Managing retracts from ID toolpaths

Recently we've heard from Mastercam Lathe users who are concerned that at the end of an ID operation, Mastercam appears to retract directly to the home position while seemingly still buried in the part, without first pulling out of the part in Z. We thought this would be a good opportunity to clear up some confusion about when and how Mastercam automatically creates such moves, and when it does not.

Simple ID toolpath

Consider the following toolpath. It simply creates an ID finish pass along a single chained line. The actual part isn’t represented, only enough geometry to create the desired operation.

Without creating a stock model or defining any reference points,
Mastercam does not create any tool motion following the initial move away from the part. Once the tool pulls off the part as shown here, it will therefore move directly to the home position. When you post it out, the code will look like this:

```
14 G50 X0 Y0 Z0
15 G96 S8200
16 G99 G1 Z-.5 P.005
17 Z-.4.
18 X1.3585 Z-3.9293
19 G28 U0, W0, D05
20 T08200
21 X30
22 %
```

You can see that line 18 is the last programmed position. Since Mastercam doesn’t know anything more about your part or application, this is all it can do.

**Adding reference points**

The more information you give Mastercam to work with, the better job it can do figuring out what you want. The most direct way to handle the above situation is to include a reference point that pulls the tool out of the part when the operation is finished. (You can also create reference points for approach moves. These can be different points than for retracts.)

One point of confusion for some users is that they do not understand that unless the checkbox in front of the **Ref point** button is selected, Mastercam will not create a
reference point, even if one has been defined.

Once they are in the **Reference Points** dialog box, users can select checkboxes to enable reference locations along one or both axes; and for approach moves, retracts, or both. In the example above, we have chosen to create only a Z-axis retract move. At the end of the toolpath, the tool will retract to the selected Z clearance distance at whatever diameter position it was at when the operation ended.

Adding the reference point as shown above results in the following toolpath:

When you post it out, you will see the following:

```
16 G99 G1 Z-.5 R.005
17 Z-4.
18 X1.3586 Z-3.9293
19 G8 Z.2
20 G28 U0. W0. M05
21 T0200
22 M30
23 G
```

The retract move out of the part is on line 19. Some users might expect this move to appear as the U-W that is output with the G28, but Mastercam outputs the reference point in the NCI as a regular toolpath move, and our generic posts will output it as shown above.

The above toolpath is an improvement, but Mastercam still doesn’t know enough about your part to know if the reference point will do the job. For example, if your part actually looks like this, even though you have a reference point in Z+, the tool will still rip through the part.
Adding a stock model

The best solution is to define an accurate stock model and use it for all your lathe toolpaths. Mastercam will add additional moves so that the tool does not collide with the stock model, and warns you if any other tool movements will result in a collision. It does this regardless of which geometry has been chained or whether reference points have been defined.

Defining a stock model and regenerating the toolpath results in the following:

As you can see, Mastercam adjusts the retract move so that it actually clears the part. Mastercam will add extra clearance moves either before or after the move to the reference point, as required. In this example, it also adjusted the entry move so that it clears the part, even though no approach reference point was even defined. Collision avoidance moves are applied to all moves between toolpaths when you define stock. As a best practice, of course, most users would want to use a reference point to pull the tool out even farther from the stock than the minimum distance shown here, but that additional distance is your own judgment.
When you post out this toolpath, you will see the following:

```
17 G99 G1 Z-.5 F.005
18 Z-.4.
19 X1.3586 Z-.3.9293
20 G60 Z.2
21 Z1.2594
22 G20 Y0. MD. MD.
23 T0200
24 M30
25 S
```

Notice that the move out of the part is actually output as two moves. The first, on line 20, is the original reference point. The second, on line 21, is the additional move that Mastercam automatically added to clear the stock model.

**Not posting out all the operations (X5 and earlier)**

For users who are using Mastercam X5 and earlier, another circumstance that can cause problems is when you create several operations, but do not post them all. In this case, if Mastercam has automatically created additional clearance moves as part of several operations—such as the first or last operations—and you do not post those out, it is possible that the operations you DO post might still collide with the part.

This is particularly likely to happen if several consecutive operations use the same tool. When this happens, Mastercam might not include a retract to the tool change position between the operations. If you post only one of the operations, you might not get the expected retract move to a tool change position.

This problem has been corrected in Mastercam X6 when stock has been defined.

**Enable home position clearance moves (X4 and earlier)**

Make sure the **Write home position clearance moves** option is selected in the **Machine Group Properties**. If this is turned off, even if you have a stock model defined, Mastercam will not write the extra clearance moves. (This setting was removed in X5.)

---

**Did you know?**

CNC Software, Inc. is just about ready to release Mastercam Swiss Expert 2012. For more information contact your local Reseller.
Summary

Keep in mind the following lessons:

- Many Lathe users assume Mastercam automatically creates a retract move out of an ID toolpath because they usually work with a stock model. Then when they do not use a stock model, they are surprised that the move isn’t created as expected.

- Creating a stock model is the safest way to work. However, if this isn’t practical, make sure to use reference points, and to set them at appropriate locations.

- Understand that the chained geometry might not accurately represent the extent of the part. Set reference locations accordingly.

- Moves to/from a reference location will typically be output as standalone G0 moves, and not as part of the G28 U-W line.

- With X5 and earlier, be especially careful when you do not post out all the operations. To be safe, use reference points and carefully Backplot/Verify the individual operations.

Bonus tip

Did you know that in the machine definition, you can define default reference points for both ID and OD operations?

Did you know?

Mastercam will be at SolidWorks World 2013 in Orlando, Florida. For more information click on the link below:

http://www.solidworks.com/sww/
1. In the **Machine Definition Manager**, click the **Edit axis combinations** button.

2. If necessary, select the desired axis combination.

3. Click **Reference Points**.

4. Set default reference positions for ID and OD work, for either approach or retract moves.
Because these are specific to the axis combination, you can even define different locations for main spindle/subspindle, and upper/lower turrets. Important—because you are in the machine definition, the coordinates in the above dialog box are in world X-Y-Z coordinates, not lathe X-Z coordinates. TT0112

File > Open differences between Mastercam X-style and Windows-style dialogs

If you're running Mastercam X6 on Windows VISTA or Windows 7, you may think there are no differences between Mastercam X and Windows standard styles for the File > Open dialog box. You can choose the style in the Control Panel applet:

However, there are two subtle differences.

- The Mastercam X style has the Options button which appears automatically when certain file types are selected:

- It also automatically displays the Mastercam-specific folders in the navigation pane:

Did you know?

Mastercam attended the International Manufacturing Technology Show this year and Mastercam also had nearly 100 partners in attendance as well.
The difference in styles is more apparent when running Windows XP because you cannot run the Windows VISTA–style File dialogs.

**Tapered tools support in surface toolpaths**

Have you ever wondered which surface toolpaths support tapered tools?

The good news is that positive taper tools are supported in all surface toolpaths since Mastercam X3. This includes all styles of 3D HST, surface rough, and surface finish toolpaths.

The only caveat is with finish and rough flowline toolpaths. The positive taper tool is supported for gouge checking and for check geometry. However, the raw flowline passes do not support positive taper tools. These are the passes that are positioned based on the tool diameter and shape of the part. This means that you might not be able to machine with the taper portion of the tool.

*Did you know?*

Mastercam has online training courses called Mastercam University. Click on the link below to see the available classes and start enhancing your skills now.

[www.mastercamu.com](http://www.mastercamu.com)
Negative taper tools (often called dove mills) and two other undercut shapes—lollipop and slot mill—are also supported, but in a limited undercutting fashion. Support is limited to the following toolpaths:

- surface finish flowline
- surface finish contour
- surface rough flowline
- surface rough contour
- surface rough pocket

Pay special attention to motion between cuts. Disable gouge checking and select the geometry in such a way as to not gouge when transitioning between cuts.

Also, when creating a surface finish contour with these tools, consider whether a surface rough pocket operation might be a better choice. This may be especially useful in parts where the area available to approach the part is limited.

- Surface rough pocket operations let you use a start point, so you know that the tool will move to that point for each depth cut.
- If you turn off the **Rough passes** option, Mastercam will only create a finish toolpath.
- Depending on the geometry, you might be able to use gouge checking in this toolpath. ■ TT0312

---

**Turning off stock model display**

With the addition of the stock model in Mastercam X6, a question that has been coming up quite often is, "How do you display and hide the stock model?" Since the stock model is actually an operation, you can show or hide it like any toolpath.

- First, select the stock model(s) so that there is a green check mark on the icon:
HELP DESK

Use Update Folder instead of Migration Utility

Instead of using the Migration Utility, we suggest using Update Folder to convert the following file types:

- machine and control definitions
- posts
- tool libraries

Here is a simple process for updating X5 files to X6. This process works if the X5 files are in the standard folder locations.

1. Select File > Update Folder from the menu.
2. Set the options as shown here:

Did you know?

Mastercam just released Blade Expert. For more information on these products click here:

3. Select the `shared mcamx5` folder as the source folder and the `shared mcamx6` folder as the destination folder.

4. Check the **Apply this action to all subsequent conflicts** option or you will need to respond to this prompt repeatedly.
5. Choose the **Update and replace original file in destination folder** option so that only the updated file will be created in the X6 folder.

While you have the option of having Mastercam automatically make a backup copy of the original file in the destination folder, we recommend simply replacing it.

6. If you see the following warning when the MD/CD files are updated, click **OK**.

   ![Warning dialog](image)

   This does not harm or change the MD or CD file and nothing needs to be changed in the file. There may be several of these warnings, depending on the number of files in the X5 folder.

7. When the posts are updated, the Editor will display a log file with the UpdatePost results. Close the Editor.

The update is complete. ■ **TT0312**
“File > Save As” restriction for Windows 64-bit systems

You may have noticed that on Windows 64-bit operating systems, Mastercam X6 does not let you save a file to a previous-version Mastercam file, such as *.MCX-5.

File changes were needed for 64-bit systems that are not easily converted back to 32-bit format. To avoid confusion and possible part file corruption, when you are using Mastercam X6 on a 64-bit system you can only save X6 Mastercam part files.

To save an X6 file back to an earlier version, use File > Save As and save the geometry to an appropriate file type, such as IGES or Parasolid®. Then, use the earlier version of Mastercam to open that file. Toolpaths cannot be saved. ■ TT0112

Did you know?

Mastercam corporate office in Tolland, CT holds different training classes throughout the year. The next class features a 4-day course on Multiaxis on December 10-13. Click here:

Displaying the Art toolbar

Follow these steps to configure Mastercam so the Art toolbar is automatically displayed on startup.

To display the Art toolbar for only the current session, complete steps 1–4 only.

1. In Mastercam, select **Settings > Customize** from the menu.
2. Click the **Open** button:
3. Select **Mastercam-art.mtb**.
4. Click OK until you return to Mastercam.
5. Select **Settings > Configuration** from the menu.
6. Go to the **Start/Exit** page and click the **Select** button next to the **Toolbars** option:

![System Configuration](image)

1. Select **Mastercam-art.mtb**:

![Startup settings](image)
2. Click **OK** to return to Mastercam.

3. Select **Yes** when prompted to save the file.

---

**Updating C-Hooks to Mastercam X6**

To build a C-Hook that will run in Mastercam X6, you must use Visual Studio 2010 Pro or higher. Do not use the free Express versions, because those versions do not include the necessary MFC components.

Please contact your local Mastercam Reseller for the document that explains how to update an existing C-Hook to X6.

Since Mastercam X6 is now available for either Windows 32-bit or 64-bit operating systems, you need to build the C-Hook with the correct “bitness.”

You can use the Express versions of Visual Studio for creating NET-Hooks, but please realize that the functionality available in the NET-Hook API is a fraction of what is available for a C-Hook.

---

**Fixed segment length in “Refine toolpath” settings**

We’ve noticed that users are not using the **Refine toolpath > Smooth Settings > Use Fixed Segment Length** option. They seem to think that it would increase the distance between points, so if they didn’t get the segment length exactly right, the accuracy and finish of their part would suffer.

First, how do you access the Refine Toolpath dialog?

1. Choose a **Surface Rough**, **Surface Finish**, or **Surface High Speed** toolpath from the Mastercam Toolpath menu or toolbar.

2. Choose one of the following actions based on your toolpath selection:

   a. For tree-style toolpaths:
      
      i. Choose the Arc Filter/Tolerance page.
      
      ii. Enter a **Total tolerance** value.
      
      iii. Then click the **Refine Toolpath** button.
b. For tab-style toolpaths:
   
i. Choose the toolpath's parameters tab (for example, Rough radial parameters or Finish project parameters).
   
ii. Enter a **Total tolerance** value.
   
iii. Then click the **Total tolerance** button.

Note: To access the Refine Toolpath dialog box, the 3D Advanced Toolpath Refinement feature must first be activated for your Mastercam configuration. To activate the feature, run the Mastercam applet and enable it, or contact your local Mastercam Reseller for assistance.

Now that we're able to run the Refine Toolpath dialog, let's look at the Segment Length option:

However, note that the **Segment Length** value is the **maximum** distance between points.

Mastercam will still create additional points wherever the curvature requires it.

---

**Did you know?**

Mastercam has a YouTube page. Check out what some other customers are doing while using Mastercam. **Subscribe today by clicking here:**

[http://www.youtube.com/user/MastercamCadCam](http://www.youtube.com/user/MastercamCadCam)
**Canned rough and finish cycles only outputting longhand code**

If your Lathe canned rough and finish cycles are outputting longhand code, you need to select the **Control supports subprograms** option in the control definition. Even though no subprograms are added to the NC output file, the post uses these routines for back-end processing, so they must be enabled in the control definition.

Make sure you change the hard disk copy of the control definition so that all future files that use it will have the correct setting.

- To correct the hard disk copy, select **Control Definition Manager** from the **Settings** Menu. Then enable the **Control supports subprograms** option as shown.

- To correct an existing file, open the **Files** page in the Machine Group Properties and click the **Replace** button. Then enable the **Control supports subprograms** option.
Canned cycles will now output canned code. ■ TT0412

**Change in X6 for Mastercam event logging**

One change in X6 that you might have missed is an update to event logging.

In Mastercam X5 and earlier, the event logs were saved to the `\my mcax5\Event logs` folder. Each log was written to a new .xml file.

In Mastercam X6, however, Mastercam does not accumulate event log files. It clears out and reuses the same file every session.
Also, the event log has been moved to the user’s Local\Temp folder. For Windows 7, this is:

C:\users\<user>\AppData\Local\Temp

For Windows XP, it is:

C:\Documents and Settings\<user>\Local Settings\Temp  TT0212

**Updating to Wire TECH machine definitions**

Mastercam X5 introduced the new power libraries feature for Wire. This feature was implemented via new Mitsubishi and Makino machine definitions that had the word TECH in their names. For wirepaths created with these machines, the Wire/Power page lets you load settings from a technology library:

**Did you know?**

Mastercam now has a corporate office in China.
In Mastercam X6 MU2, it is important to realize that for existing Wire parts, it is NOT possible to replace an earlier wire machine definition with one of the new TECH machine definitions. This is not a bug or defect; it is caused by the addition of the new features.

If the machine definition was replaced in X5, X6, or X6 MU1, the operations will look OK; they will Backplot OK; and when you look at the Wire/Power page you will see the new Tech settings; but when you post the operations, the Tech settings will not be written to the NCI and the posted output will be incorrect.

In MU2, if you try to replace a non-TECH machine definition with a TECH machine definition, you will see the following message:
Users with legacy parts need to follow this workflow if they want to use those parts with the new TECH machine definitions:

1. Open the legacy part.
2. Create a new machine group with one of the new TECH machine definitions.
3. Delete all the operations from the original machine group.
4. Delete the original machine group.
5. Recreate the operations in the new machine group. Do not copy the operations from one machine group to the other. ■ TT0512

Hot Topics

**Mastercam for SolidWorks X6 released**

Mastercam for SolidWorks X6 was released to all customers on March 7, 2012 as a download from Mastercam.com. ■ TT0312

**What’s New DVD for Mastercam X6 does not install**

As has been discussed previously, Mastercam X6 features a new multimedia What’s New DVD instead of the PDF documents supplied with previous versions. By design, this DVD deliberately does not have an option to install on the local workstation. Since it includes many video clips, it would take a large amount of disk space were it to be installed. However, you can download an installer for the What’s New DVD from our website:


**POCO support files available on website**

The support files for the Pickoff/Cutoff utility are now available on our main website where they can be downloaded by any customer:

The package includes sample part files, a properly modified post, machine/control definitions, and PDF documentation.

**SolidWorks files with multiple configurations**

You may have noticed that when opening a SolidWorks file that has multiple configurations, sometimes you are asked to choose a configuration and sometimes not.

Just like with Mastercam's operations, the SolidWorks file needs to be clean to work. Mastercam only recognizes SolidWorks configurations that are clean, meaning they do not need to be regenerated using the SolidWorks' **Rebuild** option.

Finding the dirty configurations is easy:

1. Open the file in SolidWorks.
2. Select each configuration and look at the file name in the title bar:

   ![SolidWorks file with rebuild and rebuild options](image)

   - If a configuration has an asterisk (*) next to its name, it needs to be rebuilt.

**Toolpath transform methods: NCI vs Geometry**

Have you ever wondered what the differences are between using the **NCI** or **Geometry** options when transforming a toolpath?

**Did you know?**

Mastercam will support the following translators in the next Maintenance update:

- SolidWorks2013
- SolidEdge ST5
- Rhino 5
- ACIS R23
NCI transforms

The NCI option will copy and transform the source operation’s NCI lines. This is the way that X4 worked when the Create new operations and geometry option was off. Mastercam reads each NCI line from each source operation, extracts all the values from each line, and transforms them based on the transform parameters.

It’s fast when the transform method is Coordinate; even faster when transforming by Toolplane.

For Translate and Rotate transforms, the NCI option is accurate, but there are many issues when using this option with Mirror transforms, such as:

- maintaining cutting direction
- applying cutter compensation direction
- avoiding creating invalid left-handed views
- maintaining depth cut order
- converting rapid retracts into feed moves
- converting feed moves in Z into rapid moves

Contour and pocket toolpaths have their depth cuts marked as such in the NCI, but
multi-surface toolpaths do not. This means that a mirror transform of a multi-surface toolpath with the Reverse option selected would cut from bottom to top.

Beginning with X5, the NCI option for Mirror transform operations is not available; you must use the Geometry option.

**Geometry transforms**

Use the Geometry method to copy and transform the source operation’s parameters, geometry, and toolpath references and to generate an entirely new NCI section.

This method copies and transforms the geometry needed for toolpath creation. For toolpaths that reference solids, Mastercam will only create and copy the surfaces, chains, and points needed to create the toolpath; it will not copy the solid itself.

The copied geometry is temporary when the Create new operations and geometry option is off, and permanent when that option is selected.

Geometry is the only mode available when mirroring operations. It may be slower to copy, mirror, and regenerate multi-surface toolpaths, but the resulting toolpath will be correct. The Geometry method ensures that all cutting directions, cutter comps, lead-ins and lead-outs, start points, etc., come out as if the user mirrored the geometry and programmed it manually. Wireframe toolpaths are quick and speed is not an issue.

We suggest always using the Geometry method for all 3 transform types—Translate, Rotate, and Mirror—on operations that either use wireframe geometry or do not have a large number of surfaces. Accuracy trumps speed in our book every time. ■ TT0312

**Automated response from Tech Support**

If you receive an automated reply from Tech Support, you may have noticed that the message has changed. Among the changes:

- We are encouraging users to contact their Reseller directly with a quick link to the Find a Reseller page on the website.
- We also want more information up-front, and list the most helpful items.
EULA functionality reminder

The End User License Agreement (EULA) is a legal document spelling out the rights and limitations of each party with regard to using Mastercam software. Beginning with X6, the EULA is now displayed several times:

- During the installation, Mastercam displays the EULA and requires that the user select **Agree** before allowing the installation to continue.
- Mastercam displays the EULA the first time it is started. There is no way to opt out of seeing the EULA.
- Mastercam also displays the EULA at start-up three more times at random.
The user must accept the EULA each time; if they click **Cancel** or click on the dialog box’s red X, Mastercam will re-set the counter, and the EULA will continue to appear.

Displaying the agreement multiple times assures CNC Software that the actual end user has seen and agreed to the EULA. ■ TT0412

---

**Support news for Autodesk 2013**

Autodesk has just released its 2013 products. Here is important information on how we will support them in Mastercam.

**AutoCAD 2013**

Due to a change in the file format of AutoCAD® DWG/DXF/DWF files, we must wait for updated libraries to be available before we can convert these files. We expect the libraries will be available to us in a few months.

Customers who receive AutoCAD 2013 files will need to request that the files be sent from an earlier version until we can support the new version.

**AutoCAD Inventor 2013**

Mastercam customers who install Autodesk® Inventor® 2013 can read Inventor 2013 files immediately. You can also download the free Inventor View 2013 program from our website, which allows you to read Inventor 2013 files.

<table>
<thead>
<tr>
<th>X6 Translator Update - AutoDesk Inventor View 2012 x86</th>
<th>inventorview-2012-x86.zip</th>
<th>12/28/11</th>
<th>327MB</th>
<th>info</th>
</tr>
</thead>
<tbody>
<tr>
<td>X6 Translator Update - AutoDesk Inventor View 2012 x64</td>
<td>inventorview-2012-x64.zip</td>
<td>12/28/11</td>
<td>508MB</td>
<td>info</td>
</tr>
</tbody>
</table>

Also note that the Mastercam Direct for Inventor add-on that we provide is now immediately capable of running in Inventor 2013. The add-on is also Autodesk Inventor 2013 “Certified.” ■ TT0412
### Operating system compatibility with Mastercam versions

The following charts have been updated for the most recent Mastercam version as well as expected future requirements:

<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Windows® 7</td>
<td>Windows Vista®</td>
<td>Windows® XP</td>
<td></td>
</tr>
<tr>
<td></td>
<td>64-bit 32-bit</td>
<td>64-bit 32-bit</td>
<td>64-bit 32-bit</td>
<td></td>
</tr>
<tr>
<td>2013</td>
<td>X7 (when released)</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>2011</td>
<td>X6</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>2010</td>
<td>X5</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>2009</td>
<td>X4</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>2008</td>
<td>X3</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>2006</td>
<td>X2</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>2005</td>
<td>X</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>2002</td>
<td>V9</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>2000</td>
<td>V8</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>1998</td>
<td>V7</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>1996</td>
<td>V6</td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

**Supported:** The selected version of Mastercam has been successfully tested with the operating system.

**Not supported:** The selected version of Mastercam has either not been tested or has been tested and failed.

**Incompatible:** The selected version of Mastercam is incompatible with the operating system and will not run.

---

*Mastercam is a registered trademark of CNC Software, Inc. Windows® is a registered trademark of Microsoft Corporation in the United States and other countries. SolidWorks® is a registered trademark of Dassault Systèmes SolidWorks Corp. All other company and product names are trademarks or registered trademarks of their respective owners.*