



PartMaker 2014

World-leading CAM software for production machining

www.partmaker.com



Now Available

What's New

PartMaker 2014



Important User Notices

Copyright © 1991-2013 Delcam plc. All rights reserved.

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

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

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

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

Acknowledgements

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

Patents

PartMaker software is subject to the following patents:

Patent granted: US 6, 112, 133 Visual system and method for generating a CNC program for machining parts with planar and curvilinear surfaces

Patent granted: US 6, 741, 905 Visual system for programming of simultaneous and synchronous machining operations on lathes

Contents

Introduction	1
Installing PartMaker	2
System requirements	3
Installing PartMaker on a stand-alone PC.....	3
Installing the license file (PAF).....	5
What's New in PartMaker 2014	7
Tool enhancements	7
New tool types	7
End Mill tool shapes	9
Manage Tools Files dialog.....	10
Working with solid models	11
Setting transparency for solid models	11
Setting the Z Face coordinate in PartMaker/Mill	12
Calculating the Tombstone Index Angle for dependent faces	17
Projecting non-cylindrical surfaces in a Mill Cylinder Face window	19
NC Program files	21
NC Program Info dialog	21
NC Program Window toolbar	22
Aligning multi-channel NC Programs.....	23
Searching NC Program files	25
Setting global coordinates for 5-axis NC Program output.....	26
New method of calculating pocketing toolpaths	27
User interface changes	28
Undoing and redoing actions.....	28
Leads In and Leads Out	30
Displaying unlicensed tools	31
Disabling invalid settings for Turning tools	32
Profile Group Parameters, Turning dialogs	32
Displaying holes and profiles from the work group	33
New layout in the View menu	33
Keyboard shortcuts for menu options	34
Changes to the Setup dialog in PartMaker/Mill.....	34
Advanced Surface Machining (ASM)	35
Vortex-style toolpaths.....	36
Step cutting settings	37
Changes to the Pattern finishing strategy.....	43
ConfigPost enhancements.....	43

Help and support

44

Help and documentation	44
Training for PartMaker and ConfigPost.....	45
Technical support for PartMaker.....	45

Introduction

PartMaker is the world's leading CAM system for automating the programming of production machining on CNC milling machines, lathes, Wire EDMs, Turn-Mill centers and Swiss-type lathes.

PartMaker automates the tedious task of manual part programming by providing an easy-to-use graphical interface and powerful tools and features.

With PartMaker you can quickly define part geometry, cycle, tool, and material information, and all other parameters associated with machining a part. What's more, you can also save frequently used tool, cycle, and material information in a database to recall and reuse at any time.

Acquired by Delcam plc in 2006, PartMaker Inc. is a wholly owned subsidiary of Delcam plc, the world's leading developer and supplier of complete CAD/CAM software solutions.

About this guide

Use this guide to find out about:

- Installing PartMaker (see page 2).
- What's new in PartMaker 2014 (see page 7).
- Accessing help and support (see page 44).

Installing PartMaker

To install PartMaker on a stand-alone PC:

- 1 Before installing PartMaker, make sure you meet the system requirements (see page 3).
- 2 Install PartMaker (see page 3).
- 3 Install your license file (PAF) (see page 5).



*If you use post processors for PartMaker/SwissCAM or PartMaker/Turn-Mill, you need to install the post processors you require (using the **Help > Download Licensed Post Config Files** menu option) before starting to use PartMaker.*

If you need further details about the installation process, or you are installing PartMaker for use on a network, refer to the *Installation Guide* on your PartMaker Documentation CD for more information.

System requirements

The system requirements for PartMaker 2014 and higher are:

	Standard PartMaker	Parasolid Import Module (Solids)*	Full Machine Simulation (FMS)	Advanced Surface Machining (ASM)
Processor	Pentium 3 800 MHz or higher	Pentium 3 800 MHz or higher	Pentium 4 3 GHz or higher	Intel Core Duo processor or higher
Memory	512 MB or more	1 GB or more	2 GB or more	4 GB or more
Hard drive	40 GB	40 GB	40 GB	80 GB
Independent video card	128 MB NVIDIA	512 MB NVIDIA	512 MB NVIDIA	1 GB NVIDIA

Operating systems

64-bit and 32-bit versions of PartMaker are available and are supported on Windows 7.



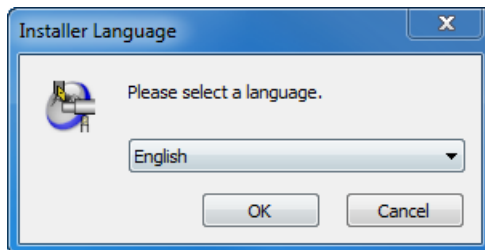
PartMaker 2014 does not support Windows XP or Windows Vista. Although you may be able to run PartMaker 2014 on these operating systems, Delcam will not address any software performance issues found on Windows XP or Windows Vista that cannot be reproduced on Windows 7 or higher.

Installing PartMaker on a stand-alone PC

To install PartMaker:

- 1 When you have started the computer and Microsoft Windows is running, insert the PartMaker Installation CD into the CD drive.

-
- 2 The Installer prompts you to select the language used on the dialogs displayed during the installation. The default selection is English. If PartMaker is available in the language you select, the translated version is installed. Select the language and click **OK**.



-
-
- 3 The PartMaker Setup program starts and the PartMaker Setup screen is displayed:



*If the Setup program does not start automatically, use **My Computer** (or **Computer** for users of Windows Vista and higher) to browse the contents of the CD and double-click **setup.exe** to start the Setup program.*



*Using Windows 7, you may see an **AutoPlay** dialog when the Installation CD is inserted. Click **Install** or **Run Program** to start the installation. If the **Account Control Manager** screen is displayed, click **Allow** to continue with the installation.*

-
-
-
- 4 Follow the on-screen installation instructions to complete the installation.
- 5 When the installation is complete, install your license file (see page 5).

Installing the license file (PAF)

PartMaker is protected using a combination of:

- a dongle, which plugs into a USB or parallel port on the computer:
- a Product Authorization File (PAF) license file, which you install onto the computer.

PAFs are usually sent in an email from Delcam plc or from the PartMaker, Inc. division of Delcam. The email is sent to the email address that was registered when your company purchased PartMaker. You may receive the PartMaker installation software before you receive the PAF file. Because you can only use PartMaker if a valid PAF is installed, Delcam recommends that you do not install PartMaker until you have received your PAF.

To install the license file (PAF):

- 1 Close PartMaker and any other Delcam programs running on the computer.
- 2 Open the email containing your PAF. The email has several attachments.



If the attachments are in a zip file (so they are accepted by your email's virus scanning software), unzip the file before continuing.

- 3 Follow the instructions in the email to install the PAF, making sure you place your PAF in the correct folder:
 - On a 32-bit computer, place your PAF in `C:\Program Files\Common Files\Delcam`
 - If you have installed a 32-bit version of PartMaker on a 64-bit computer, place your PAF in `C:\Program Files (x86)\Common Files\Delcam`
 - If you have installed a 64-bit version of PartMaker on a 64-bit computer, place your PAF in `C:\Program Files\Common Files\Delcam`
- 4 When you have installed the PAF and plugged in the dongle, you can start PartMaker by double-clicking the PartMaker desktop shortcut. For example, to start the 64-bit version of PartMaker/SwissCAM, double-click:





*If you use post processors for PartMaker/SwissCAM or PartMaker/Turn-Mill, you need to install the post processors you require (using the **Help > Download Licensed Post Config Files** menu option) before starting to use PartMaker.*

What's New in PartMaker 2014

PartMaker 2014 contains many new features and enhancements.

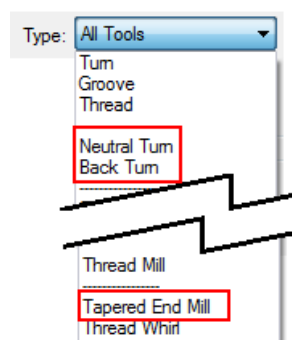
Tool enhancements

PartMaker 2014 includes the following enhancements to tools:

- New tool types for Neutral Turn, Back Turn and Tapered End Mill tools (see page 7).
- A new way of specifying the shape of an End Mill tool (see page 9).
- Changes to the **Manage Tools Files** dialog to support the new tool types and tool shapes (see page 10).

New tool types

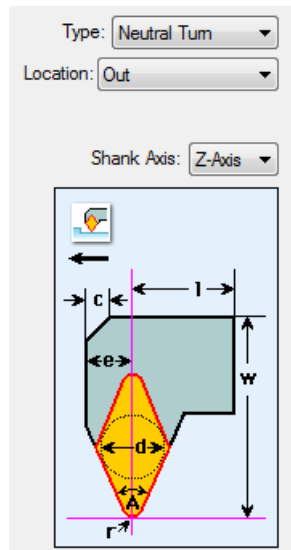
PartMaker 2014 includes new **Neutral Turn**, **Back Turn** and **Tapered End Mill** tool types in the **Tool Data** dialog:



New values are available for the `<tool-type>` reserved word in ConfigPost (see page 43) to support these new tools types.

Neutral Turn tool

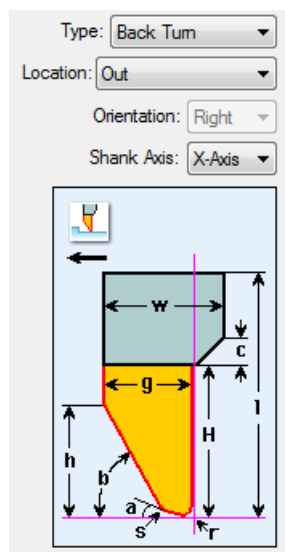
The **Neutral Turn** tool is available for use with roughing and finishing operations in PartMaker/SwissCAM, PartMaker/Turn-Mill and PartMaker/Turn:



In the NC Program, the output point of a Neutral turn tool is the tip of the tool.

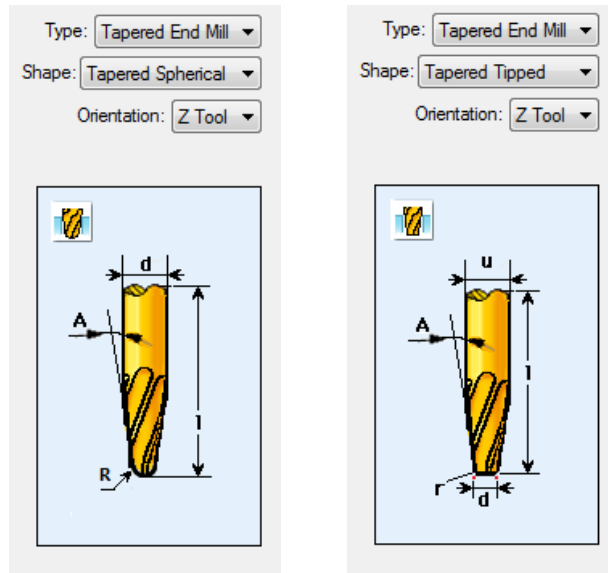
Back Turn tool

The **Back Turn** tool is available with the Advanced Tools licensing option. When licensed, the **Back Turn** tool is available for finishing operations in PartMaker/SwissCAM, PartMaker/Turn-Mill and PartMaker/Turn:



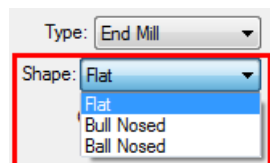
Tapered End Mill tool

A **Tapered End Mill** tool (with a **Tapered Spherical** or a **Tapered Tip** shape) is available with the Advanced Tools licensing option. When licensed, the **Tapered End Mill** tool is available for Advanced Surface Machining (ASM), Advanced Milling Toolpaths (AMT) and for pocketing and contouring in PartMaker/SwissCAM, PartMaker/Turn-Mill and PartMaker/Mill:



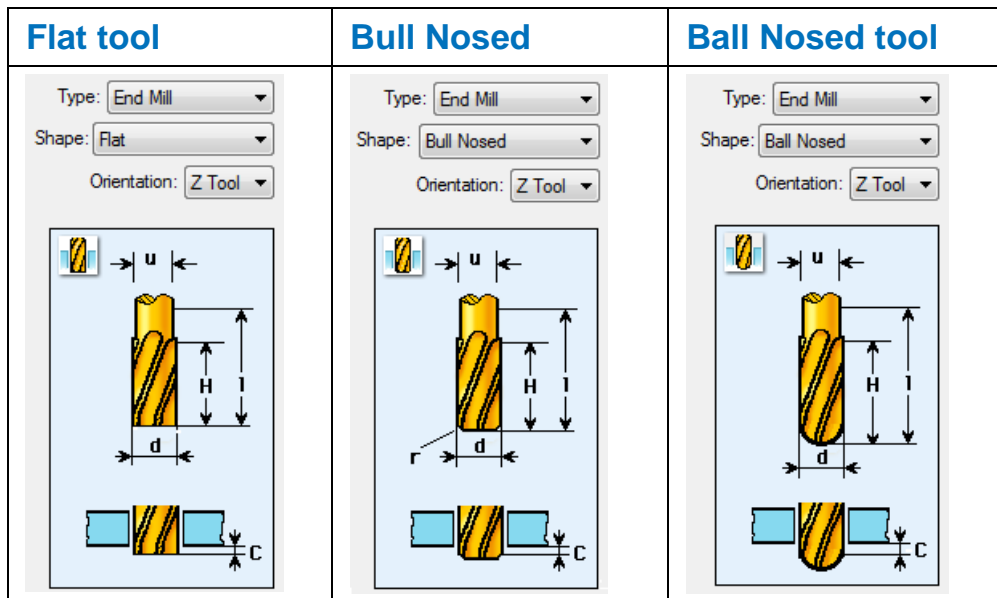
End Mill tool shapes

A new **Shape** selection box on the **Tool Data** and **Edit Tool** dialogs makes it easier to specify the shape (Flat, Bull Nosed or Ball Nosed) of an End Mill tool:



In previous versions of PartMaker, you had to enter specific values in the **Corner Radius** field to specify the shape of the tool.

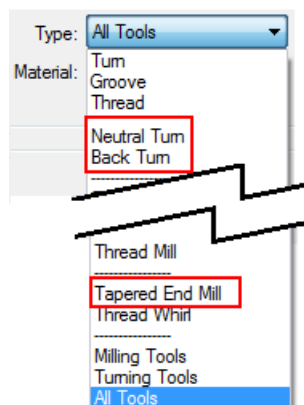
The tool images displayed in PartMaker also reflect the shape you select. For example, on the **Tool Data** dialog:



Manage Tools Files dialog

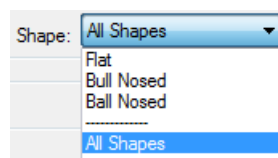
The **Manage Tools Files** dialog includes options for the new tool types and tool shapes introduced in PartMaker 2014:

- The new **Neutral Turn**, **Back Turn** and **Tapered End Mill** tool types are available in the **Type** selection box:

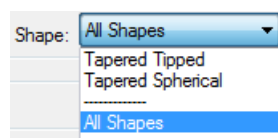


- A new **Shape** field is available to select the tool shape for:

- End Mill tools:



- Tapered End Mill tools:



Working with solid models

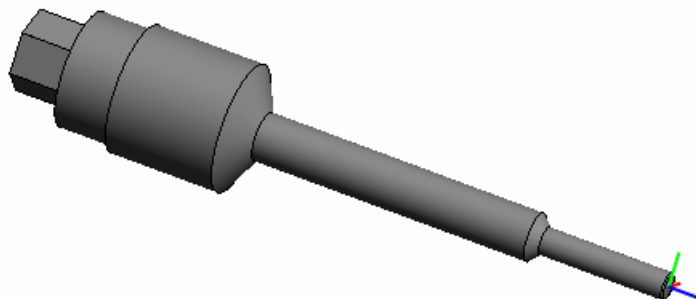
PartMaker 2014 contains the following enhancements for working with solid models:

- Ability to set the transparency of solid models (see page 11).
- PartMaker/Mill now displays the Face Plane and axis at the **Z Face** coordinate in the Solids window (see page 12).
- When defining the Face plane of a dependent face from a solid model, PartMaker/Mill now calculates the **Tombstone Index Angle** for the dependent face automatically (see page 17).
- When using the **Mill Cylinder** machining function in PartMaker/SwissCAM, PartMaker/Turn-Mill and PartMaker/Mill, you can now project the outlines and edges of non-cylindrical surfaces from a solid model onto the Face plane (see page 19).
- When importing solid models, PartMaker 2014 is compatible with:
 - Solidworks 2014
 - Parasolid v24.0


Setting transparency for solid models

You can now set the transparency of a solid model displayed in the Solids window.

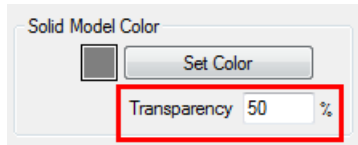
Previously, PartMaker always displayed solid models as opaque (no transparency). For example:



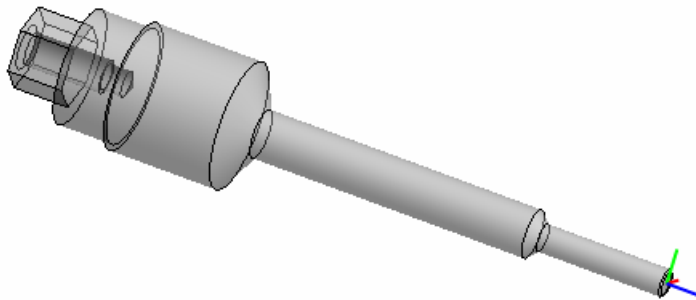
To set the transparency of a solid model:

- 1 Click  on the **Solids Window** toolbar or select **Solids > Solids Options** to display the **Solids Options** dialog.

- 2 Enter a value in the **Transparency** field. Use a value between **0** and **100**, where **0** displays a solid (opaque) model and **100** displays a fully transparent (invisible) model:

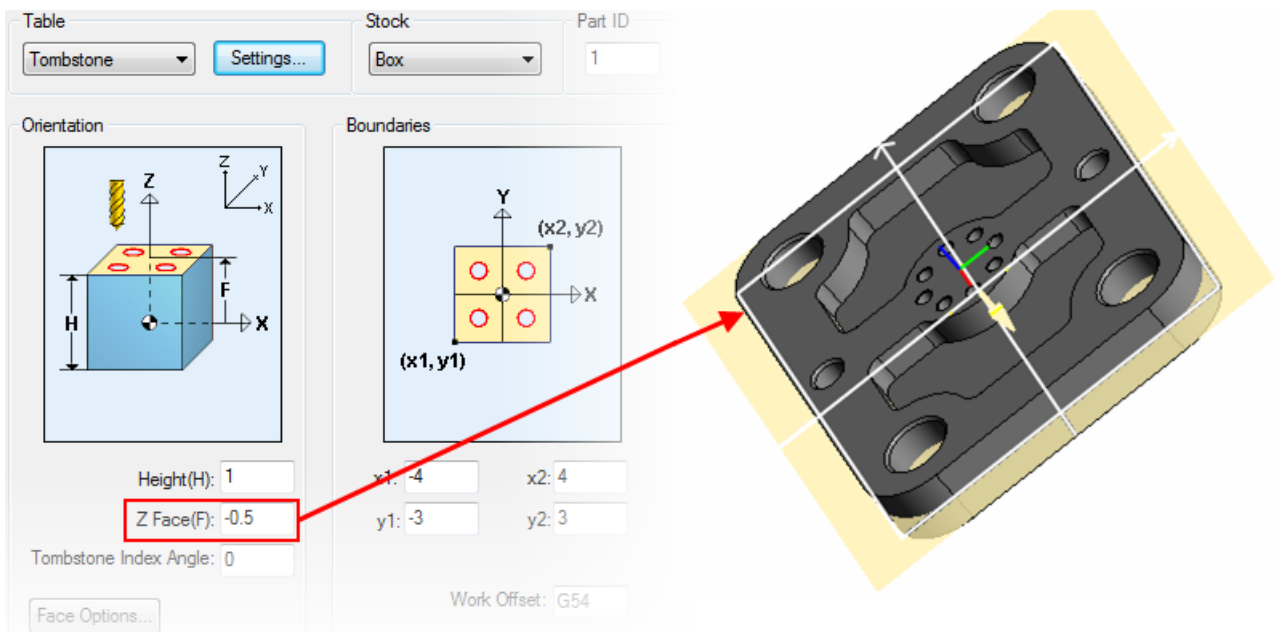


- 3 Click **OK** to close the **Solids Options** dialog. PartMaker updates the display of the solid model to reflect the value you entered:



Setting the Z Face coordinate in PartMaker/Mill

PartMaker/Mill now displays the Face Plane and axis at the **Z Face** coordinate in the Solids window for all machining function/stock/milling table combinations that display this field. For example:



You can set the Z Face coordinate using the following dialogs:

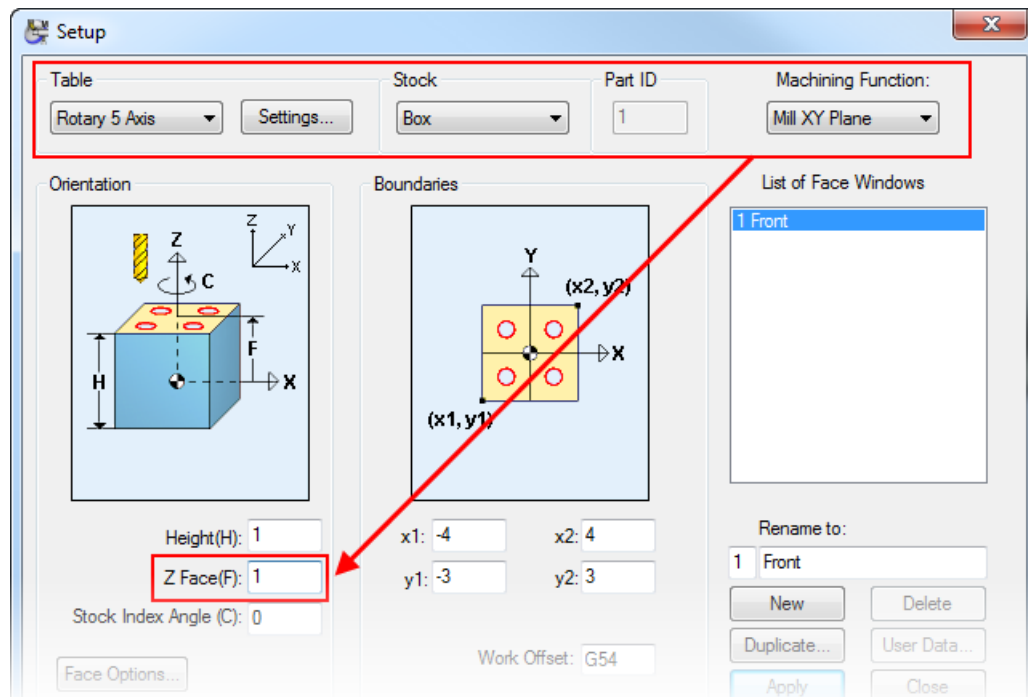
- **Setup** dialog (see page 13).
- **Define Face Plane** dialog (see page 14).
- **Surface Info** dialog (see page 16).

Changes to the Setup dialog

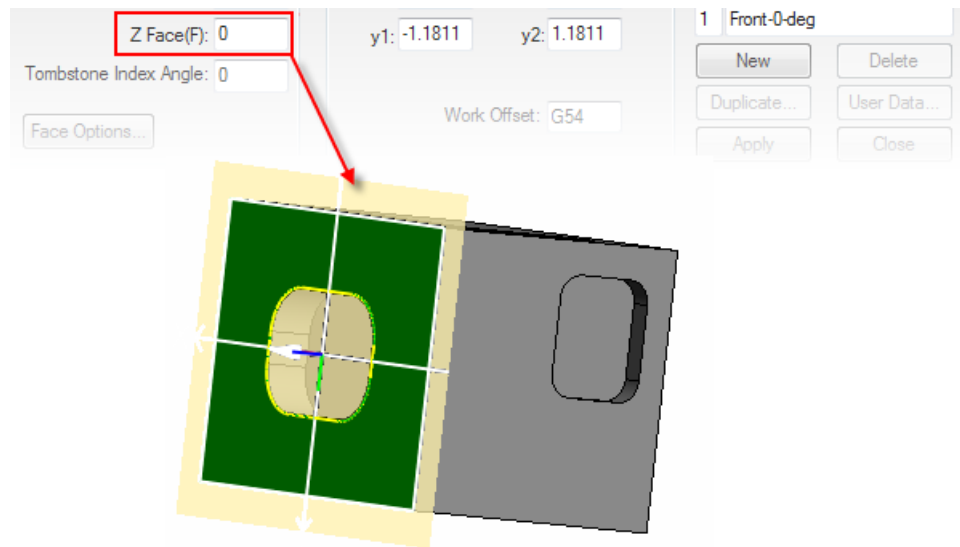
Previously, the **Z Face** field was available on the **Setup** dialog only when using the:

- **Mill XY Plane** machining function for a **Standard** table.
- **Mill 4-Axis Plane** machining function for a **4-Axis** or **5-Axis Rotary** table.

In PartMaker/Mill 2014, you can now also enter a **Z Face** coordinate on the **Setup** dialog when using the **Mill XY Plane** machining function for **Tombstone** or **5-Axis Rotary** tables. For example:



PartMaker displays the Face Plane and axis at the **Z Face** coordinate specified:

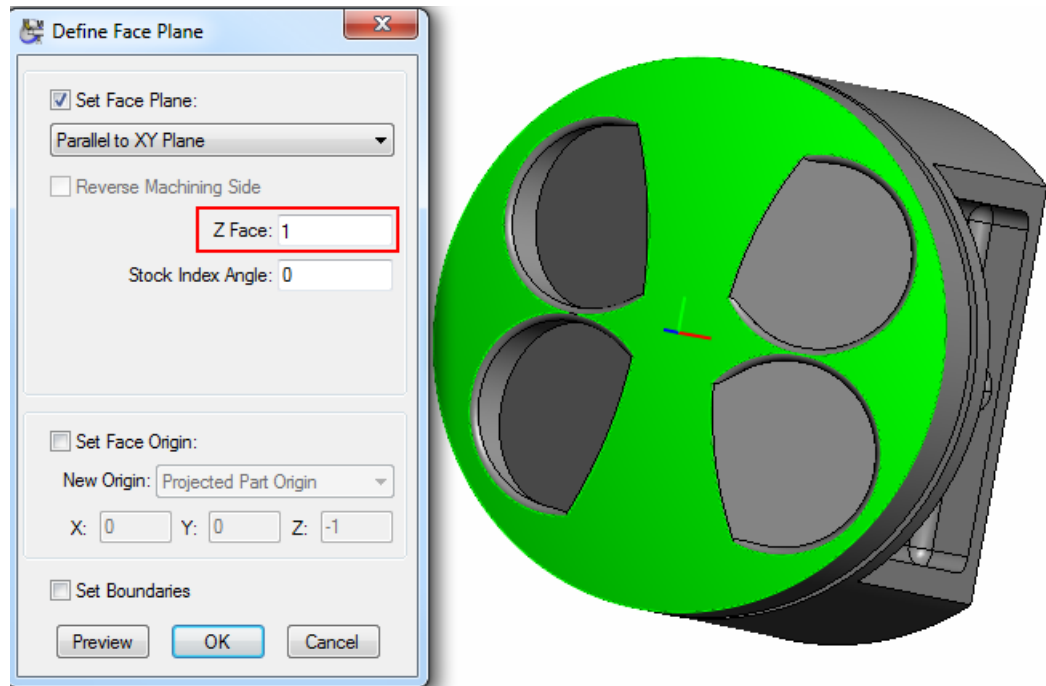


Using the Define Face Plane dialog

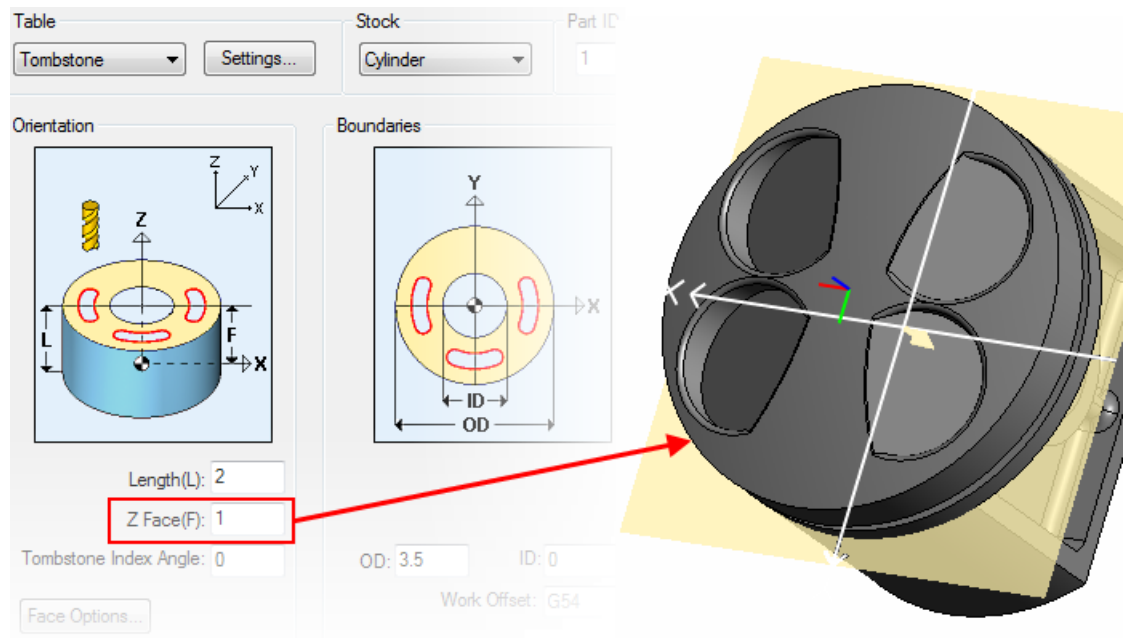
When defining the Face Plane from a solid model, you can now specify a **Z Face** coordinate on the **Define Face Plane** dialog. The **Z Face** field is available when using the **Parallel to XY Plane** option with the following machining function/stock/table combinations:

- **Mill XY Plane** machining function with **Box** or **Cylinder** stock on **Standard** or **Tombstone** tables.
- **Mill XY Plane** machining function with **Box** stock on **5-Axis Rotary** tables.
- **Mill 4-Axis Plane** with **Box** stock on **4-Axis Rotary** or **5-Axis Rotary** tables.

For example, if you select the following surface (shown in green) and enter a **Z Face** value of **1**:



PartMaker updates the Solids window to display the Face Plane at the specified **Z Face** coordinate value and updates the **Z Face** field value on the **Setup** dialog to reflect the new value:

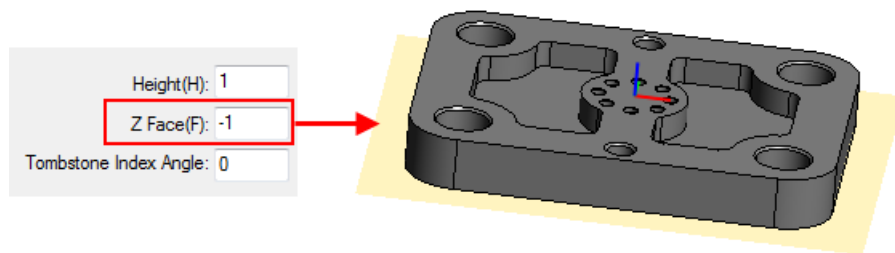


Using the Surface Info dialog

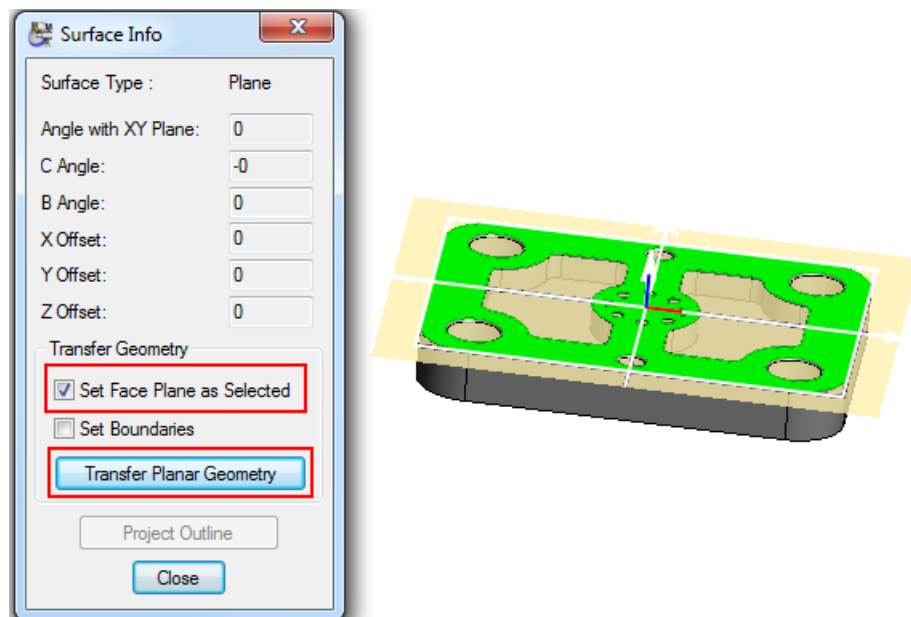
When using the **Surface Info** dialog to define the Face Plane from the solid model, PartMaker/Mill 2014 now calculates the **Z Face** coordinate value from the selected surface when you use the following machining function/stock/table combinations:

- **Mill XY Plane** machining function with **Box** or **Cylinder** stock on **Standard** or **Tombstone** tables.
- **Mill XY Plane** machining function with **Box** stock on **5-Axis Rotary** tables.
- **Mill 4-Axis Plane** with **Box** stock on **4-Axis Rotary** or **5-Axis Rotary** tables.

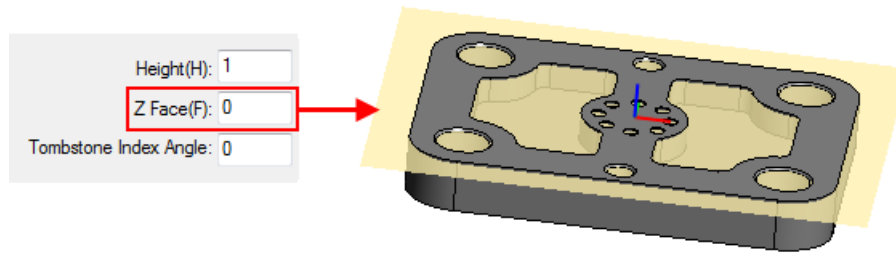
For example, when the following Face Plane is defined at the bottom of the part, the **Z Face** value on the **Setup** dialog is **-1**:



You can change the Face Plane by double-clicking another surface on the part to display the **Surface Info** dialog. You can then select **Set Face Plane as Selected** and click **Transfer Planar Geometry**.



When transferring the planar geometry into the Face window, PartMaker resets the Face Plane and updates the **Z Face** coordinate value on the **Setup** dialog to reflect the new **Z Face** value:



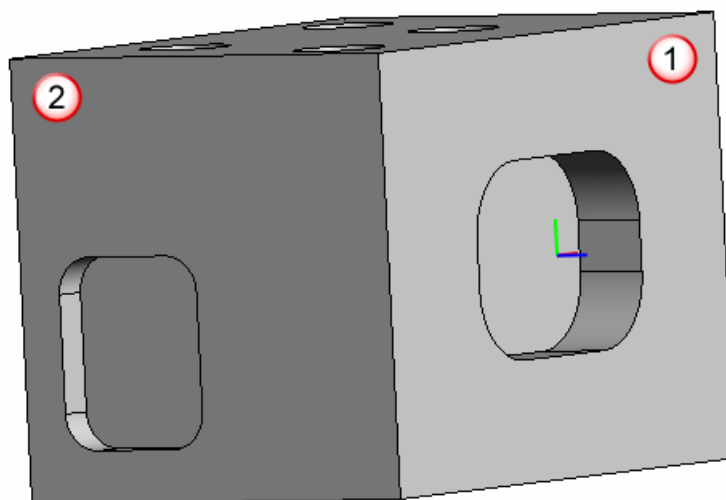
Calculating the Tombstone Index Angle for dependent faces

When defining the Face plane of a dependent face from a solid model, PartMaker/Mill now calculates the index angle for the dependent face and displays this in the **Tombstone Index Angle** field on the **Setup** dialog.

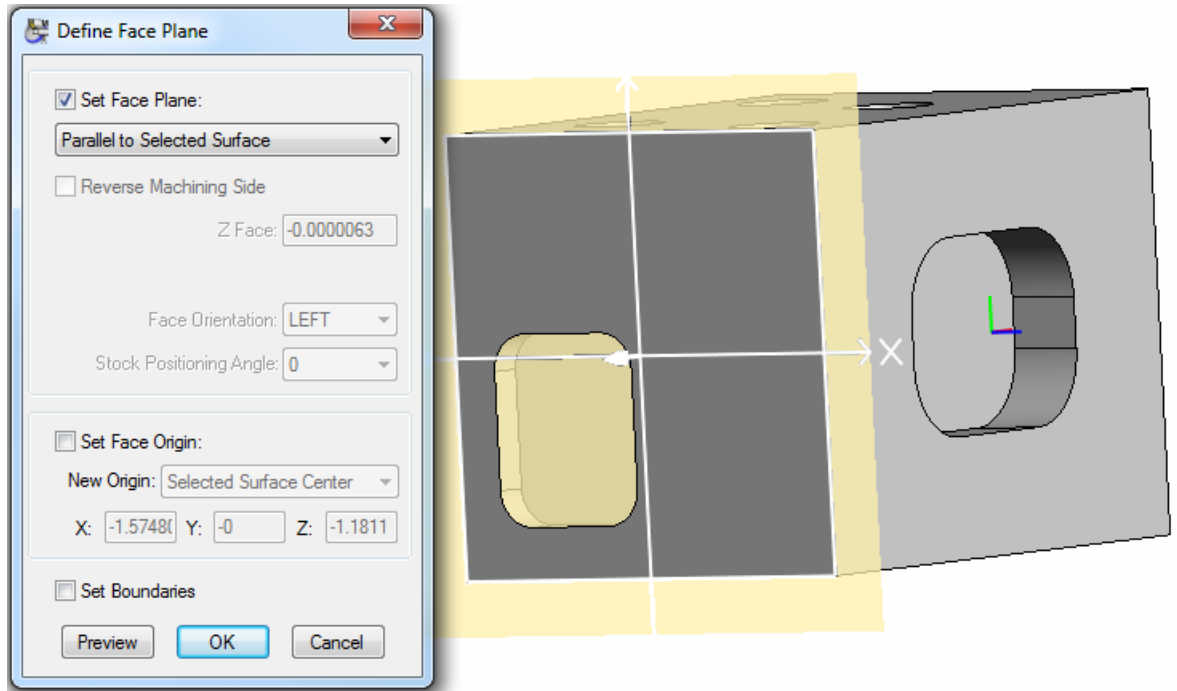
In previous versions of PartMaker/Mill, you had to calculate the index angle required and then enter it into the **Tombstone Index Angle** field on the **Setup** dialog.

The following example shows how PartMaker/Mill 2014 calculates the **Tombstone Index Angle**:

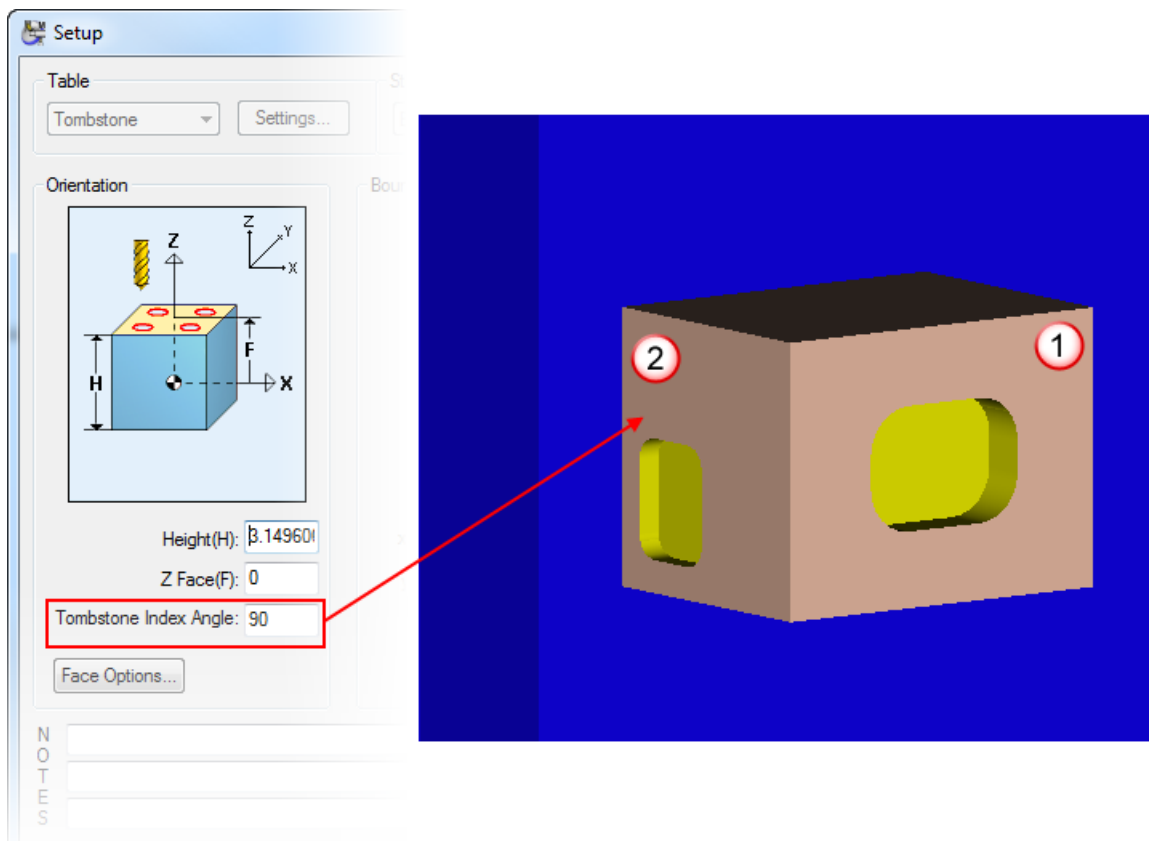
- 1 Create two Face windows — an Independent Face (FRONT, see ①) and a Dependent Face (LEFT see ②):



- 2 On the solid model, select the plane to use as the Face plane for the LEFT Face, then right-click and select **Define Face Plane** to display the **Define Face Plane** dialog:



- 3 When you click **OK** to confirm the Face plane, PartMaker calculates the **Tombstone Index Angle** required and updates this value on the **Setup** dialog. Select **View > Setup** to view the index angle that PartMaker has calculated:



Projecting non-cylindrical surfaces in a Mill Cylinder Face window

In PartMaker/SwissCAM, PartMaker/Turn-Mill and PartMaker/Mill, you can now project the outlines and edges of non-cylindrical surfaces from a solid model onto the Face plane when using the **Mill Cylinder** machining function.

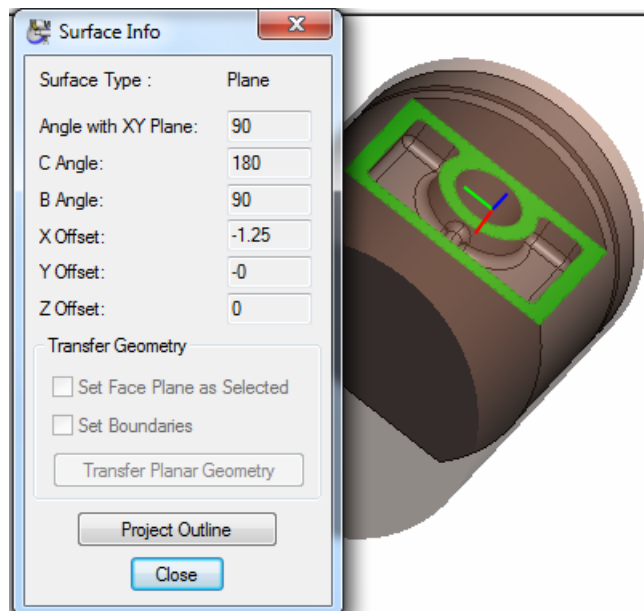
Previously, this feature was available for other machining functions, but not for **Mill Cylinder**.

When working in the Solids window, use the **Solids > Selection Options** menu to specify whether you want to select surfaces and/or edges on the solid model. You can then make your selection on the solid model and can project:

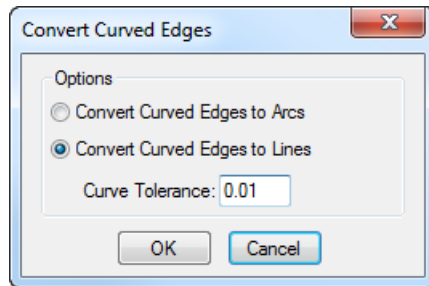
- **Outlines** by clicking the **Project Outline** button on the **Surface Info** dialog or selecting the **Solids > Project > Project Outline** menu option.
- **Edges** by clicking the **Project** button on the **Curve Info** dialog or selecting the **Solids > Project > Project Edges** menu option.

For example, to project an outline of a planar surface into a Face window that uses the **Mill Cylinder** machining function:

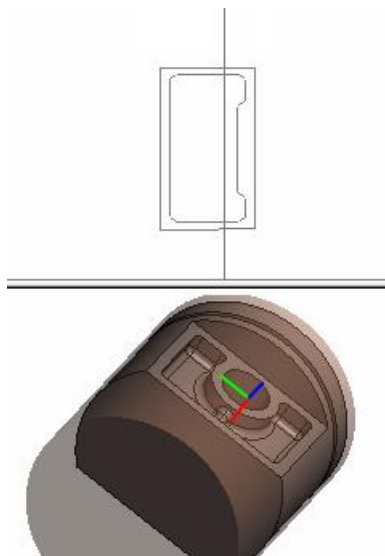
- 1 Double-click a planar surface on the solid model in the Solids window to display the **Surface Info** dialog:



- 2 Click **Project Outline**. PartMaker displays the **Convert Curved Edges** dialog, where you can specify how PartMaker converts the curved edges of the surface when projecting it into the Face window:




- 3 Click **OK** to close the **Convert Curved Edge dialog**. PartMaker projects an outline of the selected surface into the Face window:



NC Program files

PartMaker 2014 includes the following enhancements for working with NC Program files:

- In PartMaker/SwissCAM and PartMaker/Turn-Mill, a new **NC Program Info** dialog replaces the NC Program Info window (see page 21).
- A new NC Program window replaces the NC Program Viewer window(s) and includes:
 - A toolbar for accessing common tasks more easily (see page 22).
 - The ability to align multi-channel NC Program files in PartMaker/SwissCAM and PartMaker/Turn-Mill (see page 23).
 - The ability to search for text in NC Program files (see page 25).
 - The ability to set a user-defined point as the origin for the Global Coordinate system when using a 5-Axis Rotary table (see page 26).
- On the **Main** toolbar, a new **Show NC Program Window**  button displays the NC Program window.

NC Program Info dialog

PartMaker/SwissCAM and PartMaker/Turn-Mill include a new **NC Program Info** dialog that replaces the NC Program Info window in previous PartMaker versions.

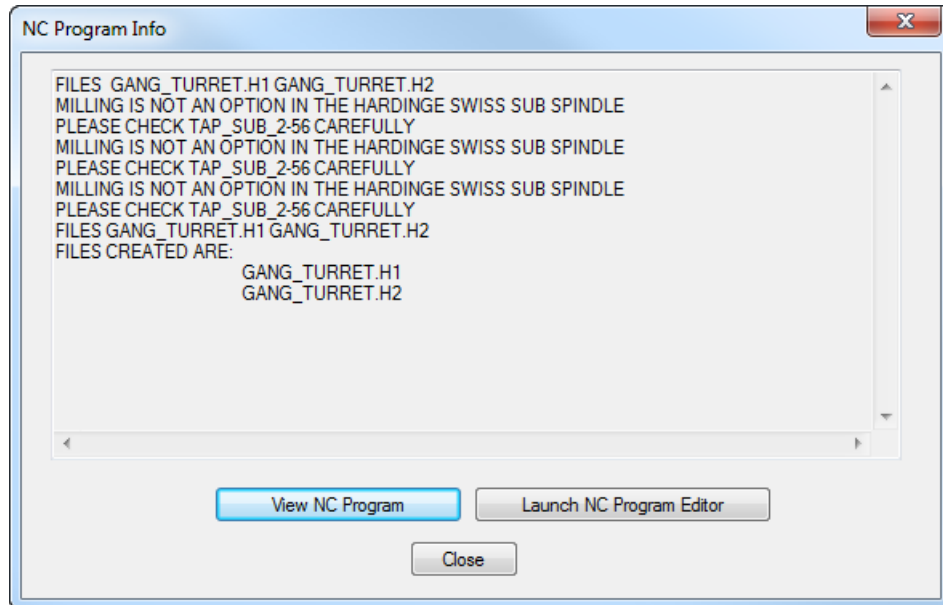
When you generate an NC Program file, PartMaker now displays:

- The **NC Program Info** dialog if any informational messages are generated during postprocessing.
- The NC Program window if no informational messages are generated during postprocessing.



*The **NC Program Info** dialog is not displayed in PartMaker/Mill, PartMaker/Turn or PartMaker/WireEDM. These applications always display the NC Program window when you generate an NC Program file successfully.*

The messages displayed in the **NC Program Info** dialog depend on the Post Configuration (.pst) file you are using. Both informational and error messages can be displayed. For example:







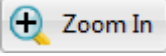
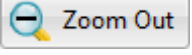


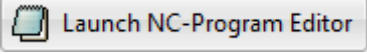
When you have viewed the messages, you can select:

- **View NC Program** to close the **NC Program Info** dialog and view the NC Program in the **NC Program** window.
- **Launch NC Program Editor** to close the **NC Program Info** dialog and open the NC Program Editor application (as specified on the **Preferences** dialog) so you can edit the NC Program file.
- **Close** to close the **NC Program Info** dialog.

NC Program Window toolbar

The new NC Program window includes a toolbar for accessing frequently used features:

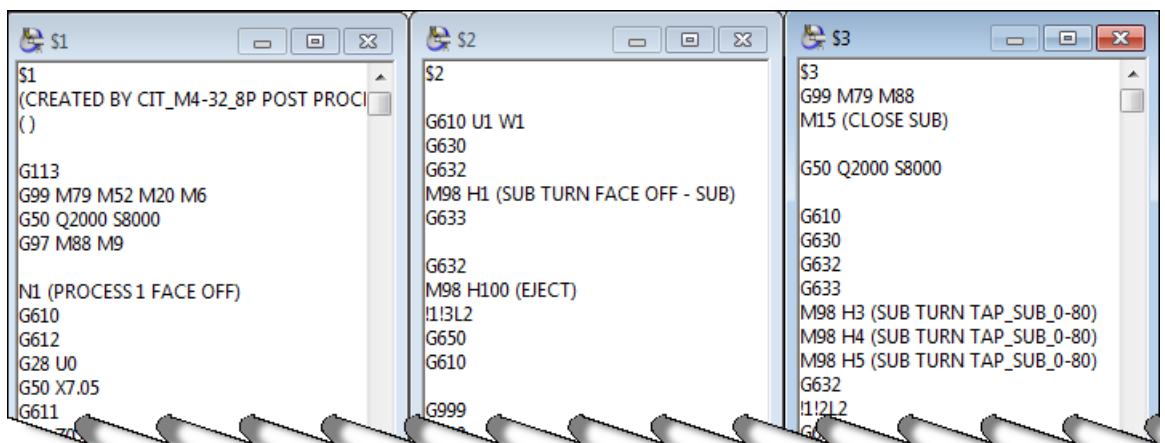
Button	Description
 Post Options	Displays the Post Options dialog, where you can specify how PartMaker creates NC Programs for the current part.
 Generate NC Program	Displays the Save NC Program file as dialog for you to specify a name for the NC Program file, and then generates an NC Program file for the current part.
 Find	Displays the Find dialog, where you can enter the text you want to search for in the NC Program file(s).

Button	Description
 Print	Displays the Print Document dialog, where you can specify the page orientation and preview before printing.
 Zoom In	Zooms in to increase the size of the text in the NC Program window.
 Zoom Out	Zooms out to decrease the size of the text in the NC Program window.
 Align	Aligns the different channels in the NC Program file(s) according to the 'wait' and 'sync' codes (see page 23).  <i>Available only in PartMaker/SwissCAM and PartMaker/Turn-Mill.</i>
 Launch NC-Program Editor	Opens the NC Program file in the NC Program Editor (as specified on the Preferences dialog) for editing.


Aligning multi-channel NC Programs

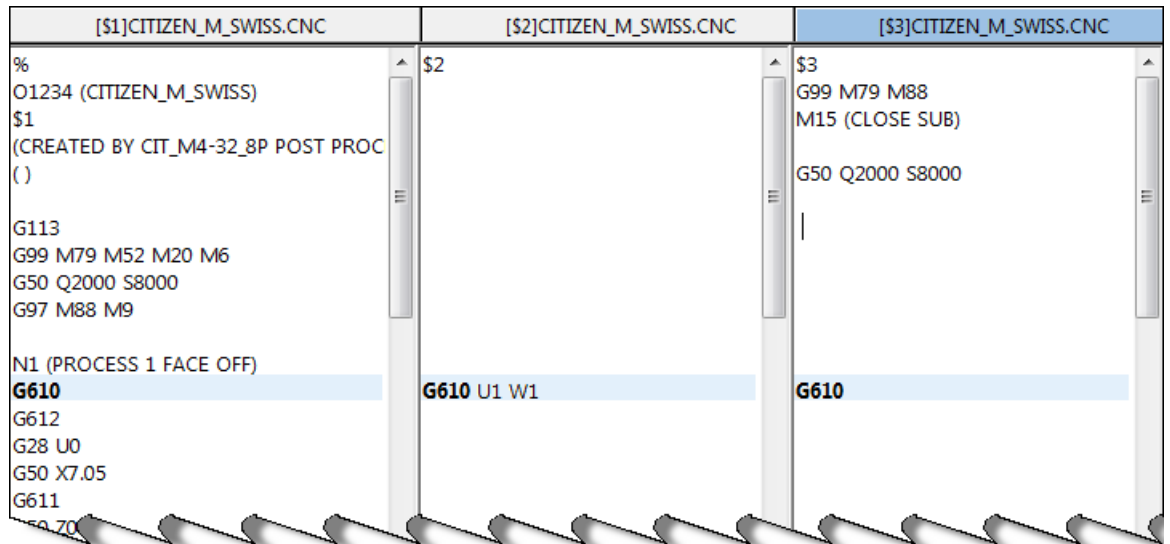
In PartMaker/SwissCAM and PartMaker/Turn-Mill, some machines require multi-channel NC Program output. Depending on the machine and postprocessor used, PartMaker combines the channels into a single NC Program file or stores them in separate files.

In previous versions, PartMaker displayed the NC Program output for multiple channels in separate NC Program windows. For example:



In PartMaker 2014, PartMaker can now display up to four channels within a single NC Program window and can align the content in all the channels using 'wait' and 'sync' codes.

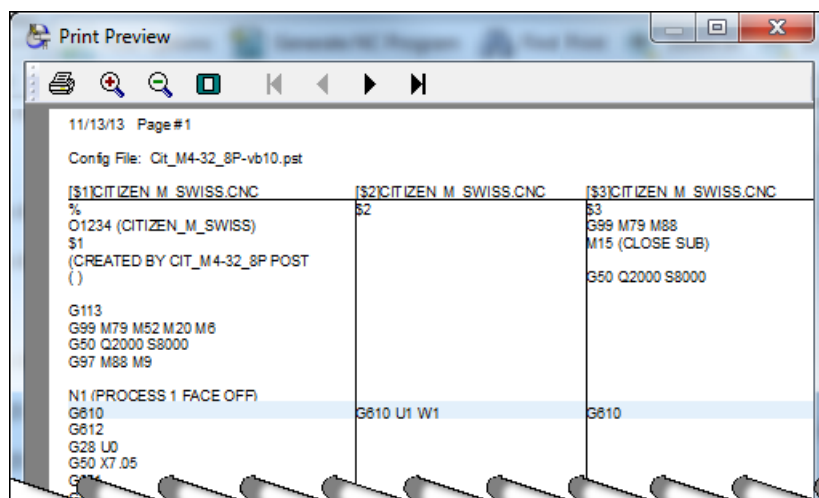
To align the different channels, click  **Align**. For example:



*PartMaker aligns channels in the NC Program by matching identical 'wait' and 'sync' codes across the different channels. The majority of multi-channel machines for which PartMaker can provide licensed Post Configuration files use this method of synchronization. However, if you are using a machine that synchronizes using other methods (for example, by matching pre-defined, non-identical wait codes), the **Align** button is unavailable.*

Printing multi-channel output


When the NC Program window displays aligned, multi-channel output, PartMaker also reflects this layout in printed output:



Searching NC Program files

You can now search the NC Program file(s) displayed in the NC Program window. Previously, you could search only in the NC Program Editor.

To search NC Program file(s):

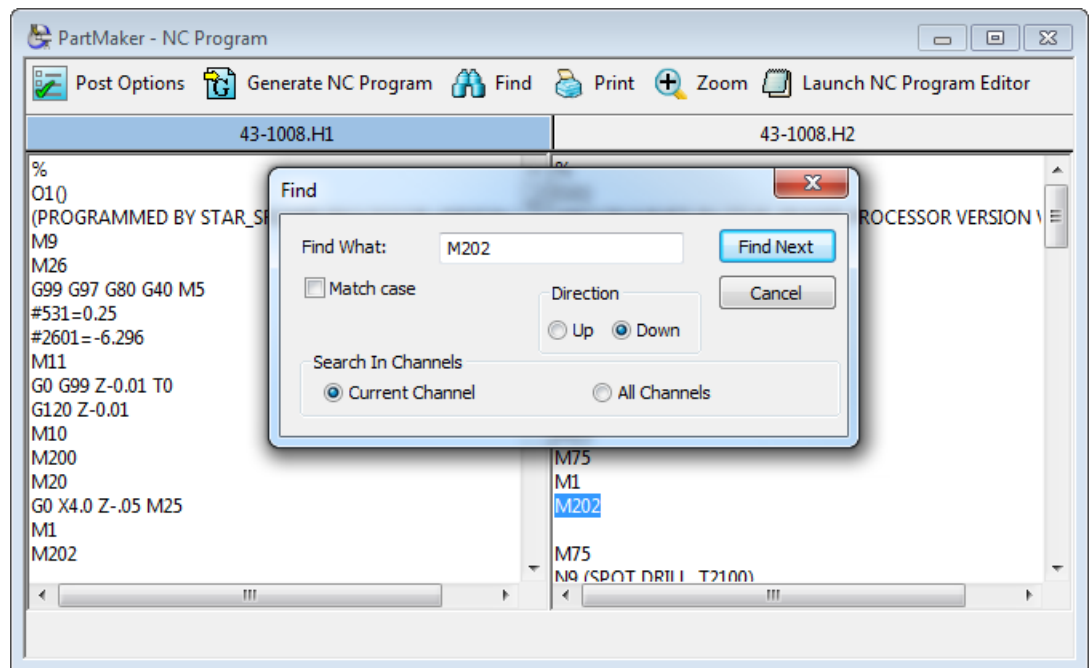
- 1 With the NC Program file(s) open in the NC Program window, display the **Find** dialog by:
 - Selecting **Edit > Find** from the menu; or
 - Clicking  on the NC Program Window toolbar.
- 2 Enter the settings for your search in the **Find** dialog:
 - **Find What** — Enter the text you want find. If you selected text in the NC Program window before opening the **Find** dialog, the selected text is displayed in this field.
 - **Match Case** — Select to search only for text that matches the combination of upper- and lower-case characters used in the **Find What** field.
 - **Direction** — Select the direction (**Up** or **Down**) in which PartMaker searches the NC Program file(s).
 - **Search In Channels** — Specify whether PartMaker searches only the currently selected channel, or searches all channels, in the NC Program file(s).



***Search In Channels** is available only when viewing a multi-channel NC Program file in PartMaker/SwissCAM or PartMaker/TurnMill.*

- 3 Click **Find Next** to search the NC Program file(s).

PartMaker highlights the text that matches your search criteria:

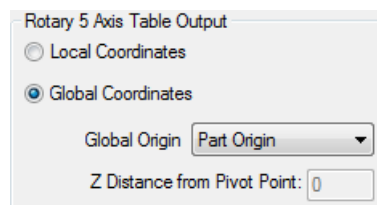


Setting global coordinates for 5-axis NC Program output

In PartMaker/Mill, you can now set a user-defined point as the origin for the Global Coordinate system when using a 5-axis rotary table. PartMaker/Mill outputs the NC coordinates in the NC Program file with respect to this point.

To specify a user-defined point for NC coordinates:

- 1 On the **Post Options** dialog, select **User Defined** from the **Global Origin** list in the **Rotary 5 Axis Table Output** area of the dialog.
- 2 Specify the location of the point by entering its distance from the table's pivot point in the **Z Distance from Pivot Point** field:

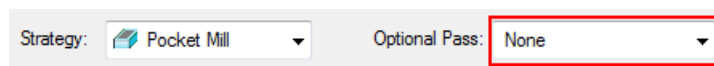


- 3 Click **OK** to confirm your selections and close the **Post Options** dialog.

New method of calculating pocketing toolpaths

PartMaker 2014 includes a new, improved method of calculating pocketing toolpaths in PartMaker/SwissCAM, PartMaker/Turn-Mill and PartMaker/Mill. PartMaker automatically uses the new method to calculate all pocketing toolpaths when you create a pocketing toolpath using the **Pocket Mill** strategy on the **Profile Group Parameters** dialog.

The new method automatically detects where material is left and generates clearance toolpaths accordingly. Therefore, it is unlikely that the toolpath will require a clean-up pass (as set using the **Optional Pass** option):



The new-style pocketing toolpaths automatically attempt to keep the tool at the cutting level. Therefore, the **Stay at Depth** option has been replaced with a new **Minimize Air Moves** option:

PartMaker 2013	PartMaker 2014
Linear Extension (L): <input type="text"/> %d <input type="checkbox"/> Stay at Depth <input checked="" type="checkbox"/> Center Entry	Linear Extension (L): <input type="text"/> %d <input type="checkbox"/> Minimize Air Moves <input checked="" type="checkbox"/> Center Entry

When **Minimize Air Moves** is selected, PartMaker machines each area with one plunge at the start of the toolpath to minimize the number of lifts.



Minimize Air Moves is not available for linear-style pocketing toolpaths.

Handling pocketing toolpaths created in earlier versions of PartMaker

When using PartMaker 2014 to open a .job file that contains pocketing toolpaths created in an earlier version of PartMaker, a message is displayed prompting you to re-verify all the pocketing toolpaths before creating a NC program.

You should re-verify all these pocketing toolpaths to ensure they produce the expected results.

User interface changes




PartMaker 2014 includes the following improvements to the PartMaker user interface:


- You can now undo and redo multiple actions when working in PartMaker (see page 28).
- Leads In and Leads Out settings are now available in a single **Leads** dialog (see page 30).
- PartMaker now displays unlicensed tools in the user interface (see page 31).
- The **Tool Data** and **Edit Tools** dialogs now disable invalid settings for turning tools (see page 32).
- The **Profile Group Parameters**, **Turning** dialogs have a new layout (see page 32).
- PartMaker now remembers the **View > Show Work Group Only** menu option setting as you move between different work groups (see page 33).
- The **View** menu has a new layout (see page 33), which is also reflected in the Face Window toolbar.
- Some menu options have new keyboard shortcuts (see page 33).
- The **Setup** dialog in PartMaker/Mill now displays the **X Face** field only when it is needed (see page 34).

Undoing and redoing actions

PartMaker 2014 enables you to undo and redo multiple actions. In previous versions of PartMaker, you could undo and redo only your most recent action.

This enhancement also changes how some aspects of the **Undo/Redo** options work:

	PartMaker 2013	PartMaker 2014
Menu options and toolbar buttons	A single toolbar button and menu option were available to toggle between undoing and redoing the most recent action:  button and Edit > Undo	Different toolbar buttons and menu options are available for undoing and redoing multiple actions: Undo  button and Edit > Undo Redo  button and Edit > Redo

	PartMaker 2013	PartMaker 2014
Using in Face windows	You could switch between Face windows and undo the most recent action in each window.	You can undo multiple actions in the active Face window.
Actions you can undo/redo	You could undo and redo many types of actions performed by using a menu option, a toolbar button, a keyboard key, or selected dialogs.	You can redo and undo the same actions as in previous PartMaker versions, as well as some new actions, including: <ul style="list-style-type: none"> ▪ In CAD mode, you can undo actions performed using the Line Info, Arc Info and Circle Info dialogs. ▪ In CAM mode, you can undo actions performed using the Hole Properties dialog.
Action history	PartMaker kept a history of the actions that could be undone/redone. PartMaker cleared the history when certain actions (such as opening a new job file or generating the Process Table) were performed.	PartMaker stores and clears the undo/redo history as in previous releases, but there are now additional actions that clear the history, including: <ul style="list-style-type: none"> ▪ Importing any file. ▪ Using options from the View > Change Spindle and Convert Data menu. ▪ Using the Zero  button in CAD or CAM mode.

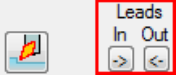
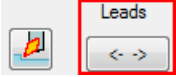


*These changes do not affect the **Undo** buttons available on the **Hole Group Parameters** dialog, **Profile Group Parameters** dialogs for Contour and Pocket Cycles and the **Edit Coordinate System** dialog. These **Undo** buttons continue to work as in previous releases.*

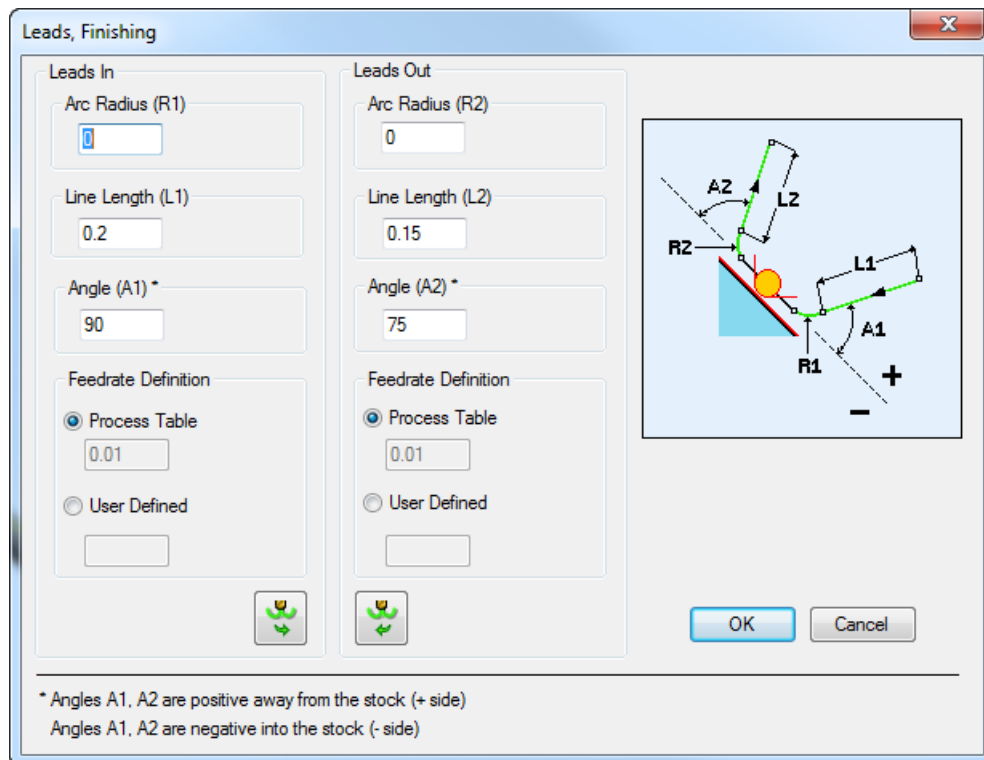
Leads In and Leads Out

PartMaker 2014 combines the settings for controlling the movement of the tool as it approaches (Leads In) and leaves (Leads Out) the stock into a single **Leads** dialog. Previously, these settings were displayed on separate **Leads In** and **Leads Out** dialogs.



To display the **Leads** dialog, click the **Leads** button on the **Profile Group Parameters** dialog. This button replaces the separate **Leads In** and **Leads Out** buttons available in previous versions:

PartMaker 2013	PartMaker 2014
	

The new **Leads** dialog enables you to view Leads In and Leads Out settings alongside each other:



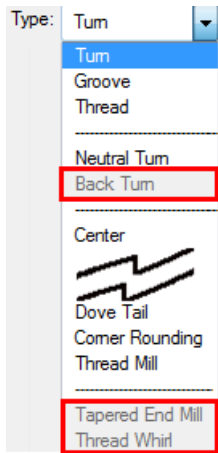
You can also copy values between the different types of moves using the following buttons:

Button	Description
	Copies the values specified for Lead In moves into the fields for Lead Out moves.
	Copies the values specified for Lead Out moves into the fields for Lead In moves.

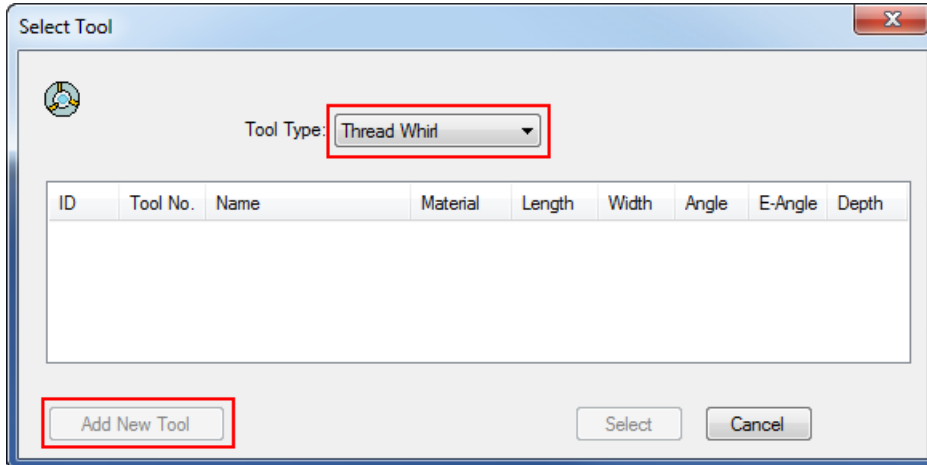
Displaying unlicensed tools

PartMaker now displays Advanced Tools in the user interface, even if they are unlicensed. However, you still require the Advanced Tools license option to create these types of tools.

For example, PartMaker/SwissCAM displays (but does not allow you to select) the Back Turn, Tapered End Mill and Thread Whirl tools in the **Tool Data** dialog when the Advanced Tools option is not licensed:



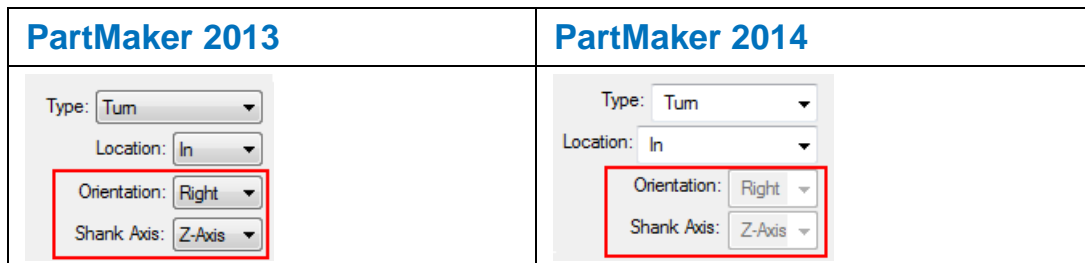
Similarly, PartMaker also displays unlicensed tool types on the **Select Tool** dialog, but does not allow you to select them using the **Add New Tool** button (Turning) or **Add** button (Milling):



Disabling invalid settings for Turning tools

The **Tool Data** and **Edit Tool** dialogs for Turn, Groove or Thread tools in PartMaker/SwissCAM, PartMaker/Turn-Mill and PartMaker/Turn now update to reflect your previous selections.

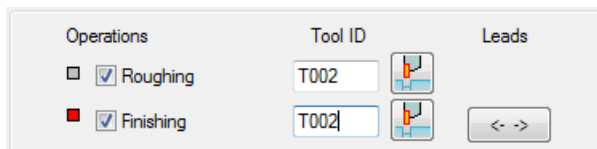
For example, ID Thread tools must have an **Orientation** of **Right** and a **Shank Axis** of **Z-Axis**, so PartMaker automatically selects these values and disables all other **Orientation** and **Shank Axis** values.



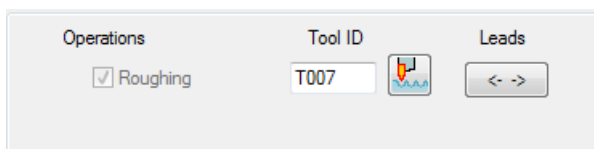
Profile Group Parameters, Turning dialogs

The **Profile Group Parameters, Turning** dialogs have a new layout to make them consistent with the **Profile Group Parameters, Milling** dialogs and improve usability.

The new layout organizes **Operations**, **Tool ID** and **Leads** settings as follows:



For strategies (such as the Cut-Off strategy) that perform only roughing operations, PartMaker now displays, and automatically selects, the **Roughing** option:

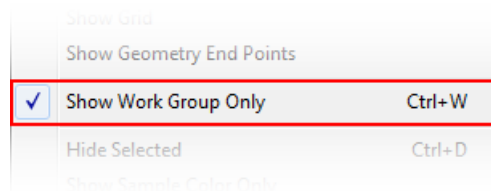


The dialog also contains a new **Leads**  button, which is described in *Leads In and Leads Out* (see page 30).

Displaying holes and profiles from the work group

The **View > Show Work Group Only** menu option enables you to view only the holes and profiles for the currently selected group (that is, the work group) in the Face window.

PartMaker 2014 remembers this menu option setting, so you can move between different work groups and view only the holes and profiles in each group in the Face window. PartMaker displays a check mark alongside this option when it is selected:



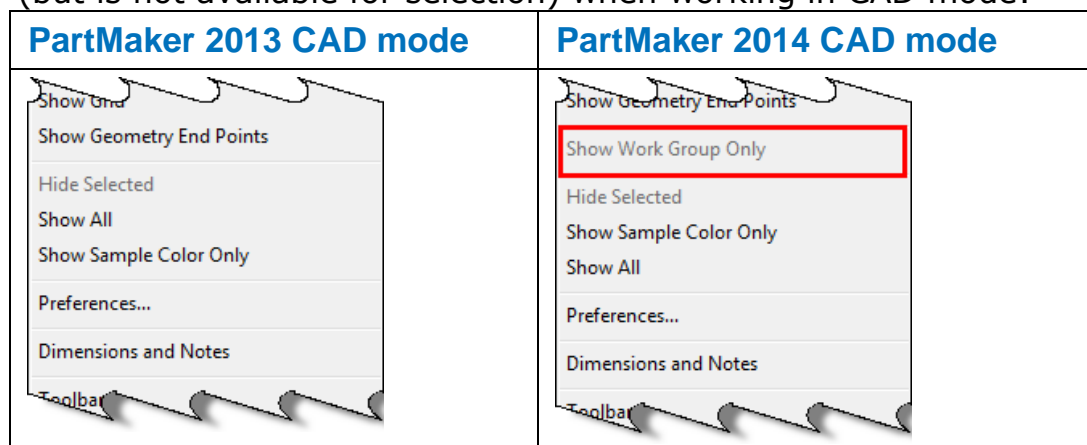
In previous versions of PartMaker, you had to re-select the **View > Show Work Group Only** menu option each time you selected a new work group.

New layout in the View menu

The **View** menu has a new layout. The menu now displays all the menu options available for working in both CAM and CAD mode.

Previously, PartMaker hid some menu options that were not applicable to your current mode.

For example, although you can use the **Show Work Group Only** option only when working in CAM mode, the option is now displayed (but is not available for selection) when working in CAD mode:



The buttons on the Face Window toolbar have also been rearranged to match the order in which the menu options appear in the **View** menu. For example, when working in CAD mode:

PartMaker 2013	PartMaker 2014
	

Keyboard shortcuts for menu options

Some menu options now have different keyboard shortcuts. These changes enable PartMaker to use standard Windows keyboard shortcuts for common operations, such as **Find** and **Redo**:

Menu option	PartMaker 2013	PartMaker 2014
Edit > Find (NC Program window only)	Menu option not available	Ctrl+F
View > Redo	Menu option not available	Ctrl+Y
View > Show Sample Color Only	Ctrl+W	No keyboard shortcut *
View > Full View	Ctrl+F	Ctrl+M
Part Features > Verify Every Tool Path	Ctrl+Y	Ctrl+B

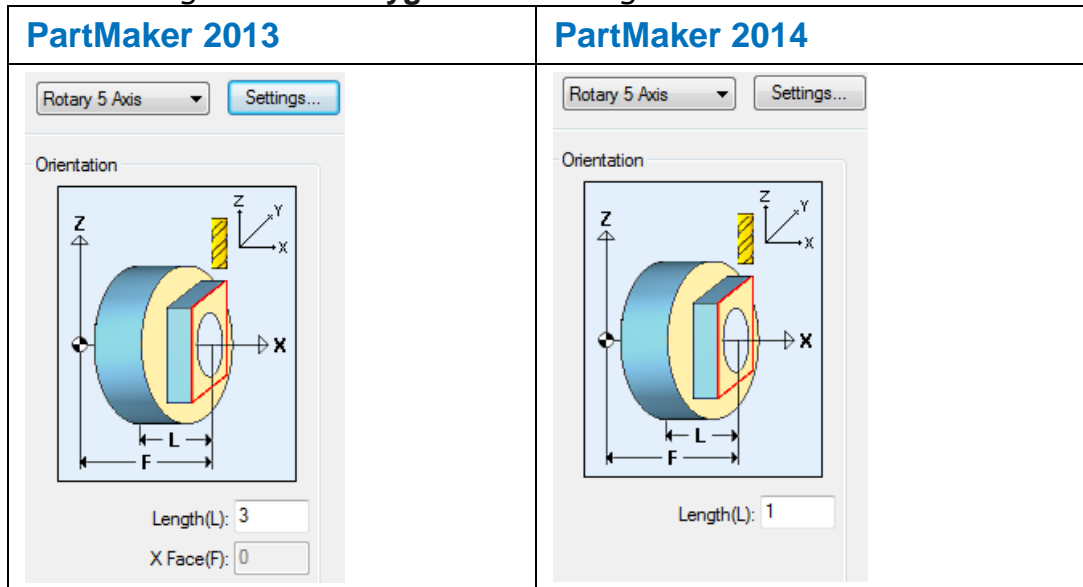
* Previously, **Ctrl+W** was used as a keyboard shortcut for both the **View > Work Group Only** and **View > Show Sample Color Only** menu options. Now, it is used only for **View > Show Work Group Only**.

Changes to the Setup dialog in PartMaker/Mill

In PartMaker/Mill, the **X Face** field is no longer displayed in the **Setup** dialog when using the following machining functions with a 5-Axis Rotary table and cylindrical stock:

- Mill 4-Axis Plane
- Mill Polygon
- Mill Diam, Index
- Mill Cylinder

Previously, the **X Face** field was displayed, but never enabled, for these machining function/stock/table combinations. For example, when using the **Mill Polygon** machining function:



Advanced Surface Machining (ASM)

PartMaker 2014 includes the following enhancements to the **Model Area Clearance** strategy available in Advanced Surface Machining (ASM):

- A new **Vortex** style is available for removing material (see page 36). This style rapidly removes material from a 3D part while controlling tool load. Vortex is best suited to solid carbide tools.
- **Step cutting** settings are now available (see page 37). Use these settings to create intermediate toolpath slices to reduce terracing.

Step cutting is particularly useful for Vortex-style toolpaths (see page 39).

- Changes to the **Tool orientation/Tool axis** settings available when using the **Pattern finishing** strategy to create 4- and 5-axis simultaneous toolpaths. (see page 43)



ASM is available as a license option for PartMaker/SwissCAM, PartMaker/TurnMill and PartMaker/Mill.

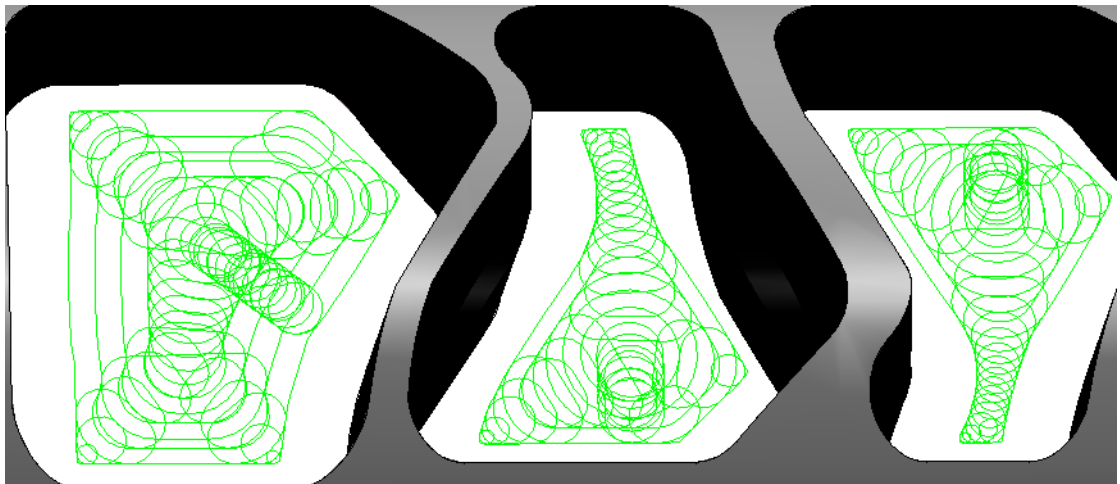


PartMaker does not support Vortex toolpaths created in PowerMILL. If you import a PowerMILL project that contains Vortex toolpaths into PartMaker, the Vortex toolpaths may not be processed correctly.

Vortex-style toolpaths

A new **Style** option of **Vortex** is available for the **Model Area Clearance** roughing strategy in Advanced Surface Machining (ASM). Using this style enables the rapid removal of material from a 3D part while controlling tool load.

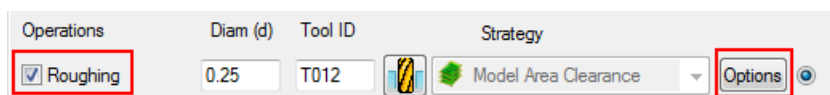
A Vortex-style toolpath produces an offset toolpath, which never exceeds the maximum tool engagement angle for optimum machining. As the tool approaches the maximum engagement angle, the toolpath changes to a trochoidal path to avoid tool overload. For example:



Vortex-style toolpaths often work well with step cutting (see page 37).

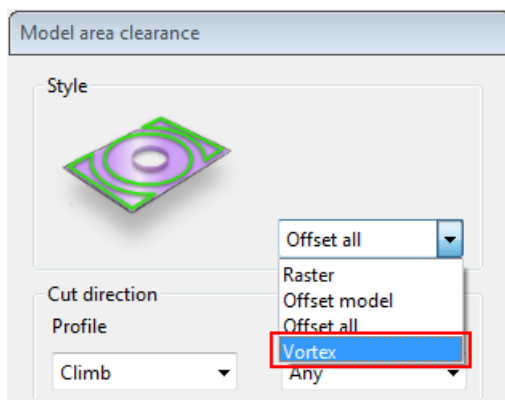
To create a Vortex-style toolpath:

- 1 Select **Roughing** on the **Surface Group Parameters** dialog, and then click the **Options** button:



PartMaker displays the **Strategy** dialog for **Model Area Clearance**.

- 2 Select **Vortex** as the **Style** on the **Model Area Clearance** page:



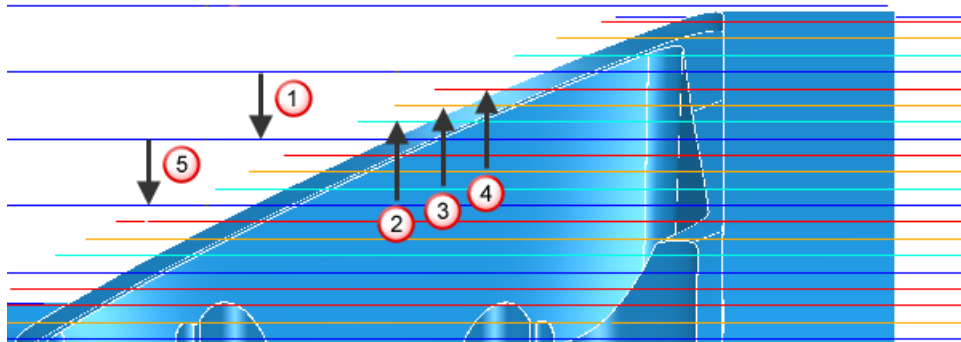
- 3 Complete the settings on the **Strategy** dialog, noting that selecting **Vortex**:
- Enables the **Vortex** page, which contains the settings required to define a Vortex toolpath.
 - Reduces the options available on the **Wall finishing** page.
 - Reduces the options available on the **High speed** page.
 - Provides additional options on the **Approach** page.
 - Disables the options on the **First lead in** page.
 - Disables the options on the **Lead in** page
 - Disables the **Inward** options on the **Extensions** page.



See the *PartMaker Reference Help* for more details about the settings in the **Strategy** dialogs.

Step cutting settings

A new **Step cutting** page is available on the **Model Area Clearance Strategy** dialog. Use the settings on this page to create intermediate toolpath slices that reduce terracing:

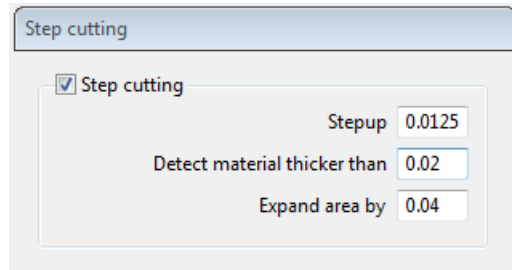


- ① The first pass is a main slice.
- ② The next pass is the lowest step-cutting slice.
- ③ The next pass is the next to lowest step cutting slice and so on up the part (shown by ④) until the previous main slice is reached.
- ④ The next main slice.

The main slices are cut from top to bottom as normal. The intermediate slices are cut from bottom to top. This is possible because the depth of cut is always less than the depth of cut (or **Stepdown**) for the main slice. This is the most efficient way of cutting the part as it reduces the number of passes and increases the depth of cut of each pass.

To use step cutting for a **Model Area Clearance** toolpath:

- 1 Select **Step cutting** on the **Model Area Clearance Strategy** dialog to display the **Step cutting** page.
- 2 Select the **Step cutting** option to enable step cutting:



- 3 Complete the following fields:
 - **Step up** — Enter the distance between intermediate cutting levels.
 - **Detect material thicker than** — Enter a threshold value. PartMaker ignores rest material that is thinner than the specified threshold. This helps to avoid thin regions being rest machined, where the benefit of a second cut is negligible. These thin regions can be caused by cusps from the previous toolpath.
 - **Expand area by** — Enter the distance by which to expand the rest areas, measured along the surface. Use with the **Detect material thicker than** value to reduce the areas to be machined to the details (for examples, corners), and then to offset these areas slightly to ensure that all the detail (for example, on the corners) is machined.

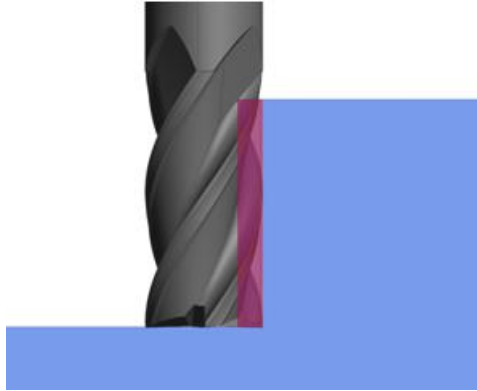


*See the PartMaker Reference Help for more details about the settings in the **Step cutting** page.*

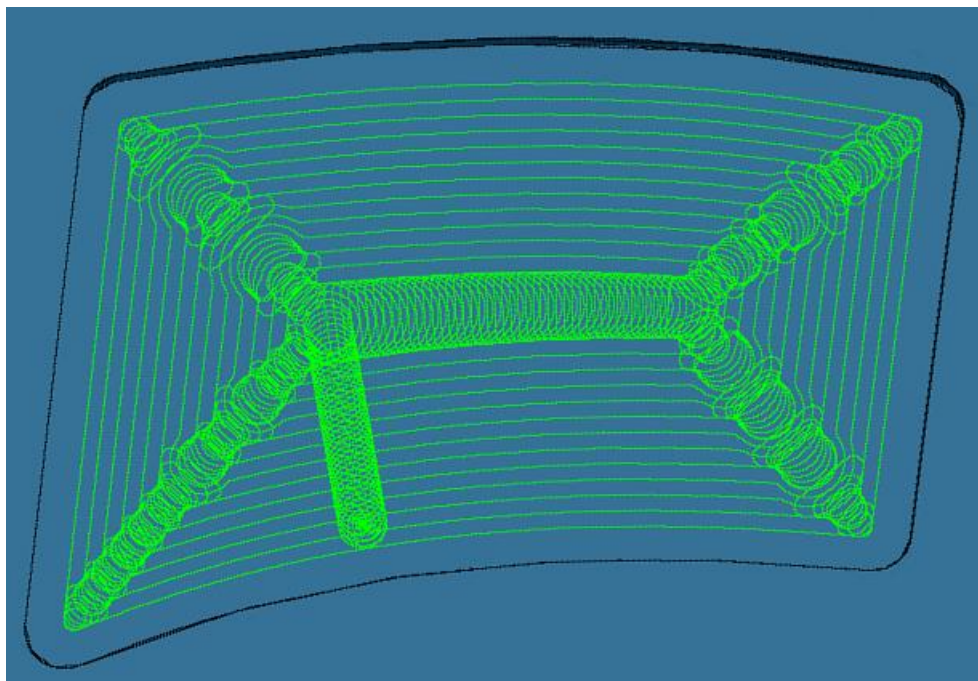
Example of using Vortex with step cutting

Vortex-style toolpaths often work well with step cutting. This example shows how to combine Vortex machining with step cutting to rapidly remove material.

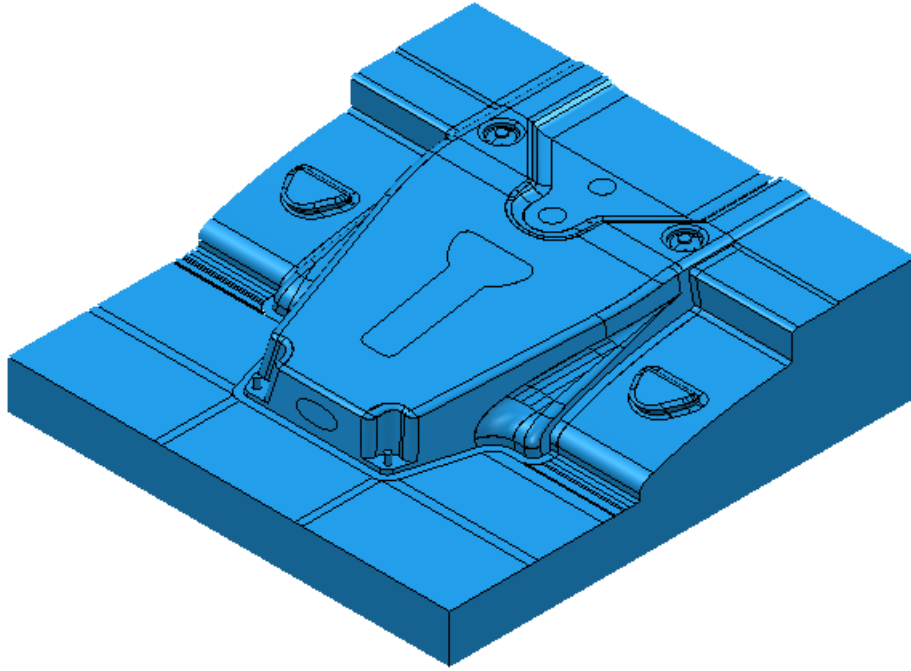
As Vortex machining cuts with the side of the tool, it is designed for solid carbide tools, but there may be other types of tools suitable for Vortex. These tools work best when taking deep cuts with a relatively small stepover.



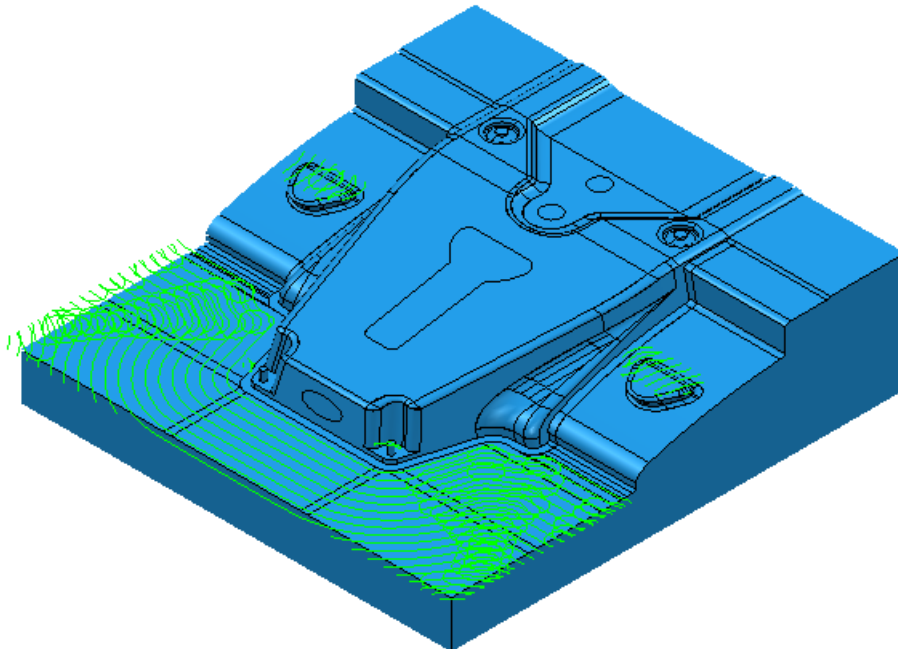
To machine effectively when taking large depths of cut, you must ensure the tool engagement angle never exceeds the specified value. This eliminates excessive tool load and all full-width cuts. PartMaker achieves this by introducing trochoidal moves to prevent the tool from exceeding the maximum tool engagement value.



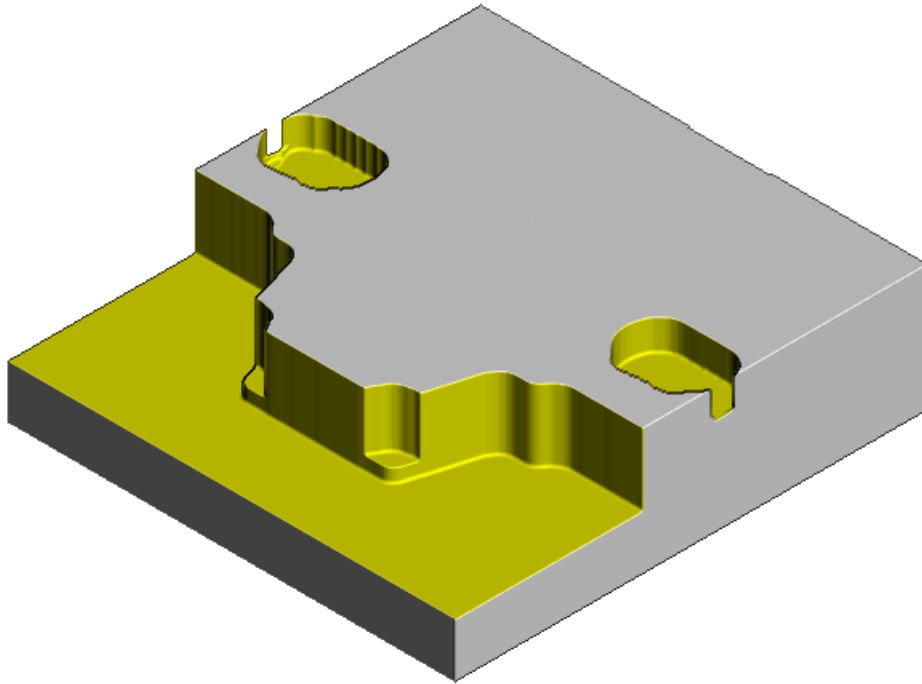
Using a 3D model:



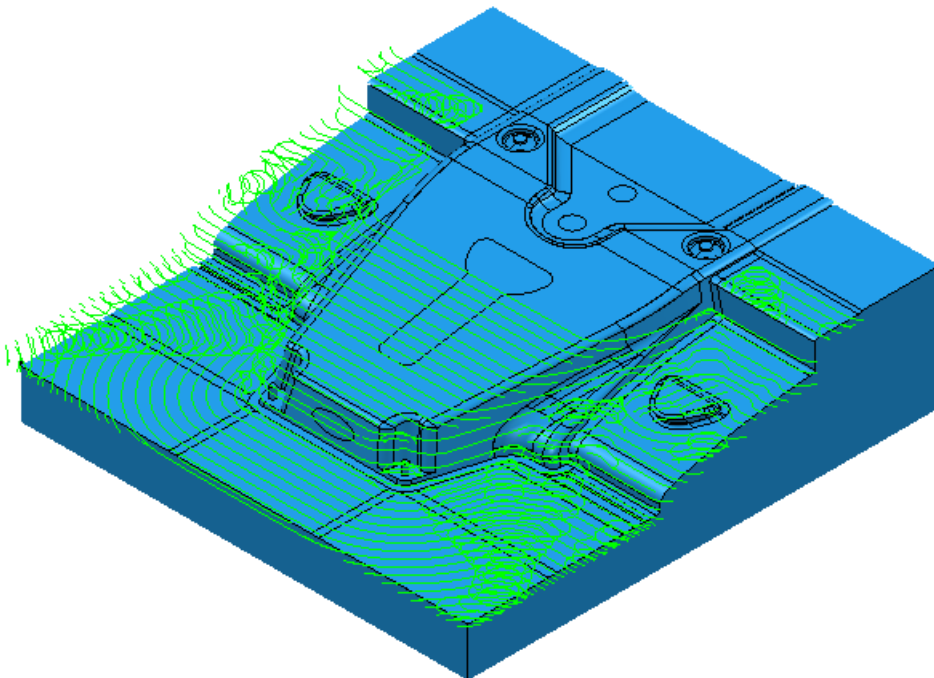
Creating a Vortex toolpath without step cutting.



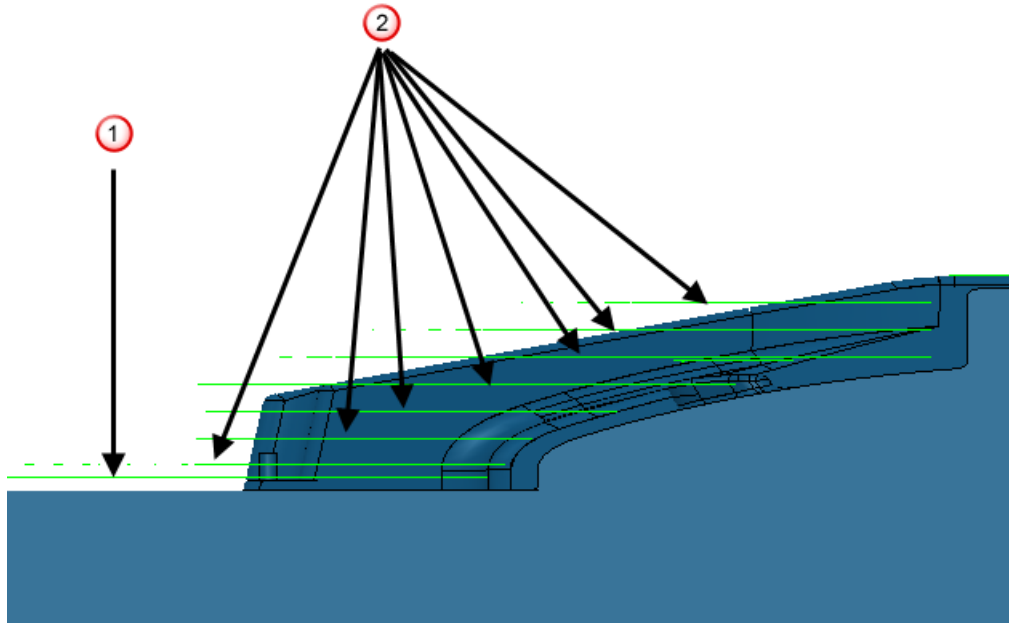
This removes vast quantities of material quickly, but leaves large terraces of unmachined stock on the part.



You can minimize the size of these terraces using the **Step cutting** options.



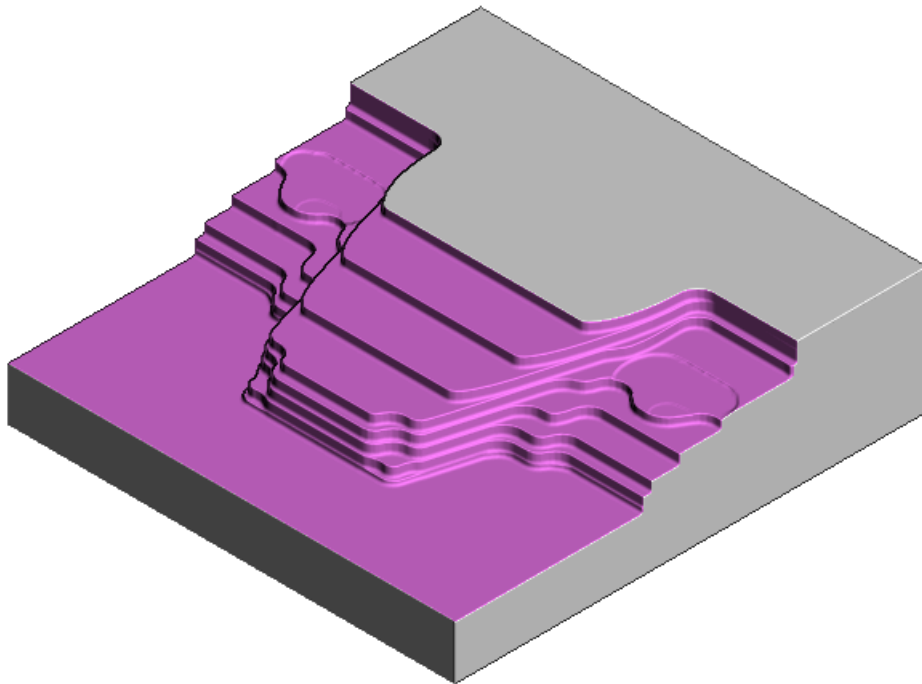
This adds extra machining slices up the part. Looking at a detail of the side view:



① Original Vortex pass

② Step cutting passes

It also machines more of the excess stock.

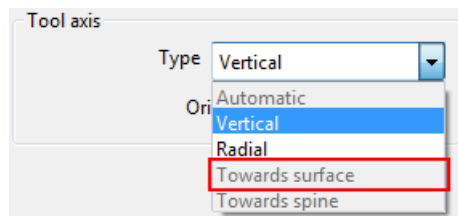


Changes to the Pattern finishing strategy

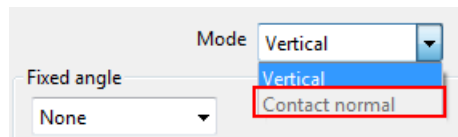
To ensure **Pattern finishing** toolpaths are calculated correctly for 4- and 5-axis simultaneous toolpaths, PartMaker 2014 now disables certain combinations of settings.

When a **Base position** of **Drive curve** is selected (so the lowest position of the toolpath follows the pattern), the following options are no longer available:

- For 4-axis simultaneous toolpaths, the **Towards surface** option is not available on the **Tool orientation** page.



- For 5-axis simultaneous toolpaths, the **Contact normal** option is not available on the **Tool axis** page.



If you open a .job file created in a previous version of PartMaker that contains these combinations of settings, PartMaker displays a message prompting you to re-verify these toolpaths to ensure they produce the expected results.

ConfigPost enhancements

In ConfigPost 2014 additional values are available for the `<tool-type>` reserved word to support the new tool types in PartMaker 2014 (see page 7):

- NT** – Neutral Turn
- BT** – Back Turn
- TE** – Tapered End Mill

Help and support

If you are a new PartMaker user, you may want to use the following resources to help you get started:

- Help and documentation (see page 44).
- Training (see page 45).

If you experience any problems when installing or using PartMaker, contact technical support (see page 45).

If you require any modifications to the Distribution Post files you have licensed, refer to the Post Processor modification policy.

Help and documentation

To help you get started, PartMaker is supplied with a comprehensive online Reference Help, which you can access from the **Help** menu. Further documentation is also provided on the PartMaker Documentation CD.

You can also download documentation from the Delcam Customer Downloads Center (<http://updates.delcam.com/login.php>), as described in *Technical support for PartMaker* (see page 45).

Training for PartMaker and ConfigPost

A wide variety of PartMaker and ConfigPost training programs are available from PartMaker, Inc., Delcam plc and their sales agents as follows:

In North America

- PartMaker, Inc, see the Training section of the PartMaker website (www.partmaker.com).

Outside of North America

- Delcam plc, see the Training section of the Delcam website (www.delcam.com).
- Delcam plc sales agents, contact your sales agent directly. The Delcam website (www.delcam.com) contains a list of sales agents and their contact details.

Technical support for PartMaker

Customers with an annual maintenance agreement can obtain technical support for PartMaker. This support may be provided by phone, email or by accessing technical support information available on the Internet.

Before contacting technical support, please refer to the documentation and help supplied with PartMaker to see if you can find information to solve your query.

Internet support

You can also access software releases, documentation, technical notes and videos from the Delcam Customer Downloads Center (<http://updates.delcam.com/login.php>).

Phone and email support

If you are unable to resolve your query, make sure you do the following before contacting technical support:

- You know the name and version of the PartMaker product you are using.
- You are able to describe the problem in detail. You may find it useful to document exactly what happened.
- If possible, be sure you can reproduce the problem. A support technician may ask you do this while you are on the phone.

- Be prepared to submit job (.job), tools (.tdb), cycles (.cdb) and Post Configuration files (.pst and .sub), as well as sample NC programs as attachments to an email message. Always try to zip your files using the WinZip program to make it easier and quicker to send the files by email.



*PartMaker will automatically zip the files if you use the **Help > Technical Support** option within PartMaker and select the **Send Files** button.*

To contact your local provider of technical support:

- For PartMaker, Inc in the United States, email support@partmaker.com or phone 215-643-5077.
- For Delcam plc in the United Kingdom, email support@delcam.com or call 0121 683 1010.
- For other countries, contact your local sales agent. Contact details for Delcam sales agents are available on the Delcam website.