PartMaker 2015

Important User Notices

Copyright © 1991-2014 Delcam Ltd. All rights reserved.

Delcam Ltd 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 the 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 Ltd 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

The Raceline smoothing functionality is subject to patent applications.

Patent granted: GB 2374562 Improvements Relating to Machine Tools

Patent granted: US 6,832,876 Machine Tools

The Vortex machining functionality is subject to patent applications.

Patent application: 1121277.6 Adaptive Clearance

PartMaker 2015. Published on 24 November 2014

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 2015 7

File compatibility in PartMaker ... 7
Post Processor Reference Guides ... 8
    Creating Post Processor Reference Guides ... 9
Using feeds and speeds ... 11
    Tool-based feeds and speeds ... 11
    Locking feeds and speeds ... 13
    Setting the spindle direction ... 15
    Feed and speed factors moved to a new dialog ... 17
Improvements to hole canned cycles ... 18
    Edit Hole Operation dialog ... 19
    Using the CHIP BREAK canned cycle ... 23
    Using the VARIABLE STEP canned cycle ... 25
    Using the HELICAL MILL canned cycle ... 27
    Changes to other dialogs ... 29
Improvements to canned cycles for Turning ... 34
    New Output Profile option ... 35
Trimming turning toolpaths using stock profile boundaries ... 36
Vertical Rotary table support in PartMaker/Mill ... 39
    Changes to other dialogs when a Vertical Rotary table is selected ... 40
Using Turning in PartMaker/Mill ... 42
    Changes to other dialogs when Turning is enabled in PartMaker/Mill ... 43
    Changing the orientation of the stock ... 47
Updates to the Machine Data dialog in PartMaker/Mill ... 48
    Tool Head Properties dialog ... 50
    Vertical Rotary Table Properties dialog ... 51
Support for compound rotary holders ... 53
Applying nose radius compensation to each process ... 56
Axial step control ... 57
Simulation Options dialog ... 59
    Fixing invalid solid models ... 60
    Increasing simulation speed ... 60
User interface changes ... 61

Profile Group Parameters, Turn dialog 62
Define Custom Shape dialog 63
Thread Mill settings 64
Table Settings dialog 65
5-Axis Rotary Table Properties dialog 65
Viewing tool parameters 66
Updated Setup dialog in PartMaker/Mill 67
Using the Manage Tools Files dialog 69
ConfigPost enhancements 71
New and updated reserved words 71
Postprocessing canned cycles for holes 73
User Data Names dialog 77
Variables Names dialog 78
File compatibility in ConfigPost 78
New Machine Settings option in ConfigPost 79
Naming convention for Post Files 80

Help and support 81

Help and documentation 81
Training for PartMaker and ConfigPost 81
Technical support for PartMaker 82

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 Ltd in 2006, PartMaker Inc. is a wholly owned subsidiary of Delcam Ltd, 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 2015 (see page 7).
- Accessing help and support (see page 81).

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.
3. Install your license file (PAF).

*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 2015 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 versions of PartMaker are available and are supported on Windows 7 and higher.

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

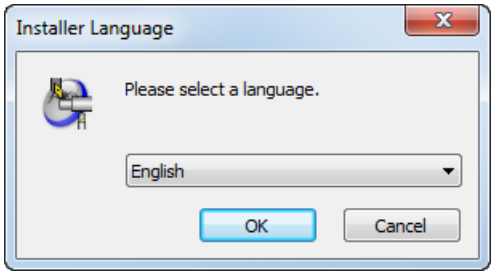
PartMaker 2015 does not support Windows XP or Windows Vista. Although you may be able to run PartMaker 2015 on these operating systems, Delcam will not address any software performance issues found on Windows XP or Windows Vista that cannot be reproduced on Windows 7 or higher.

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 or higher, 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.

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; and
- a Product Authorization File (PAF) license file, which you install onto the computer.

PAFs are usually sent in an email from Delcam Ltd 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 2015

PartMaker 2015 contains many new features and enhancements.

File compatibility in PartMaker

ConfigPost file compatibility

- .pst and .sub files produced in ConfigPost 2014 are compatible with PartMaker 2015.
- .pst and .sub files produced in ConfigPost 2015 are not compatible with PartMaker 2014. If you try to open such files in PartMaker 2014, a message dialog is displayed saying that there is a problem opening the Post Configuration file.
- .pst and .sub files produced in ConfigPost 2015 are recognized by a '-*ve*' suffix placed after the name, for example, Citizen_M3_32-ve122.pst.

PowerMill Project file compatibility

PartMaker/Turn-Mill and PartMaker/SwissCAM 2015 can import PowerMILL Projects created in PowerMILL V2015 R1 SP6 and lower.

Inventor *.IPT file compatibility

All PartMaker 2015 modules can import files created in Autodesk Inventor V2015 and lower.

SolidWorks *.SLDPRT file compatibility

All PartMaker 2015 modules can import files created in SolidWorks V2015 and lower.

Parasolid *.X_T, *.X_B file compatibility

All PartMaker 2015 modules can import Parasolid files V26 and lower.

Pro/Engineer *.prt file compatibility

All PartMaker 2015 modules can import files created in Wildfire 5 and lower.

Creo *.prt file compatibility

All PartMaker 2015 modules can import files created in Creo 2.0 and lower.

Post Processor Reference Guides

In PartMaker/SwissCAM and PartMaker/Turn-Mill 2015 you can now create Post Processor Reference Guides that describe the CNC programming capabilities of licensed post processors (Post Configuration files).

Previously, Delcam provided machine-specific addendum documents to describe licensed post processors. These addendums were downloaded to your machine when you used the **Help > Download Licensed Post Config Files** option to download licensed Post Configuration files.

Addendums typically contained both the CNC programming capabilities of the Post Configuration file and specific information related to using the Post Configuration file in conjunction with the PartMaker interface. Some Post Processor Reference Guides contain a link to a document called a Reference Guide Supplement. Reference Guide Supplements contain only information related to using a specific Post Configuration file in conjunction with the PartMaker interface.

Over time, Post Processor Reference Guides and Reference Guide Supplements will replace machine-specific addendum documents. However, during this transition PartMaker will continue to provide addendum documents for licensed Post Configuration files.

Delcam recommends that you use Post Processor Reference Guides whenever possible for information relating to the CNC programming capabilities of licensed post processors, as they offer several advantages over machine-specific addendum documents:

- All Post Processor Reference Guides use a standard layout and contain the same level of information.

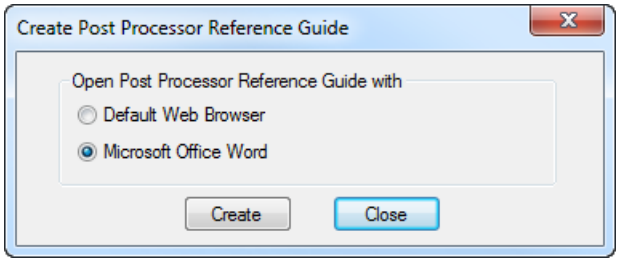
- A Post Processor Reference Guide is specific to the currently loaded post processor and machine (not to a range of machines).
- You can create a Post Processor Reference Guide at any time, by selecting the **Job Optimizer > Create Post Processor Reference Guide** option, so you can be sure it contains the most up-to-date information available about the post processor.

If you modify a Post Configuration file supplied by Delcam, you must ensure that the modifications do not prevent PartMaker from creating an accurate Post Processor Reference Guide for that post processor. Refer to a Post Processor Reference Guide for more details on how your modifications may affect the creation of Post Processor Reference Guides.

Creating Post Processor Reference Guides

To create a Post Processor Reference Guide for the currently loaded Post Configuration file in PartMaker/SwissCAM or PartMaker/Turn-Mill:

1. Select **Job Optimizer > Create Post Processor Reference Guide** to display the **Create Post Processor Reference Guide** dialog:



2. Specify how PartMaker will open the Post Processor Reference Guide.
   - **Default Web Browser** — Displays the Post Processor Reference Guide in your default web browser.
   - **Microsoft Office Word** — Displays the Post Processor Reference Guide in Microsoft Word.

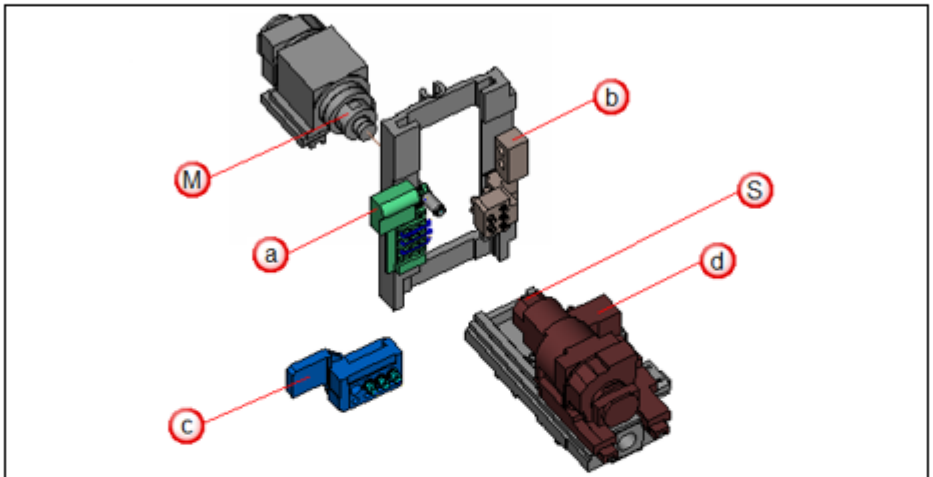
   PartMaker creates all Post Processor Reference Guides as .html files, but you can choose to view them in a web browser or in Microsoft Word.
3. Click **Create**. PartMaker displays the **Save Post Processor Reference Guide** dialog.
4. Select a folder and filename for the Guide, and then click **Save**.

PartMaker creates and displays the Post Processor Reference Guide. For example:

PartMaker/SwissCAM Post Processor Reference Guide

Machine Manufacturer	Tsugami	Machine Model	S206
Control Manufacturer	Fanuc	Control Series	31i-A

Machine Layout



	Tool Post	Tool Range*	(M) Main Spindle	(S) Sub-Spindle
(a)	Gang Slide	1-8	Yes	Yes
(b)	Gang Slide #2	11-11, 13-13, 15-19, 21-23, 45-49	Yes	Yes
(c)	Back Working	31-38	No	Yes
(d)	End Working	91-92	Yes	No

Machining Functions Support

Machining Function	Main Spindle	Sub-Spindle
Turn	Yes	Yes
Mill End, Index	Yes	Yes
Mill End, Polar	Yes	Yes

Using feeds and speeds

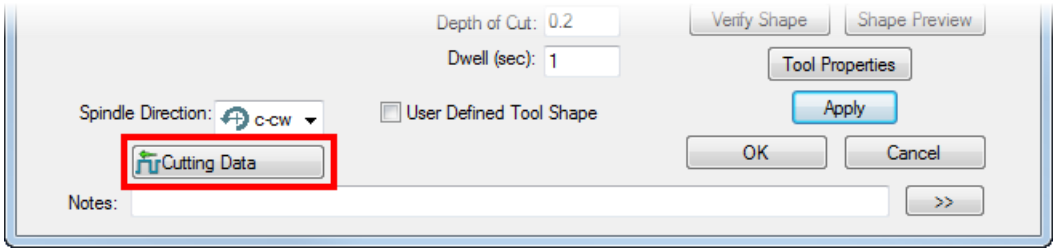
There are several changes to how you can use feeds and speeds in PartMaker.

- Set feeds and speeds per tool (see page 11).
- Lock feeds and speeds per process (see page 13).
- Specify the spindle direction (see page 15).
- Feed and speed factors moved to a new dialog (see page 17).

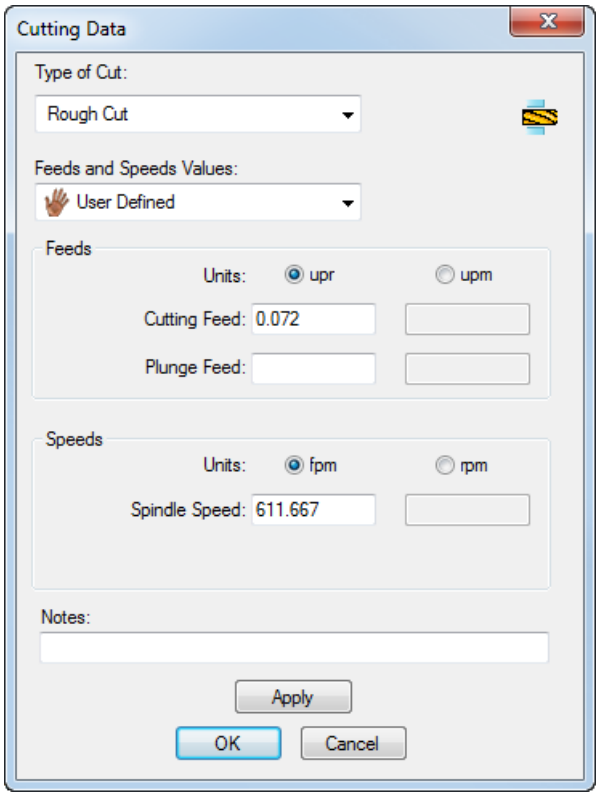
Tool-based feeds and speeds

In PartMaker/SwissCAM, PartMaker/Turn-Mill, PartMaker/Mill, and PartMaker/Turn, you can specify machine feed rates and spindle speeds for each tool using the new **Cutting Data** dialog.

To display the **Cutting Data** dialog, click the **Cutting Data** button on either the **Tool Data** or **Edit Tool** dialogs.



The settings available on the **Cutting Data** dialog depend on the type of tool you are using, and not all settings are available for every tool.



Type of Cut — Select whether the tool is used for a roughing or finishing operation. If the tool can be used only for one type of operation, this field is unavailable.

Rough Cut — Select if the tool is used for a roughing operation.

Finish Cut — Select if the tool is used for a finishing operation.

Feeds and Speeds Values — Select whether to use the values calculated by PartMaker, or to define your own feeds and speeds values.

Calculated — When selected, PartMaker calculates values based on the Material file.

User Defined — When selected, you can enter your own values in the available fields.

Units (Feeds) — Select the units used to specify the feed rates.

upr — Select if you want to specify the value in units-per-revolution.

upm — Select if you want to specify the value in units-per-minute.

*For Turning tools, only **upr** is available because **upm** depends on the stock diameter, which is not defined per tool.*

Cutting Feed — Enter a value for the tool's cutting feed rate.

Plunge Feed — Enter a value for the tool's plunge feed rate.

Units (Speeds) — Select the units used to specify the spindle speed.

fpm / mpm — Select if you want to specify the value in feet-per-minute (fpm) if you are working in the inches system or meters-per-minute (mpm) if you are working in the metric system.

rpm — Select if you want to specify the value in revolutions-per-minute.

*For Turning tools, only **fpm** is available because **rpm** depends on the stock diameter, which is not defined per tool.*

Spindle Speed — Enter a value for the spindle speed.

Tool Speed — Enter a value for the tool speed. This field is available only for Thread Whirl tools.

Notes — Enter any additional information about the cutting data.

Apply — Click to recalculate the units. If you are using a Turning tool, this field is unavailable because you cannot change the units.

OK — Click to apply your changes and close the dialog.

Cancel — Click to discard any changes that have not been applied and close the dialog.

Locking feeds and speeds

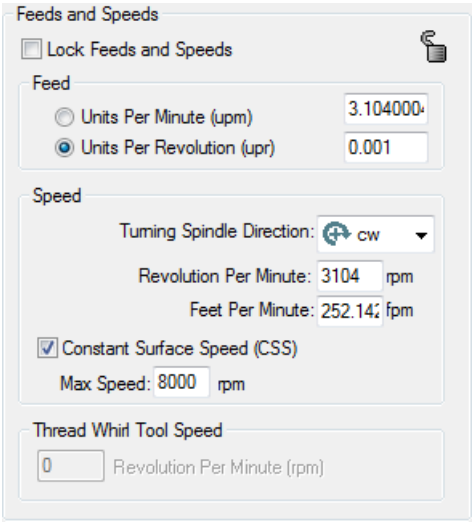
In PartMaker 2015, the feeds and speeds settings on the **Process Parameters** dialogs are grouped together in a **Feeds and Speeds** area. Within this area, a **Lock Feeds and Speeds** option enables you to lock all these settings, which means the feeds and speeds can still be retained when:

- generating a Process Table with **Retain Process Table Modifications** *deselected*.

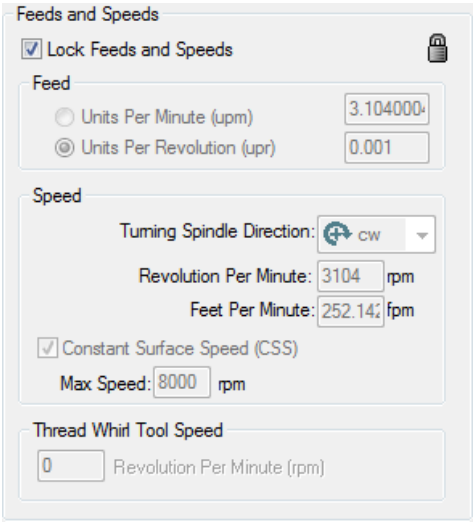
- generating a Process Table with **Retain Process Table Modifications** *selected*, and a different material file loaded.

The **Lock Feeds and Speeds** option is deselected by default when a Process Table is generated for the first time. When selected, all values within the **Feeds and Speeds** area are locked.

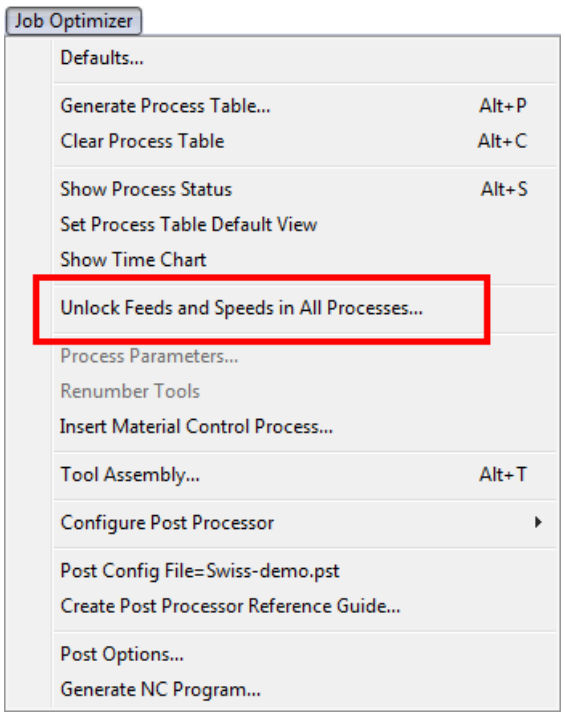
Process Parameters (Turn): unlocked



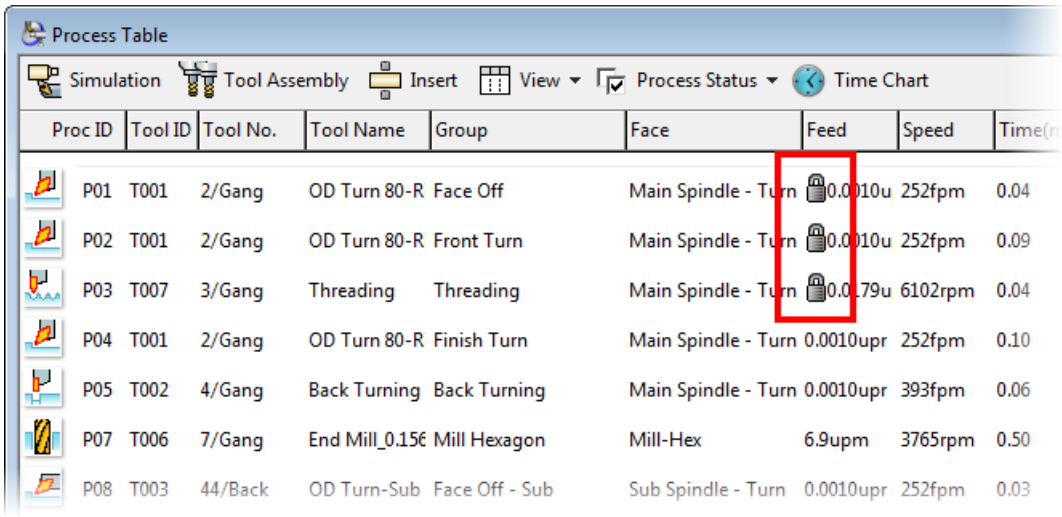
Process Parameters (Turn): locked



There is an **Unlock Feeds and Speeds in All Processes** option on the **Job Optimizer** menu. This option is disabled if there are no processes with locked feeds and speeds.



On the **Process Table**, a padlock icon is displayed beside feeds and speeds that are locked.



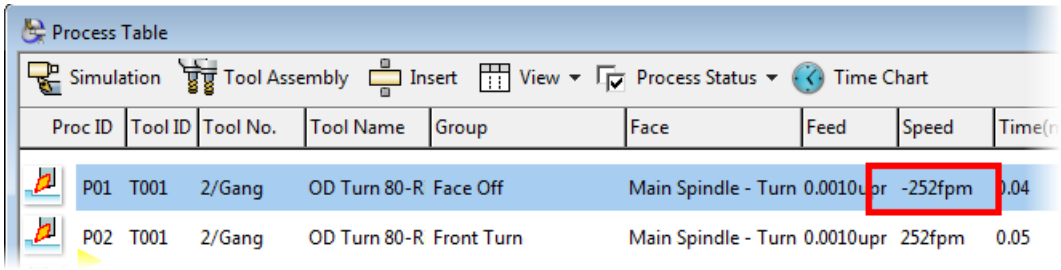
Setting the spindle direction

In PartMaker/SwissCAM, PartMaker/Turn-Mill, PartMaker/Turn, and PartMaker/Mill, a drop-down list is added to the **Tool Data**, **Edit Tool**, and **Process Parameters** dialogs, which enables you to set the direction of the spindle, either per tool, or per process. This means you no longer have to enter a **Speed Factor** of -1 in the **Tool Data** and **Edit Tool** dialogs, or enter a negative speed in the **Process Parameters** dialog to reverse the direction of the spindle.

The following settings are available from the **Spindle Direction** drop-down list:

cw — Select to set a clockwise rotation for the spindle.

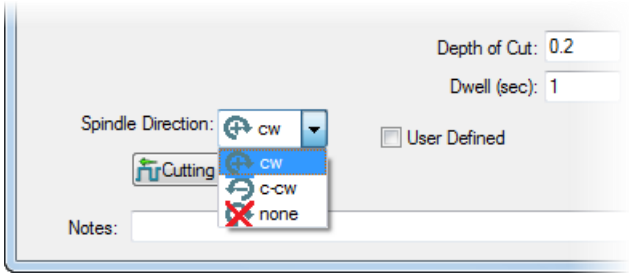
c-cw — Select to set a counter-clockwise rotation for the spindle. When you select a counter-clockwise spindle direction, a minus symbol is displayed in front of the **Speed** value in the Process Table.



none — Select this option to stop the spindle rotating. The **Spindle Speed** is set to 0, and the field is disabled.

Setting the spindle direction for a tool

In the **Tool Data** and **Edit Tool** dialogs, select an option from the **Spindle Direction** drop-down list to set the the direction of the spindle per tool.



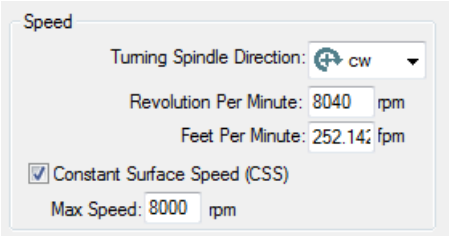
*When you open the **Edit Tool** dialog from the Process Table and change the spindle direction, PartMaker will not immediately change the spindle direction in all processes where this tool is used. You must regenerate the Process Table to update the Spindle Direction.*

Setting the spindle direction for a process

When a Process Table is generated, the spindle direction setting is copied from a tool to a process. In the case of a Turning process, the spindle direction could be different between main spindle and sub-spindle processes using the same tool.

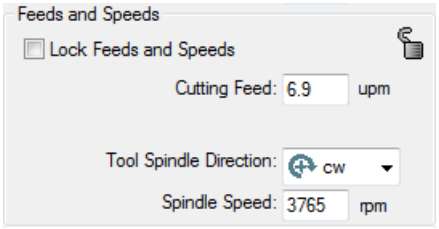
In the **Process Parameters** dialogs, select an option from the drop-down list to set the direction of the spindle for a process.

- For turning processes, select an option from the **Turning Spindle Direction** drop-down list to set the spindle direction:

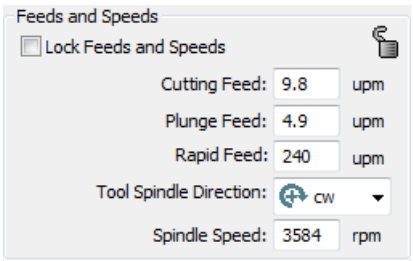


- For milling and surfacing processes, select an option from the **Tool Spindle Direction** drop-down list to set the spindle direction:

Process Parameters (milling)

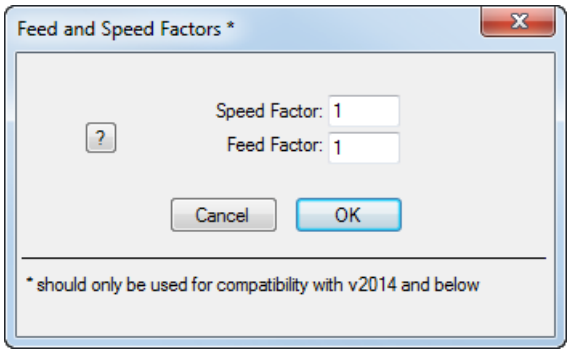


Process Parameters (ASM)

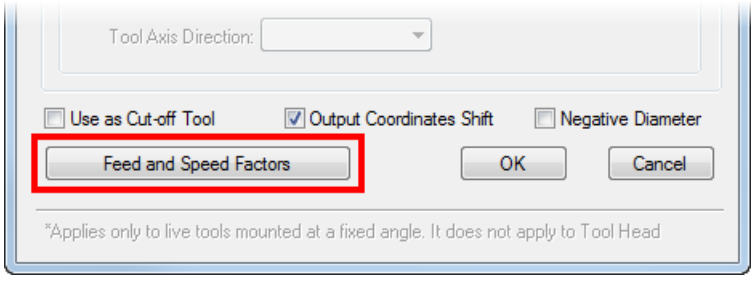


Feed and speed factors moved to a new dialog

The **Feed Factor** and **Speed Factor** settings are now located on the **Feed and Speed Factors** dialog. Previously, these settings were available from the **Tool Data** and **Edit Tool** dialogs.



To display the **Feed and Speed Factors** dialog, click **Feed and Speed Factors** on the **Tool Properties** dialog.



Feed and speed factors are kept for compatibility with old job files. They should no longer be used in PartMaker 2015 because you can now enter cutting data per tool with the new **Cutting Data** dialog (see page 11), and set the spindle direction with the **Spindle Direction** option on the **Tool Data** and **Edit Tool** dialogs (see page 15).

Improvements to hole canned cycles

Canned cycles simplify the NC Program for the part by using a single G-code to initiate a sequence of machine operations.

In PartMaker 2015 the handling of canned cycles is improved, and several areas are updated as a result of these improvements:

Edit Hole Operation dialog (see page 19) — There is a new **Edit Hole Operation** dialog, where you can view, and in some cases modify, settings for machining holes.

New hole canned cycles — There are three new canned cycles for machining holes:

Turning and milling:

- CHIP BREAK canned cycle (see page 23)
- VARIABLE STEP canned cycle (see page 25)

Milling only

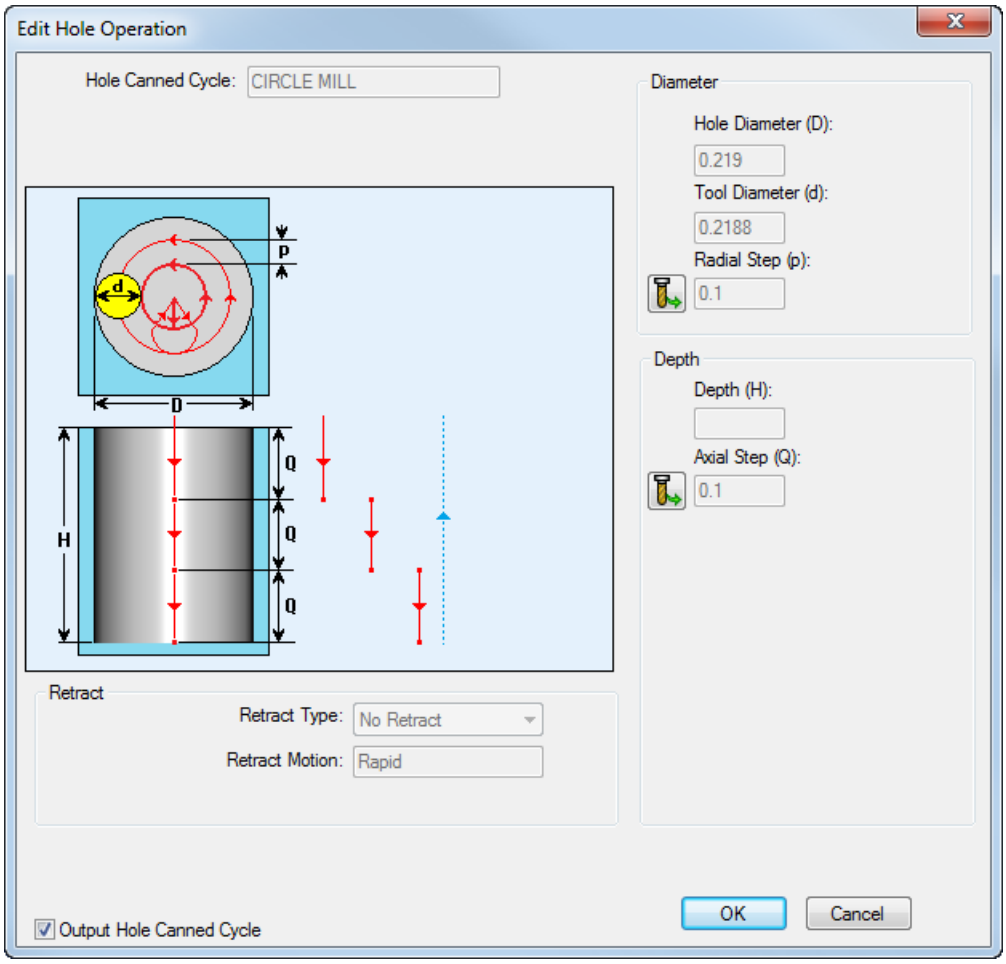
- HELICAL MILL canned cycle (see page 27)

Postprocessing canned cycles (see page 73) — There are several changes to ConfigPost to support the improvements to hole canned cycles.

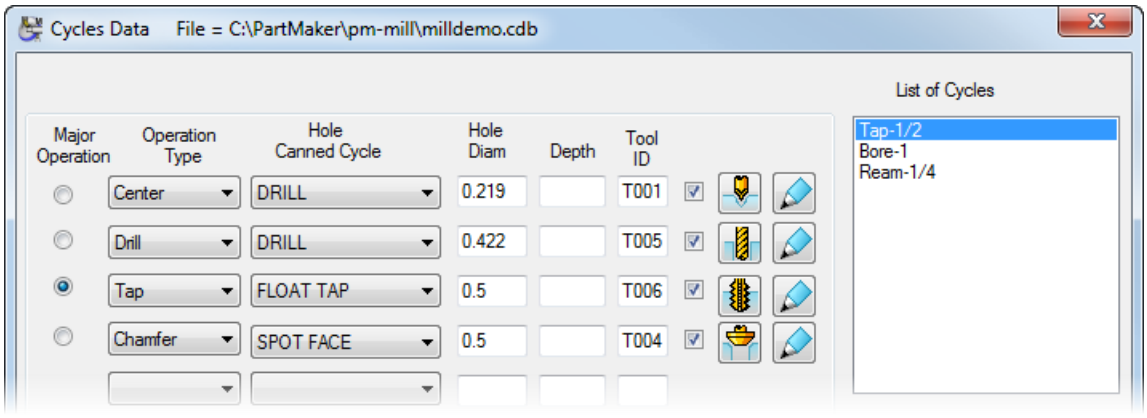
Changes to other dialogs (see page 29) — There are a number of changes to other dialogs as a result of the improvements to canned cycles.

Edit Hole Operation dialog

In PartMaker 2015, there is a new **Edit Hole Operation** dialog that enables you to view and specify settings for machining a hole canned cycle.



To display the **Edit Hole Operation** dialog, click the new **Edit Hole Operation** button beside the operation you want to edit on either the **Cycles Data** or **Edit Cycle** dialogs . This button is displayed only beside operations that have a tool selected.



When the **Edit Hole Operation** dialog is displayed, the button that you clicked to open it turns orange to make it clear which operation you are editing.

The settings available on the dialog depend on which hole canned cycle you have selected on the **Cycles Data** or **Edit Cycle** dialog.

Hole Canned Cycle — This field displays the name of the hole canned cycle selected on the **Cycles Data** or **Edit Cycle** dialogs.

Style — Select the style for machining the hole.

For RIGID TAP hole canned cycles, the following options are available:

- **Single Step** — Select to tap the hole using a single step.
- **Multiple Steps** — Select to tap the hole using multiple steps.

For THREAD MILL hole canned cycles, the following options are available:

- **Bottom Up** — Select so the tool starts at the bottom of the hole and threads up to the top of the hole.
- **Top Down** — Select so the tool starts at the top of the hole and threads to the bottom of the hole.

Thread Direction — Select the direction of the thread:

- **Right Hand** — Select to cut a right-handed thread, which is standard in threading applications.
- **Left Hand** — Select to cut a left-handed thread.

This field is displayed only for THREAD MILL hole canned cycles.

Helical Step Defined By — Select how the step is defined:

- **Pitch** — Select to define the step by the pitch of the helix. The **Pitch (p)** field is displayed in the **Depth** area.
- **Angle** — Select to define the step by the angle of the helix. The **Angle (A)** field is displayed in the **Depth** area.

This field is displayed only for HELICAL MILL hole canned cycles.

Hole Diameter (D) — This field displays the diameter of the hole. The value is taken from the **Hole Diam** field on the **Cycles Data** or **Edit Cycle** dialogs.

Tool Diameter (d) — This field displays the diameter of the tool. The value is taken from the **Tool Data** or **Edit Tool** dialogs.

This field is displayed only for CIRCLE MILL, THREAD MILL, and HELICAL MILL hole canned cycles.

Radial Step (p) — Enter the amount of material removed in each circular pass. If the **Tool Parameter** button is displayed, the value is taken from either the **Tool Data** or **Edit Tool** dialogs. If the **User Defined Parameter** button is displayed, you can modify the value. Click to change it to .

This field is displayed only for CIRCLE MILL hole canned cycles.

Depth (H) — This field displays the depth of the hole. The value is taken from the **Depth** field on the **Cycles Data** or **Edit Cycle** dialogs.

Axial Step (Q) — Enter the depth of the steps used to drill the hole. If the **Tool Parameter** button is displayed, the value is taken from the **Tool Data** or **Edit Tool** dialogs. If the **User Defined Parameter** button is displayed, you can modify the value. Click to change it to .

This field is displayed only for CHIP BREAK, DEEP HOLE, VARIABLE STEP, RIGID TAP (with Multiple Steps selected), and CIRCLE MILL canned cycles.

Step Decrease (s) — After the initial step, the step decreases until a minimum step is reached.

This field is displayed only for VARIABLE STEP hole canned cycles.

Minimum Step (m) — The step does not decrease below this value.

This field is displayed only for VARIABLE STEP hole canned cycles.

Feed Factor for Initial Step — Enter a value to modify the feed rate entered in the Process Table by a factor on the initial step.

This field is displayed only for VARIABLE STEP hole canned cycles.

Pitch (P)

- For THREAD MILL hole canned cycles — Enter the distance between the teeth of the thread when drilling the hole. The default value of this option is calculated from the Threads Per Inch value (Inch units) or taken from the Pitch value (metric units) from the tool's properties in the **Tool Data** dialog.

- For HELICAL MILL hole canned cycles — Enter the pitch of the helix. This field is displayed only if **Pitch** is selected in the **Helical Step Defined By** drop-down list.
- For both THREAD MILL and HELICAL MILL hole canned cycles — If the **Tool Parameter** button is displayed, the value is taken from the **Tool Data** or **Edit Tool** dialogs. If the **User Defined Parameter** button is displayed, you can modify the value. Click to change it to .

Angle (A) — Enter the angle of the helix. This field is displayed only if **Angle** is selected in the **Helical Step Defined By** drop-down list.

This field is displayed only for HELICAL MILL hole canned cycles.

Number of Spring Passes — Enter the number of clean-up passes taken at the bottom of a hole.

This field is displayed only for HELICAL MILL hole canned cycles.

Retract Type

- **Full Retract** — After each step, the tool retracts completely out of the hole.
- **Partial Retract** — After each step, the tool retracts up the hole to the distance defined in the **Retract Distance (r)** field.

Retract Type is programmable only for VARIABLE STEP hole canned cycles.

Retract Motion

- **Rapid** — For all cycles except BORE and BORE,DWELL, the tool makes a rapid move out of the hole when it retracts.
- **Linear** — For BORE and BORE,DWELL hole canned cycles, the tool makes a linear move out of the hole at the feed rate entered on the Process Table when it retracts.

Retract Distance (r) — If **Partial Retract** is specified in the **Retract Type** drop-down list, enter the distance that the tool retracts up the hole between steps.

This field is available only for CHIP BREAK and VARIABLE STEP (with Partial Retract selected) hole canned cycles.

Output Hole Canned Cycle — This option is selected by default so the machining operation for the hole is output as a canned cycle in the NC Program file. If your postprocessor does not support all of the parameters for this canned cycle, deselect this option.

Thread Mill Options — Click to display the **Thread Mill Options** dialog, where you can control the start angle of the thread and whether a tapered thread is required.

OK — Click to save any changes and close the dialog.

Cancel — Click to discard any changes and close the dialog.

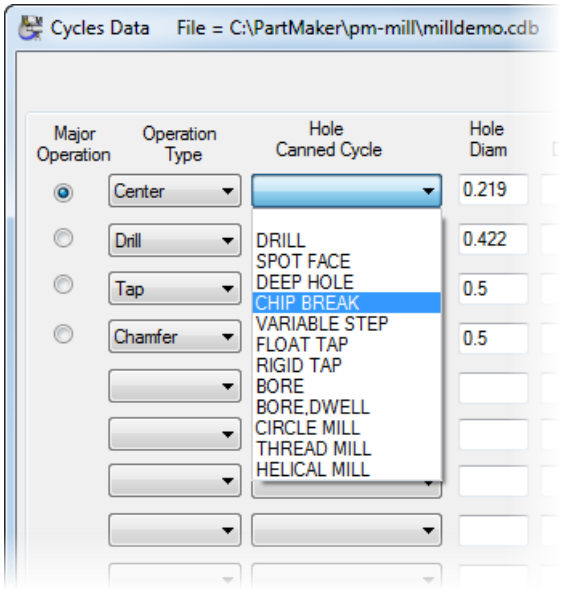
Using the CHIP BREAK canned cycle

When creating, or editing, a cycle, you can now select a CHIP BREAK canned cycle for machining holes.

The CHIP BREAK canned cycle drills a hole in several stages (multiple step), where the tool partially retracts after each step.

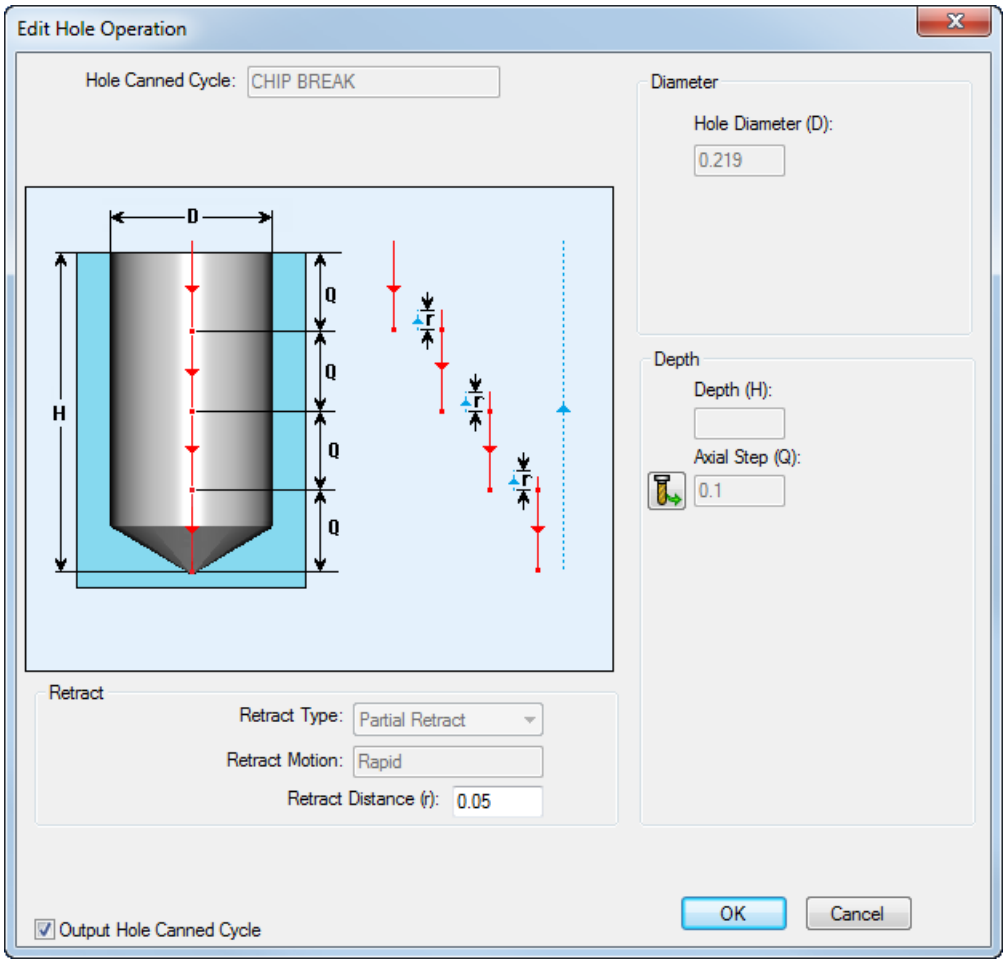
To create a cycle that uses the CHIP BREAK hole canned cycle:

1. Select **ToolMinder > Cycles** to display the **Cycles Data** dialog.
2. Complete the settings on the dialog and select *CHIP BREAK* as the **Hole Canned Cycle**:



3. Click the **Edit Hole Operation** button to display the **Edit Hole Operation** dialog for the CHIP BREAK canned cycle.

This button is displayed only if you have selected a tool.



4 On the **Edit Hole Operation** dialog, specify the following settings to control how holes are machined when using the CHIP BREAK cycle:

- **Axial Step (Q)** — Enter the depth of the steps used to drill the hole. If the **Tool Parameter** button is displayed, the value is taken from the **Tool Data** or **Edit Tool** dialog. If the **User Defined Parameter** button is displayed, you can modify the value. Click to change it to .
- **Retract Distance (r)** — Enter a value to control how far the tool retracts up the hole between each step.
- **Output Hole Canned Cycle** — This option is selected by default so the machining operation for the hole is output as a canned cycle in the NC Program file. If your postprocessor does not support any of the parameters for this canned cycle, deselect this option.

5 Click **OK** to close the **Edit Hole Operation** dialog and return to the **Cycles Data** dialog.

6 When you have completed creating, or editing, the cycle, click **Close** on the **Cycles Data** dialog.

Using the VARIABLE STEP canned cycle

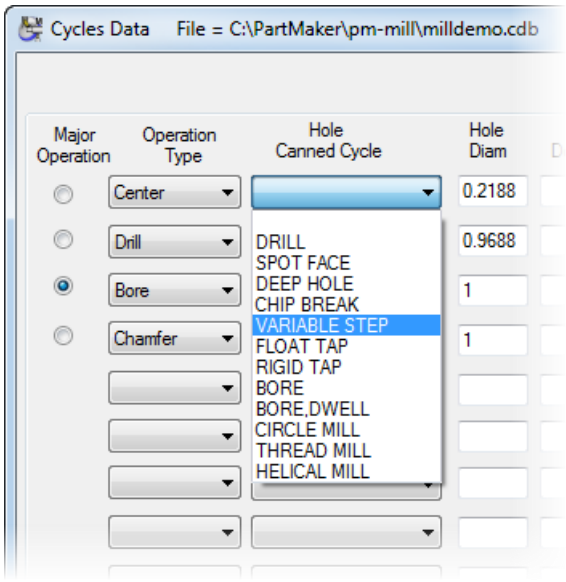
When creating, or editing, a cycle, you can now select a VARIABLE STEP canned cycle for machining holes.

The VARIABLE STEP canned cycle drills a hole where the step amount decreases after each step until a minimum step is reached. The minimum step is maintained until the bottom of the hole is reached.

To create a cycle that uses the VARIABLE STEP hole canned cycle:

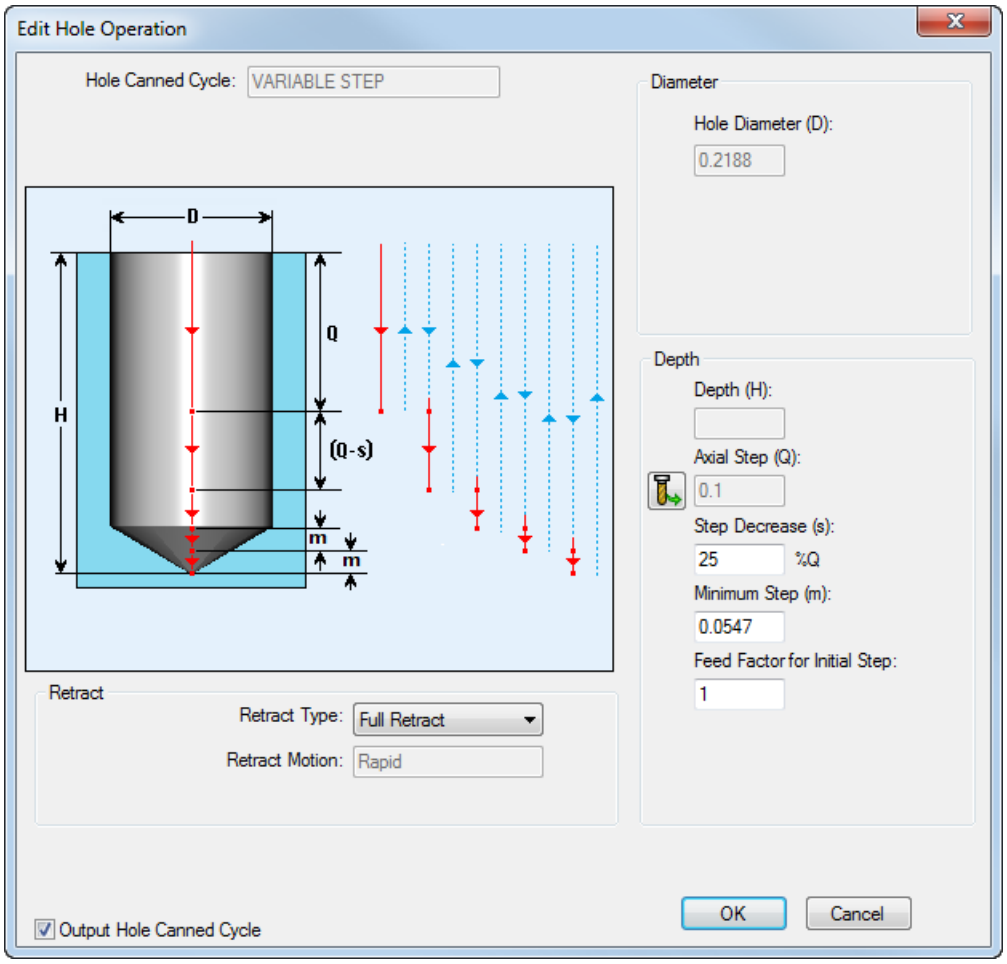
1 Select **ToolMinder > Cycles** to display the **Cycles Data** dialog.

2 Complete the settings on the dialog and select *VARIABLE STEP* as the **Hole Canned Cycle**.



3 Click the **Edit Hole Operation** button to display the **Edit Hole Operation** dialog for the *VARIABLE STEP* canned cycle.

This button is displayed only if you have selected a tool.



4 On the **Edit Hole Operation** dialog, specify the following settings to control how holes are machined when using the VARIABLE STEP canned cycle:

- **Axial Step (Q)** — Enter the depth of the steps used to drill the hole. If the **Tool Parameter** button is displayed, the value is taken from the **Tool Data** or **Edit Tool** dialogs. If the **User Defined Parameter** button is displayed, you can modify the value. Click to change it to .
- **Step Decrease (s)** — After the initial step, the step decreases until a minimum step is reached.
- **Minimum Step (m)** — The step does not decrease below this value.
- **Feed factor for Initial Step** — Enter a value to modify the feed rate entered in the Process table by a factor on the initial step.

- **Retract Type** — Select a retract type:

  Full Retract — After each step, the tool retracts completely out of the hole.

  Partial Retract — After each step, the tool retracts up the hole to the distance defined in the **Retract Distance (r)** field.

- **Output Hole Canned Cycle** — This option is selected by default so the machining operation for the hole is output as a canned cycle in the NC Program file. If your postprocessor does not support all of the parameters for this canned cycle, deselect this option.

5 Click **OK** to close the **Edit Hole Operation** dialog and return to the **Cycles Data** dialog.

6 When you have completed creating, or editing, the cycle, click **Close** on the **Cycles Data** dialog.

Using the HELICAL MILL canned cycle

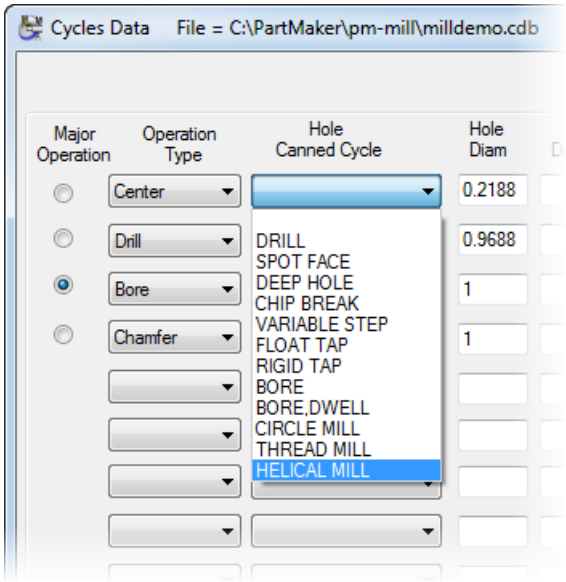
When creating, or editing, a cycle, you can now select a HELICAL MILL canned cycle for machining holes.

The HELICAL MILL canned cycle machines a hole with an endmill tool that is smaller than the hole.

To create a cycle that uses the HELICAL MILL hole canned cycle:

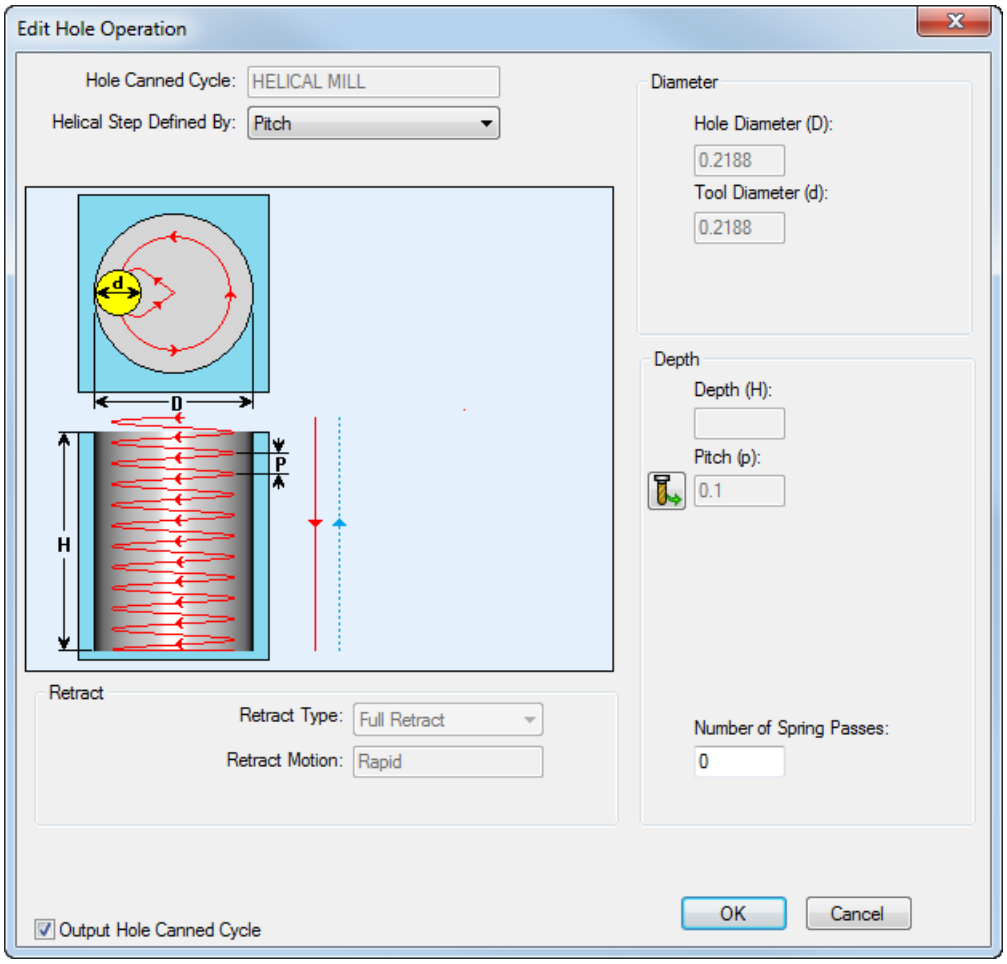
1 Select **ToolMinder > Cycles** to display the **Cycles Data** dialog.

2 Complete the settings on the dialog and select *HELICAL MILL* as the **Hole Canned Cycle**.



3 Click the **Edit Hole Operation** button to display the **Edit Hole Operation** dialog for the *HELICAL MILL* canned cycle.

This button is displayed only if you have selected a tool.



4 On the **Edit Hole Operation** dialog, specify the following settings to control how holes are machined when using the HELICAL MILL canned cycle:

- **Helical Step Defined By** — Select how the step is defined.
- **Pitch (P)** — Enter the pitch of each helical step. This field is displayed only if **Pitch** is selected in the **Helical Step Defined By** drop-down list.
- **Angle (A)** — Enter the angle of each helical step. This field is displayed only if **Angle** is selected in the **Helical Step Defined By** drop-down list.
- **Number of Spring Passes** — Enter the number of clean-up passes taken at the bottom of a hole. This is not used for through-holes.
- **Output Hole Canned Cycle** — This option is selected by default so the machining operation for the hole is output as a canned cycle in the NC Program file. If your postprocessor does not support any of the parameters for this canned cycle, deselect this option.

5 Click **OK** to close the **Edit Hole Operation** dialog and return to the **Cycles Data** dialog.

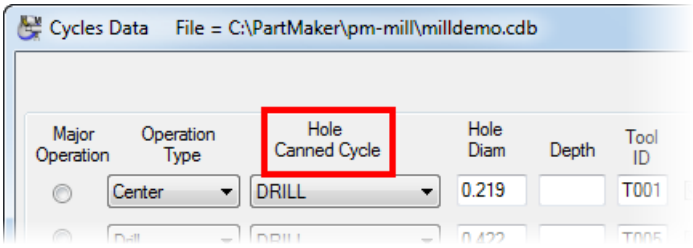
6 When you have completed creating, or editing, the cycle, click **Close** on the **Cycles Data** dialog.

Changes to other dialogs

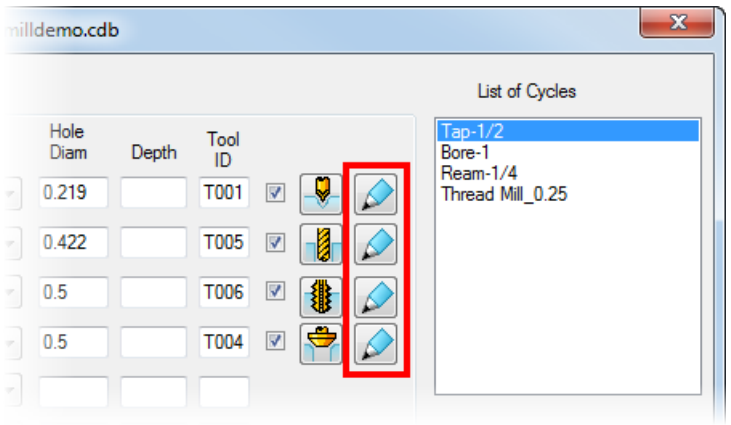
The improvements to hole canned cycles have resulted in a number of minor changes to other dialogs:

In the **Cycles Data** and **Edit Cycles** dialogs:

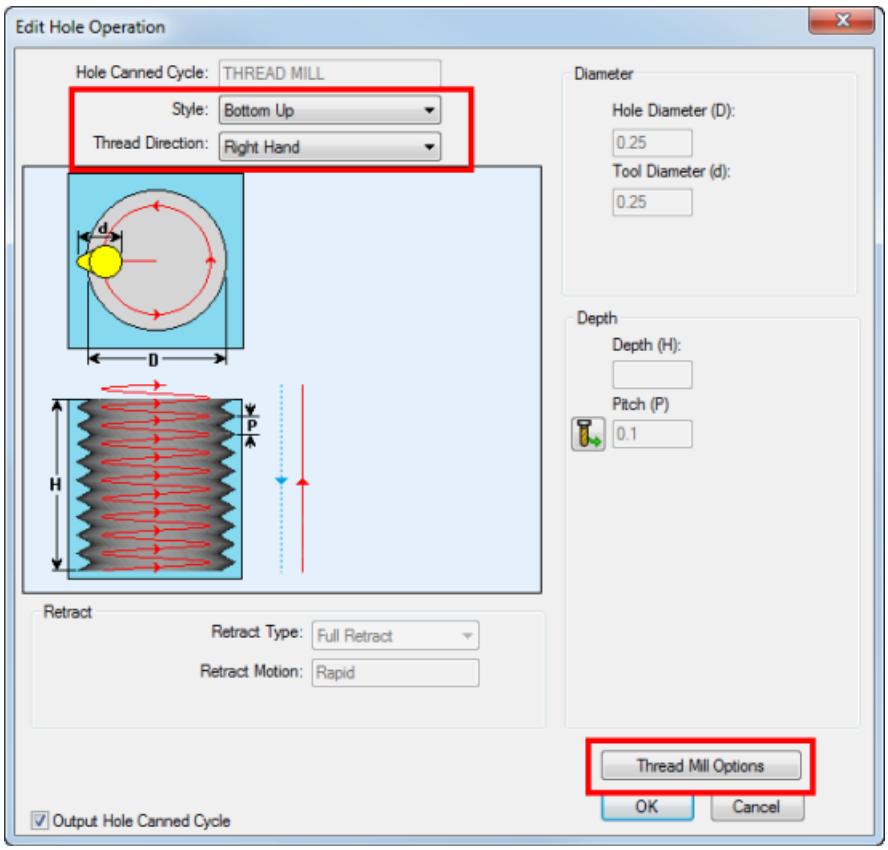
- the **Canned Cycle** column is renamed to **Hole Canned Cycle**.



- beside each operation with a tool assigned, there is a new Edit Hole Operation button.

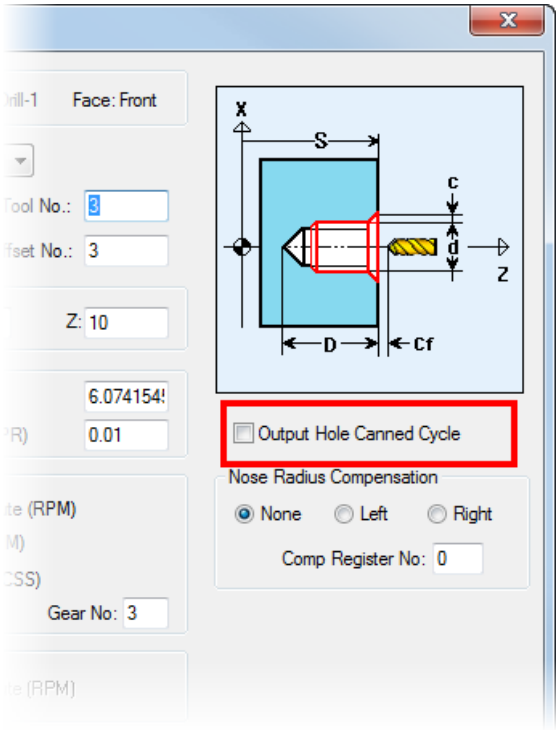


Thread Mill settings are now defined in the **Edit Hole Operation** dialog for a THREAD MILL hole canned cycle (see page 64). Previously, these settings were defined in the **Hole Group Parameters** dialog.

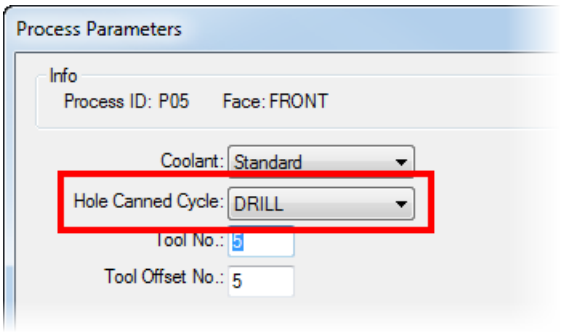


In the **Process Parameters** dialog:

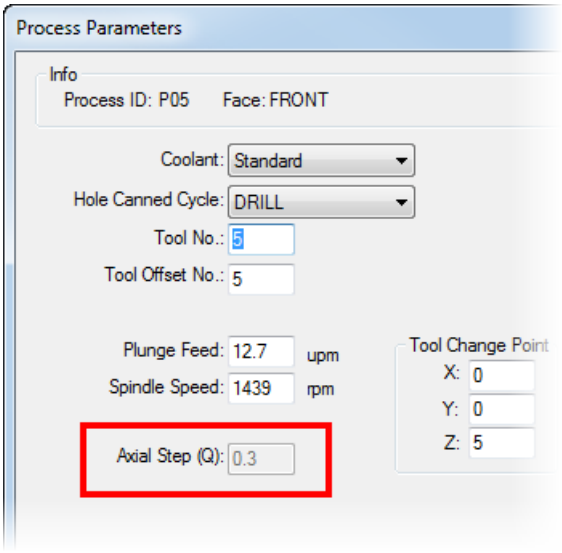
- The **Output Hole Canned Cycle** check box replaces the **Canned Output** check box.



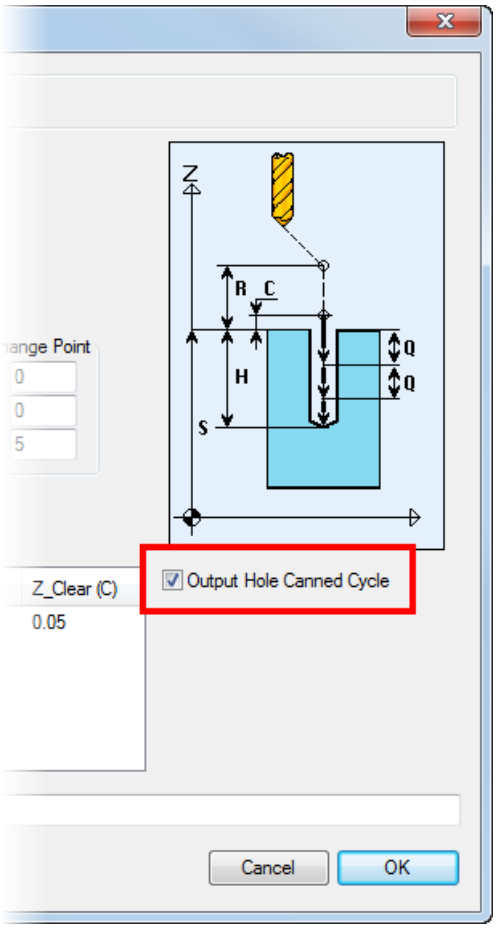
- The **Hole Canned Cycle** drop-down list replaces the **Canned Cycle** drop-down list.



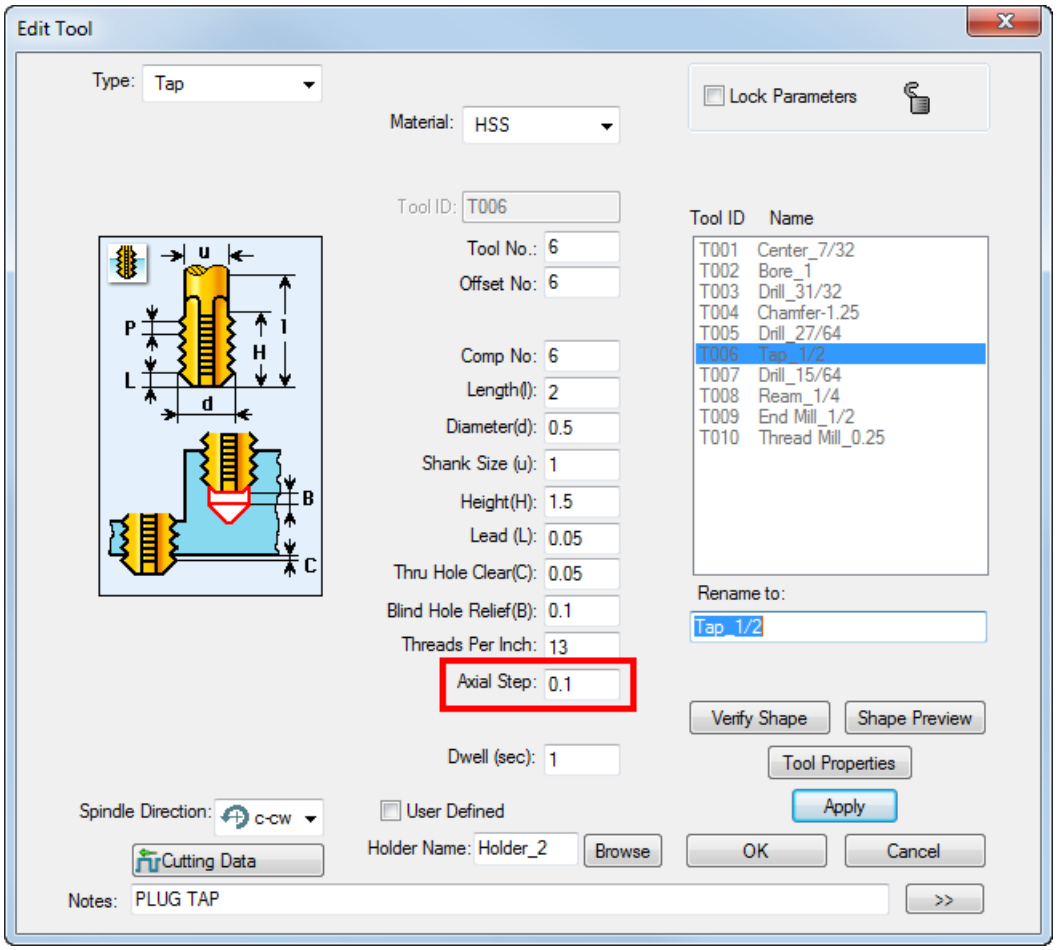
- The **Axial Step (Q)** field is disabled.



- There is a new **Output Hole Canned Cycle** check box.



In the **Tool Data** and **Edit Tool** dialogs, when a Tap tool is selected, an **Axial Step** field is added. This enables you to program an Axial Step for RIGID TAP cycles.

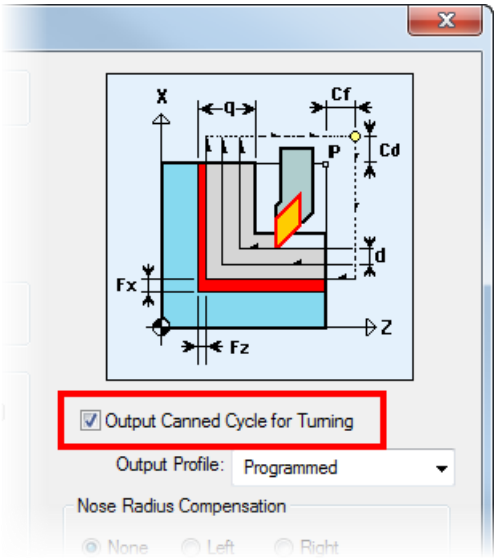


Improvements to canned cycles for Turning

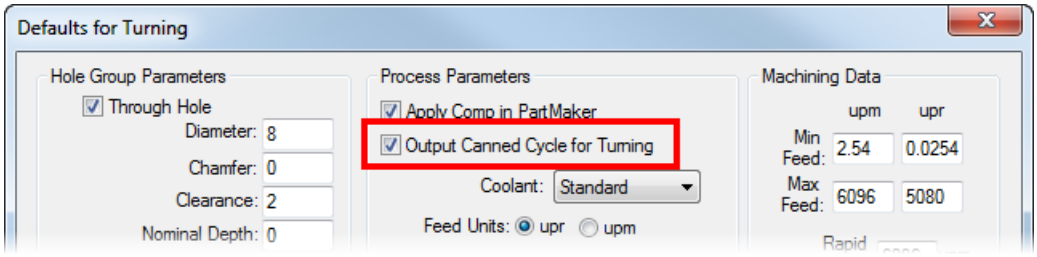
There are some improvements to canned cycles for Turning:

In the **Process Parameters** dialog:

- There is a new **Output Profile** option, where you can select whether to output the profile as programmed or compensated (see page 35).
- The **Output Canned Cycle for Turning** check box replaces the **Canned Output** check box.



In the **Defaults for Turning** dialog, the **Output Canned Cycle for Turning** check box replaces the **Output Canned Cycles** check box and is moved below the **Apply Comp in PartMaker** option in the **Process Parameters** area.



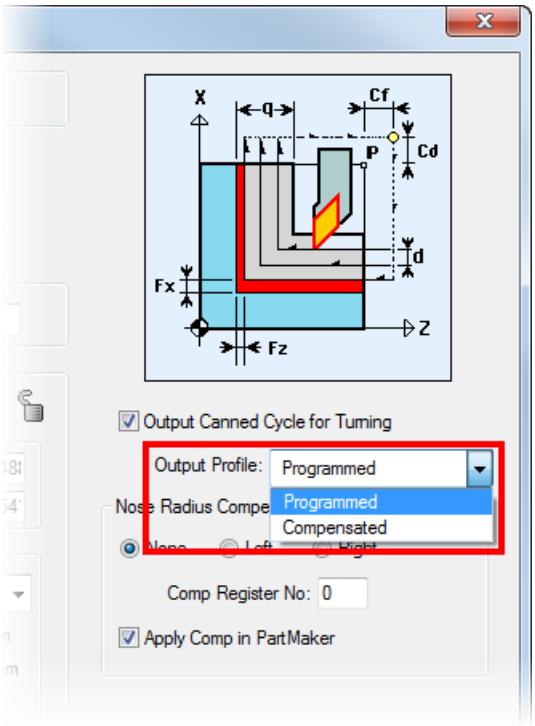
New Output Profile option

In PartMaker/SwissCAM, PartMaker/Turn-Mill, and PartMaker/Turn, there is a new **Output Profile** option on the **Process Parameters** dialog that enables you to select how the Profile is output to the NC Program for a canned cycle for Turning.

The **Output Profile** option is available for the following canned cycles for Turning:

- Turning: Finishing operations
- Turning: Roughing operations
- Contouring: Finishing operations
- Contouring: Roughing operations

For Grooving, Threading, and Thread Whirling processes, and for processes using back turning tools, this option is not applicable.



The following settings are available:

Programmed — Select this option when you enter the tool nose radius in the Machine Tool's control.

Compensated — Select this option when you do not enter the tool nose radius in the Machine Tool's control.

*If **Output Canned Cycle for Turning** is deselected, **Output Profile** is unavailable.*

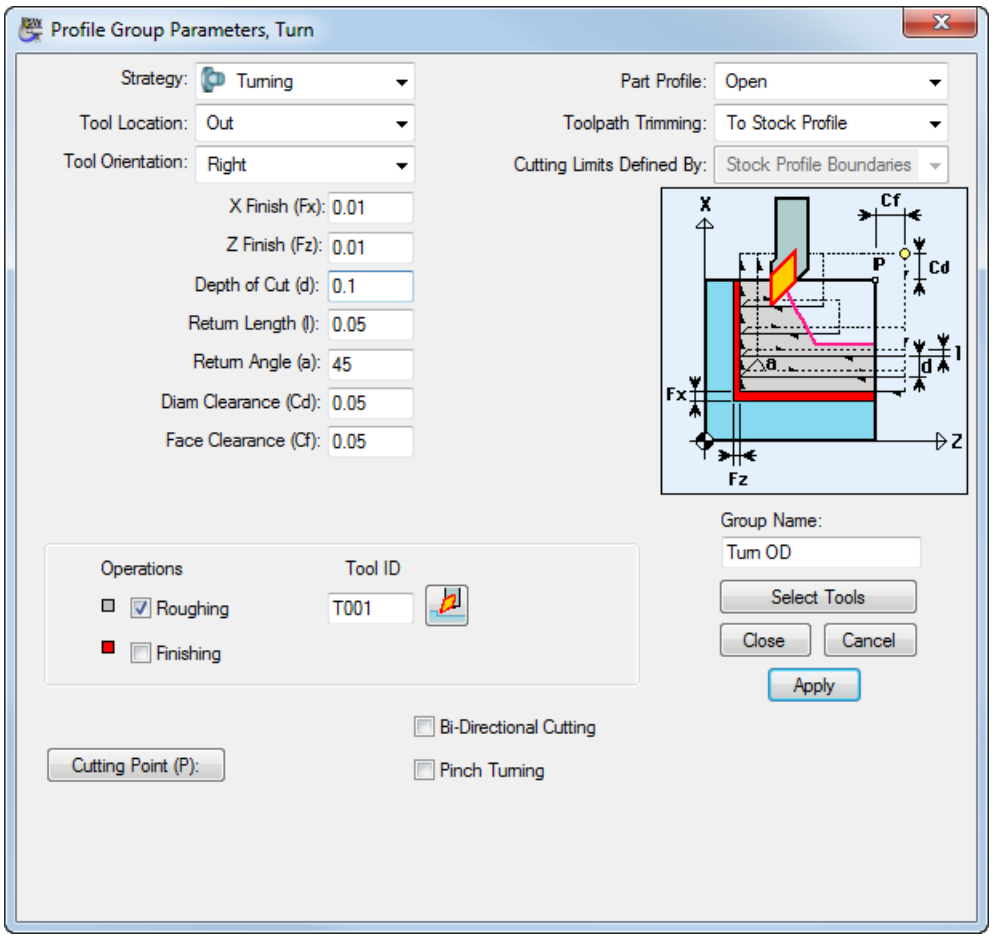
Trimming turning toolpaths using stock profile boundaries

Cutting limits define the area in which all cutting moves occur when machining a Turning toolpath. In previous versions of PartMaker, cutting limits defined only rectangular cutting areas.

In PartMaker/SwissCAM, PartMaker/Turn-Mill, and PartMaker/Turn, you can now define non-rectangular cutting areas (for example, areas of unmachined stock) using stock profile boundaries. Trimming the toolpath to this cutting area reduces air-cutting, which, in turn, reduces machining time.

To use a stock profile to define the cutting limits for a Turning profile group and trim the toolpath to this area:

1. Click in the Face window toolbar to display the **Profile Group Parameters** dialog.



2. Specify settings for the toolpath. To use a stock profile to define cutting limits, you must:

a. Select *Turning* from the **Strategy** drop-down list. Cutting limits are not available for other toolpath strategies.

b. Select the **Roughing** check box in the **Operations** area of the dialog. Stock profiles are applicable only to roughing operations.

c. Select *Open* from the **Profile Shape** drop-down list. Stock profiles must be open on one side.

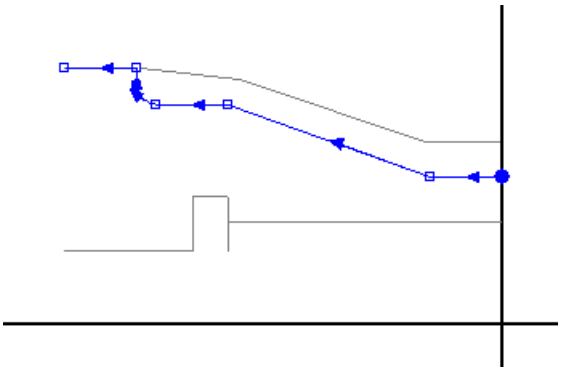
d. Select *To Stock Profile* from the **Toolpath Trimming** drop-down list to specify that you want PartMaker to use the stock profile to set the cutting limits for the toolpath. PartMaker also updates the **Cutting Limits Defined By** drop-down list to select *Stock Profile Boundaries* to reflect your selection.

3. Complete the remaining settings on the **Profile Group Parameters** dialog, then click **Apply** to create the profile group.

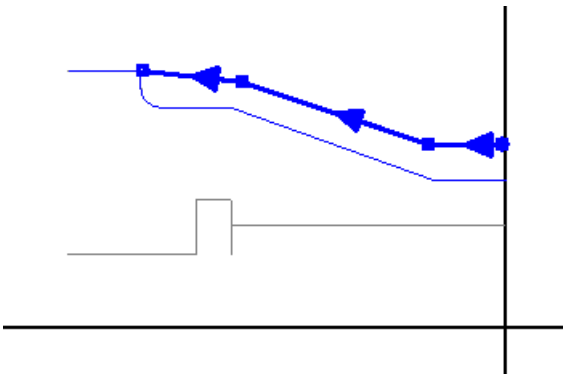
4. Click **Close** to close the dialog.

5. You can now create two profiles for the group: the part profile and the stock profile. By default, PartMaker uses the first profile you create as the part profile, and the second profile as the stock profile. For example:

   - Click to create a profile for the **part profile**, then click and define the profile on a contiguous piece of geometry:

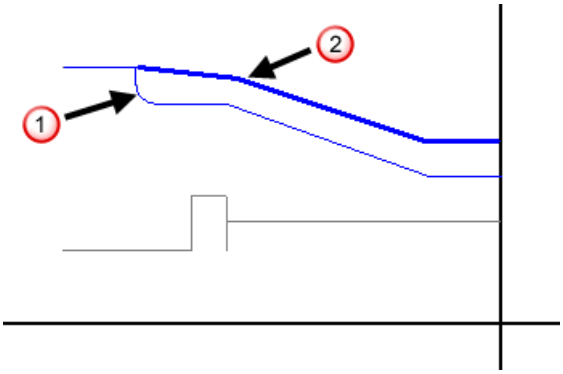


   - Click to create a profile for the **stock profile**, then click and define the profile on a contiguous piece of geometry:



PartMaker displays both profiles in the profile group color and uses a:

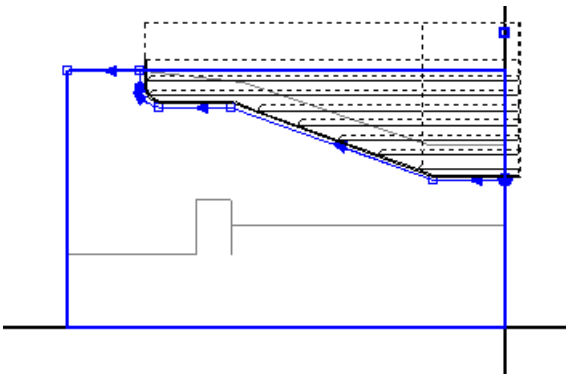
- thin line for the part profile (see 1)
- thicker line for the stock profile (see 2)



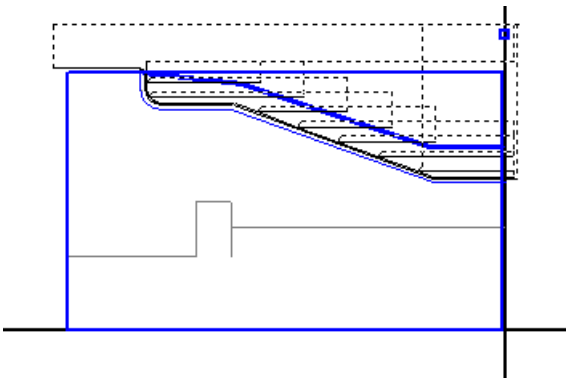
*To change whether a profile represents a part profile or stock profile, double-click a profile line to display the **Profile Properties** dialog and change the profile type.*

6 To verify the toolpath for the profile group, select the profile group in the Job Explorer and select **Part Features > Verify Work Group Tool Path**. PartMaker verifies the toolpath. The following example compares the toolpaths created when a stock profile defines the cutting limits and when a part profile defines the cutting limits:

Cutting limits defined by the part profile



Cutting limits defined by the stock profile



Vertical Rotary table support in PartMaker/Mill

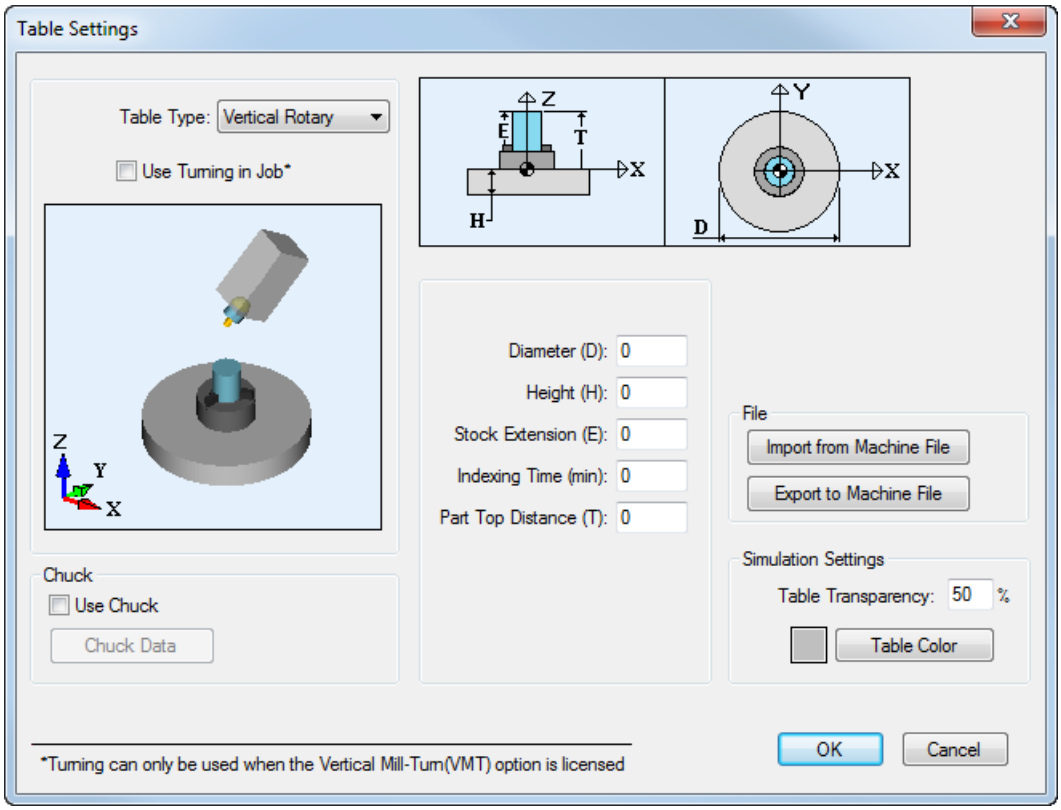
A Vertical Rotary table is used on a 5 axis head-table Vertical Milling Center with a rotary table and a tilting head. These tables are usually used for milling large, heavy, parts since the table does not tilt, and tilt is instead achieved by the tool head.

PartMaker/Mill 2015 includes support for head-table Vertical Milling Centers, and you can select a Vertical Rotary table when defining your table settings.

For the Vertical Rotary table option to be available, you must have PMM (Product Milling Module) licensed.

To select a Vertical Rotary Table:

1. In the **Setup** dialog, click **Settings**. A **Table Settings** dialog for the currently selected table is displayed.
2. In the **Table Type** drop-down list, select *Vertical Rotary*. The Vertical Rotary Table settings are displayed on the dialog.



3. The **Table Settings** dialog controls how the table appears in basic simulation. Enter the settings for your table in the available fields.

4 If you have Vertical Mill-Turn(VMT) licensed, and have selected a Cylinder stock, select **Use Turning in Job** to enable you to use Turning in PartMaker/Mill (see page 42).

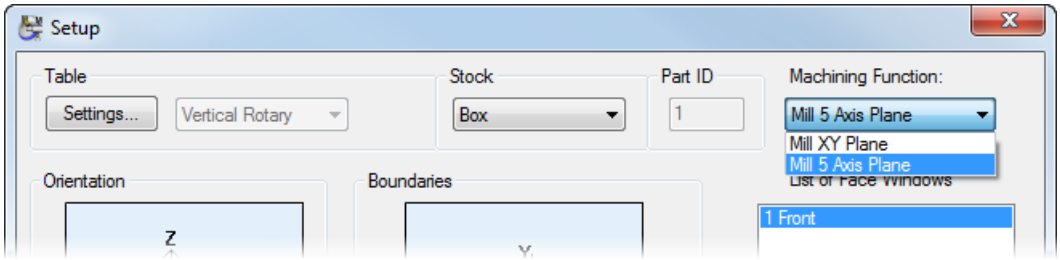
5 Click **OK**. Several dialogs are updated (see page 40).

Changes to other dialogs when a Vertical Rotary table is selected

When you have a Vertical Rotary table selected on the **Table Settings** dialog, there are changes to several other dialogs:

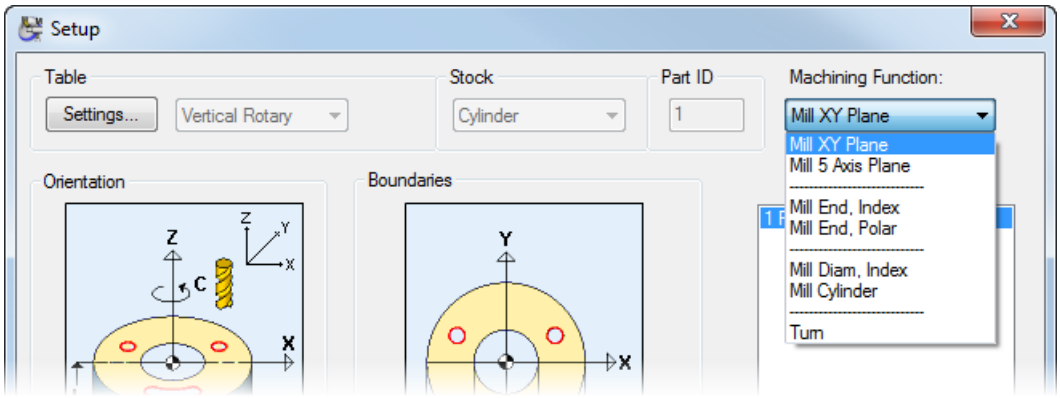
- In the **Setup** dialog:

  With a **Box** stock selected, the following options are available from the **Machining Function** drop-down list:



  - Mill XY Plane
  - Mill 5 Axis Plane.

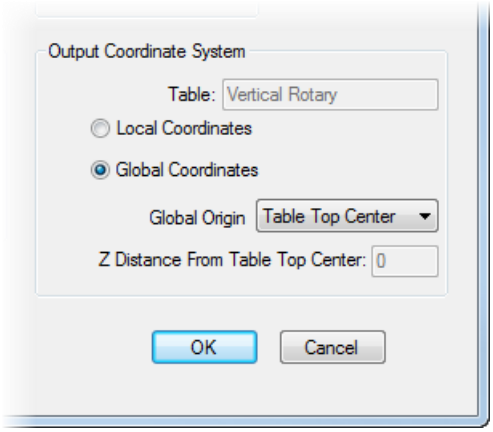
  With a **Cylinder** stock selected, the following options are available from the **Machining Function** drop-down list:



  - Mill XY Plane
  - Mill 5 Axis Plane
  - Mill End, Index
  - Mill End, Polar
  - Mill Diam, Index
  - Mill Cylinder

- Turn (This is available only if **Use Turning in Job** is selected on the **Table Settings** dialog).

- In the **Post Options** dialog, the **Output Coordinate System** area displays options for Vertical Rotary tables.



- There are some changes to the **Machine Data** dialog. These are described in *Updates to the Machine Data dialog* (see page 48).

Using Turning in PartMaker/Mill

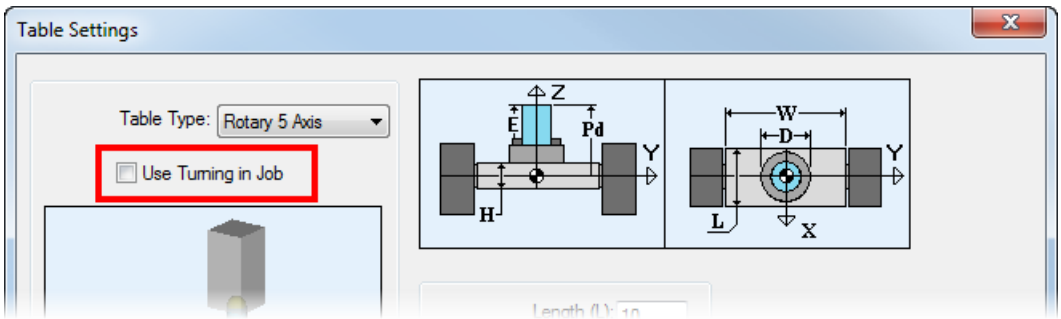
PartMaker/Mill 2015 includes programming, simulation and postprocessing capabilities for Vertical Mill-Turn (VMT) machines.

A VMT machine is a Vertical Milling machine that has the ability to perform a range of turning functions by enabling the machine's vertical rotary table to rotate at the speed of a lathe spindle. VMTs combine the turning capabilities of a Vertical Turret Lathe (VTL) and the machining capabilities of a 5-axis Vertical Machining Center (VMC).

For Vertical Mill-Turn support, you must have the Vertical Mill-Turn(VMT) module licensed.

To use turning in PartMaker/Mill:

1. In the **Setup** dialog, select *Cylinder* from the **Stock** drop-down list.
2. Click **Settings** to display the **Table Settings** dialog.
3. From the **Table Type** drop-down list, select either a *Rotary 5 Axis* or *Vertical Rotary* table. The **Use Turning in Job** option is displayed below the drop-down list.

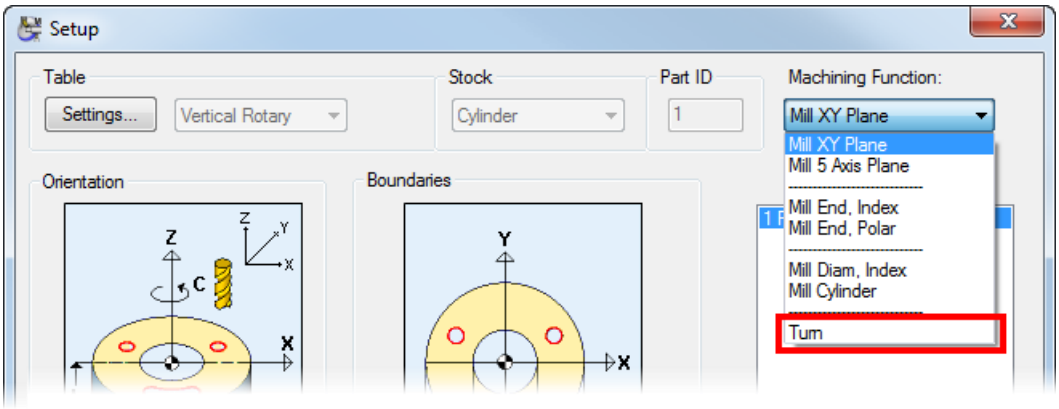


4. Select **Use Turning in Job** and click **OK**. Several dialogs are updated (see page 43).

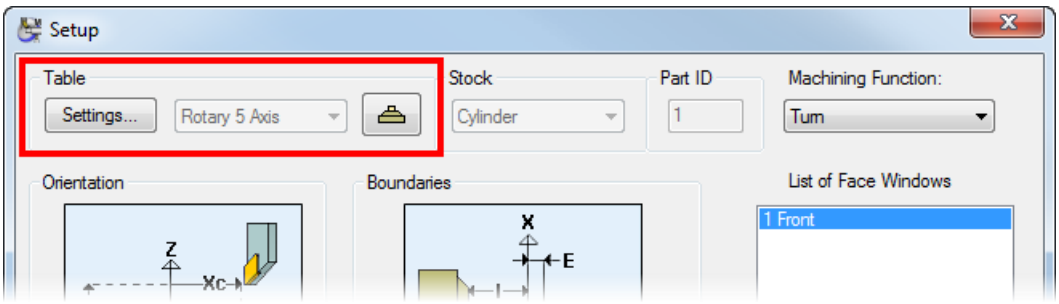
Changes to other dialogs when Turning is enabled in PartMaker/Mill

When you select **Use Turning in Job** to enable Turning in PartMaker/MILL, the following changes are made to the user interface:

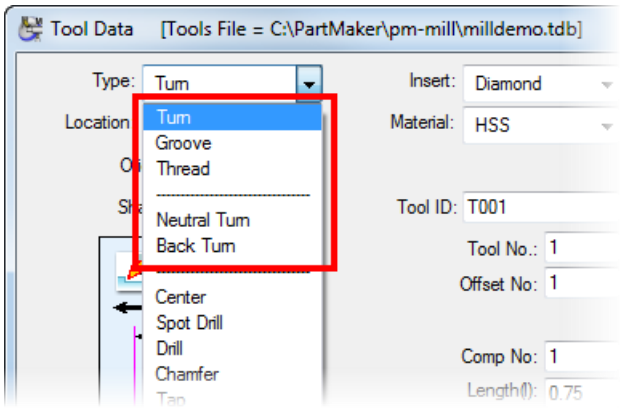
- In the **Setup** dialog:
  - *Turn* is added to the **Machining Functions** drop-down list.



  - If you are using a *Rotary 5 Axis* table and have selected *Turn* from the **Machining Function** drop-down list, the **Turning Machine Position** button is displayed in the **Table** area. Click this button to display the **Turning Machine Position** dialog and select the orientation of the stock (see page 47).

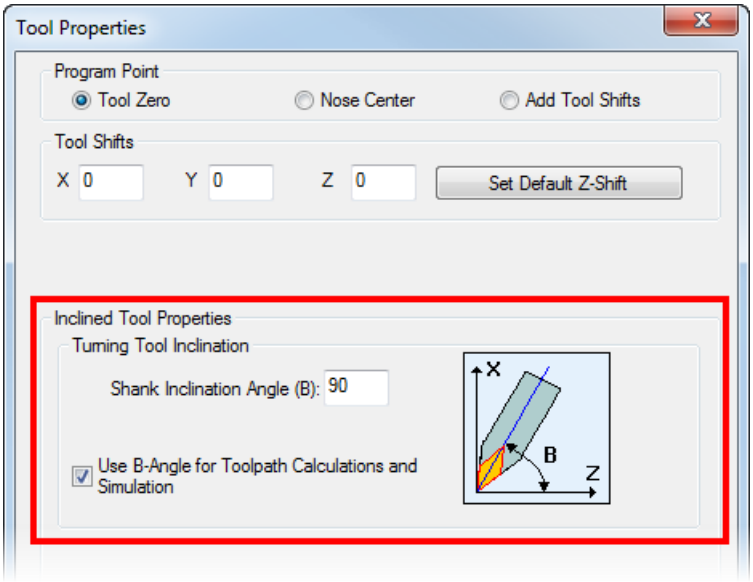


- In the **Tool Data** and **Edit Tool** dialogs, Turning tools are available from the **Type** drop-down list.



When you select a Turning tool, the settings available on the **Tool Data** and **Edit Tool** dialog are the same as when you select a Turning tool in PartMaker/SwissCAM and PartMaker/Turn-Mill, except there is no **Tool Post** field.

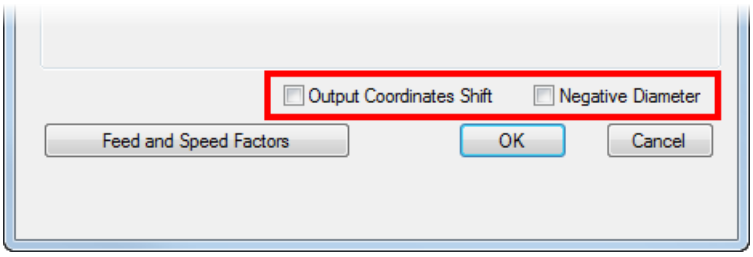
- In the **Tool Properties** dialog, when you have a *Vertical Rotary Table* selected in the **Table Settings** dialog:
  - The **Inclined Tool Properties** area is available.



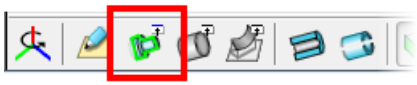
Use B-Angle for Toolpath Calculations and Simulation is unavailable if *Z-Axis* is selected in the **Shank Axis** list on the **Tool Data** or **Edit Tool** dialogs.

There are different Head-Table machine architectures that use inclined tools. PartMaker 2015 only supports inclined turning tools for machines where the tool head remains in the ZX plane while rotating about the Y-axis.

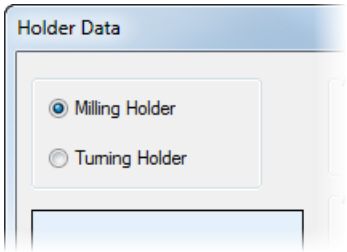
- The **Output Coordinates Shift** and **Negative Diameter** options are available.



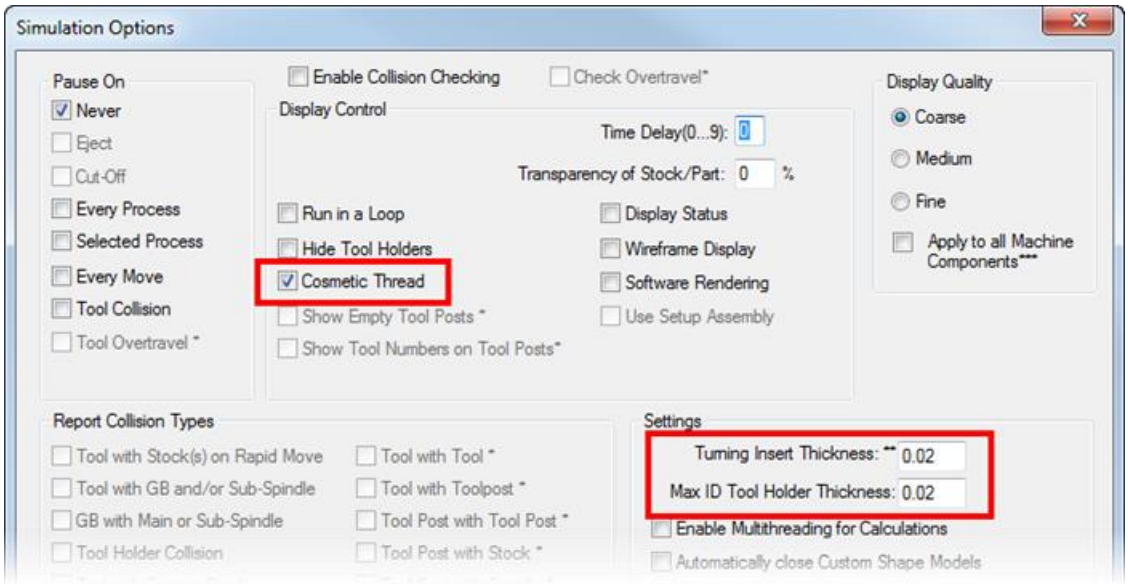
- In the **Solids Window** toolbar, the **Extract Turn Geometry** button is available if you select *Turn* from the **Machining Function** drop-down list in the **Setup** dialog.



- In the **Holder Data** dialog, new options enable you to choose between a Milling holder and a Turning Holder.

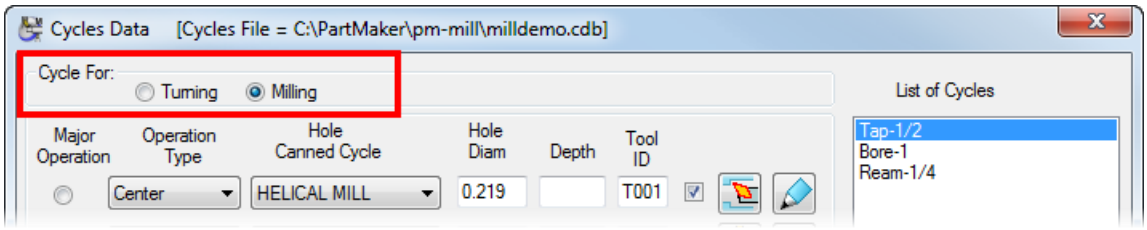


- In the **Simulation Options** dialog, the **Cosmetic Thread**, **Turning Insert Thickness**, and **Max ID Tool Holder Thickness** settings are available. These settings were previously available only in PartMaker/SwissCAM, PartMaker/Turn-Mill, and PartMaker/Turn.

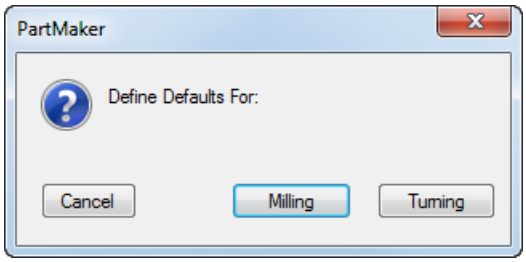


For more information on these options, see the Reference Help.

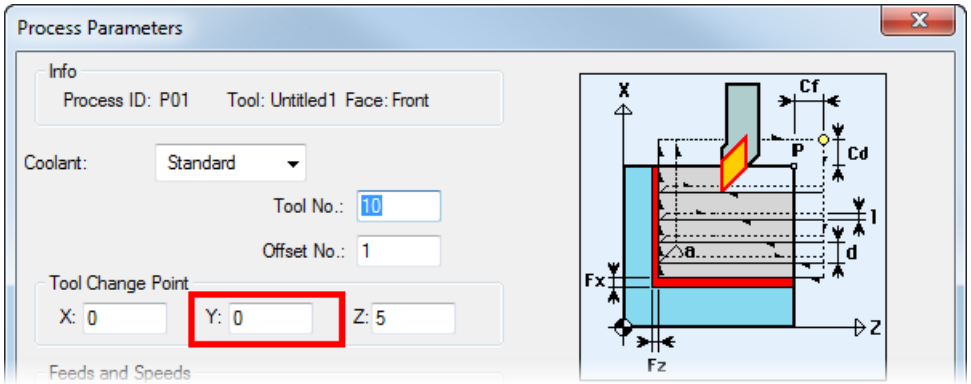
- In the **Cycles Data** dialog, the **Cycle For** area is added, which enables you to select whether the cycle is for Turning or Milling.



- When you select **Job Optimizer > Defaults**, the **Define Defaults For** dialog is displayed where you can choose to display the **Defaults** dialog for Milling or Turning.



- In the **Process Parameters** dialog for Turning processes, the **Tool Change Point Y** parameter is available. The values for **Tool Change Point X**, **Y**, and **Z** are taken from the **Defaults for Milling** dialog.

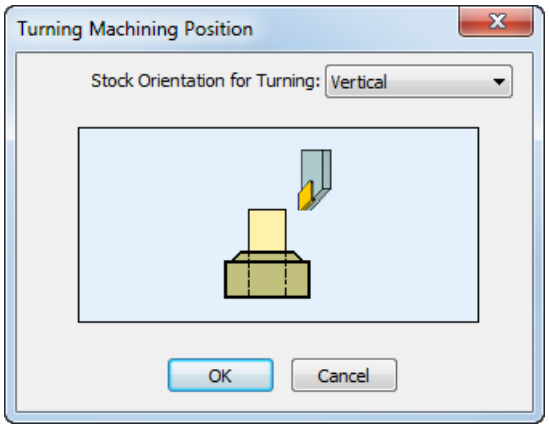


In PartMaker/SwissCAM, PartMaker/Turn-Mill, and PartMaker/Turn, the **Y** value is set to **0** and the field is unavailable.

- There are some changes to the **Machine Data** dialog. These are described in *Updates to the Machine Data dialog* (see page 48).

Changing the orientation of the stock

In PartMaker/Mill, you can use the **Turning Machining Position** dialog to change the orientation of the stock for turning operations when you are using a Rotary 5 Axis table.

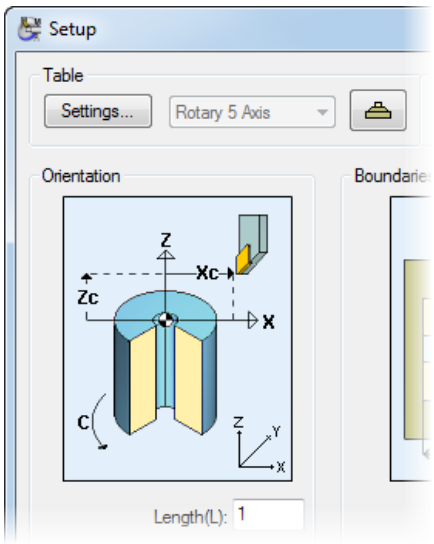


To display this dialog, click the button in the **Table** area of the **Setup** dialog. This button is only available when you have the following:

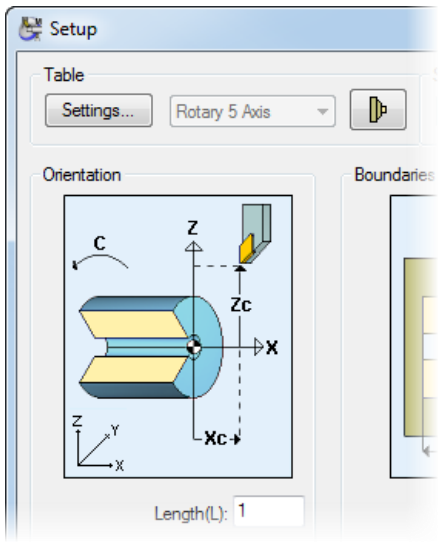
- *Rotary 5 Axis* selected from the **Table** drop-down list in the **Table Settings** dialog.
- **Use Turning in Job** selected in the **Table Settings** dialog.
- *Turn* selected in the **Machining Functions** drop-down list in the **Setup** dialog.

If you change the orientation from vertical to horizontal, the graphic in the **Orientation** area of the **Setup** dialog is updated, and the button changes from to .

Vertical stock orientation



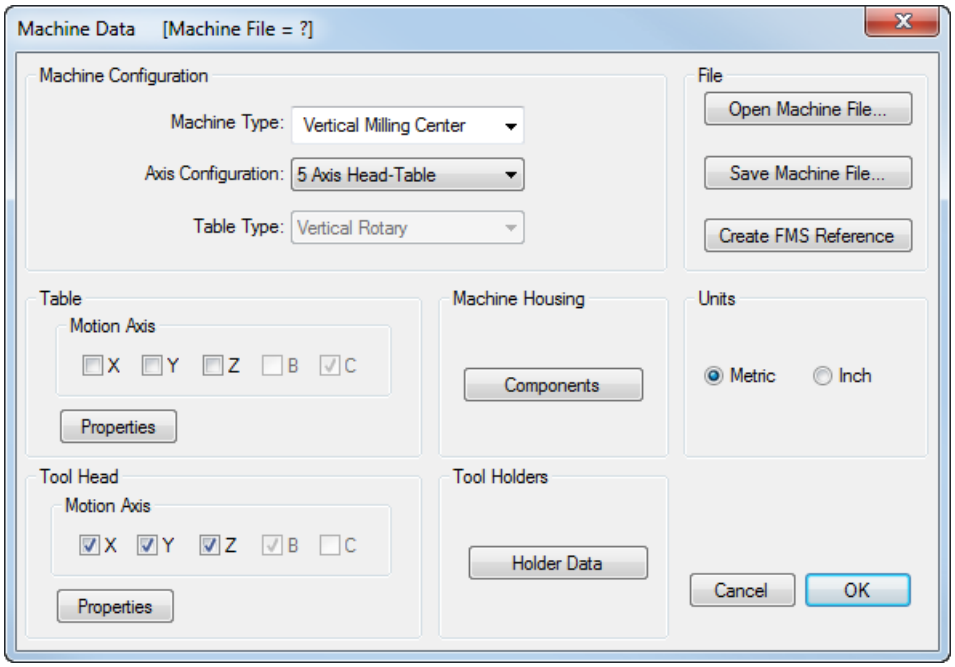
Horizontal stock orientation



If you select a *Horizontal* stock rotation, the **New Hole Group** button on the CAM window toolbar is unavailable.

Updates to the Machine Data dialog in PartMaker/Mill

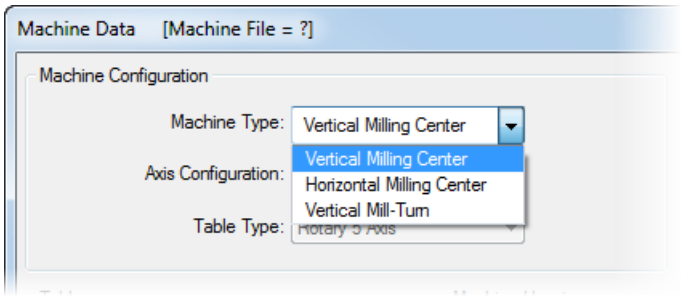
The **Machine Data** dialog enables you to specify information about how your machine is represented during Full Machine Simulation.



In PartMaker/Mill 2015, there are several changes to the **Machine Data** dialog as result of PartMaker/Mill now supporting Vertical Rotary tables and Vertical Mill-Turn machines, and to bring terminology in line with industry standards.

- The **Milling Machine Configuration** area is renamed to **Machine Configuration**.
- In the **Machine Configuration** area:

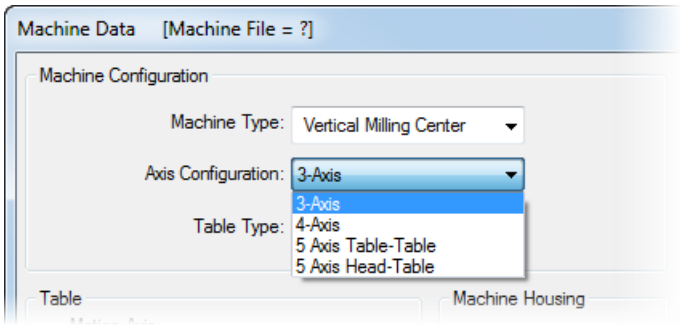
  The **Type** drop-down list is renamed to **Machine Type**, and the available options are updated:



  - *Vertical Machine* is renamed to *Vertical Milling Center*.
  - *Horizontal Machine* is renamed to *Horizontal Milling Center*.

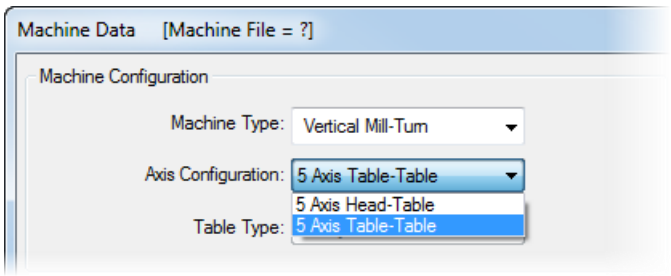
- *Vertical Mill-Turn* is added. This option is available only if you are licensed for the **Vertical Mill-Turn (VMT)** module (see page 42).

When you select *Vertical Milling Center*, the options available in the **Axis Configuration** drop-down list are updated:



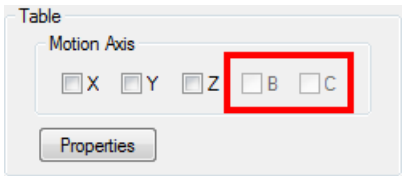
- *3-Axis Machine* is renamed to *3-Axis*.
- *4-Axis Machine* is renamed to *4-Axis*.
- *5-Axis Machine* is replaced by *5 Axis Head-Table* and *5 Axis Table-Table*.

When you select *Vertical Mill-Turn*, only *5 Axis Table-Table* and *5 Axis Head-Table* are available from the **Axis Configuration** drop-down list.

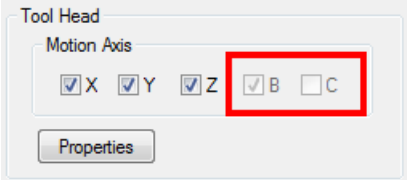


The **Table Configuration** drop-down list is renamed to **Table Type.**

- In the **Table** area, there are two additional motion axes displayed. When you have a Vertical Rotary table selected in the Table Type drop-down list, click Properties to display the new **Vertical Rotary Table Properties** dialog (see page 51).



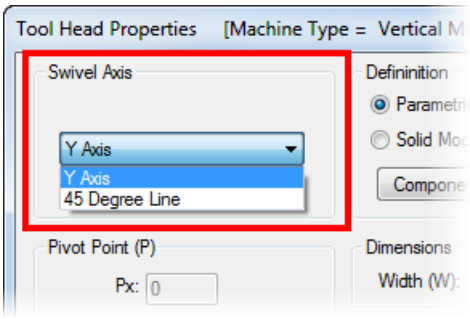
- The **Tool Head** area replaces the **Tool Spindle** area, and there are two additional motion axes displayed. There are also updates to the **Tool Head Properties** dialog (see page 50).



Tool Head Properties dialog

In the **Tool Head Properties** dialog:

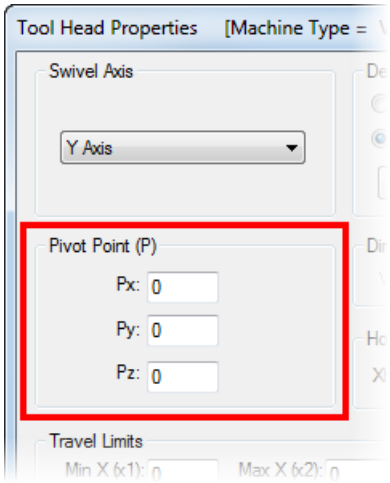
- There is a new **Swivel Axis** area. This is the 4th rotational axis in a 5-axis Head-Table machine, and is the axis about which the tool head rotates.



  - **Y Axis** — Select this option to indicate that the rotary axis of the tool head is along the Y-axis of the machine. When selected, PartMaker displays horizontal tools along the X-axis during Basic and Full Machine Simulation.
  - **45 Degree Line** — Select this option to indicate that the rotary axis of the tool head is along a 45 degree line to the machine Y-axis lying in a ZY plane. When selected, PartMaker displays horizontal tools along the Y-axis during Full Simulation.

PartMaker programs all horizontal tools on a Vertical Rotary table as X tools. Although, PartMaker displays the horizontal tool along the Y-axis when 45 Degree Line is selected, it is still programmed as an X tool.

- There is a new **Pivot Point (P)** area. The Pivot Point is a point on the tool head through which the tool head rotary axis passes. Specify the **X**, **Y**, and **Z** distance from the tool head edge to the center of rotation.



Vertical Rotary Table Properties dialog

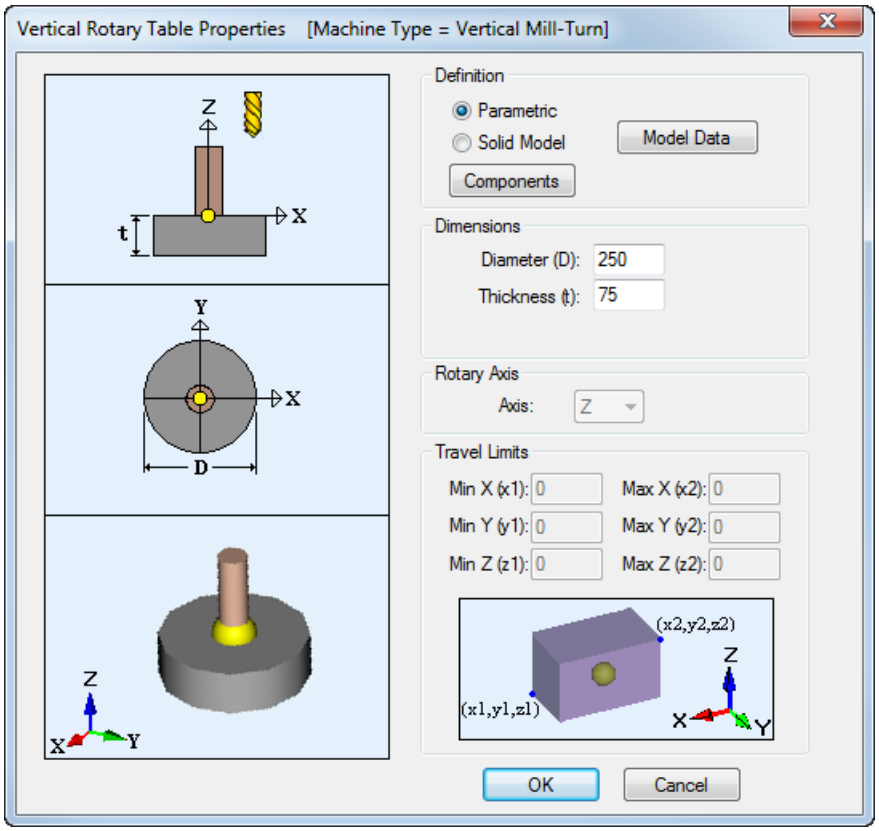
Use the **Vertical Rotary Table Properties** dialog to specify details about the table and how it is displayed in Full Machine Simulation.

To specify how the Vertical Rotary Table is displayed in Full Machine Simulation:

1. Select **ToolMinder > Machine** to display the **Machine Data** dialog.
2. In the **Machine Configuration** area, select either *Vertical Milling Center* or *Vertical Mill-Turn* from the **Machine Type** drop-down list.

   The Vertical Mill-Turn option is available only if you have Vertical Mill-Turn(VMT) licensed.
3. Select *5 Axis Head-Table* from the **Axis Configuration** drop-down list. *Vertical Rotary* is selected automatically in the **Table Type** drop-down list.

4 In the **Table** area, click **Properties** to display the **Vertical Rotary Table Properties** dialog.



Support for compound rotary holders

Compound rotary holders are holders that have a fixed part (base model) mounted to the tool post, and a rotating part (rotary model).

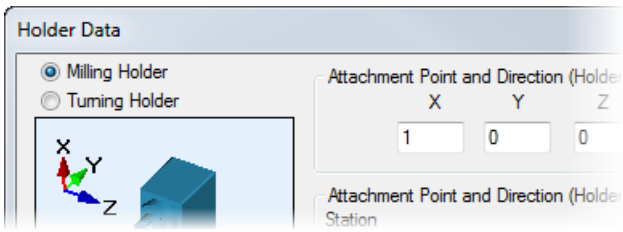
In PartMaker/SwissCAM and PartMaker/Turn-Mill you can now define the base model and rotary model of a compound rotary holder separately. Previously compound rotary holder models were created as one piece. They were created at common angles so you would have to choose a holder with an angle that was closest to your tool angle for a given process.

As a result of this improvement, there are changes to the **Holder Data** dialog, and there is a new **Set Holder Models** dialog.

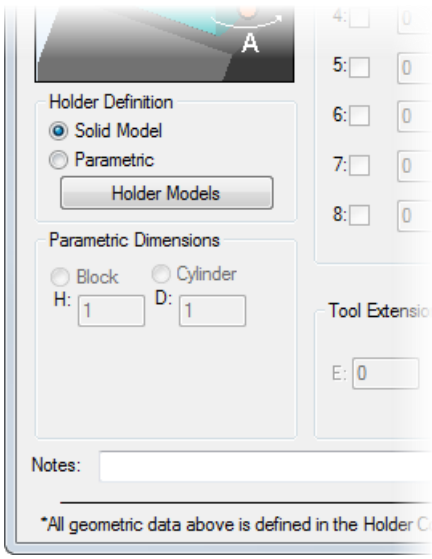
Holder Data dialog

In the **Holder Data** dialog:

- There are two new radio buttons so you can select whether you have a **Milling Holder** or a **Turning Holder**. Only Milling holders can be compound rotary holders.



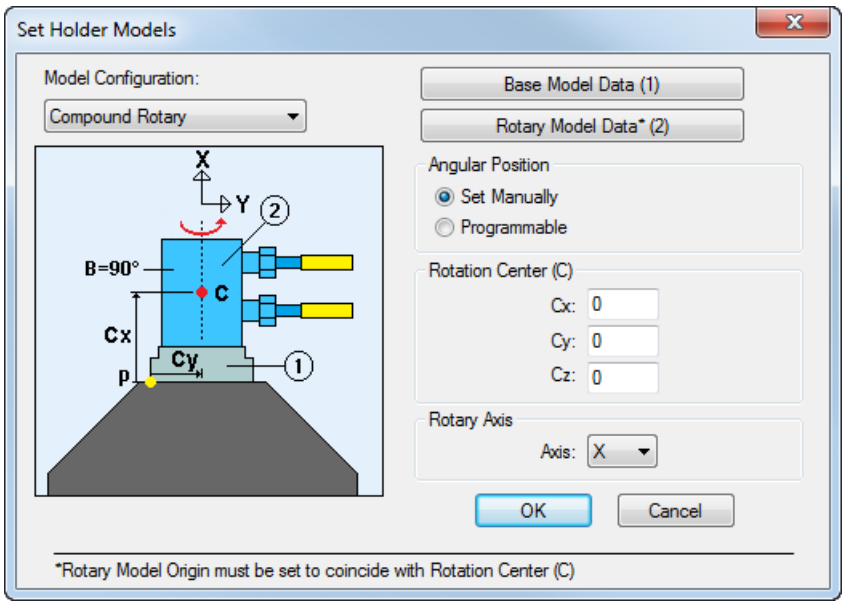
- The **Model Data** button is removed and replaced with the **Holder Models** button. Click this button to display the **Set Holder Models** dialog.



*If you select **Parametric**, the **Holder Models** button is unavailable because the dimensions of the holder are defined in the **Parametric Dimensions** area.*

Set Holder Models dialog

Use the **Set Holder Models** dialog to control how the holder is displayed during Full Machine Simulation.



To display this dialog, click the **Holder Models** button on the **Holder Data** dialog.

The following settings are available:

Model Configuration — Select what type of holder you have. If you selected **Turning Holder** on the **Holder Data** dialog, this drop-down list is unavailable.

Standard Fixed — Select this option if you have a standard fixed holder. Only the **Model Data** button is available.

Compound Rotary — Select this option if you have a compound holder. This option is not available for Turning holders.

Model Data — Click to display the **Model Data** dialog.

Base Model Data (1) — Click to display the **Model Data** dialog and define the properties of the compound holder's base model.

Rotary Model Data (2) — Click to display the **Model Data** dialog and define the properties of the compound holder's rotary model.

Angular Position

Set Manually — Select this option if your compound holder is rotated manually by the machine operator prior to running the NC program.

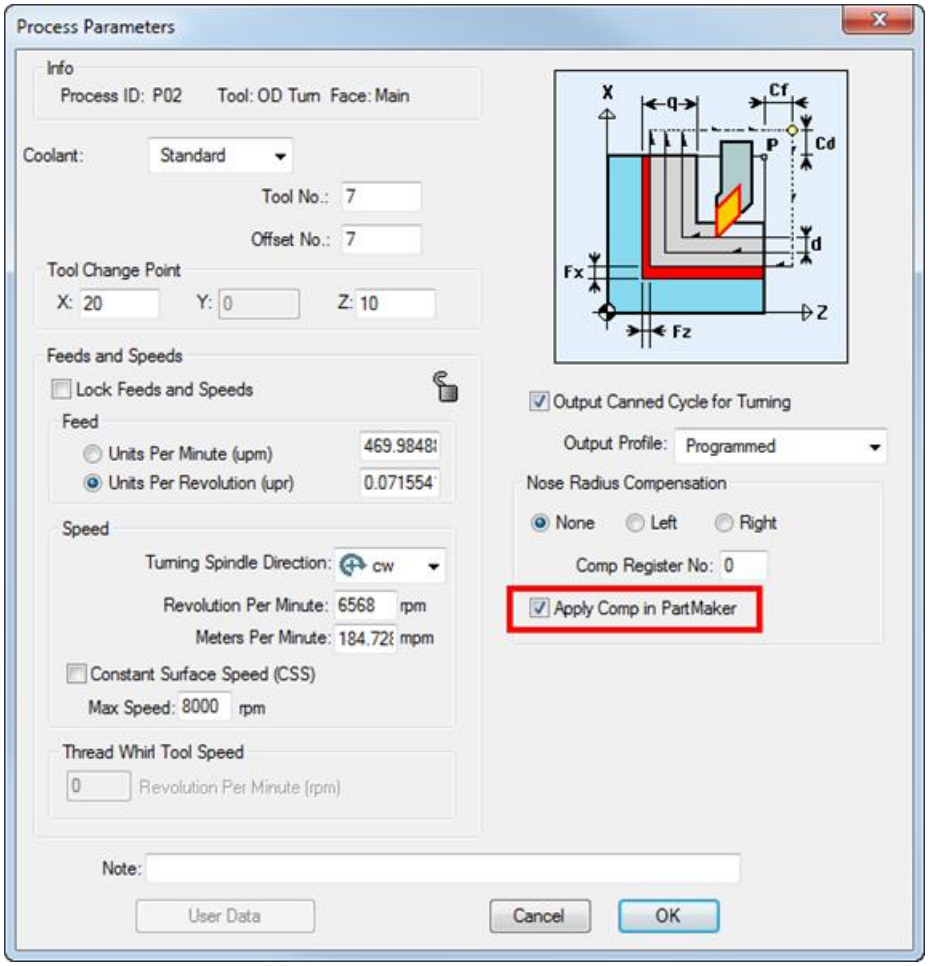
Programmable — Select this option if the angle of your holder is programmed in the NC code.

Rotation Center (C) — Enter the position of the Rotation Center. The rotation center is measured from the model origin of the Base Model to the model origin of the Rotary Model.

Rotary Axis — Select the axis in which the Rotary Model rotates about to get to a specific angular position.

Applying nose radius compensation to each process

The **Apply Comp in PartMaker** option controls whether PartMaker compensates the toolpath output to the NC Program based on the tool nose radius, so that you do not need to enter the tool nose radius in the control. In PartMaker 2015, you can choose to apply cutter compensation per process for turning, rather than using the **Defaults for Turning** dialog as a global setting for every turning process on the Process Table.

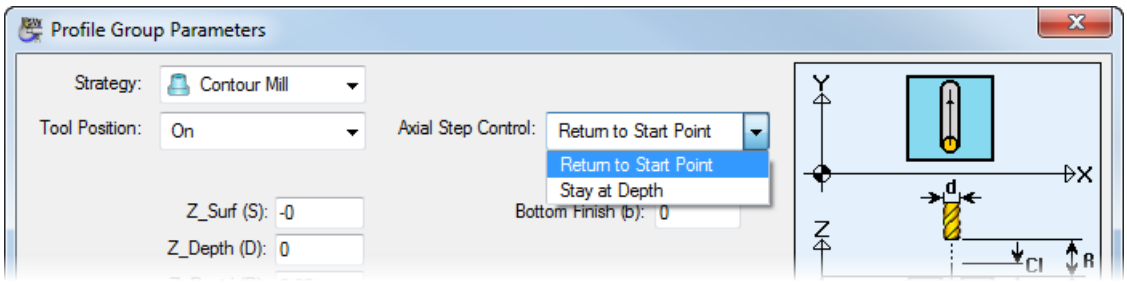


Apply Comp in PartMaker can be programmed for the following operations:

- Turning: Finishing operations
- Contouring: Roughing operations
- Contouring: Finishing operations
- Grooving: Finishing operations

Axial step control

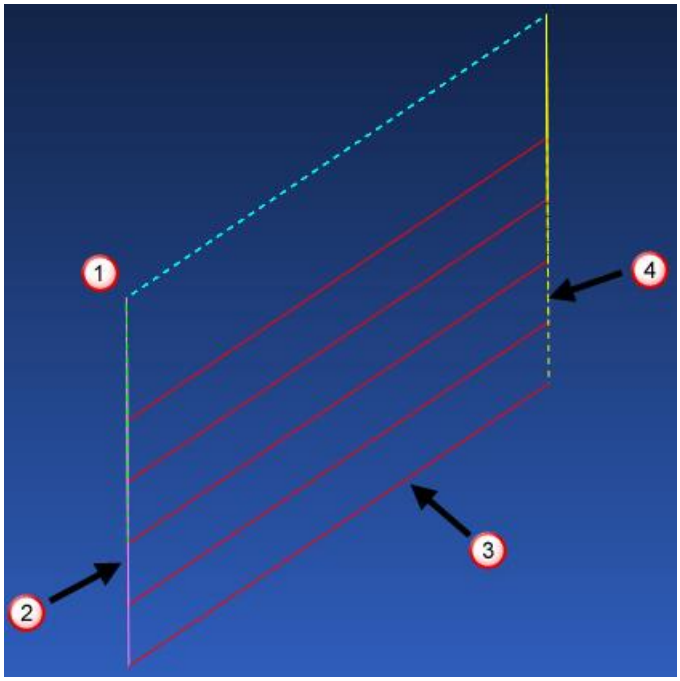
In PartMaker/SwissCAM, PartMaker/Turn-Mill, and PartMaker/Mill, there is a new **Axial Step Control** drop-down list that enables you to control whether or not the tool retracts back to the start position between Z moves in a **Contour Mill** strategy with a **Tool Position** of *On*. This means you can avoid unnecessary rapid moves back to the start of the profile between passes.



The **Axial Step Control** option is available only when you have a **Mill XY Plane**, **Mill ZY Plane**, **Mill 5 Axis Plane**, **Mill End Polar**, or **Mill 4 Axis Plane** machining function selected.

The **Axial Step Control** drop-down list contains the following options:

Return to Start Point — If selected, the tool retracts to the start position (see ①) before the next Z step down.

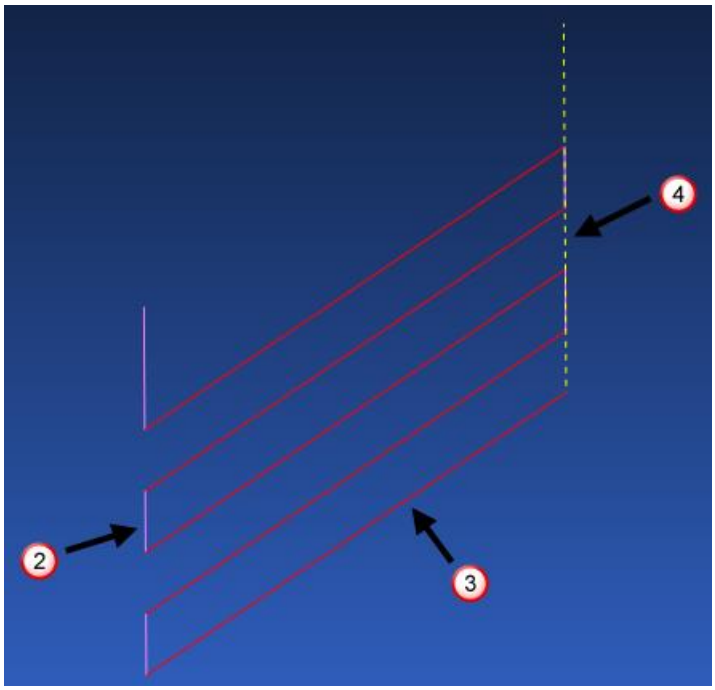


② Plunge moves (purple).

③ Cutting moves (red).

④ Retract moves (yellow).

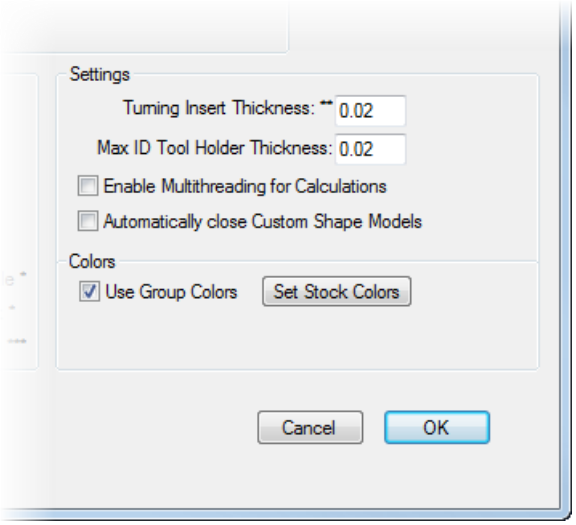
Stay at Depth — If selected, the tool stays at depth without retracting until the final depth is reached.



2. Plunge moves (purple).
3. Cutting moves (red).
4. Retract moves (yellow).

Simulation Options dialog

There are some changes to the **Simulation Options** dialog.



There are two new options in the **Settings** area of the dialog:

Enable Multithreading for Calculations — Select this option to increase simulation speed (see page 60).

Automatically Close Custom Shape Models — Select this option to fix invalid custom solid models (see page 59).

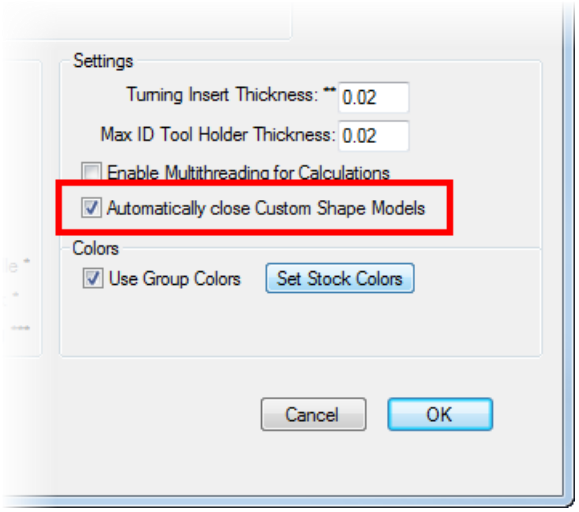
The **Colors and Textures** area is renamed to **Colors**.

Fixing invalid solid models

In PartMaker 2015, you can fix custom shape solid models that have missing or self intersecting faces so that these solid models can be used in simulations.

To fix invalid custom shapes:

1. Select **Simulation > Simulation Options** from the menu bar.
2. In the **Settings** area, select **Automatically close Custom Shape Models**.

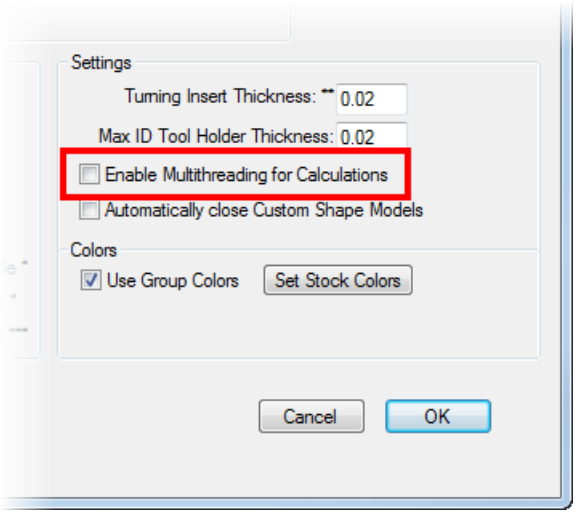


3. Click **OK** to close the dialog.

Increasing simulation speed

In PartMaker 2015, you can take advantage of your computer's multithreading capabilities to increase the speed of your simulation.

To use multithreading when running a simulation, select **Enable Multithreading for Calculations** in the **Settings** area of the **Simulation Options** dialog. This option is deselected by default.



User interface changes

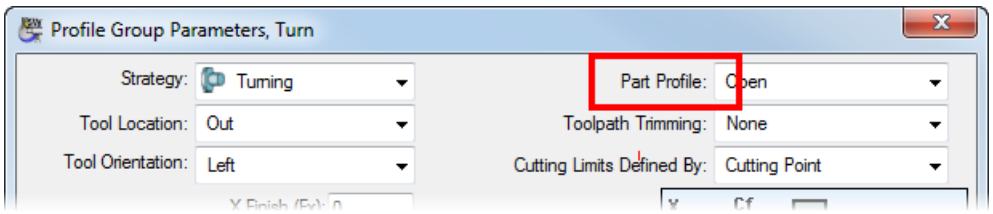
PartMaker 2015 includes the following improvements to the PartMaker user interface:

- **Profile Group Parameters, Turn dialog** (see page 62) — There are several changes to this dialog when you have a *Turning* strategy selected.
- **Define Custom Shape dialog** (see page 63) — There are changes to the terminology and layout of the **Define Custom Shape** dialog.
- **Thread mill settings** (see page 64) — The thread mill settings are now located on the **Edit Hole Operation** dialog.
- **5-Axis Rotary Table Properties** (see page 65) — The Primary and Secondary Rotary axes are renamed.
- **Viewing tool parameters** (see page 66) — Tool parameters of unlicensed tools can be viewed but not edited in the **Tool Data** and **Edit Tool** dialogs.
- **Using the Manage Tools Files dialog** (see page 69) — There are several changes to the Manage Tools Files dialog.
- In PartMaker/Mill, the following dialogs are updated:
  - **Machine Data** dialog (see page 48).
  - **Setup** dialog (see page 67).
  - **Table Settings** dialog (see page 65).
- The following options retain the same function, but are renamed:
  - In the **Tool Data** and **Edit Tool** dialogs, the **User Defined Shape** option replaces the **User Defined** option.
  - The **Tool Head Properties** dialog replaces the **Tool Spindle Properties** dialog.
  - In the **Holder Data** dialog, *Tool Head* replaces *Tool Spindle*.

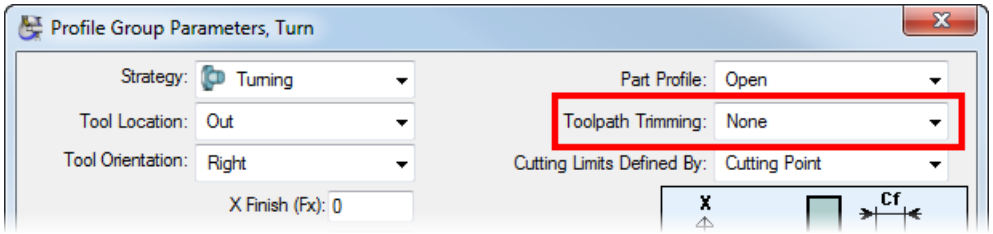
Profile Group Parameters, Turn dialog

When you have a *Turning* strategy selected in the **Profile Group Parameters, Turn** dialog, there are the following changes to the dialog in PartMaker 2015:

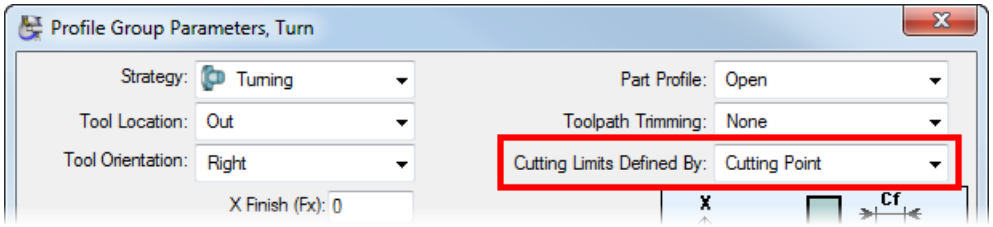
- The **Profile Shape** option is renamed to **Part Profile**.



- A new **Toolpath Trimming** drop-down list is available, which enables you to trim the toolpath to a cutting area defined by a stock profile (see page 36).



- The **Cutting Limits Defined By** area of the dialog has been removed and replaced by a drop-down list:



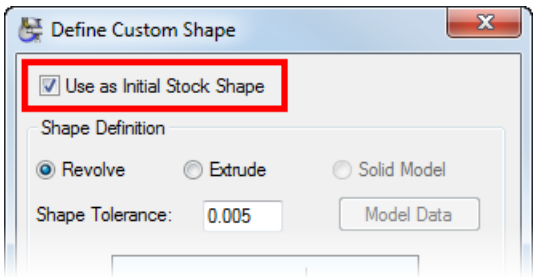
- The options available for defining cutting limits are now:
  - *Cutting Point* (same as PartMaker 2014).
  - *Part Profile Boundaries* (known as *Profile Boundaries* in PartMaker 2014).
  - *Stock Profile Boundaries* (new in PartMaker 2015).

    If you select *None* from the **Toolpath Trimming** drop-down list, the *Stock Profile Boundaries* option is unavailable. If you select *To Stock Profile* from the **Toolpath Trimming** drop-down list, *Stock Profile Boundaries* is selected automatically and the other options are unavailable.

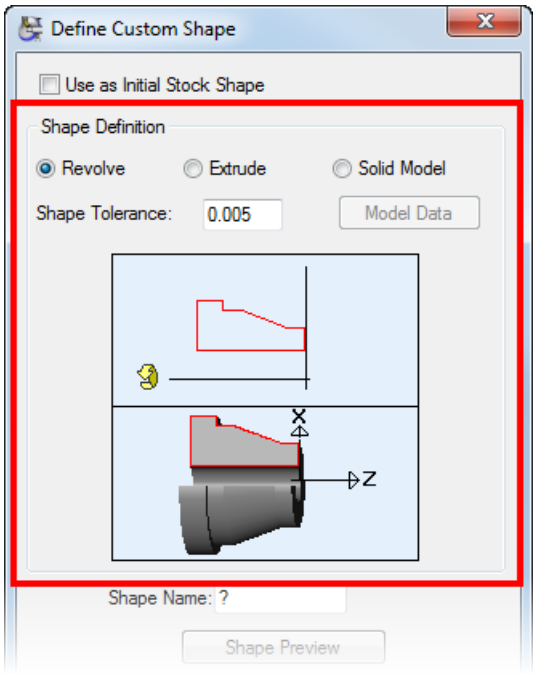
Define Custom Shape dialog

The **Define Custom Shape** dialog includes the following changes:

- The **Use as Stock Shape** option is now called **Use as Initial Stock Shape.** This change emphasizes that PartMaker uses the custom shape as the initial stock shape only during simulation.



- The **Define Custom Shape** dialog now groups the settings for defining a custom shape together in a **Shape Definition** area:



Thread Mill settings

In PartMaker 2015, the thread mill settings for holes are located on the **Edit Hole Operation** dialog when you have a THREAD MILL hole canned cycle selected (see page 19). Previously these settings were located on the **Profile Group Parameters** dialog when you selected a Thread Mill strategy.

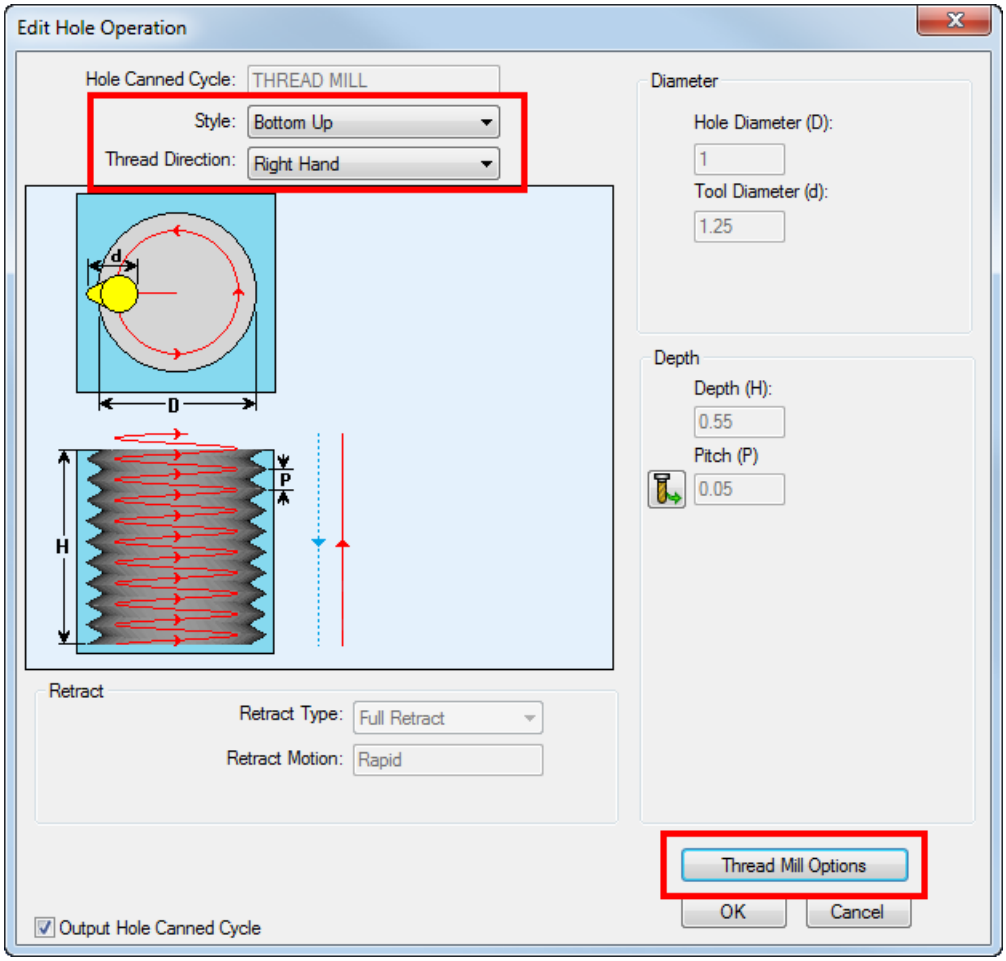
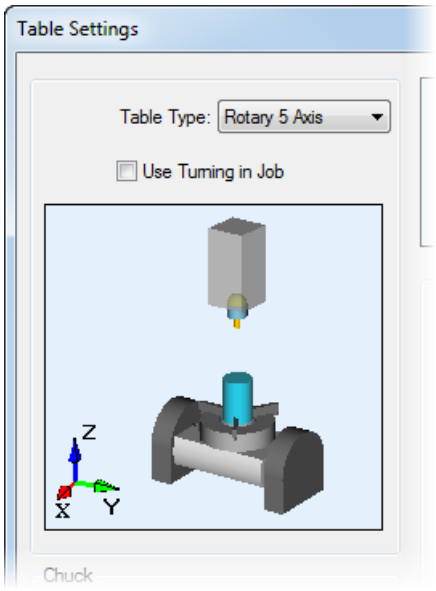


Table Settings dialog

In PartMaker/Mill, you can now select your Table type from the **Table Settings** dialog, rather than on the **Setup** dialog. Each **Table Settings** dialog now has a **Table** area that includes a drop-down list of Table types and a picture illustrating the selected table. When you select a table from the drop-down list, the properties on the dialog are updated to reflect the selected table.



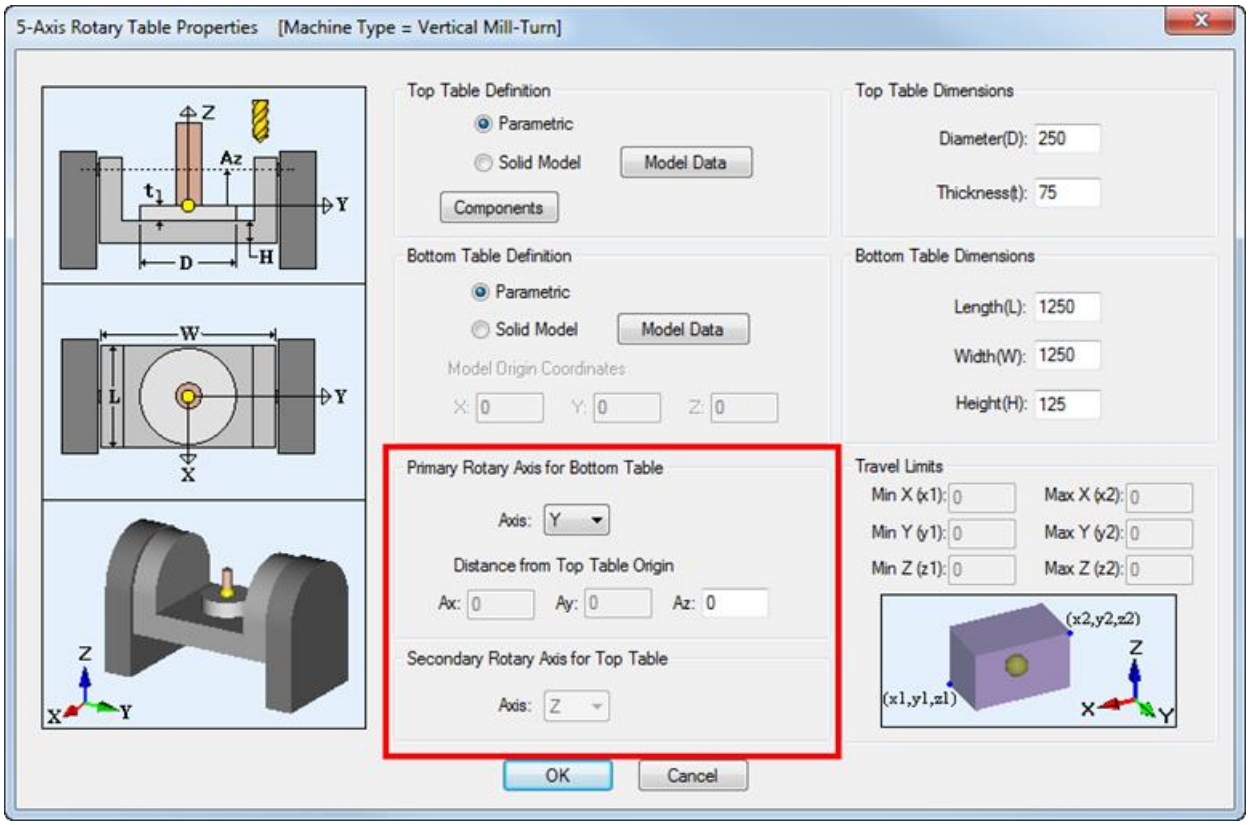
When you have a *Rotary 5 Axis* table or *Vertical Rotary* table selected, there is a new **Use Turning in Job** option. Select this option to enable turning in PartMaker/Mill. **Use Turning in Job** is available only if you have the Vertical Mill-Turn (VMT) module licensed (see page 42).

5-Axis Rotary Table Properties dialog

In the **5 Axis Rotary Table Properties** dialog, the Primary Rotary Axis and Secondary Rotary Axis are renamed to bring PartMaker in line with standard naming conventions. The position of these settings in the dialog is also reversed so that the Primary Rotary Axis is positioned before the Secondary Rotary Axis.

Secondary Rotary Axis for Top Table replaces **Primary Rotary Axis for Top Table**.

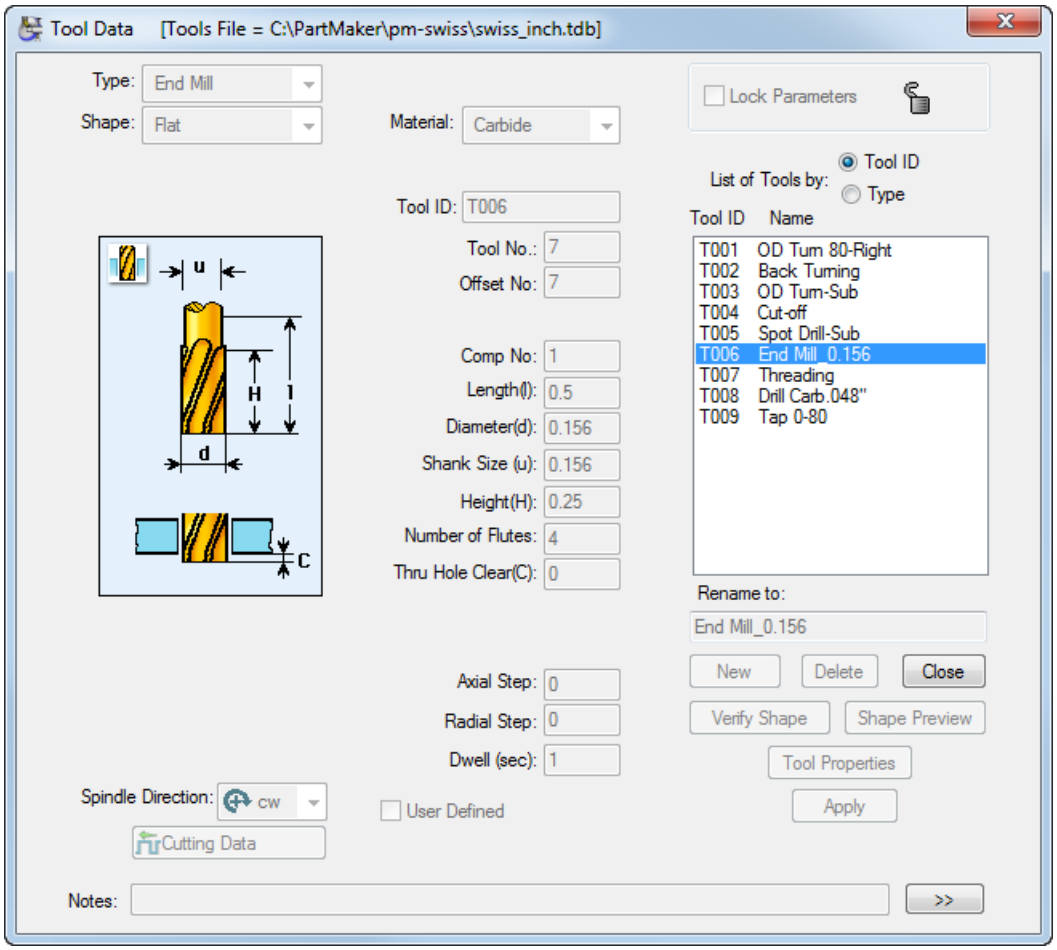
Primary Rotary Axis for Bottom Table replaces **Secondary Rotary Axis for Bottom Table**.



Viewing tool parameters

When you load a Tool Database (.tdb) file that contains tools which are not licensed by your version of PartMaker or are not applicable to your module, the parameters of a tool are displayed when you select it from the drop-down list, but the fields are disabled so they cannot be edited or selected.

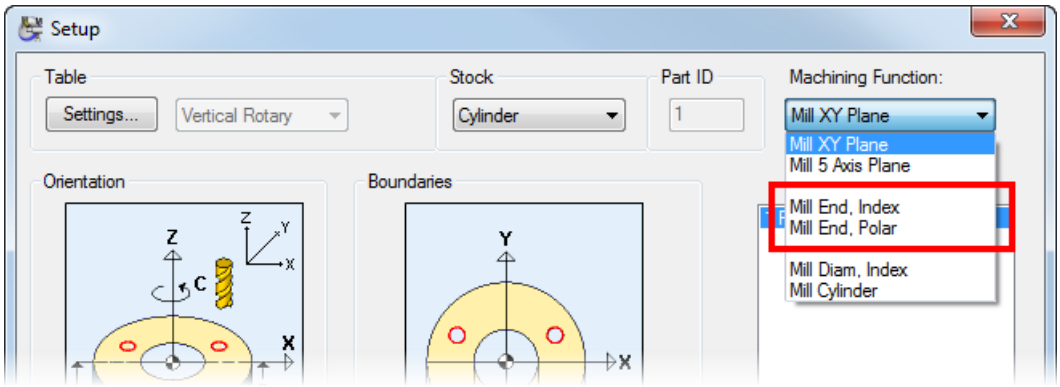
For example, if you select a milling tool when using PartMaker/Turn, the **Tool Data** dialog looks as follows:



Updated Setup dialog in PartMaker/Mill

There are the following changes to the **Setup** dialog in PartMaker/Mill:

- *Mill End, Index* and *Mill End, Polar* are added to the **Machining Function** drop-down list when either a *Rotary 5 Axis* table or *Vertical Rotary* table with *Cylinder* stock is selected. These options were previously available only in PartMaker/SwissCAM and PartMaker/Turn-Mill.



- When you have a *Rotary 5 Axis* table selected with a *Mill Cylinder* or *Mill Diam, Index* **Machining Function**, 2D coordinates in the **Boundaries** window are now displayed as Z, C to ensure a uniform method of programming that is also consistent with machine programming conventions.
- *Turn* is added to the **Machining Function** drop-down list when **Use Turning in Job** is selected on the **Table Settings** dialog (see page 42).
- In the **Table** area, the **Settings** button and table drop-down list have swapped positions. You now select your table in the **Table Settings** dialog (see page 65).
- There is a new **Turning Machine Position** button in the **Table** area when a *Rotary 5 Axis* table is selected on the **Table Settings** dialog (see page 47).
- The **Z Face (F)** field is removed if you select a *Mill XY Plane*, *Mill End, Index*, or *Mill End, Polar* face from the **Machining Function** drop-down list (see page 68).

Z Face field in PartMaker/Mill

In PartMaker/Mill, when you have a *Rotary 5 Axis* table selected with a cylindrical stock, the **Z Face (F)** field is removed from the **Setup** dialog if you select a *Mill XY Plane* face from the **Machining Function** drop-down list.

The **Z Face** field is also removed from the **Define Face Plane** dialog if you have a *Rotary 5 Axis* table selected with cylindrical stock.

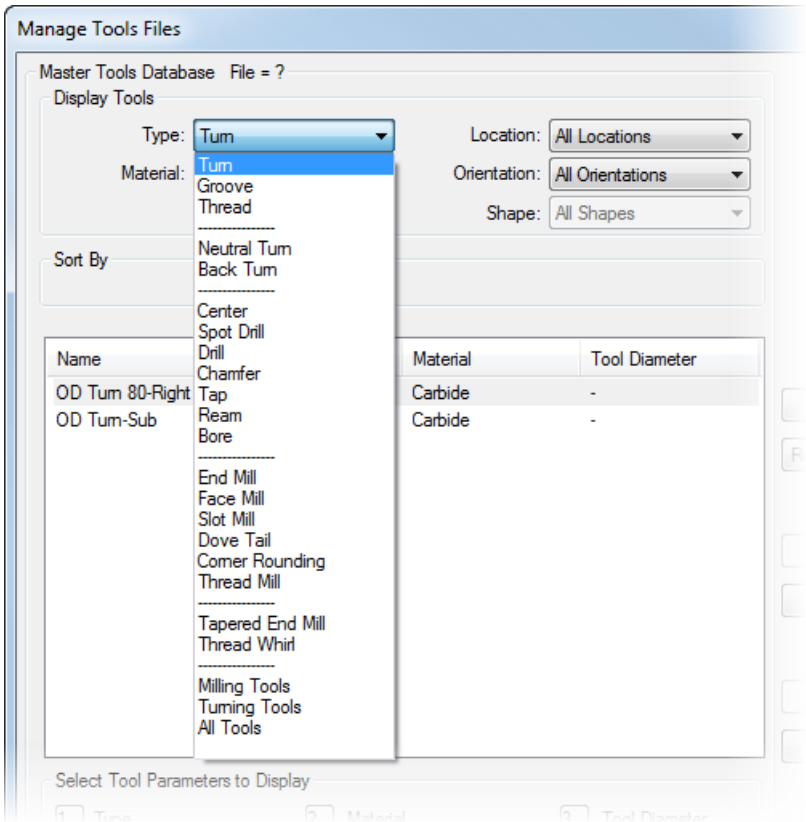
In PartMaker 2015, if you open an old job with a non-zero **Z Face (F)** value for a *Mill XY Plane* face, a message is displayed asking you to adjust the Z_Surf value for all groups.

Using the Manage Tools Files dialog

There are several changes to the **Manage Tools Files** dialog.

Availability of tools in the Type drop-down list

In PartMaker 2015, all tool types are included in the **Type** drop-down list, no matter which version of PartMaker you are running. Previously, the **Type** drop-down list only included tools that were applicable to the version of PartMaker you were running.



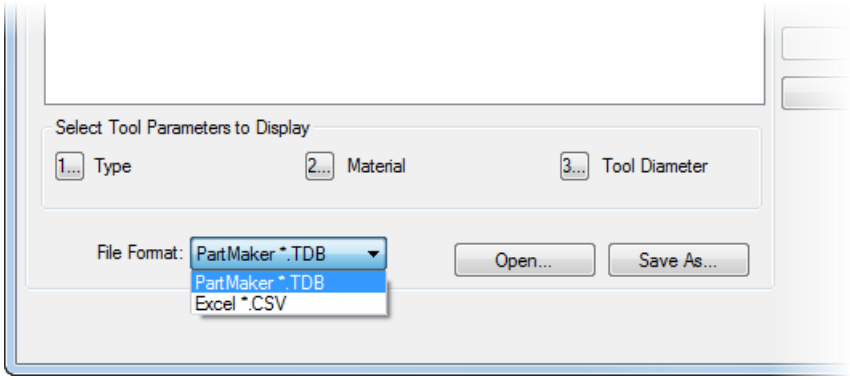
Editing tools

If you select a tool from either the Tool Database or Master Tools Database, which you are not licensed to use in your version of PartMaker, the **Edit** button is unavailable.

In PartMaker/Mill, you can edit turning tools if you have Turning enabled (see page 42).

Saving the Master Tools Database

You can now choose to save the Master Tools Database file to *PartMaker .TDB* or *Excel .CSV* format. If you save to .tdb, any cutting data associated with individual tools in the database is preserved (see page 11). If you save to .csv, a message is displayed warning you that cutting data will not be saved.



ConfigPost enhancements

ConfigPost 2015 includes the following enhancements:

- **New and updated reserved words** (see page 71) — There are several new and updated Reserved words.
- **Postprocessing canned cycles for holes** (see page 73) — There are enhancements to support the new hole canned cycles in PartMaker.
- **User Data Names dialog** (see page 77) — More user conditions and user parameters fields are added to the **User Data Names** dialog.
- **Variables Names dialog** (see page 78) — A third tab of variables is added to the **Variable Names** dialog.
- **File compatibility** (see page 78) — There are changes to the compatibility of .pst/.sub/.wri files created in previous versions of ConfigPost.
- **Machine Settings in ConfigPost** (see page 79) — There is a new **Machine Settings** option in the **Settings** menu to enable you to indicate the type of Post File used
- **Naming Convention for Post Files** (see page 80) — The naming convention for Post Files in ConfigPost is changed.

New and updated reserved words

ConfigPost 2015 includes new several new and updated reserved words:

New reserved words

The following reserved words have been added:

- `<face-tool>` — Used to specify whether a Face turning tool is being used.
- `<is-final-thread-pass>` — This word is TRUE during the final entire pass when a Threading Strategy is selected in the Profile Group Parameters, Turn dialog. If spring passes are used, this word will be true on the final spring pass. This word should not be referenced when a THREAD canned cycle is used.
- `<is-z-shank-axis>` — Used to specify the orientation of the OD tool. This word is set to true when the **Shank Axis** is set to **Z Axis** for turning tools in the **Tool Data** dialog.

- <local-origin-nrt-x> — Used primarily for PartMaker/Mill to specify non-rotated local origin coordinates.
- <local-origin-nrt-y> — Used primarily for PartMaker/Mill to specify non-rotated local origin coordinates.
- <local-origin-nrt-z> — Used primarily for PartMaker/Mill to specify non-rotated local origin coordinates.
- <num-thread-passes> — Used to specify the number of threading passes in a Threading group.
- <right-orient-in-group> — Used to specify the tool orientation from the profile group parameters. This word is set to true if **Tool Orientation** is set to **Right** in the **Profile Parameters, Turn** dialog.
- <thread-mill-internal> — This word is set to true when thread milling an ID thread.
- <time> — Used to reproduce the time when the part program was post processed. The time format matches the computer's regional settings.
- <tool-next-on-toolpost> — Used to specify the next tool number on the same tool post. This word will return the next tool on the tool post if it is different from the current tool.
- <toolpost-next-on-spdl> — Used to specify the next tool post to be used on the current spindle. PartMaker/SwissCAM and PartMaker/Turn-Mill only.

Updated reserved words

The following reserved words have been updated:

- <cw-spindle> — The description is updated because the spindle direction is now set using the **Spindle Direction** field.
- <diam-sign> — You can now use this word in ConfigPost when the Target Application is PartMaker/Turn.
- <local-origin-x> — This word is renamed to <local-origin-rot-x>. It is used primarily for PartMaker/SwissCAM and PartMaker/Turn-Mill to specify rotated local coordinates.
- <local-origin-y> — This word is renamed to <local-origin-rot-y>. It is used primarily for PartMaker/SwissCAM and PartMaker/Turn-Mill to specify rotated local coordinates.

- <local-origin-z> — This word is renamed to <local-origin-rot-z>. It is used primarily for PartMaker/SwissCAM and PartMaker/Turn-Mill to specify rotated local coordinates.
- <next-tl-num-on-chnl> — This word is renamed to <tool-num-next-on-chnl>.
- <num-passes> — This word is renamed to <num-spring-passes>.
- <rapid-abs> — This word is replaced with <rapid-abs-x> (X coordinate for X-oriented tools) and <rapid-abs-z> (Z coordinate for Z-oriented tools).
- <tool-nose-com> — The description is changed to explain that you can use this word to specify the nose compensation value for a mill as well as a lathe.
- <tool-nose-rad> — The description is updated because this word works in Milling as well as Turning.
- <tool-post> — This word is renamed to <toolpost>.
- <tool-post-next> — This word is renamed to <toolpost-next>.

Canned cycles for holes

There are several new reserved words, and some changes to existing reserved words, to support the new canned cycles for holes. See *New and changed reserved words for canned cycles* for more details (see page 75).

Postprocessing canned cycles for holes

ConfigPost includes the following changes to support the enhancements to canned cycles for holes in PartMaker/SwissCAM, PartMaker/Turn-Mill, PartMaker/Turn, and PartMaker/Mill:

- The ability to configure how a Post file uses the new canned cycles for holes (see page 73).
- New and changed reserved words (see page 75).

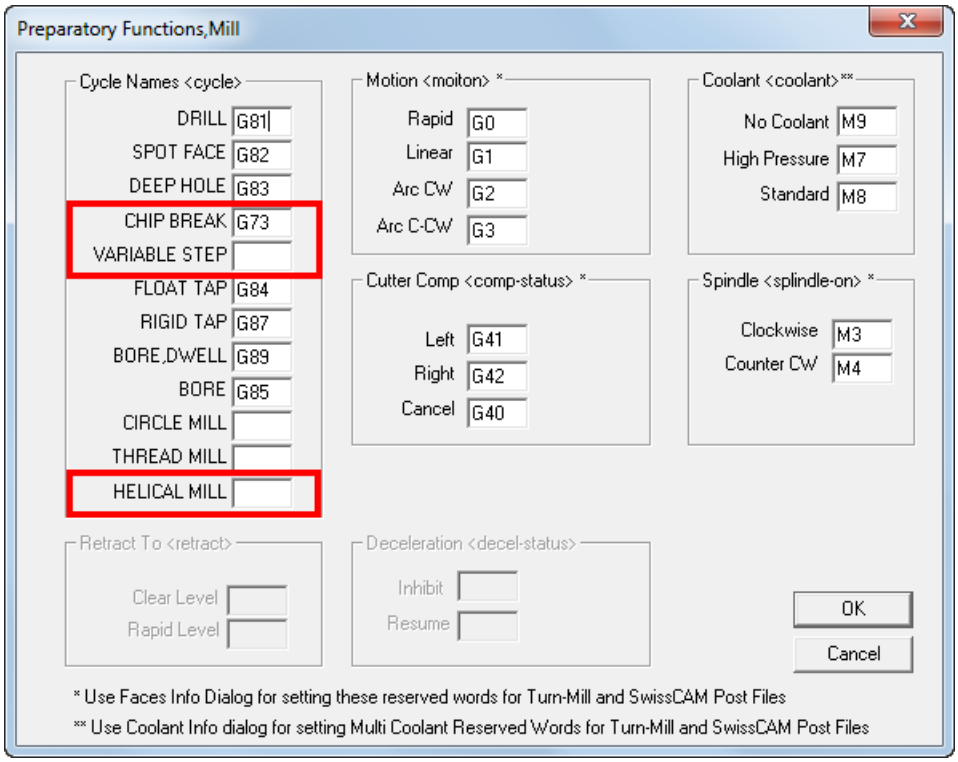
Configuring canned cycles for holes

PartMaker 2015 includes the following new canned cycles for holes:

- CHIP BREAK (turning and milling) (see page 23)
- VARIABLE STEP (turning and milling) (see page 25)
- HELICAL MILL (milling only) (see page 27)

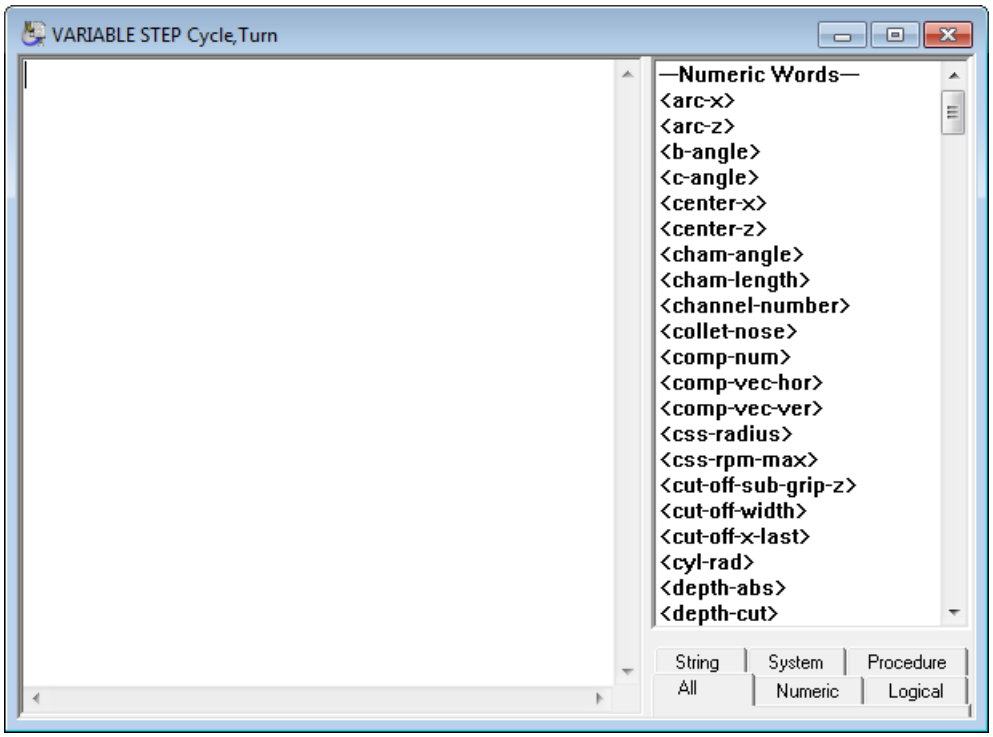
To specify how a Post Configuration file processes these canned cycles you can:

- Specify the machining codes for cycles on the **Preparatory Functions** dialogs. For example, on the **Preparatory Functions, Mill** dialog:



- Use the new options that are available in the **Configure Mill > Canned Cycle Formats** and **Configure Turn > Canned Cycle Formats** menus to configure the new CHIP BREAK, VARIABLE STEP, and HELICAL MILL canned cycles.

For example, select **Configure Turn > Canned Cycle Formats > VARIABLE STEP cycle** to display the **VARIABLE STEP Cycle, Turn** dialog:



New and changed reserved words for canned cycles

ConfigPost 2015 includes new reserved words, and changes to existing reserved words, to support the changes to canned cycles for holes in PartMaker (see page 18).

New reserved words

`<hole-diameter>` — Used to specify the Hole Diameter of the hole canned cycle.

`<is-full-retract>` — This word is set to TRUE if the Retract Type for a hole canned cycle is Full Retract. Not all machines support this in a canned cycle.

`<init-step-feed-fact>` — Used to specify the feed rate factor for the initial step in a VARIABLE STEP hole canned cycle.

`<radial-step>` — Used to specify the Radial Step of a CIRCLE MILL hole canned cycle.

`<retract-dist>` — Used to specify the retract distance for CHIP BREAK and VARIABLE STEP hole canned cycles. Not all machines support this in a canned cycle.

`<step-decrease>` — Used to specify the decrease in speed after the initial step for a VARIABLE STEP hole canned cycle. Not all machines support this in a canned cycle.

<step-min> — Used to specify the minimum step performed in a VARIABLE STEP hole canned cycle.

<thread-mill-left-hand> — Used to specify the direction of the thread when thread milling. This word is set to true when the **Thread Direction** is set to **Left Hand** in the **Profile Group Parameters** or **Edit Hole Operation** dialogs.

<thread-mill-top-down> — Used to specify the style of the thread when thread milling. This word is set to true when the **Style** is set to **Top Down** in the **Profile Group Parameters** and **Edit Hole Operation** dialogs.

Changes to existing reserved words

<can-cyc> — The description is updated because the **Canned Output** check box is renamed to **Output Hole Canned Cycle** for drilling processes, and **Output Canned Cycle for Turning** for turning processes.

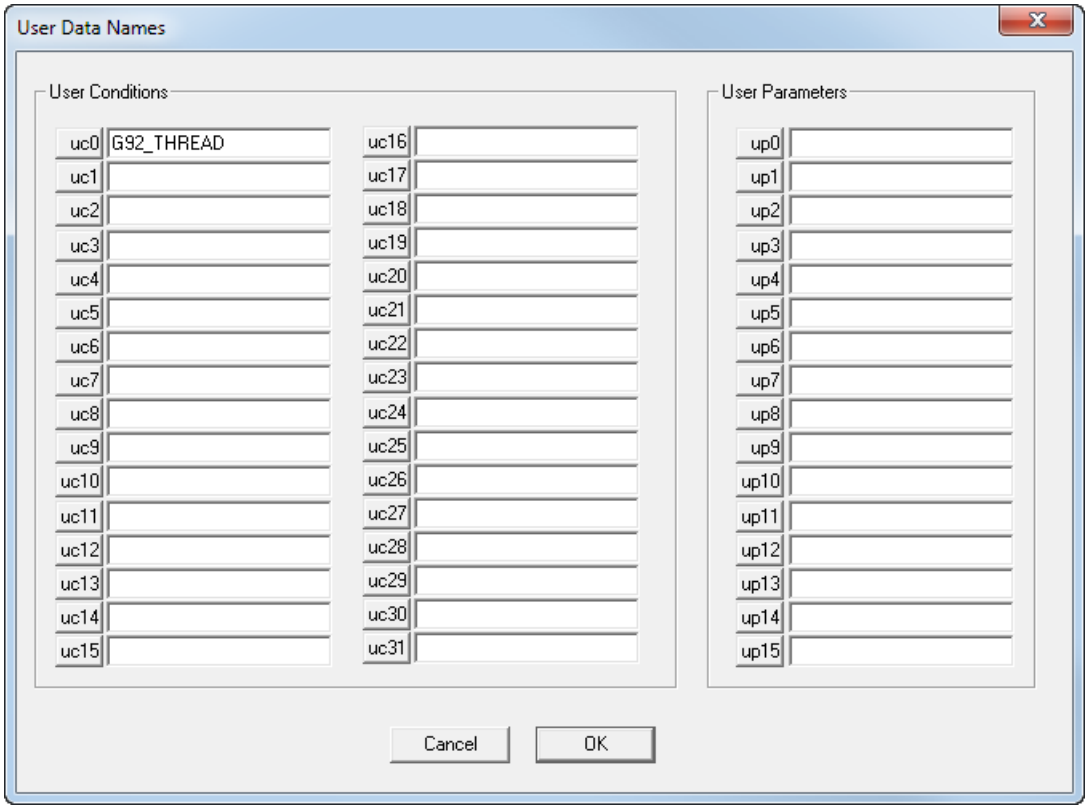
<num-spring-passes> — Replaces <num-passes>. Previously this was used only for threading to specify the number of spring passes for finishing a thread (as specified on the **Thread Options** dialog). Now, it is also used for the HELICAL MILL canned cycles to specify the number of spring passes at the bottom of a hole (as specified on the **Edit Hole Operation** dialog).

<step> — The description is updated because the **Axial Step (Q)** field is added to the **Edit Hole Operation** dialog.

<step1> — The description is updated because the **Axial Step (Q)** field is located on the **Edit Hole Operation** dialog in addition to the **Profile Group Parameters - Grooving** dialog.

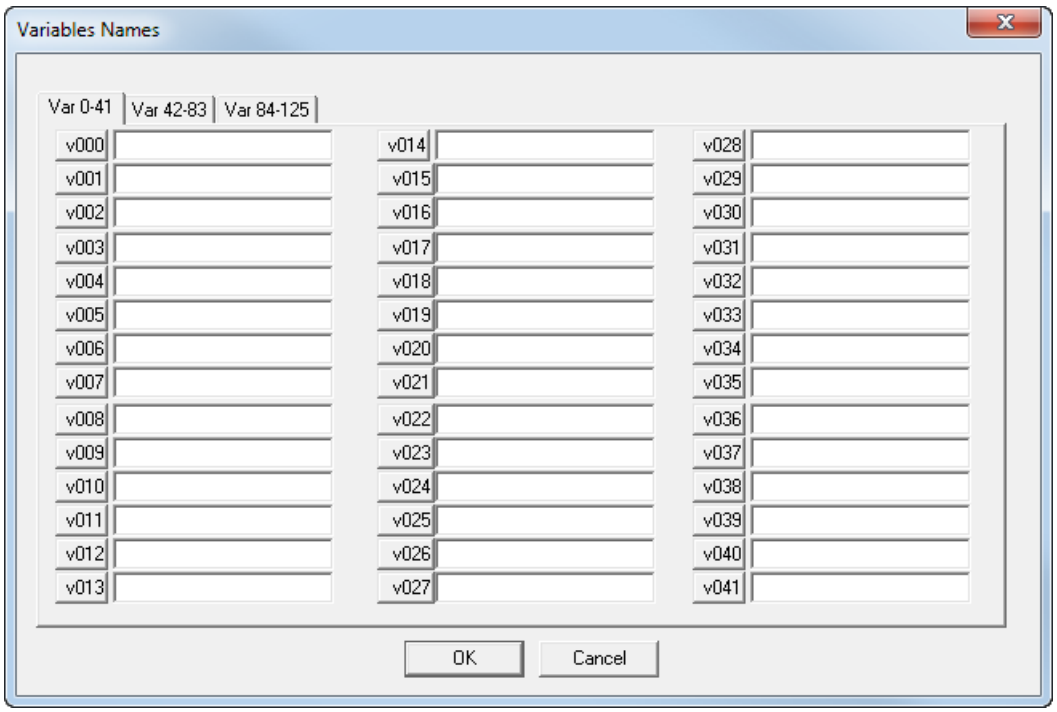
User Data Names dialog

The **User Data Names** dialog enables you to add your own logical reserved words (User Conditions) and string reserved words (User Parameters) to a Post file. In PartMaker 2015, the number of User Conditions you can add is increased from 15 to 31, and the number of User Parameters you can add is increased from 7 to 15.



Variables Names dialog

The **Variables Names** dialog enables you to enter your own variables to a Post file. In PartMaker 2015, another tab has been added to the dialog, which means that the number of variables you can add is increased from 83 to 125.



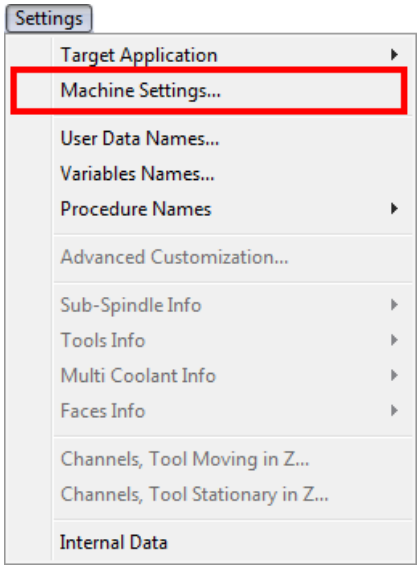
File compatibility in ConfigPost

In ConfigPost 2015, there are changes to the compatibility of .pst, .sub, and .wri files that were created in previous versions of the software:

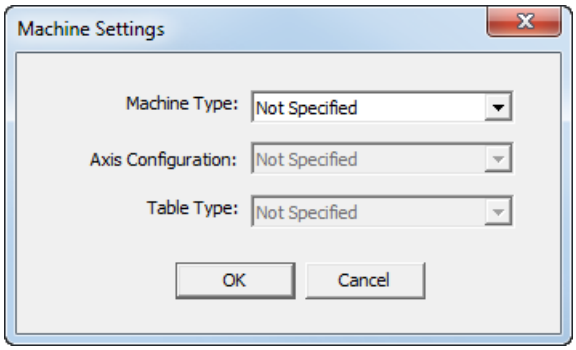
- .pst and .sub files produced in ConfigPost 2014 are compatible with ConfigPost 2015.
- .wri files produced in ConfigPost 2014 are not compatible with ConfigPost 2015.

New Machine Settings option in ConfigPost

In ConfigPost 2015, there is a new **Machine Settings** option in the **Settings** menu to enable you to indicate the type of Post file used. This option is available only when PartMaker/Mill is selected as the Target Application.



Select this option to display the **Machine Settings** dialog, where you can select your machine type, axis configuration, and table type. The options on this dialog are the same as those found in the **Machine Configuration** area of the **Machine Data** dialog in PartMaker/Mill (see page 48). For new and existing Post files, all the fields are set to **Not Specified** until you select options from the drop-down lists.



Select **Vertical Mill-Turn** from the **Machine Type** drop-down list to make the **Configure Turn** menu available on the **Main** menu bar.

Naming convention for Post Files

There are changes to the naming convention used when you create, modify, or save a distribution Post File in ConfigPost.

When you create, modify, or save distribution Post File in ConfigPost 2015 the character 'e' is appended after 'v' in the distribution Post File name. For example, Citizen_M3_32-ve122.pst.

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 81).
- Training (see page 81).

If you experience any problems when installing or using PartMaker, contact technical support (see page 82).

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 82).

Training for PartMaker and ConfigPost

A wide variety of PartMaker and ConfigPost training programs are available from PartMaker, Inc., Delcam Ltd 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 Ltd, see the Training section of the Delcam website (www.delcam.com).
- Delcam Ltd 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 Ltd 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.