
NOTCH	0.5 notch - NOTCH	
SPRUE_BUSHING_CENTRE_HOLE	0.70 sprue rough drill -...HING_CENTRE_HOLE	
SPRUE_BUSHING_CENTRE_HOLE	0.75 sprue finish bore -...HING_CENTRE_HOLE	

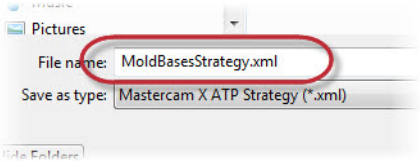
11 Reorder the operations as shown below.

Level	Operation	Edit
ASSEMBLY_SCREWS	0.4218 assembly screw... - ASSEMBLY SCREWS	
LEADER_PINS	0.75 leader pin drill - LEADER PINS	
LEADER_PINS_CBORE	0.375 leader pin counte...EADER_PINS_CBORE	
SPRUE_BUSHING_CENTRE_HOLE	0.70 sprue rough drill -...HING_CENTER_HOLE	
SPRUE_BUSHING_CENTRE_HOLE	0.75 sprue finish bore -...HING_CENTER_HOLE	
MATERIALS	0.5 outside contour - MATERIAL	
NOTCH	0.5 notch - NOTCH	

12 In Strategy Explorer's toolbar, click **Save**, and then click **Yes** if asked to save the level list.

The Save Strategy window opens.

13 Save the strategy under the name `MoldBasesStrategy`, and then click **OK** in the Strategy Explorer.

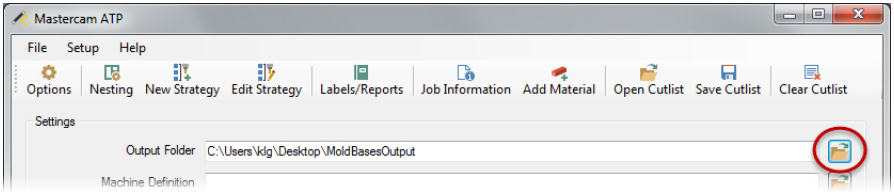


TIP: You can create multiple strategies for the same level list, allowing you to apply different operations in different situations.

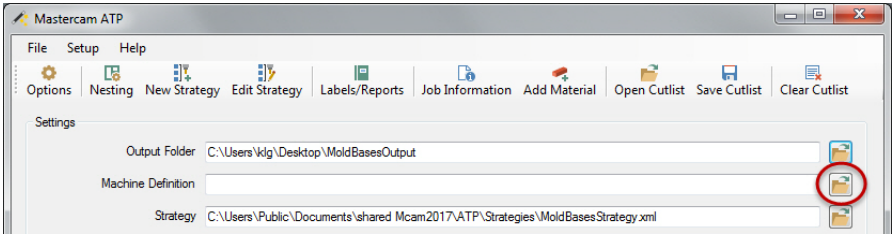
Exercise 2: Running ATP on the Files

1 Anywhere convenient, create a folder named `MoldBasesOutput`.

2 Click the **Output Folder** button, and set **Output Folder** to your `MoldBasesOutput` folder.

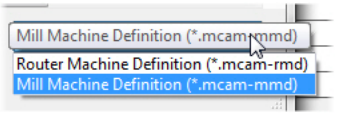


3 Click the **Select Machine Definition** button.



The **Select Machine Definition file** dialog box opens.

4 In the file type list, select the Mill machine definition type.



5 Select the default mill machine (MILL DEFAULT.MCAM-MMD), and click **Open**.

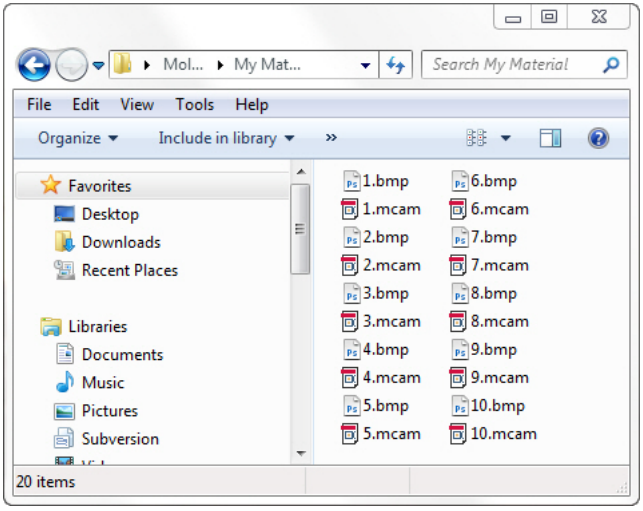
6 Click **OK** to run ATP with the settings and files you selected.

ATP automates Mastercam to create `MCAM` and `BMP` files for the parts in the `DXF` files.

IMPORTANT: Do not touch Mastercam until you see the message **Process Complete**.

7 Click **OK** to dismiss the **Process Complete** message.

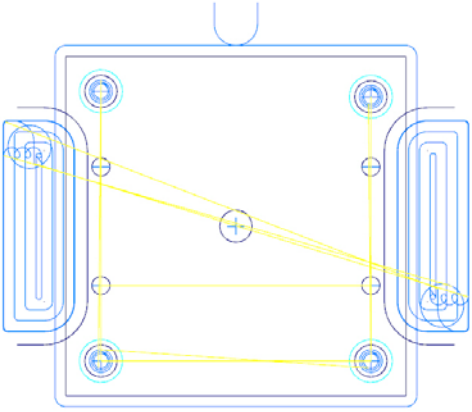
In the output folder you selected, you now have a subfolder. The folder is named according to the type of material you are using. This folder contains your files, as shown below.



NOTE: When you run ATP with nesting off, it does not create NCI and NC files. To create these files, you must post the ATP-generated part files manually.

8 Choose **File**, **Open** to open the file `1.mcam`.

This file shows the toolpaths that ATP created for the part's geometry.



9 Examine the other part files in the output folder to see how ATP applied toolpaths to the geometry.

Conclusion

Congratulations! You have completed the *Getting Started with Automatic Toolpathing* 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.

Mastercam Resources

Enhance your Mastercam experience by using the following resources:

- *Mastercam Help*— 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.
- *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.

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.

For assistance with installing Mastercam, its HASP or NetHASP, or to obtain more information on using Mastercam, contact your local Mastercam Reseller. If your Reseller is unavailable, you can call CNC Technical Support Services Monday through Friday, 8:00 a.m.–5:30 p.m., USA Eastern Standard Time.

When calling CNC Software for technical support, please follow these guidelines:

- Be sure you have already tried to contact your Mastercam Reseller.
- Provide the serial number of your HASP or NetHASP.
- Be ready to describe the problem in detail. Write down what happened, particularly if you cannot call immediately after the problem occurs.
- Be in front of your computer when you call.
- If possible, try to duplicate the problem before calling. Our Support Services technician may require you to duplicate the problem while you are on the phone.
- When you call, have ready a complete description of your hardware, including your operating system (OS), central processing unit (CPU), graphics card and settings, and memory.

You can also leave a message for CNC Support Services twenty-four hours a day, seven days a week via our email or website addresses. When sending email, please include:

- The serial number of your HASP or NetHASP
- Telephone number and contact information where you can be reached
- Files required to reproduce an issue

TIP: This utility makes it easy to provide your Reseller or CNC Support Services with a file attachment that contains the information they need. For more information on using Zip2Go, please refer to Mastercam Help.

Important Contact Information	
Address	CNC Software, Inc. 671 Old Post Road Tolland, Connecticut 06084-9970 USA
Phone	(860) 875-5006
Fax	(860) 872-1565
FTP Address	ftp://ftp.mastercam.com
Internet Address	http://www.mastercam.com
Email	support@mastercam.com

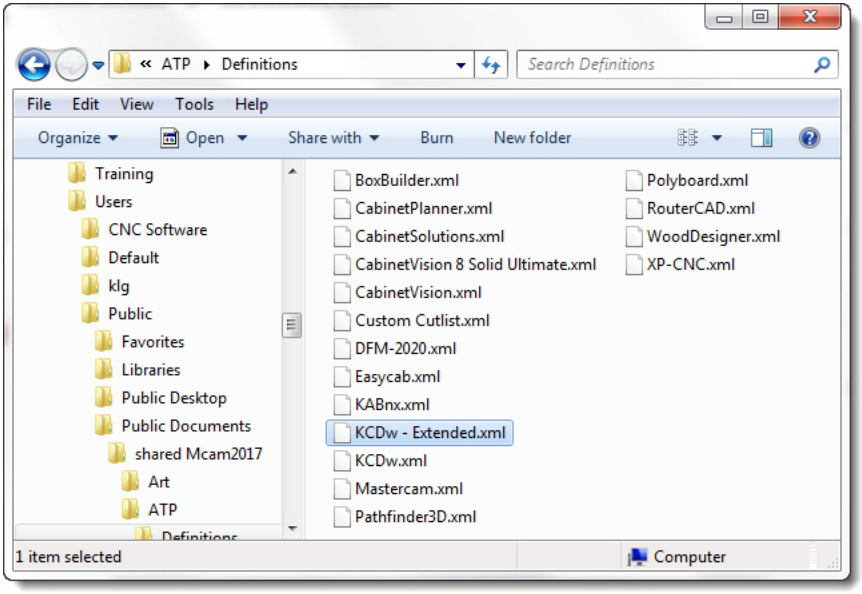
APPENDIX A
Custom Columns

There may be times when the cutlist you receive for a project has an extra field. Consider, for example, a KCDw case where your cutlist contains extra data that specifies an edge banding code. Because the KCDw definition installed with ATP does not expect this extra data, if you try to load the cutlist in ATP, you get an error.

To solve this problem, you can add a custom column (or columns) to the cutlist specification in the KCDw definition. The following exercises show how to do this for a KCDw project. The steps are similar for other ATP definitions.

Exercise 1: Adding a Custom Column to an ATP Definition

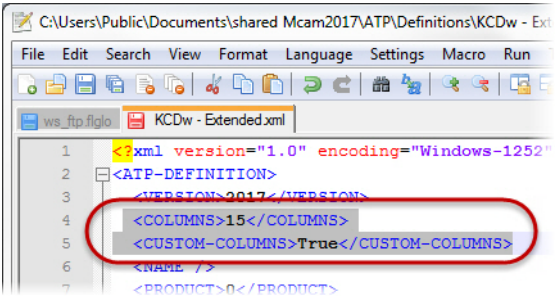
1. In Windows Explorer, browse to the folder that contains the ATP definitions (usually `C:\Users\Public\Documents\shared Mcam2017\ATP\Definitions`).
2. Make a copy of the `KCDw.xml` file, and name it `KCDw - Extended.xml`.



3. Open the copy into an ASCII text editor, such as Notepad.

4 In the **<COLUMNS>** tag, change the value from **14** to **15**, which adds a custom column to the cutlist. (See the image below.)

5 Change the **<CUSTOM-COLUMNS>** tag from **False** to **True**.



6 Save the file.

Exercise 2: Generating the Output Files

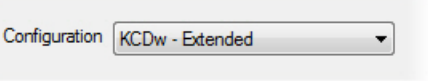
To get the new column data into your output files, you must run the cutlist containing the new information. Before you can do this, however, you must create your operations library, level list, and strategy file. You saw how to do this in Lesson 2 and Lesson 3. Rather than making you go through the steps needed to generate these files, this tutorial supplies them for you. In the following procedure, you copy the files where they belong and generate your new output.

1 From this tutorial's `CustomColumnsFinished` folder, copy `KCW_EBC_AcmeSmall.mcam-operations` to `C:\Users\Public\Documents\shared Mcam2017\router\OPS`.

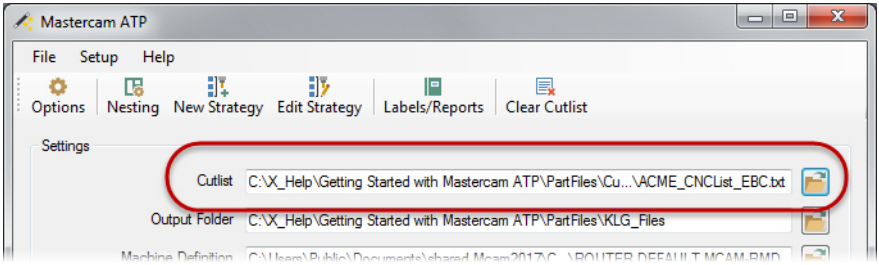
2 From this tutorial's `CustomColumnsFinished\ATP` folder, copy the `Levels` and `Strategies` folders to `C:\Users\Public\Documents\shared Mcam2017\ATP`. (If you are asked to confirm folder replacement, click **Yes**.)

The ATP folder now contains the strategy and level list files that you need to run the cutlist that contains the added column.

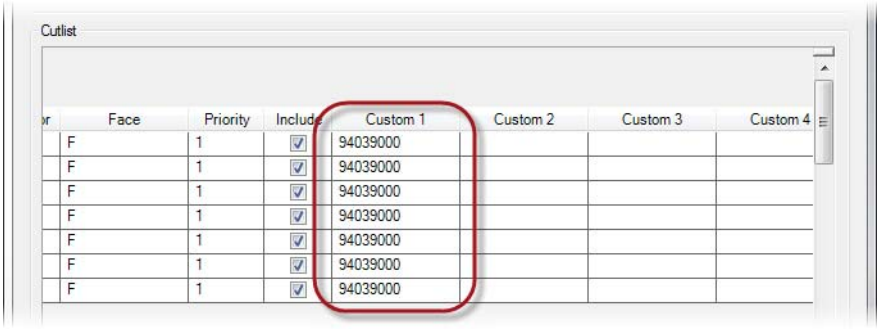
3 Start ATP, and ensure that **Configuration** is set to **KCDw – Extended**. This is the KCDw definition that you modified in the previous procedure. Remember, you need to restart ATP if you change your configuration.



4 Set **Cutlist** to the file `ACME_CNCList_EBC.txt`, which is included in this tutorial's `CustomColumns\AcmeSmall` folder. This file represents a cutlist that has additional data—in this case, an edge banding code.



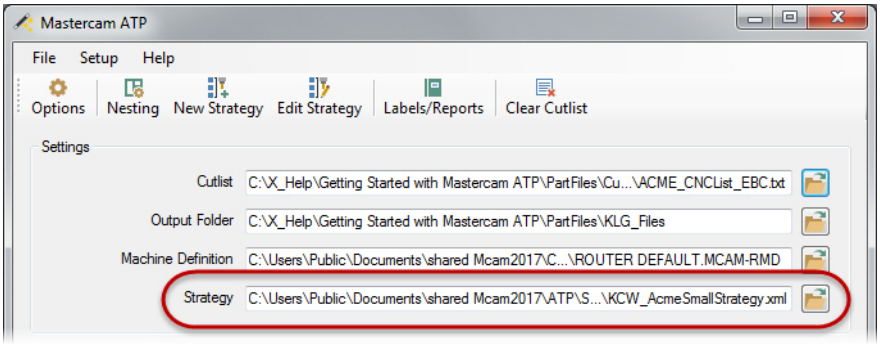
5 In the Cutlist pane, use the horizontal scrollbar to view the **Custom 1** column, and notice the edge banding code, **94039000**.



6 Set **Output Folder** to wherever you want ATP to place the generated files.

7 Set the **Machine Definition** to the `ROUTER DEFAULT.MCAM-RMD` definition supplied with Mastercam.

8 Make sure the **Strategy** is set to `KCW_AcmeSmallStrategy.xml`.



9 Click the **OK** button to run the extended cutlist.

ATP generates the output files.

10 In the output folder, find one of the material XML files (for example, `1_4 Melamine.xml`), and load it into a text editor.

Find the custom column definition containing the edge banding code. You can use this information in reports, as you see in the following steps.

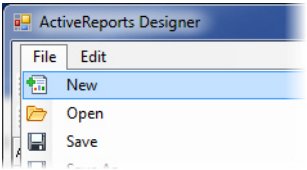
```
77  <PART-DXF>ACME_1_Back_001F.dxf</PART-DXF>
78  <PART-DXF-PATH>C:\Users\caw\Desktop\PartFiles\Custom C
79  <PART-FACE>F</PART-FACE>
80  <PART-MATERIAL>1/4 Melamine</PART-MATERIAL>
81  <PART-MCX-FILE>C:\Users\caw\Desktop\output\1_4 melamin
82  <CUSTOM-1>94039000</CUSTOM-1>
83  <CUSTOM-2 />
84  <CUSTOM-3 />
85  <CUSTOM-4 />
86  <CUSTOM-5 />
```

Exercise 3: Using Custom Data in a Report

As you learn in this exercise, the new edge banding code can now be used in your reports.

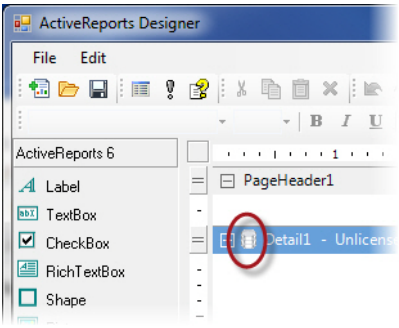
1 From the **Mastercam 2017/Utilities** folder in the Start menu, run **Active Reports Designer**.

2 Choose **File, New** to start a new report.



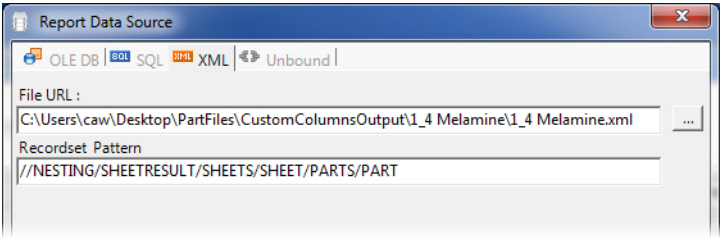
3 Click the data source icon on the **Detail1** header.

The Report Data Source dialog box opens.



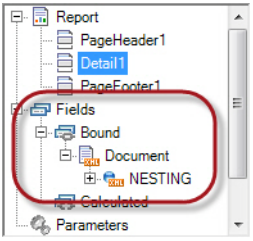
4 On the XML tab, set **File URL** to the XML file you located in Step 10.

5 Set **Recordset Pattern** to **//NESTING/SHEETRESULT/SHEETS/SHEET/PARTS/PART/** (must be uppercase), and click **OK**.

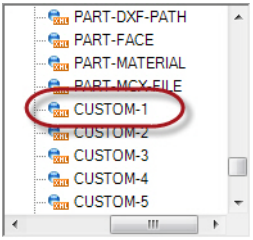


6 In Report Explorer, open the **Fields** node and then the **Bound/Document** node.

Notice the **NESTING** node. This is the first node in the recordset pattern you entered in Step 5.



7 Follow the recordset path (open **NESTING**, then **SHEETRESULT**, and so on) down to **CUSTOM-1**, which is the data element for the added edge banding code.



You can drag this data element into your report where needed.

For more information on ActiveReports, please refer to the Mastercam Help or the *Creating Setup Sheets with ActiveReports Designer* tutorial, available on 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