

FeatureCAM 2014 R3

Feature-based CAM software for mills, multi-tasking
lathes and wire EDMs

www.featurecam.com



Now Available

What's New

FeatureCAM 2014 R3

What's New



Copyright © 1995-2014 Delcam plc. All rights reserved.

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

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

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

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

Acknowledgements

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

FeatureCAM Version: 2014 R3 Date: 03 April 2014 11:17

Contents

What's New in FeatureCAM 2014 R3 1

Milling improvements	1
Support for right-angled heads	2
Better transformation of surface milling features	3
Standard thread improvements	4
Thread Milled Hole feature type	5
Manually define plunge points for Z level rough features	6
Chamfer cutter compensation default.....	7
Specify default drill diameters for Reamed Holes	8
Turning improvements	9
Separate approach and withdraw lengths for finish operations	9
Irrelevant canned cycle parameters hidden.....	10
Wire EDM improvements	11
Generate single program with program stop between each setup	11
User Interface improvements	12
Replace curves when pasting features.....	12
Tool preview improvement	13
Pick radius from cylindrical solids.....	14
Quick access to Stock settings from the Operation List	15
Import improvements	15
Positioning solids in 5 axis parts.....	15
Importing work planes	15
XBUILD improvements	16
Faster machining times when using polar coordinate interpolation	17
Post variable names.....	17
Restrict machine movements greater than 360 degrees	19
Detect Euler angle changes	20
Detect if alternative 5 axis position is enabled.....	20
Add-in improvements.....	21
Setup Sheet add-in tags.....	21

What's New in FeatureCAM 2014 R2 22

Vortex machining	23
Creating a 3D Vortex toolpath (3D HSM)	24
3D Vortex example (3D HSM).....	25
Creating a 2.5D Vortex toolpath (REC)	30
2.5D Vortex example (REC).....	30
User Interface and work-flow improvements.....	32
Feature Properties dialog icons.....	33
Operation List cursor changes.....	33

Turret numbering	34
Renaming items in the Part View	35
UCS Translate pick locations	35
Tool Manager	36
Expanded dialog bar edit boxes	37
Delcam Exchange installation	37
Remove stock model operations	37
2.5D Milling improvements.....	38
NT toolpaths for multiple diameters.....	38
Thread Milling ID/OD depth %.....	39
Bottom finish allowance for Counterbore features.....	40
Separate cutter compensation.....	41
Finish cutter comp and Partline program independence	42
3D Milling improvements.....	42
Offset Direction.....	42
Radial Plunge Roughing.....	44
Turning improvements	45
Transfer operation changes.....	45
Wire EDM improvements	45
Project curves.....	45
Machine design improvements	47
Add Tool Location dialog improvements	47
Send Machine Design files	48
Import improvements	49
SolidWorks 2014 files.....	49
Dowel holes.....	49
SolidEdge ST5 and ST6 files.....	49
XBUILD improvements	50
Improved Error message.....	50
Add-in Improvements.....	51
Setup Sheet Cutter Compensation.....	51

What's New in FeatureCAM 2014 R1 52

User interface improvements	52
Feature names	53
Layers in Part View	53
New Pick Curve buttons	55
New Part Documentation fields	56
Setup at opposite end of cylinder	57
Send crash logs.....	58
Shaded Grey default style	59
More attractive tool display window.....	59
Licensed components activated by default.....	60
License expiry warning	60
Import improvements	61
OpenDesign AutoCAD	61
Spatial R23 SP1 Interop integrated	61
Spatial 3D R23 SP2 Interop integrated	61
Import Using Exchange logged	61

Entry-level solid wireframe import	61
2.5D Milling improvements.....	62
Automatic clamp avoidance.....	62
Chamfer before Finish	65
Solid toolholder.....	66
OLF engraving fonts.....	66
Solid improvements (SOLID)	67
Offset faces	67
Turn improvements (TURN).....	68
Automatic corner rounding	68
Steady rest and tailstock support	70
Machine simulation improvements (MSIM).....	73
Show edges.....	73
Play simulation once	75
Machine design improvements (MSIM).....	75
Replace solid.....	75
Easier relationship definitions.....	76
Simplified milling machine interface	77
Improved user interface.....	77
Probing improvements	78
Edit default values	78
Decision-making features.....	79
Turn/mill probing simulation improved.....	82
XBUILD improvements	83
New reserved words.....	83
Z index clearance optional.....	83
Force segment start option moved	84

What's New in FeatureCAM 2014 R3

FeatureCAM 2014 R3 offers all of the original features of FeatureCAM 2014 R2, but with numerous improvements. The most significant improvements are detailed in the topics that follow.



FeatureCAM has three major enhancement releases per year (R1, R2, and R3) issued once every four months. This section describes what's new for FeatureCAM 2014 R3 and covers four months of development. To see a year's worth of development, please also see the What's New topics for the previous two releases.

- **What's New in FeatureCAM 2014 R2** (see page 22)
- **What's New in FeatureCAM 2014 R1** (see page 52)

Milling improvements

FeatureCAM 2014 R3 includes these Milling improvements:

- Support for right-angled heads (see page 2).
- Better transformation of surface milling features (see page 3).
- You can use the standard thread sizes more easily (see page 4).
- There is a new Thread Milled Hole feature type (see page 5).
- You can manually define plunge points for Z level rough features (see page 5).
- You can use cutter compensation by default for chamfer operations (see page 7).
- You can specify default drill diameters for Reamed Holes (see page 8).

Support for right-angled heads

You can use right-angled machine tool heads for 2.5D Milling and Turn/Mill parts:



This enables you to create difficult features, such as internal pockets, and enables you to create multiple features without repositioning the part.

Selecting a tool holder for right-angled heads



You must use a solid to describe the tool holder shape for right-angled heads.

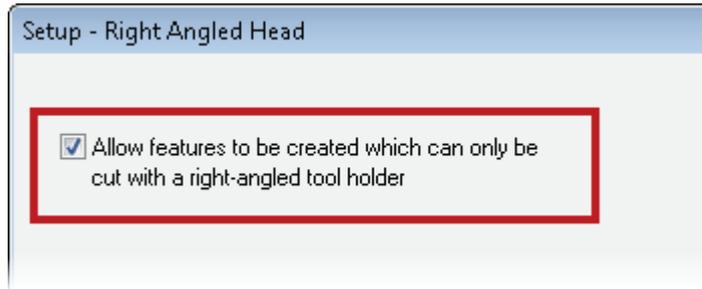
To specify a tool holder for right-angled heads:

- 1 In the **Tool Holder Properties** dialog, select **Use solid to describe holder shape**.
- 2 Select the solid from the list.
- 3 Select **Angled Head**.
- 4 Select whether the **Azimuth angle** is **Variable** or **Fixed**.

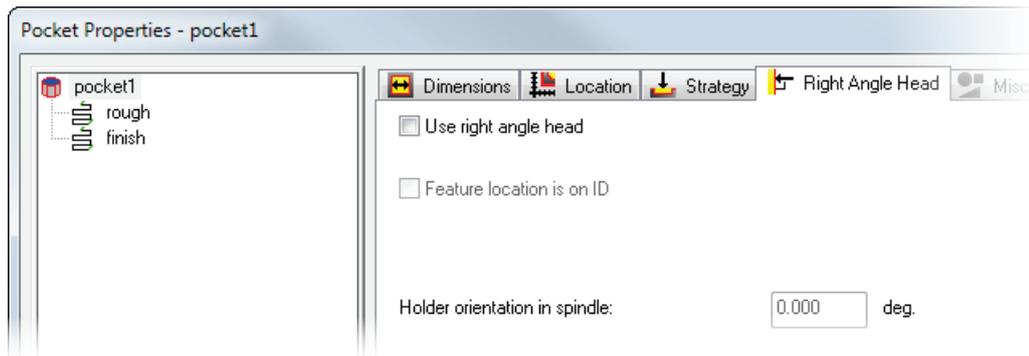
Using the right-angled head tool

There is a new page of the Setup wizard.

Select the option on the **Right Angled Head** page of the **Setup** wizard:



When selected, the **Right Angled Head** tab is displayed in the **Feature Properties** dialog for features which can use a right-angled head:



Use right angle head — Select to use a right-angled head tool.

Feature location is on ID — Select if the feature is on the inside diameter of the part.

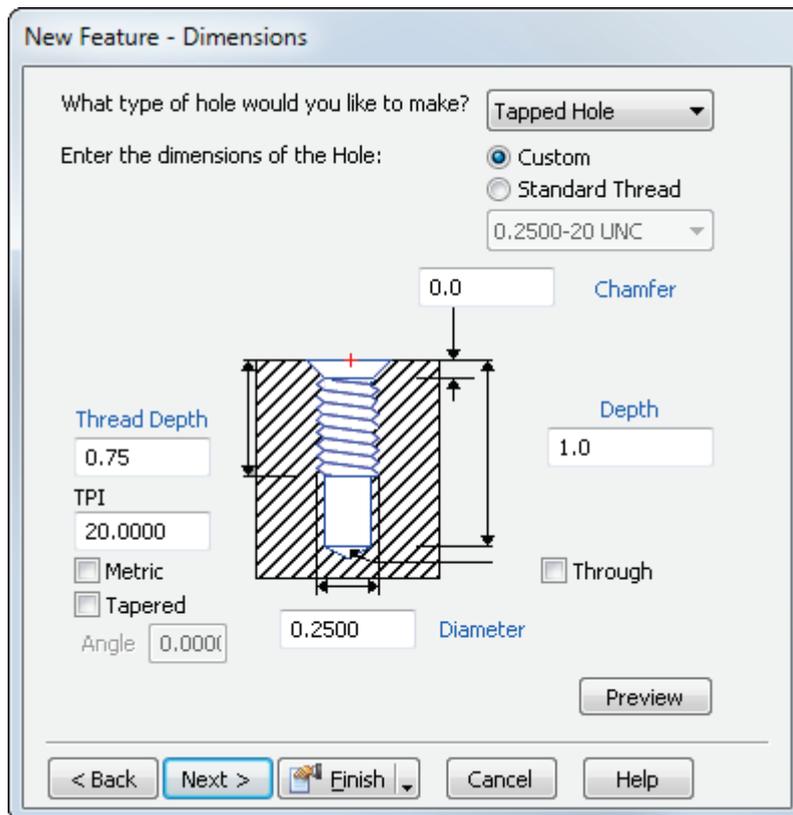
Holder orientation in spindle — Enter the holder orientation in the spindle in degrees.

Better transformation of surface milling features

You can now transform surface milling features which are created from solid faces.

Standard thread improvements

You can use the standard thread sizes more easily in the **New Feature** wizard and **Feature Properties** dialog:



To use a standard thread size, select **Standard Thread** and select a thread size from the list. You cannot edit the standard dimensions when **Standard Thread** is selected.

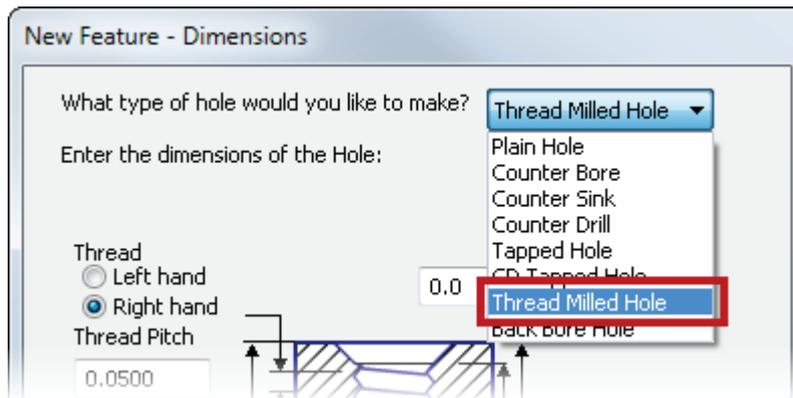
You can click the blue dimensions labels to pick the values from the graphics window.

Thread Milled Hole feature type

There is a new Thread Milled Hole feature type, which you can use to easily create Thread Milled Hole features and recognize them with Automatic Feature Recognition.

Previously, you would have to create separate Hole and Thread features to create a Thread Milled Hole.

On the **Dimensions** page of the **New Feature** wizard, select **Thread Milled Hole** from the type list to display the Thread Milled Hole parameters:



You can add a thread to an existing Hole feature by changing the **Type** on the **Dimensions** tab of the **Hole Feature Properties** dialog.

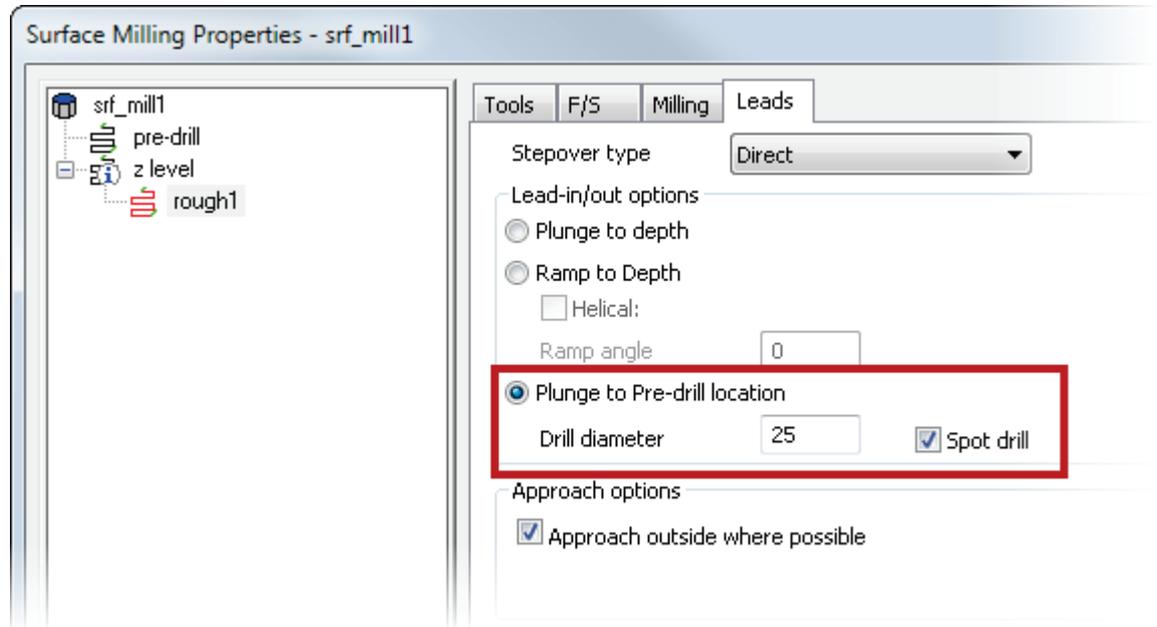
In the **Hole Feature Properties** dialog, you can use the **Strategy** tab to edit the Hole settings, and the **Thread Mill** tab to edit the Thread settings.

Manually define plunge points for Z level rough features

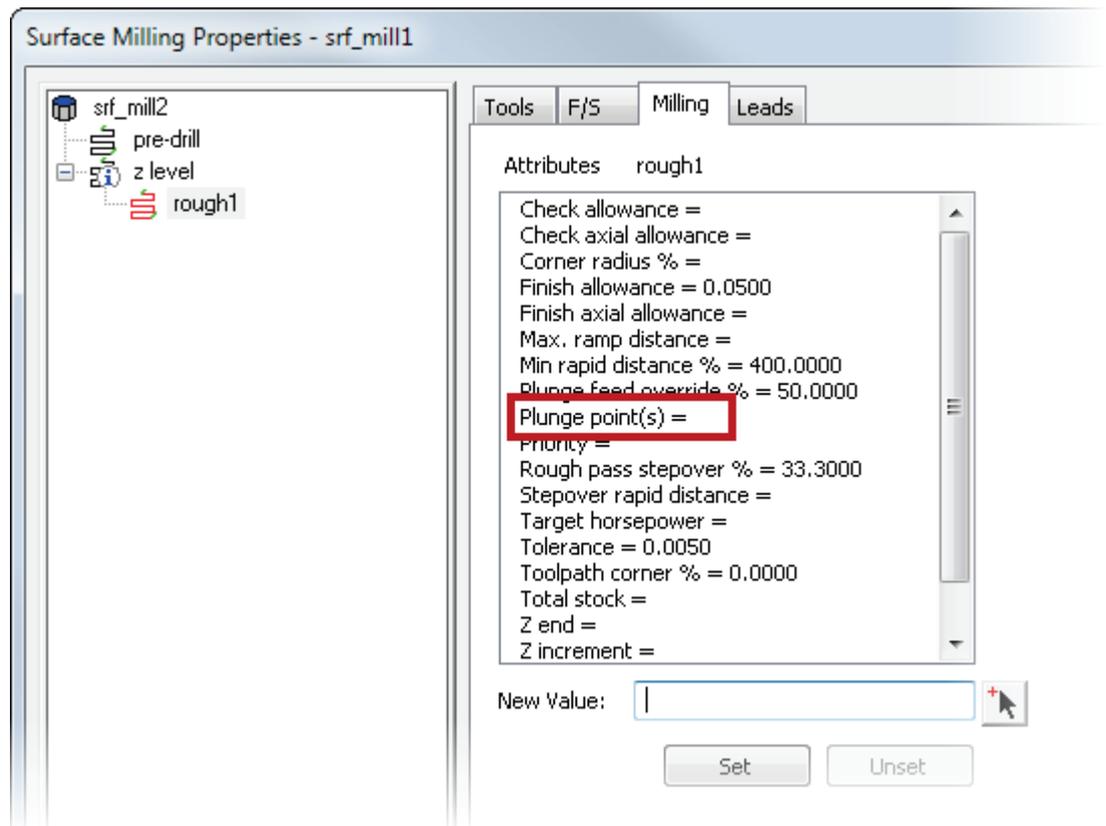
You can manually define plunge points for Z level rough features.

To define the plunge point:

- 1 On the **Leads** tab of the **Surface Milling Properties** dialog, select **Plunge to Pre-drill location**, and enter a **Drill diameter**:



- 2 Select the **Plunge point(s)** parameter on the **Milling** tab:



- 3 Enter the coordinates of a point, or click **Pick location**  and select a point in the graphics window.

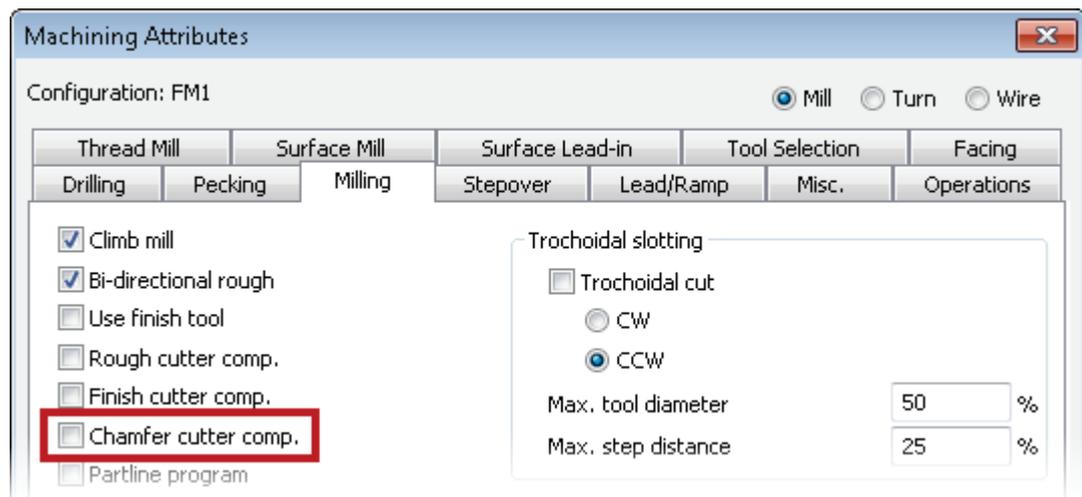
Alternatively, enter the name of a curve in the **New Value** field to place the plunge point at a point on the curve.

- 4 Click **Set** to accept the selected point.

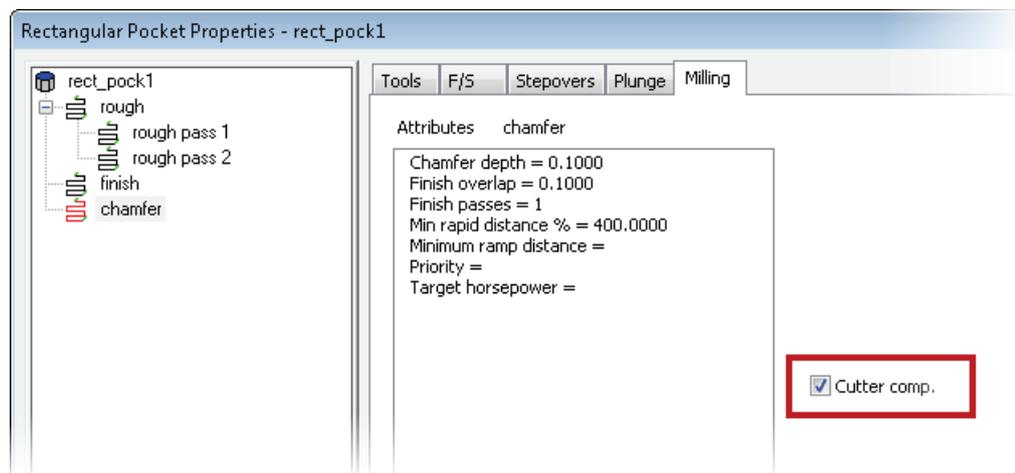
Chamfer cutter compensation default

You can use cutter compensation by default for chamfer operations.

Select **Chamfer cutter comp.** on the **Milling** tab of the **Machining Attributes** dialog:



When selected, **Cutter comp.** is enabled by default for chamfer operations on new features, and any existing features which use the default setting.



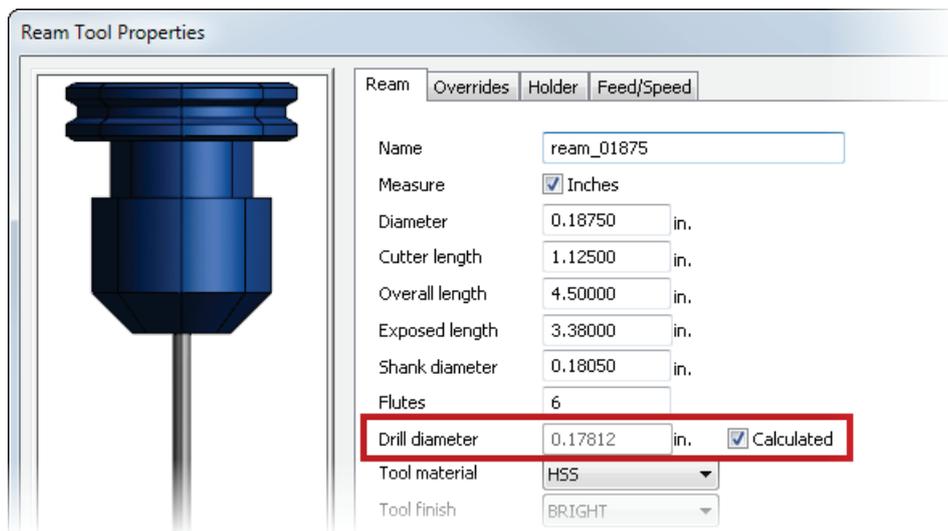
Specify default drill diameters for Reamed Holes

For Reamed Hole features, you can specify the default tool diameter for all drill and bore operations in the feature.



Previously, these were calculated as a percentage of the Ream tool diameter.

There is a new **Drill diameter** parameter on the **Ream** tab of the **Ream Tool Properties** dialog:



If **Calculated** is deselected, you can enter a default **Drill diameter** value, which is used for the drill and bore operations in the feature.

If **Calculated** is selected, the **Drill diameter** value is calculated from the **Drill % of ream/bore** attribute on the **Tool Selection** tab of the **Machining Attributes** dialog.

Turning improvements

FeatureCAM 2014 R3 includes these Turning improvements:

- You can specify separate approach and withdraw lengths for finish operations (see page 9).
- Irrelevant parameters are hidden in the **Feature Properties** dialog when using Canned cycle (see page 10).

Separate approach and withdraw lengths for finish operations

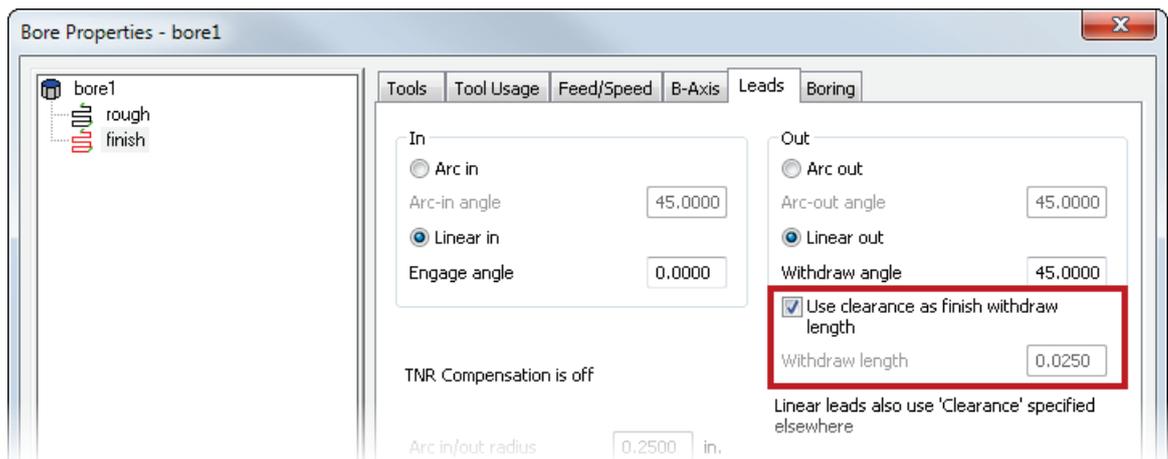
You can control the approach and withdraw lengths separately for Turning finish operations.



Previously, you could only do this for rough operations.

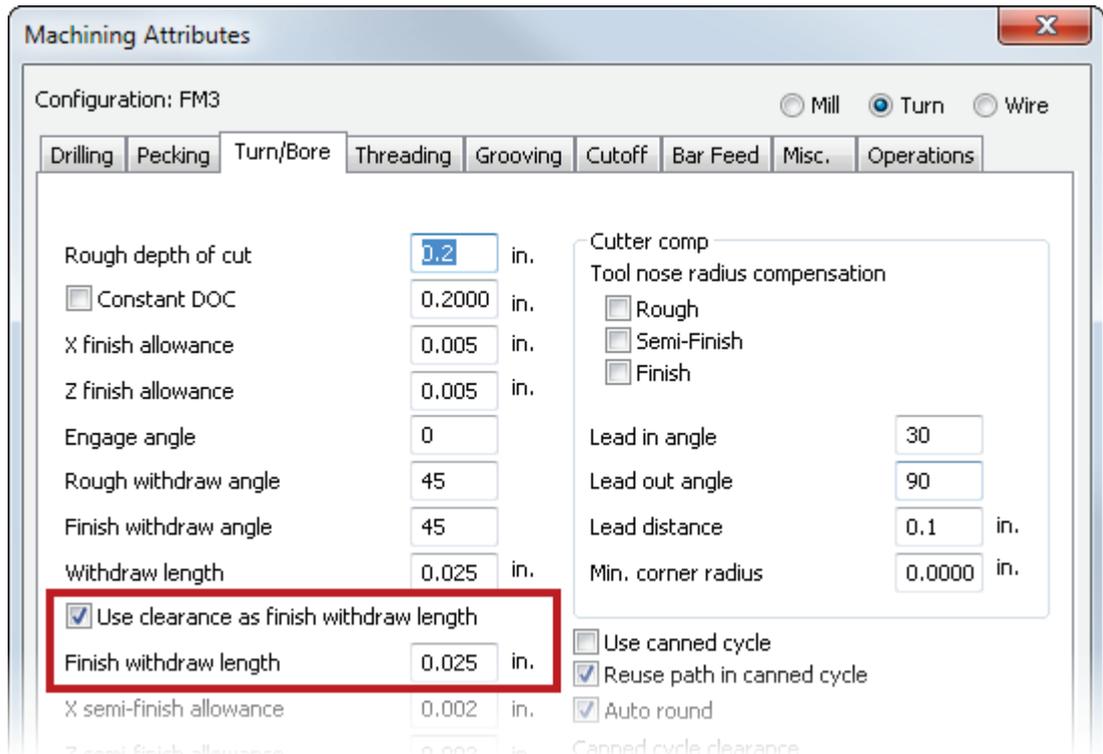
For example, this is useful for internal Bore features where you need to ensure the tool does not withdraw too far and gouge the part.

There are new options on the **Leads** tab of the Turning **Feature Properties** dialog:



When **Use clearance as finish withdraw length** is selected, the **Clearance** value is used as the approach and withdraw length for finishing moves. When deselected, you can enter a separate **Withdraw length**.

You can set the default values for these parameters on the **Turn/Bore** tab of the **Machining Attributes** dialog:



Irrelevant canned cycle parameters hidden

Irrelevant canned cycle parameters are hidden in the Turning **Feature Properties** dialog when they do not affect the toolpaths:

- When **Use canned cycle** and **TNR** are selected on the **Strategy** tab, the **Canned cycle Z clearance** option is hidden on the **Turning** tab.



*The **Lead distance** and **Canned cycle X clearance** parameters are used instead.*

- When **Face** is selected in the **Strategy** tab, the **Canned cycle X clearance** option is hidden on the **Turning** tab.



*The **Lead distance** and **Canned cycle Z clearance** parameters are used instead.*

Wire EDM improvements

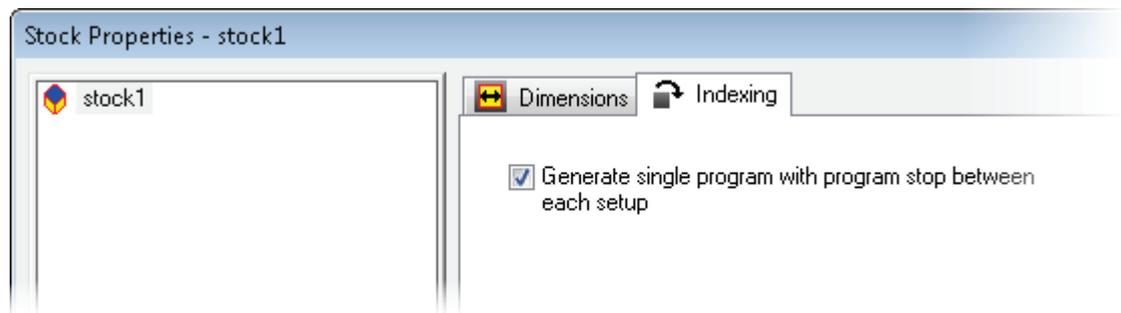
FeatureCAM 2014 R3 includes this Wire EDM improvement:

- You can output multiple Setups as a single Program with a Program stop between Setups (see page 11).

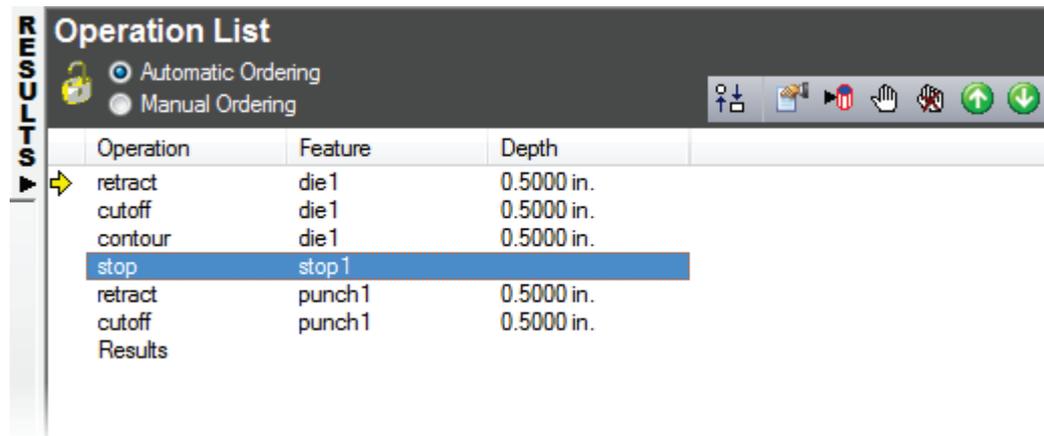
Generate single program with program stop between each setup

For Wire EDM parts, you can output multiple Setups as a single NC program, instead of a separate program for each Setup.

On the **Indexing** tab of the **Stock Properties** dialog, select **Generate single program with stop between each setup**:



Program stops are added to the **Operation List**, allowing you to reposition the part between Setups, for example:



User Interface improvements

FeatureCAM 2014 R3 includes these User Interface improvements:

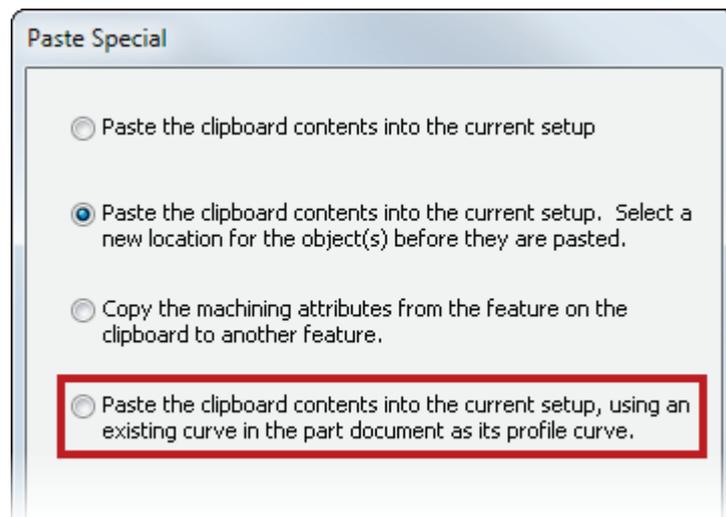
- You can replace curves when pasting features (see page 12).
- The tool preview is oriented correctly for the selected machine type (see page 13).
- You can pick radius values from cylindrical solids (see page 14).
- You can quickly access the Stock settings from the Operations List (see page 15).

Replace curves when pasting features

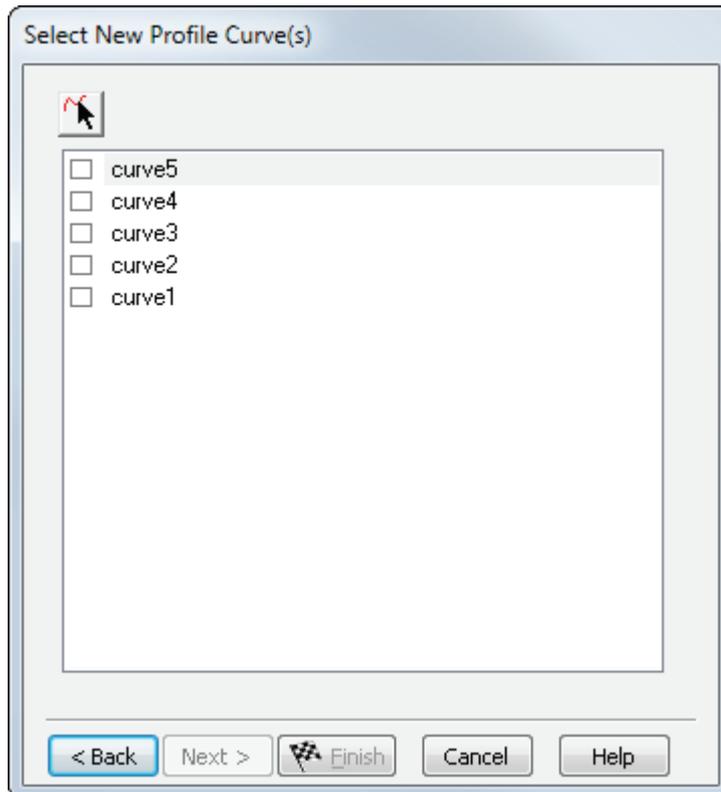
When pasting a feature, you can replace the feature's boundary curve with a curve in the document.

This enables you to quickly create custom features, without having to manually change the boundary curve for each feature.

In the **Paste Special** dialog, select **Paste the clipboard contents into the current setup...** and click **Next**:



In the **Select New Profile Curve(s)** dialog, select any curves you want to use to create the feature and click **Next**:



Use the **Paste Special - Reference** and **Paste Special - Location** dialogs to position the feature in the document.



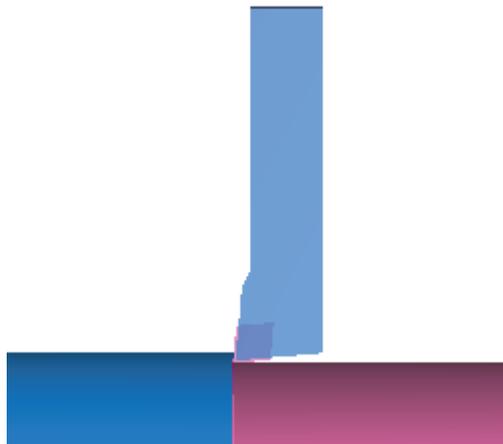
*If you select multiple curves in the **Select New Profile Curve(s)** dialog, a single feature is created for all curves, the same as if you select multiple curves when creating a feature using the **New Feature wizard**.*

Tool preview improvement

The tool previews now display the tool in the correct orientation for the selected machine type.

For example, when **Slant bed lathe** is selected on the **Machine** tab of the **Viewing Options** dialog:

3D simulation:



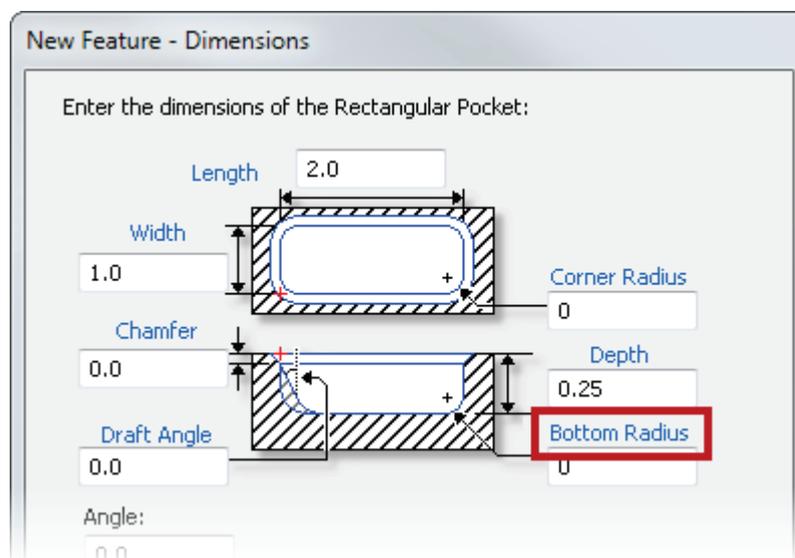
Tool preview:



Pick radius from cylindrical solids

In the **New Feature** wizard and the Feature **Properties** dialog, you can click on a blue radius label and pick the value from a cylindrical solid in the graphics window.

For example, the **Bottom Radius** parameter on the **Dimensions** page of the **New Feature** wizard:



Quick access to Stock settings from the Operation List

You can quickly access the Stock settings from the **Operation List**.

Right-click in the **Operation List** and select **Stock** from the context menu to display the **Stock** tab of the Feature **Properties** dialog for the operation.



This applies to Surface Milling features only.

Import improvements

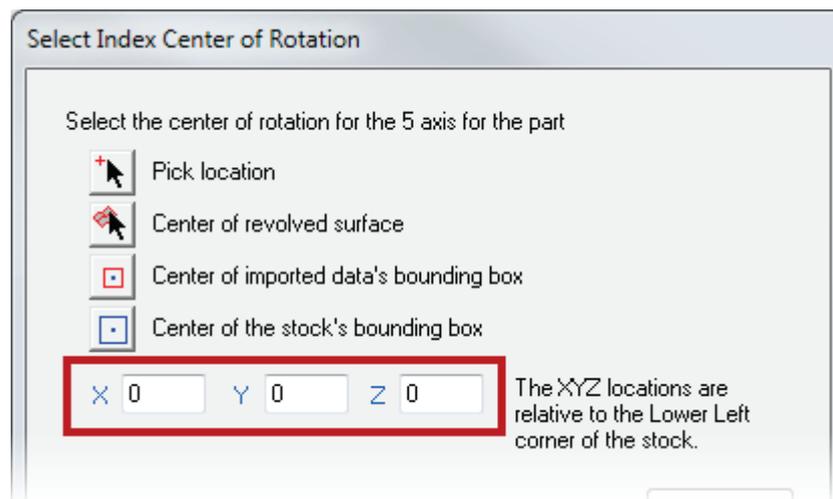
FeatureCAM 2014 R3 includes these Import improvements:

- You can position solids in 5-axis parts more easily (see page 15).
- You can choose whether to import or ignore work planes (see page 15).

Positioning solids in 5 axis parts

You can position solids in 5 axis parts more easily when importing.

On the **Select Index Center of Rotation** page of the **Import** wizard, you can enter the **X**, **Y** and **Z** coordinates of the center of rotation for 5 axis parts:

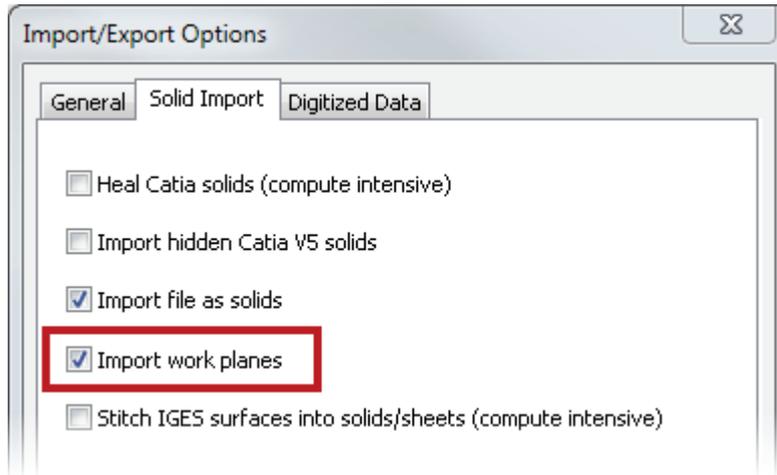


Previously, you could only enter the Y and Z coordinates.

Importing work planes

You can choose whether to import or ignore work planes when importing solids into FeatureCAM.

Deselect **Import work planes** on the **Solid Import** tab of the **Import/Export Options** dialog to ignore planar surfaces when importing solids:



XBUILD improvements

FeatureCAM 2014 R3 includes these XBUILD improvements:

- Faster machining times when using polar interpolation (see page 17).
- You can rename post variables (see page 17).
- You can restrict machine movements greater than 360 degrees (see page 19).
- There are new reserved words which determine whether the Euler angles have changed (see page 20).
- There is a new reserved word which determines whether alternative 5 axis position is enabled (see page 20).

Faster machining times when using polar coordinate interpolation

You can achieve faster machining times when using polar coordinate interpolation by canceling polar coordinate interpolation for rapid moves between cuts.

You can use the following XBUILD reserved words to cancel polar coordinate interpolation:

Reserved word	Definition
<POLAR-X>	The transformed X coordinate for polar coordinate interpolation. This gives you flexibility to turn polar coordinate interpolation on and off when needed.
<POLAR-Y>	The transformed Y coordinate for polar coordinate interpolation. This gives you flexibility to turn polar coordinate interpolation on and off when needed.

Post variable names

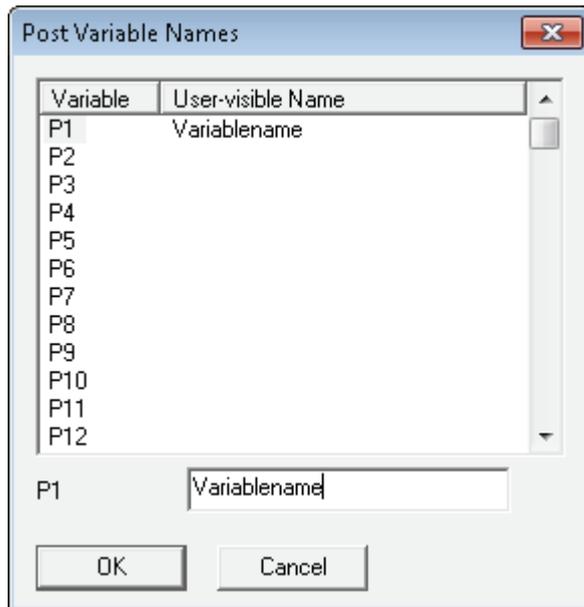
You can rename post variables to make them easier to understand.

By default, post variables are numbered sequentially:

Variable	Value
P1	
P2	
P3	
P4	
DE	

In the XBUILD dialog, select **CNC-Info > Post Variable Names** from the menu to display the **Post Variable Names** dialog.

You can enter a **User-visible Name** for each variable:



The new variable name is displayed in the following places:

- In the **Post Variables** list in FeatureCAM:

Variable	Value
variablename	
P2	
P3	
P4	
P5	
P6	

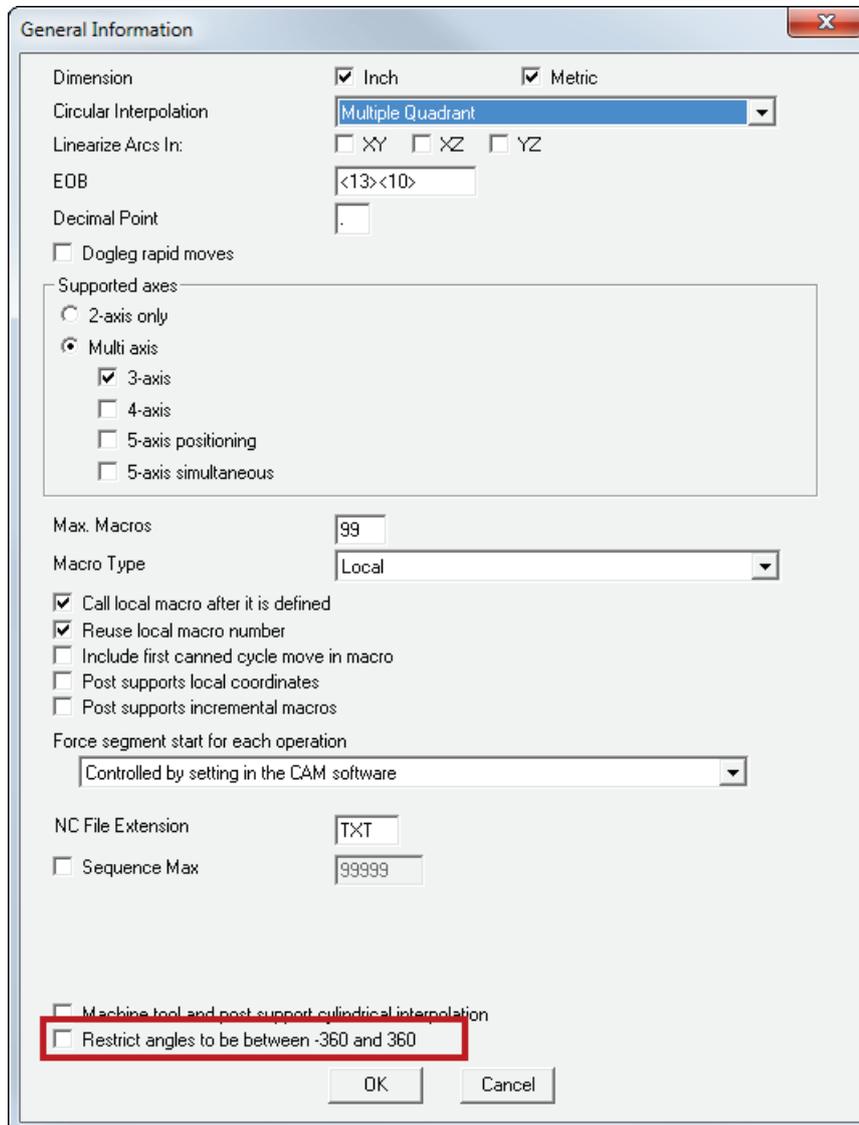
- In any Formats which contain the variable, for example:

```
XBUILD - Editing Format "Segment Start"
File Edit
==SEGMENT START==<EOB>
p1 = <PVAR:variablename><EOB>
p2 = <p2><EOB>
p3 = <p3><EOB>
p4 = <p4><EOB>
p5 = <p5><EOB>
```

Restrict machine movements greater than 360 degrees

For milling machines, you can specify whether the machine allows angles greater than 360 degrees.

Select **Restrict angles to be between -360 and 360** on the **General Information** dialog:



When selected, moves which have rotations greater than 360 degrees are split into smaller pieces.

Detect Euler angle changes

There are new reserved words which determine whether the Euler angles have changed.

Reserved word	Definition
<EULER-CHANGED>	TRUE on any move where <code>EULER-1</code> , <code>EULER-2</code> , or <code>EULER-3</code> has changed. This includes the Linear, Rapid, Canned Move, and Segment Start formats. In addition, <EULER-CHANGED> is TRUE for the Program Start and Tool Change formats if 4- or 5-axis indexing is enabled.
<ABS-SHIFT-CHANGED>	TRUE on any move where <code>ABS-SHIFTX</code> , <code>ABS-SHIFTY</code> , or <code>ABS-SHIFTZ</code> has changed. This includes the Linear, Rapid, Canned Move, and Segment Start formats. In addition, <ABS-SHIFT-CHANGED> is always TRUE for the Program Start and Tool Change formats.

Detect if alternative 5 axis position is enabled

There is a new reserved word which determines whether **alternative 5 axis position** is enabled:

Reserved word	Definition
<IS-ALT-POS>	TRUE if alternate 5-axis position is selected.

Add-in improvements

FeatureCAM 2014 R3 includes this add-in improvement:

- There are new tags in the Setup Sheet add-in for finding the operation depth and minimum Z coordinate (see page 21).

Setup Sheet add-in tags

There are new tags in the Setup Sheet add-in:

- `{operation.depth}` — Operation depth.
- `{.max_depth}` — Minimum Z coordinate.

For example, you can use `{tool.max_depth}`.

What's New in FeatureCAM 2014 R2

FeatureCAM 2014 R2 offers all of the original features of FeatureCAM 2014 R1, but with numerous improvements. The most significant improvements are detailed in the topics that follow.

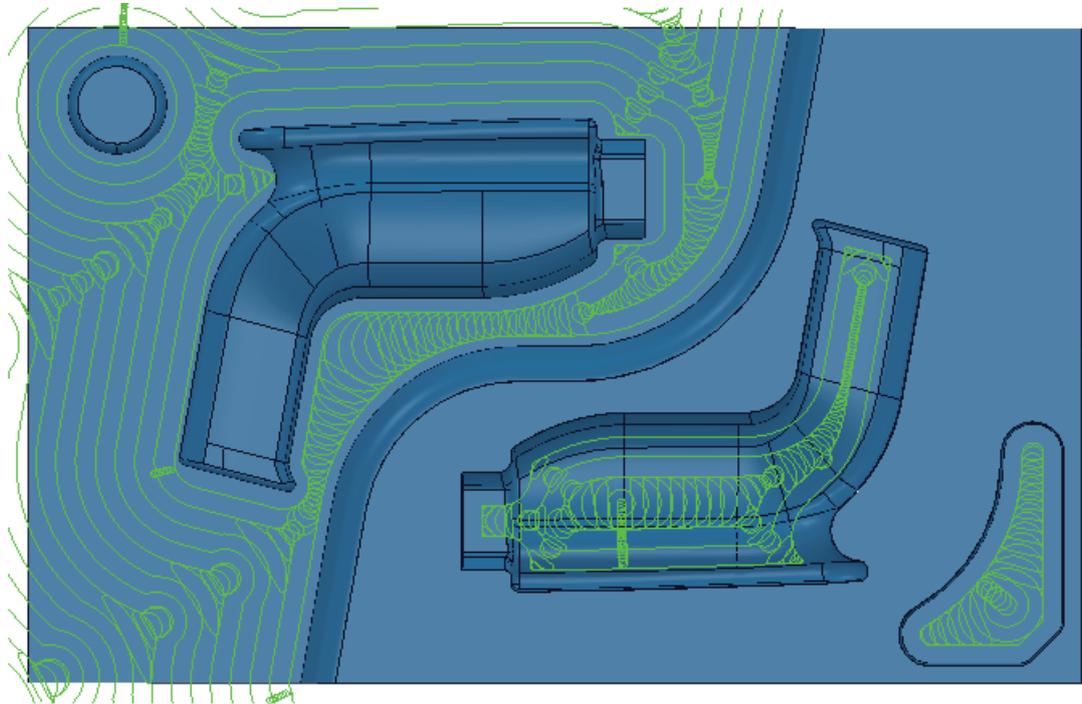


FeatureCAM has three major enhancement releases per year (R1, R2, and R3) issued once every four months. This section describes what's new for FeatureCAM 2014 R2 and covers four months of development. To see a year's worth of development, please also see the What's New topics for the previous two releases.

- **What's New in FeatureCAM 2014 R1** (see page 52)
- **What's New in FeatureCAM 2013 R3** (see page 85)

Vortex machining

Vortex machining is an area clearance strategy that rapidly removes material from a 3D part while controlling tool load. Vortex is best suited to solid carbide tools and is frequently used in combination with step cutting.



It is an offset-style toolpath and has these main features:

- The engagement angle never exceeds, by more than 15%, that produced by a straight line cut with a given stepover. This eliminates excessive tool load and all full-width cuts. This enables you to increase feed rates. For other area clearance toolpaths, the cutting values are based on the tool manufacturer's slot cutting parameters to ensure the tool can sustain full cutting engagement. As the tool approaches the maximum engagement angle for optimum machining, the toolpath changes to a trochoidal path to avoid tool overload.
- The machine tool almost always runs at the specified feed rate. With other area clearance toolpaths, the machine tool automatically slows down as it approaches a corner and the engagement angle increases. Vortex modifies the toolpath so the tool engagement angle is never exceeded and the machine tool achieves the specified feed rate. The only time the machine tool doesn't run at the specified feed rate is when the model geometry (a slot or corner) is smaller than the smallest radius that the machine can run at full speed.
- Vortex machining cuts with the side of the tool so it is designed for solid carbide tools, but you may be able to use other tools.

- Because FeatureCAM controls the tool engagement, you can increase the depth of cut, which minimizes machining time.
- Vortex machining is frequently used in combination with **Step cutting** to minimize terracing while maximizing the removal rate.
- Vortex toolpaths are automatically checked for safety. FeatureCAM checks for:
 - plunges into stock.
 - excess tool engagement.
 - excess depth of cut.
 - small arc movements.

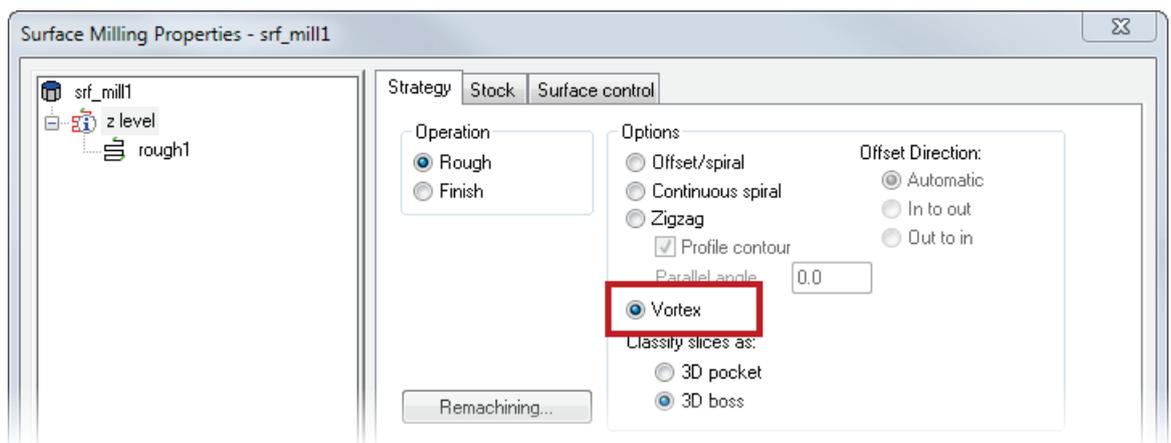
To maximize the benefits of Vortex machining:

- Configure the Vortex parameters to suit each machine tool.
- Use **Step Cutting** (see page 25) to minimize terracing caused by the increased depth of cut.

With optimum settings, Vortex machining greatly reduces machining times.

Creating a 3D Vortex toolpath (3D HSM)

To create a 3D Z-level roughing Vortex toolpath, select **Vortex** on the **Strategy** tab of the **Surface Milling Properties** dialog:



You must have the 3D HSM module to use Vortex for 3D Z-level roughing.

The following attributes are available for Vortex toolpaths in the **Feature Properties** dialog:

F/S tab:

- **Feed** — Enter the speed the tool cuts into the material. Vortex toolpaths are machined at this cutting feed rate almost all of the time.

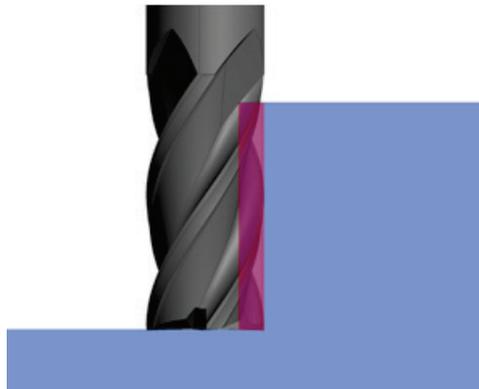
Milling tab:

- **Vortex min point spacing** — Enter the minimum point spacing at which the machine tool can move at the specified feed rate. If the machine tool has too many points to process, it cannot sustain the specified feed rate.
- **Vortex min radius** — Enter the minimum radius of the internal trochoids. Vortex toolpaths use trochoidal moves to maintain a constant feed rate. Higher feed rates require a larger minimum radius. If you do not override this value, a default value is used, which is suitable for a typical machine tool at the feed rate specified for the operation.
- **Vortex Z lift distance** — Enter a Z distance to lift the tool during trochoidal moves to avoid contact between the tool and the surface.

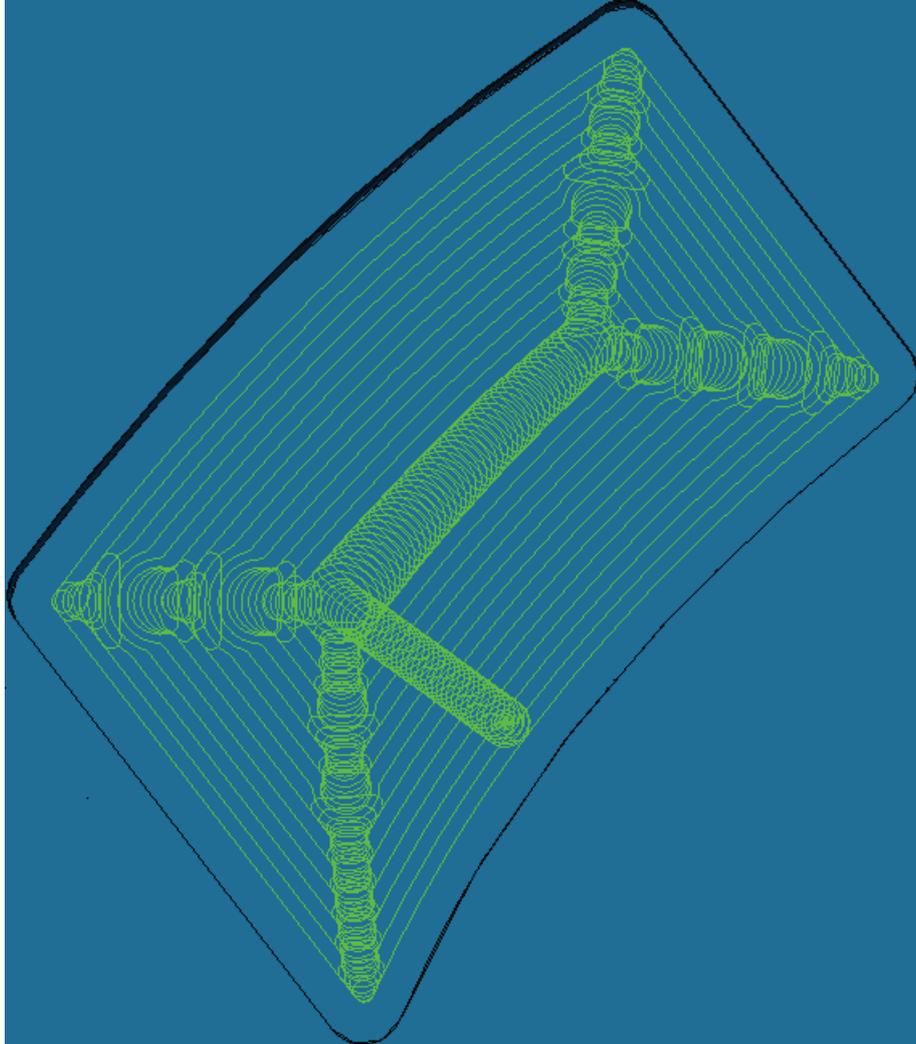
3D Vortex example (3D HSM)

This example shows you how to combine Vortex machining with step cutting to rapidly remove material.

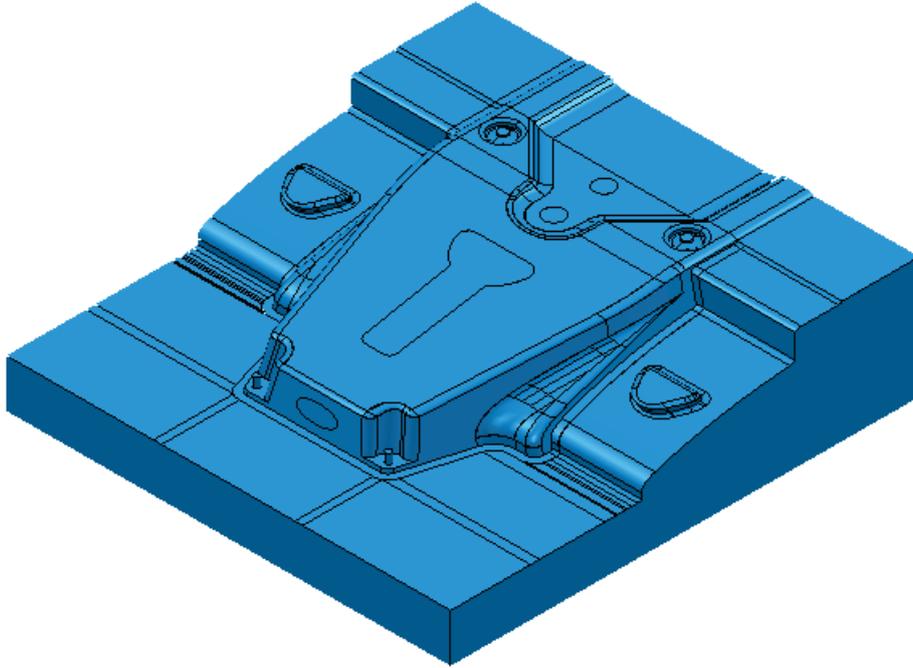
Because Vortex machining cuts with the side of the tool, it is designed for solid carbide tools, but there may be other types of tools suitable for Vortex. These tools work best when taking deep cuts with a relatively small stepover.



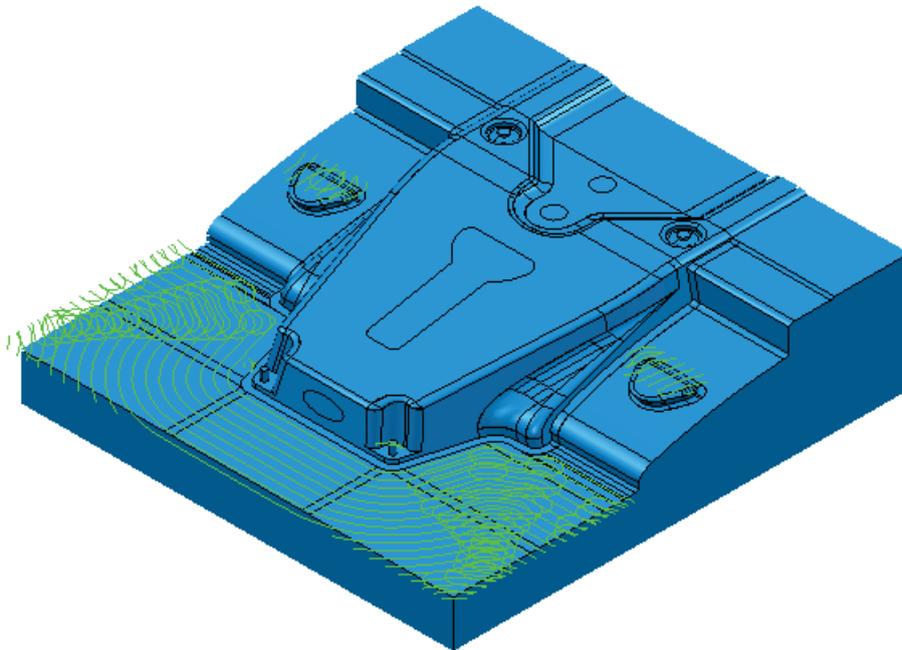
To machine effectively when taking large depths of cut, you must ensure the tool engagement angle never exceeds the specified value. This eliminates excessive tool load and all full-width cuts. FeatureCAM achieves this by introducing trochoidal moves to prevent the tool from exceeding the maximum tool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



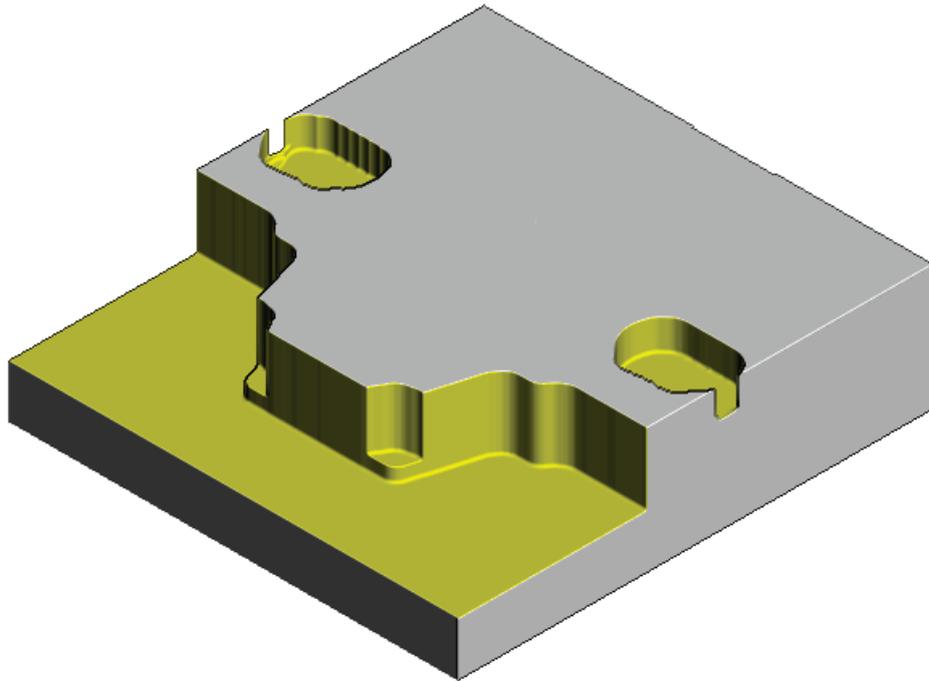
Using a 3D model:



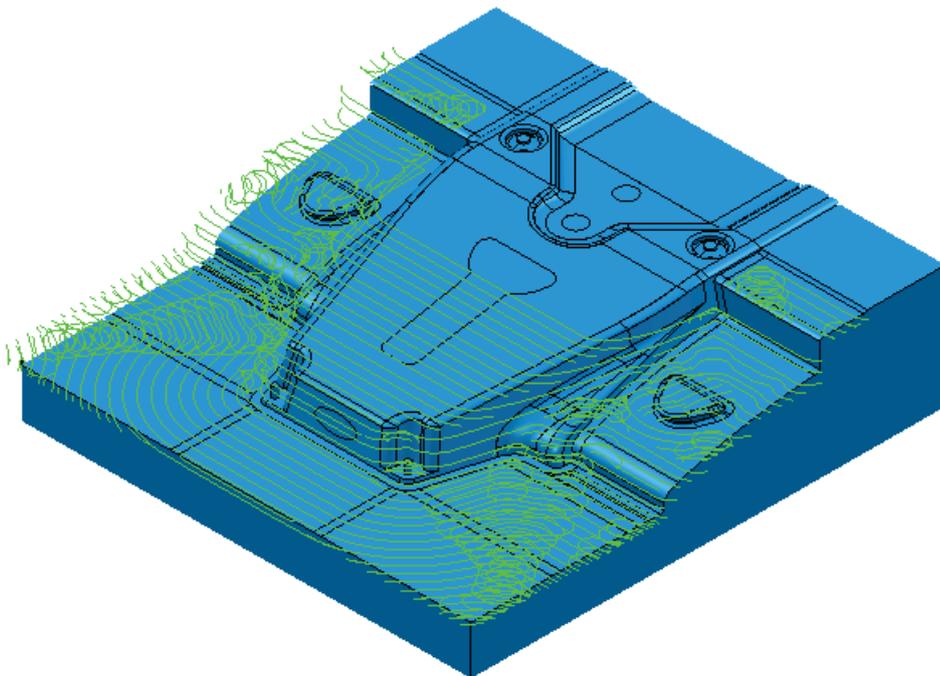
Creating a Vortex toolpath without step cutting.



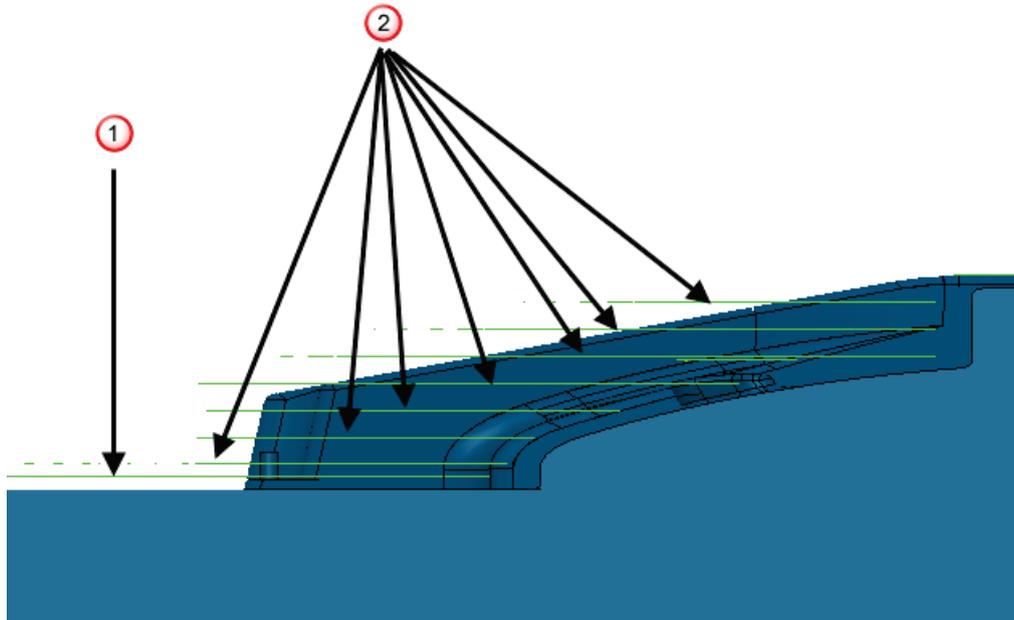
This removes vast quantities of material quickly, but leaves large terraces of unmachined stock on the part.



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



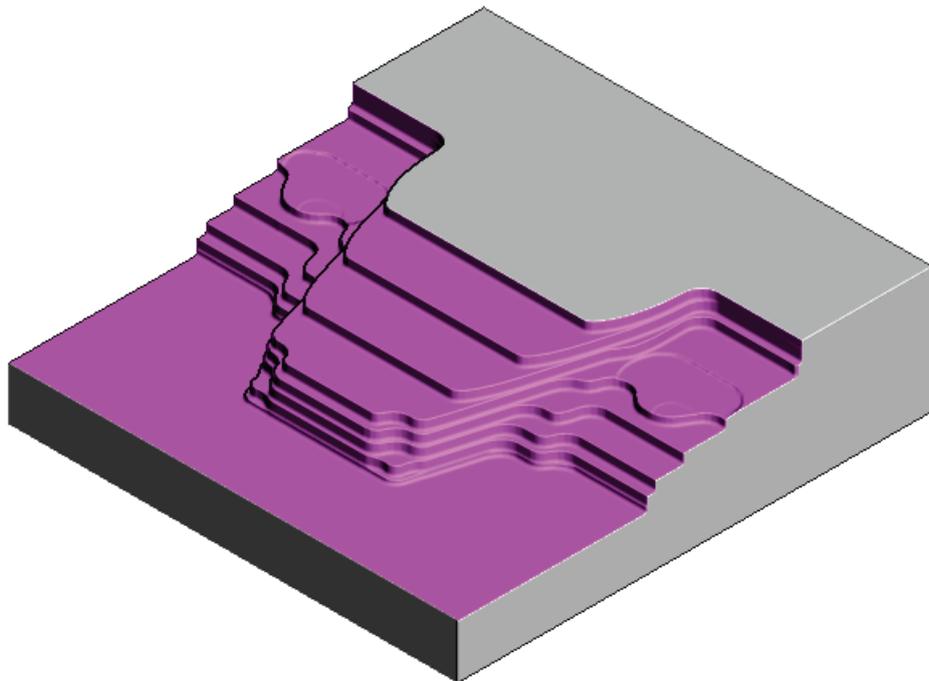
This adds extra machining slices up the part. Looking at a detail of the side view:



① Original Vortex pass

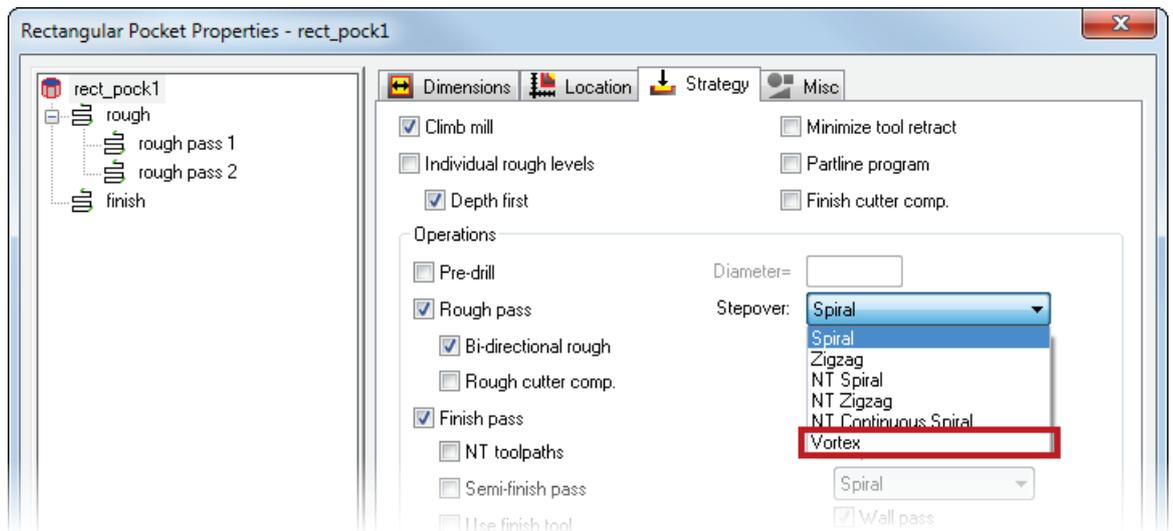
② Step cutting passes

It also machines more of the excess stock.



Creating a 2.5D Vortex toolpath (REC)

To create a 2.5D Vortex toolpath, select **Vortex** from the **Stepover** list on the **Strategy** tab of the **Feature Properties** dialog:



You must have the FeatureRECOGNITION module to use Vortex for 2.5D roughing.

The 3D HSM and 3D MX modules include FeatureRECOGNITION, 3D LITE does not.



You can use Vortex to machine most 2.5D feature types.

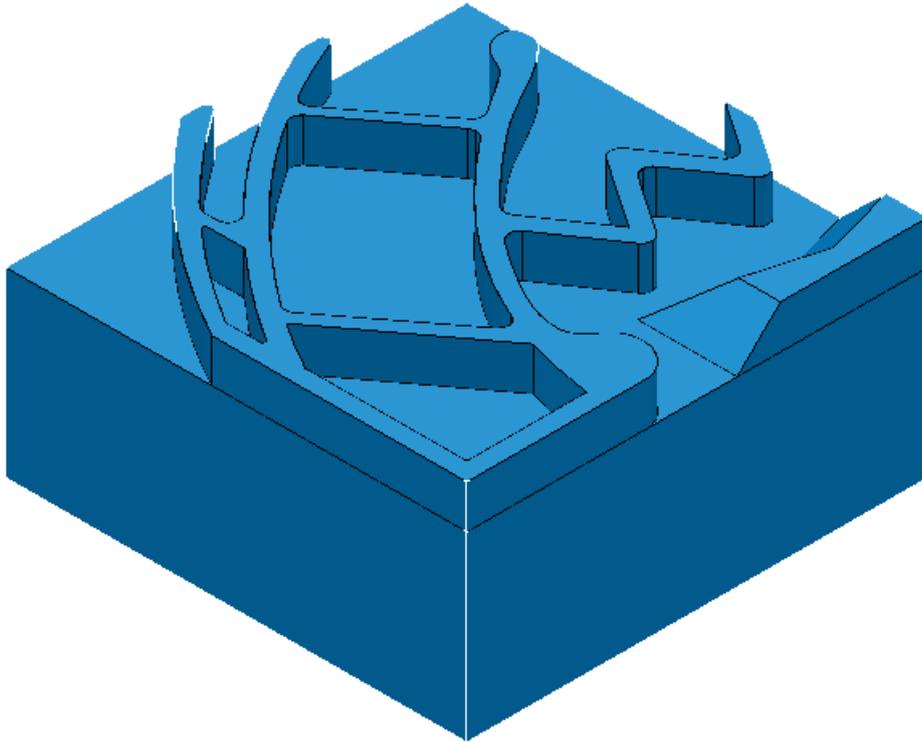
You can use Vortex to machine a Side feature only if the curve is closed. If the curve is open and you select a **Stepover** of **Vortex**, **NT Spiral** is used instead.

2.5D Vortex example (REC)

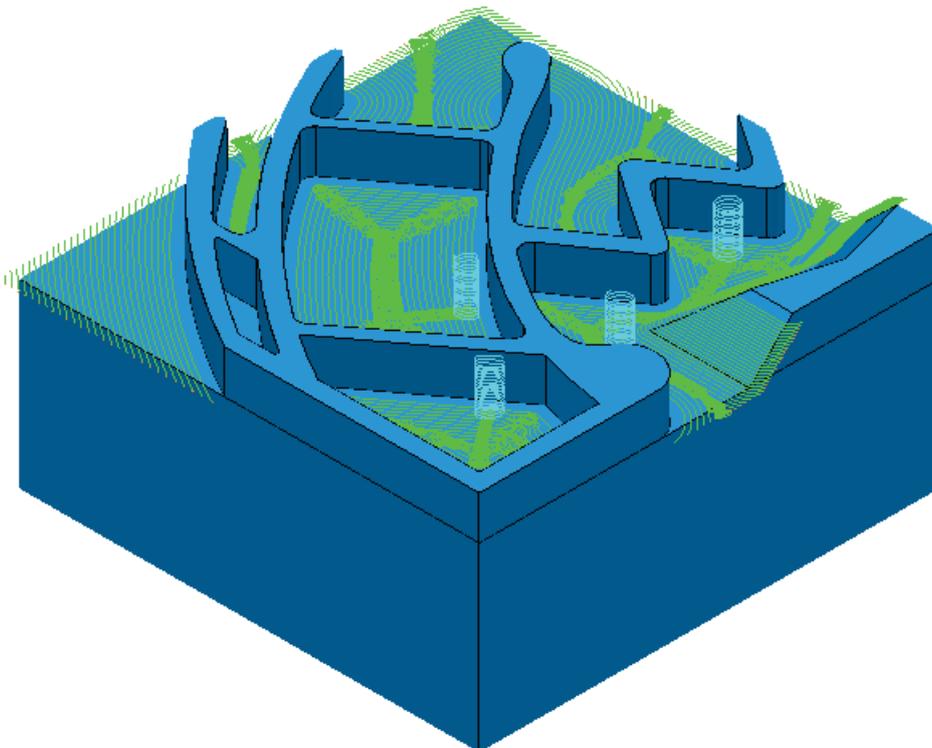
This example shows how Vortex machines pockets, channels, and narrow corners.

For more information on the general principles of Vortex machining, see the 3D Vortex and step cutting example (see page 25).

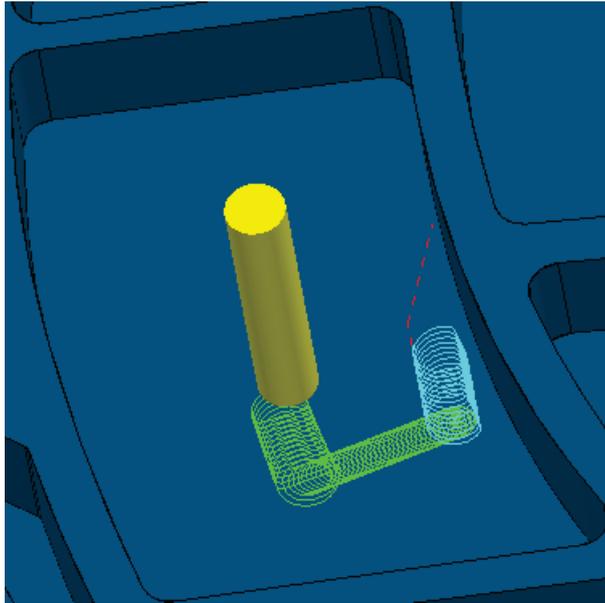
Using a 2.5D model:



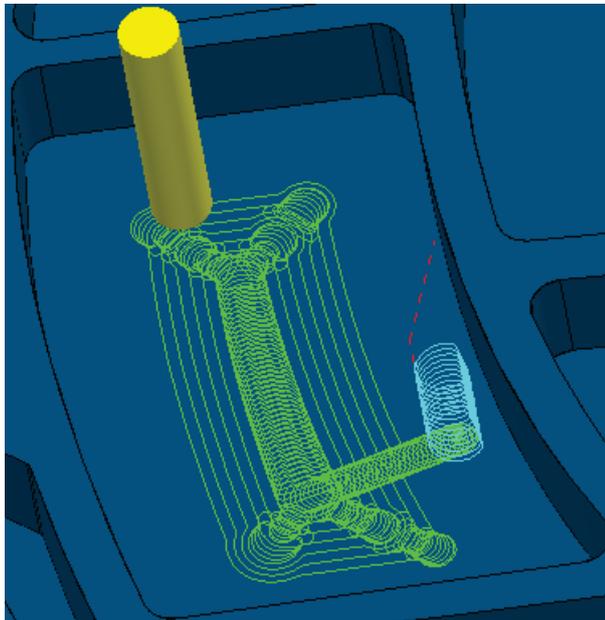
Create a Vortex toolpath:



For pockets, the tool spirals down into the pocket before using trochoidal paths over the full-width cuts.



On completion of the initial full-width cut, the trochoids are placed in the corners where the maximum tool engagement angle would otherwise be exceeded.



User Interface and work-flow improvements

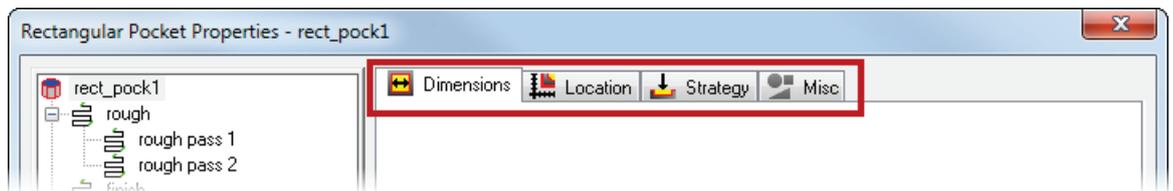
FeatureCAM 2014 R2 includes these User Interface and work-flow improvements:

- There are icons in the **Feature Properties** dialog (see page 33).

- The cursor changes in the **Operation List** to show when additional actions are available (see page 33).
- You can change the Post number of a Turret, and the new number is displayed throughout FeatureCAM (see page 34).
- You can quickly rename items in the **Part View** (see page 35).
- You can pick values from the graphics window in the **UCS Translate** dialog (see page 35).
- The **Tool Manager** remembers the previously selected **Tool Group** (see page 36).
- The edit boxes in the dialog bar are wider (see page 37).
- Delcam Exchange is installed during the standard FeatureCAM installation (see page 37).
- You can remove operations from a stock model (see page 37).

Feature Properties dialog icons

In FeatureCAM 2014 R2, the feature level tabs have icons in the **Feature Properties** dialog:

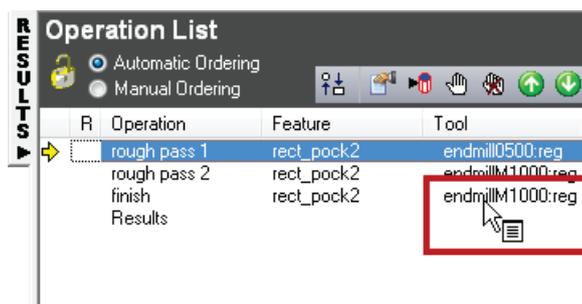


Operation List cursor changes

In FeatureCAM 2014 R2, when you move the cursor over an item in the **Op List** tab of the **Results** window, the cursor changes shape if additional actions are available.

If the cursor changes to , you can double-click the item to display its associated dialog.

For example, if you move the cursor over a tool name in the **Operation List**, the cursor changes:

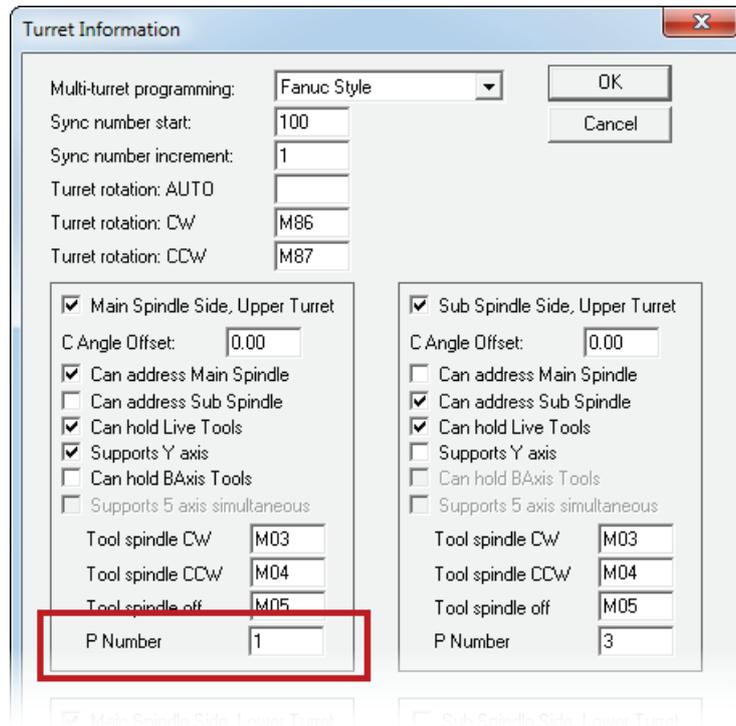


Double-click the tool name to display the **Tools** tab of the **Feature Properties** dialog for the operation.

Turret numbering

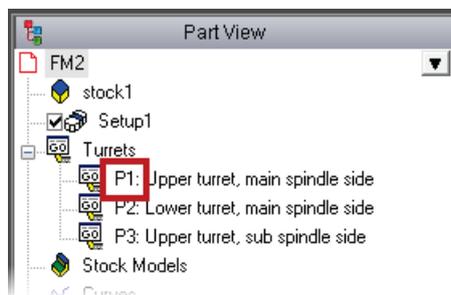
If you change the post number of a turret in FeatureCAM 2014 R2, the new value is displayed throughout FeatureCAM so you can easily identify turrets.

Enter a **P Number** in the **Turret Information** dialog:

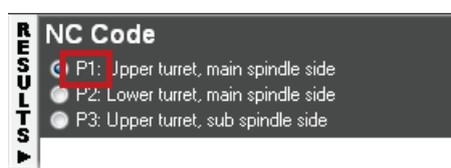


The new post number is displayed in the turret's name in the following places:

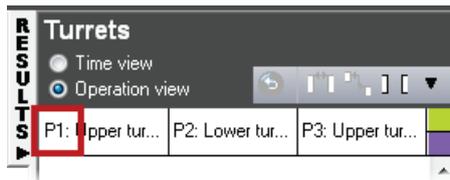
- In the **Part View** panel:



- On the **NC Code** tab of the **Results** window:



- On the **Turrets** tab of the **Results** window:

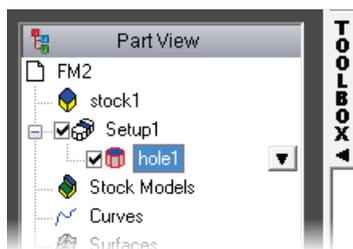


Renaming items in the Part View

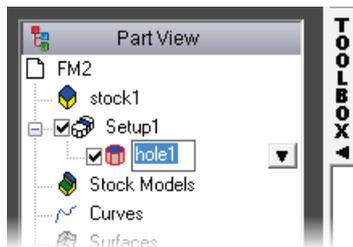
In FeatureCAM 2014 R2, you can rename items more easily in the **Part View** panel by clicking on a selected item:

To quickly rename an item:

- 1 Select an item in the **Part View** panel:



- 2 Click on the selected item's name to make it editable:



- 3 Enter a new name for the item.
- 4 To accept the changes press the **Enter** key or click in the FeatureCAM window.

To cancel the changes press the **Esc** key.

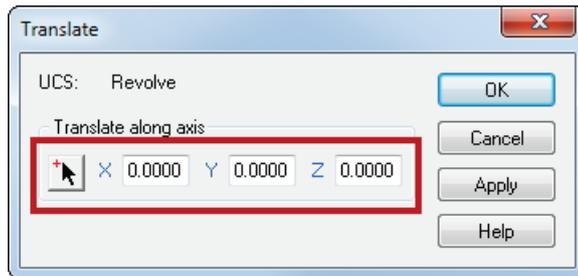


*Double-clicking an item in the **Part View** panel displays the item's dialog, and does not rename the item.*

UCS Translate pick locations

In FeatureCAM 2014 R2, in the **UCS Translate** dialog, you can pick the X, Y, and Z coordinates individually from the graphics window.

The X, Y, and Z labels are blue:



Click a blue label to minimize the dialog, then select a point in the graphics window.

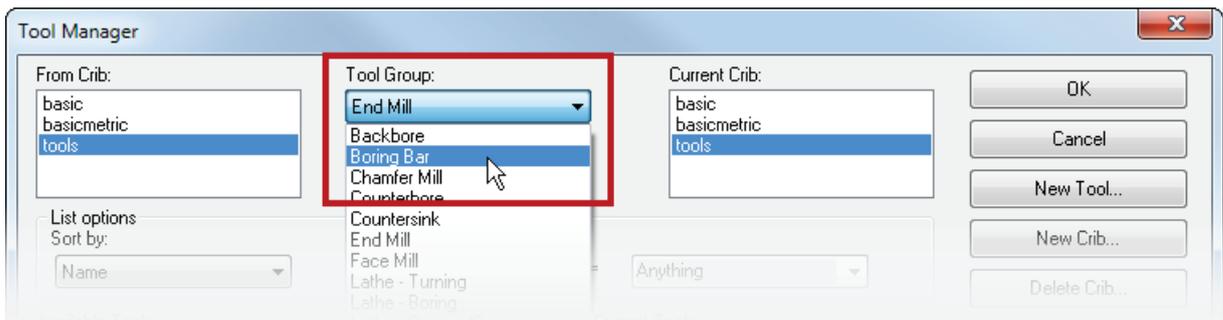
The coordinate of the point is given to the selected item.

For example, this is useful if you want to align a UCS with the center of a cylindrical face, with the depth along the cylinder (Z) defined using a different face.

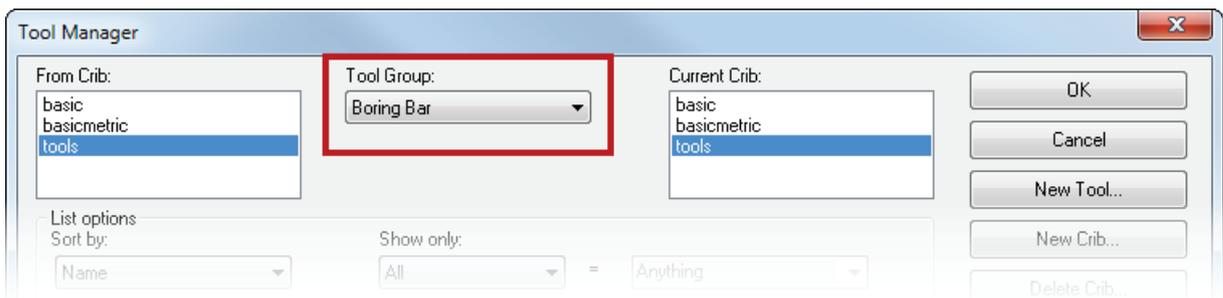
Tool Manager

In FeatureCAM 2014 R2, the **Tool Manager** dialog remembers the previously selected **Tool Group**.

If you select a **Tool Group** in the **Tool Manager** dialog, then close the dialog:



The next time you display the **Tool Manager** dialog, the previously selected **Tool Group** is still selected:





*This applies only if you display the **Tool Manager** dialog by selecting **Manufacturing > Tool Manager** from the menu. If you display the **Tool Manager** dialog by clicking **Tool manager**  on the **Tools** tab of the **Feature Properties** dialog, the feature's default **Tool Group** is selected.*

Expanded dialog bar edit boxes

In FeatureCAM 2014 R2, the edit boxes in the dialog bar are wider:



This enables you to display more decimal places.

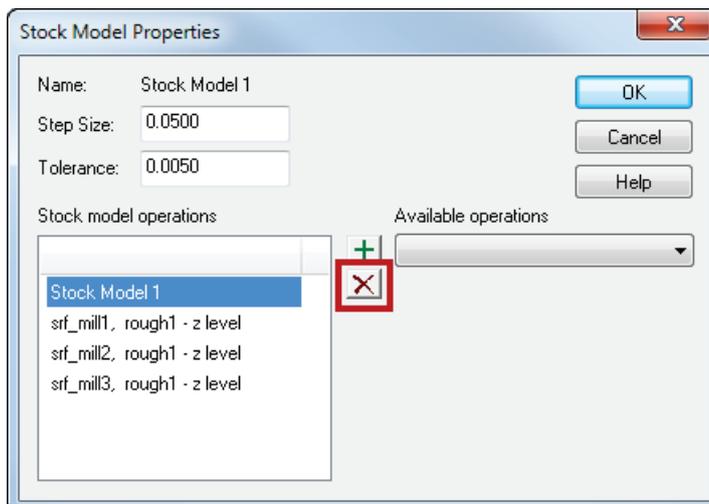
Delcam Exchange installation

In FeatureCAM 2014 R2, Delcam Exchange is installed during the standard FeatureCAM installation.

Remove stock model operations

In FeatureCAM 2014 R2, you can remove operations from a stock model.

There is a new button in the **Stock Model Properties** dialog:



Select an operation in the **Stock model operation** list and click  to remove it from the stock model. The operation is added to the **Available operations** list.

2.5D Milling improvements

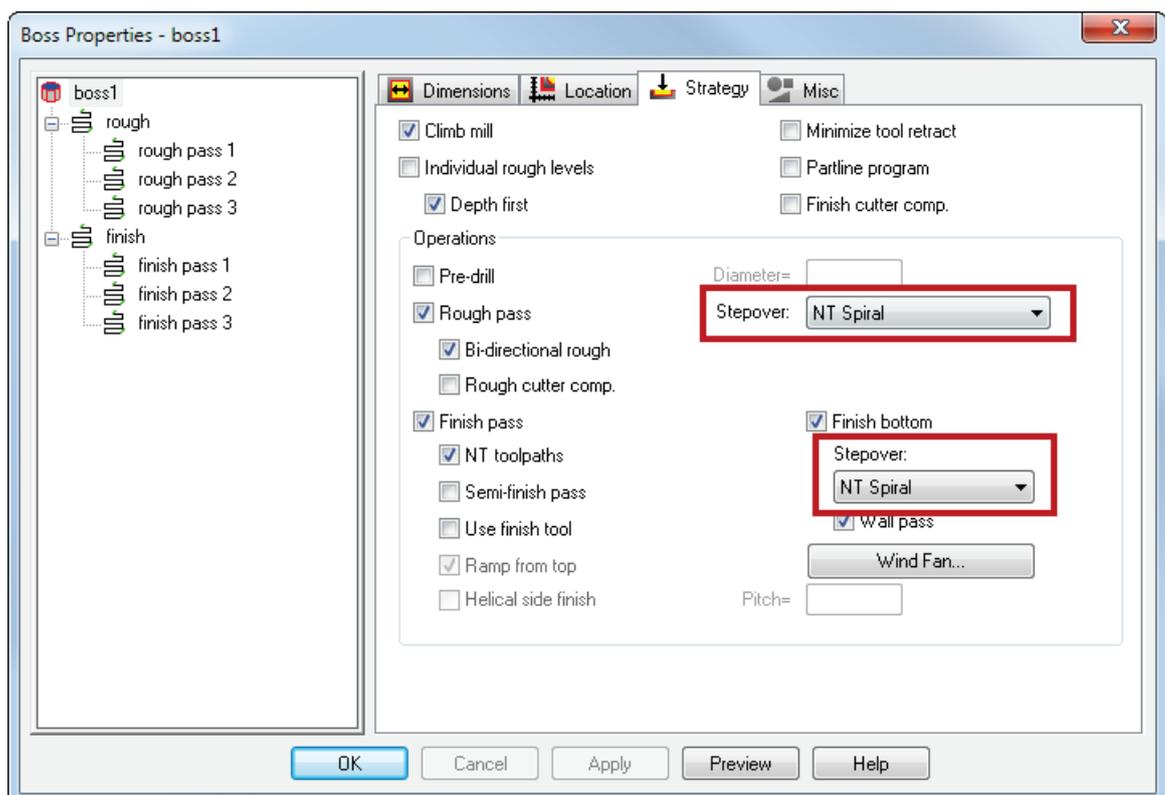
FeatureCAM 2014 R2 includes these 2.5D Milling improvements:

- You can apply an NT stepover type to multiple passes of a feature (see page 38).
- You can change the default ratio of **Pitch to Thread Height** for Thread Milling features (see page 39).
- You can machine the bottom of a Counterbore Hole feature on the finish pass, and leave a **Bottom finish allowance** after the roughing pass (see page 40).
- You can enable cutter compensation separately for a feature's operations (see page 41).
- You can select **Partline program** and **Finish cutter comp.** independently (see page 42).

NT toolpaths for multiple diameters

In FeatureCAM 2014 R2, you can apply an NT stepover type to all roughing and finishing passes of a feature.

Select an NT **Stepover** on the feature-level **Strategy** tab of the **Feature Properties** dialog to apply it to all roughing or finishing passes of the feature:





Previously, selecting an NT toolpath type would apply it to the first pass only.

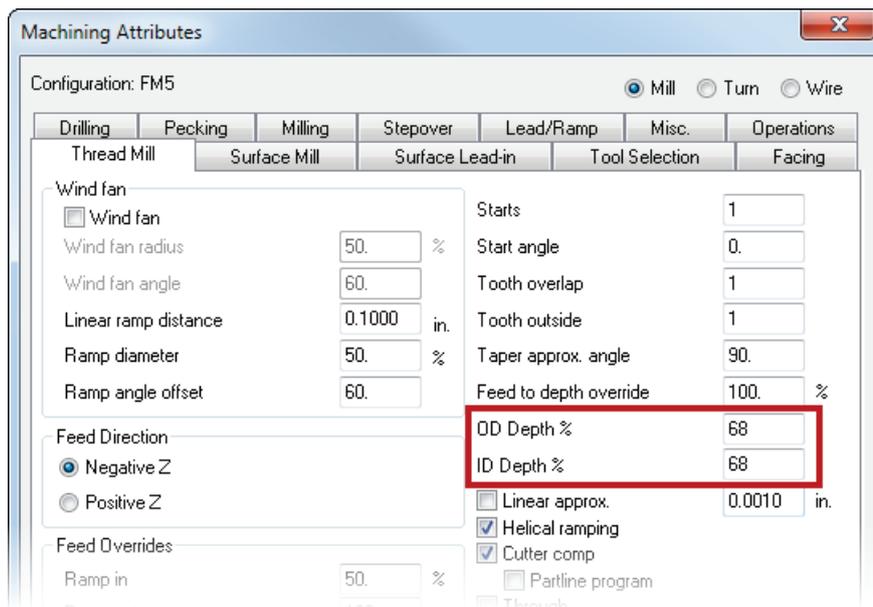


The **Stock Model Settings** dialog is available for the remachining passes on the **Milling** tab.

Thread Milling ID/OD depth %

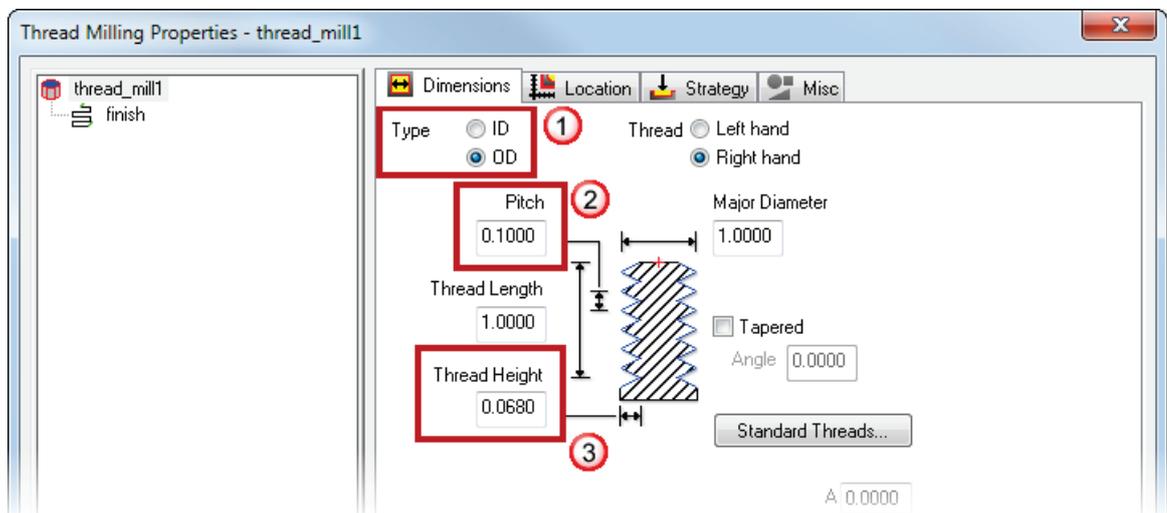
In FeatureCAM 2014 R2, you can change the default ratio of **Pitch** to **Thread Height** for Thread Milling features.

There are new options on the **Thread Mill** tab of the **Machining Attributes** dialog, **OD Depth %** and **ID Depth %**:



The default value for **OD Depth %** and **ID Depth %** is 68.

On the **Dimensions** tab of the **Thread Milling Properties** dialog:



The **Thread Height** ③ is calculated as a percentage of the **Pitch** ②, based on the **OD Depth %** or **ID Depth %** (depending on the **Type** selected).

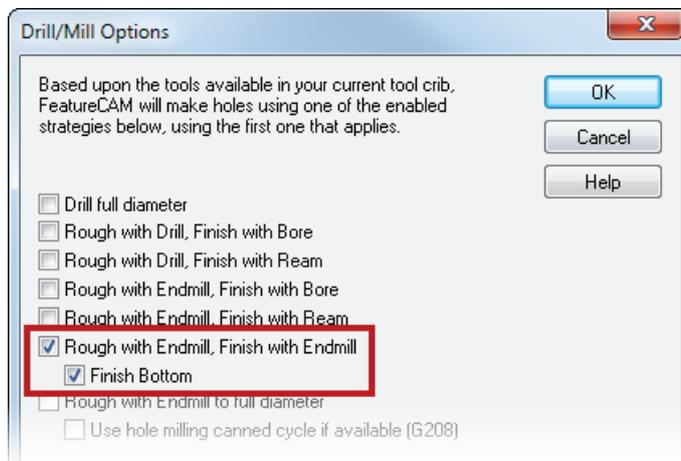
For example, if you enter an **OD Depth %** of 68, select a **Type** of **OD**, and enter a **Pitch** of 0.1, the **Thread Height** is updated to 68% of 0.1 = **0.068**.

This also applies when creating a Thread Milling feature using the **New Feature** wizard.

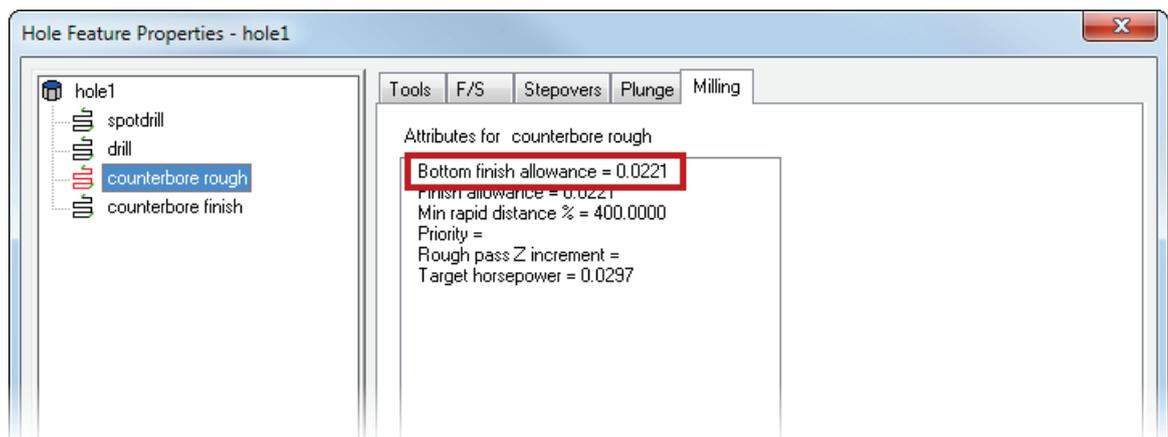
Bottom finish allowance for Counterbore features

In FeatureCAM 2014 R2, you can machine the bottom of a Counterbore Hole feature on the finish pass, and leave a **Bottom finish allowance** after the roughing pass.

In the **Drill/Mill Options** dialog, if **Rough with Endmill, Finish with Endmill** is selected, you can select **Finish Bottom**:



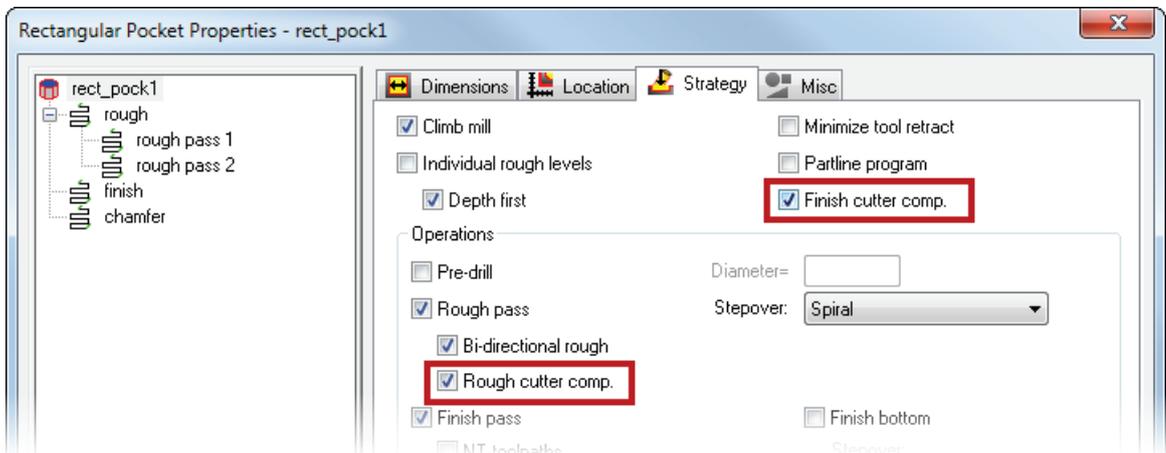
When selected, the bottom of the Counterbore feature is machined on the finish pass, and the **Bottom finish allowance** option is available on the **Milling tab** of the **Hole Feature Properties** dialog:



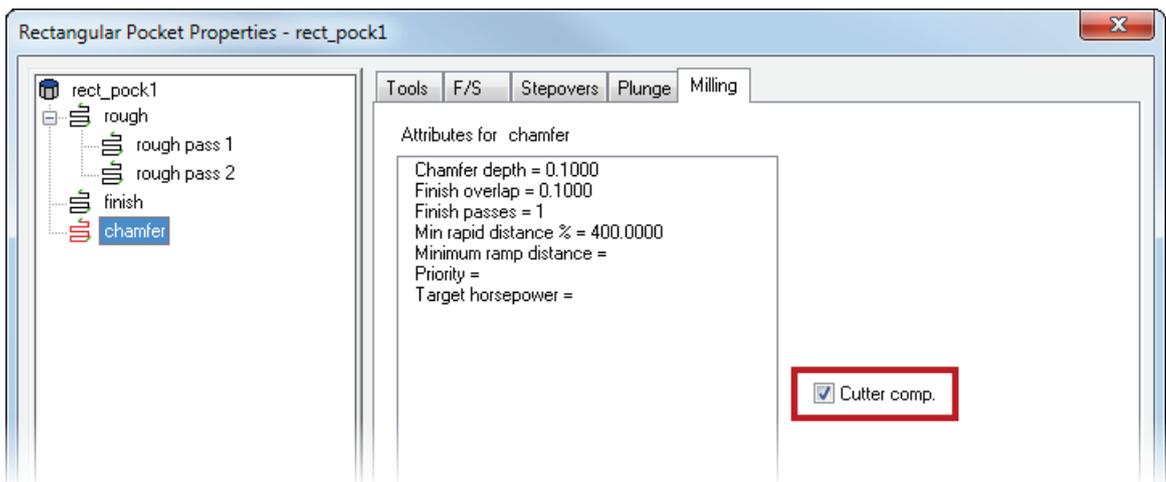
Separate cutter compensation

In FeatureCAM 2014 R2, you can enable cutter compensation separately for a feature's operations.

Selecting **Finish cutter comp.** or **Rough Cutter comp.** on the **Strategy** tab of the **Feature Properties** dialog applies cutter compensation to the finish or rough pass only:

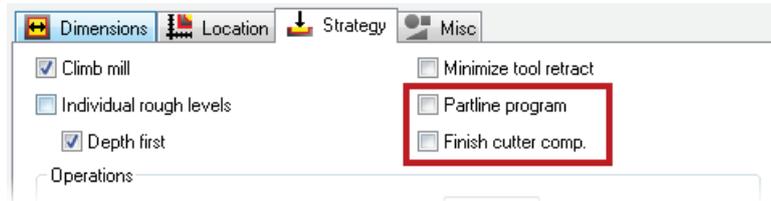


To enable cutter compensation for an operation, such as a chamfer, select **Cutter comp.** on the operation's **Milling** tab:



Finish cutter comp and Partline program independence

In FeatureCAM 2014 R2, you can select **Partline program** and **Finish cutter comp.** independently on the **Strategy** tab of the **Feature Properties** dialog.



This enables you to use partline program when cutter compensation is used for an operation in a feature, such as a chamfer, but not for the finishing operation.



*If cutter compensation is not enabled for any of the operations in a feature, selecting **Partline program** does not affect the NC Code.*

3D Milling improvements

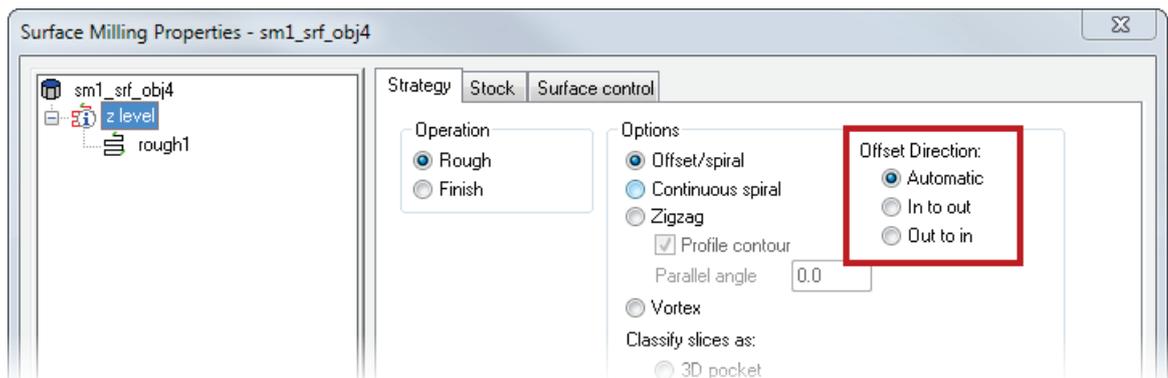
FeatureCAM 2014 R2 includes these 3D Milling improvements:

- You can specify the **Offset Direction** of a 3D offset toolpath (see page 42).
- You can perform radial plunge milling (see page 44).

Offset Direction

In FeatureCAM 2014 R2, you can select whether a toolpath cuts from the inside out or from the outside in.

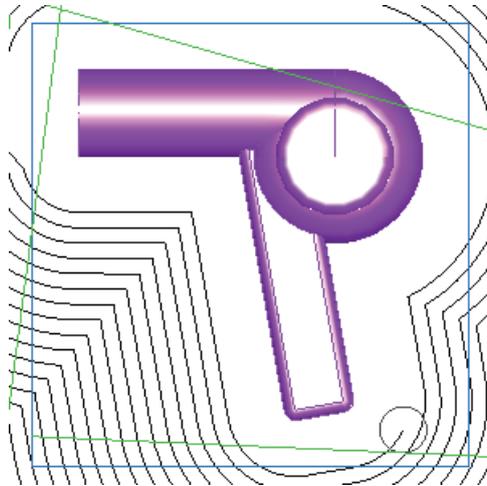
There are new **Offset Direction** options on the operation-level **Strategy** tab of the **Surface Milling Properties** dialog for Offset toolpaths:



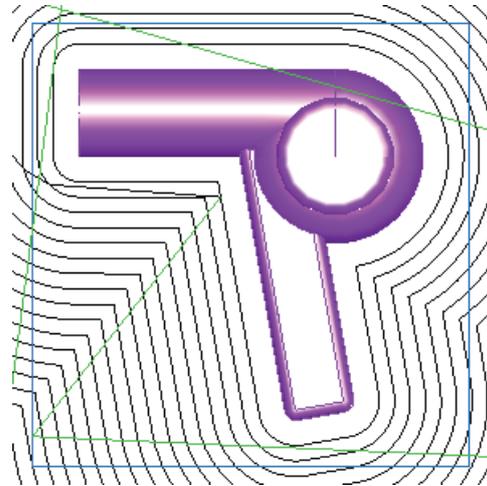
Select an **Offset Direction** from the following:

- **Automatic** — The tool cuts from the outside to the inside of the stock in a continuous radial movement. This is the default option.

During cutting:

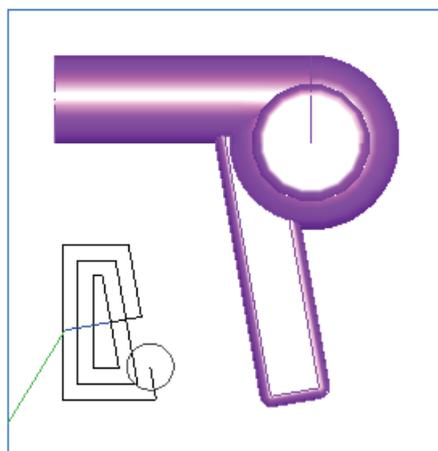


Cutting complete:

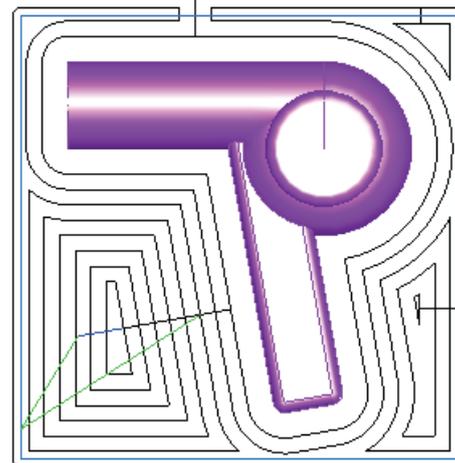


- **In to out** — The tool plunges into the stock and cuts outwards.

During cutting:



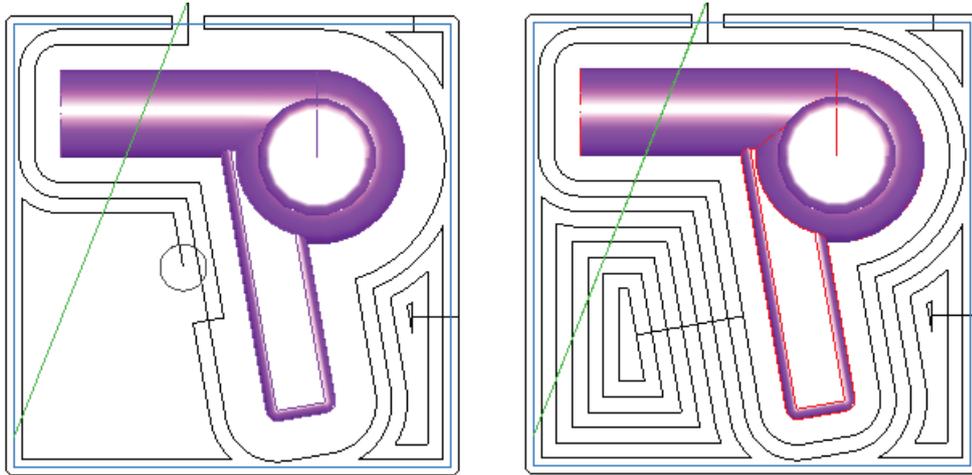
Cutting complete:



- **Out to in** — The tool cuts from the outside to the inside of the stock. The tool outlines the surfaces and cuts large sections last.

During cutting:

Cutting complete:



Radial Plunge Roughing

In FeatureCAM 2014 R2, you can create a radial plunge milling toolpath using the radialPlungeMill add-in.

To create a radial plunge milling toolpath:

- 1 Load the [radialPlungeMill.bas](#) add-in.
- 2 Create a surface milling feature using the surfaces you want to machine.
- 3 In the **Part View** panel, deselect the surface milling feature to exclude it.
- 4 In the **New Feature** wizard, select **User** in the **From Feature** section and click **Next**.
- 5 Select **RadialPlungeRough 1.0** in the **Registered features** list and click **Next**.
- 6 Select **Srf mill feature**, enter the name of the surface milling feature in the **New Value** field and click **Set**.
- 7 Follow the steps in the rest of wizard to modify the location and tool selection.
- 8 Click **Finish** to close the dialog.

New radial plunge milling features are given the name **udf_radial_plunge_rough** and numbered sequentially.

Turning improvements

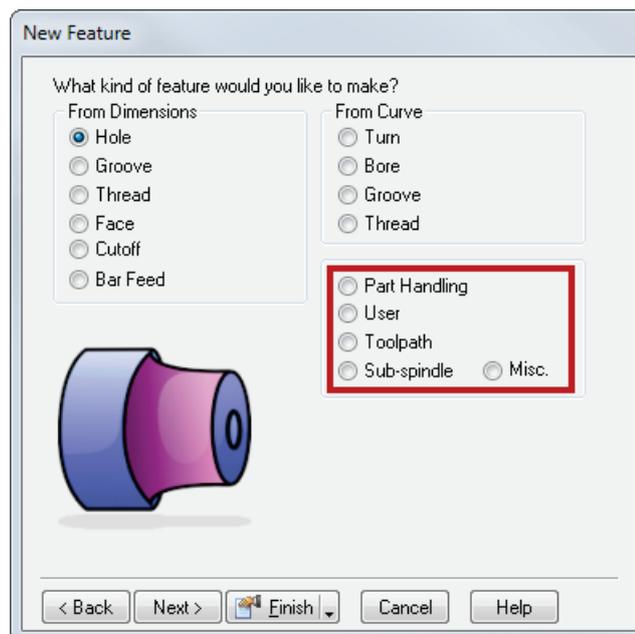
FeatureCAM 2014 R2 includes this turning improvement:

- Turning transfer operations are improved (see page 45).

Transfer operation changes

The Turning **Sub-spindle** features are being deprecated, and will be removed in a future release. For transfer operations use **Part Handling** features.

In the **New Feature** wizard, the recommended **Part Handling** option is displayed at the top of the list:



Wire EDM improvements

FeatureCAM 2014 R2 includes this Wire EDM improvement:

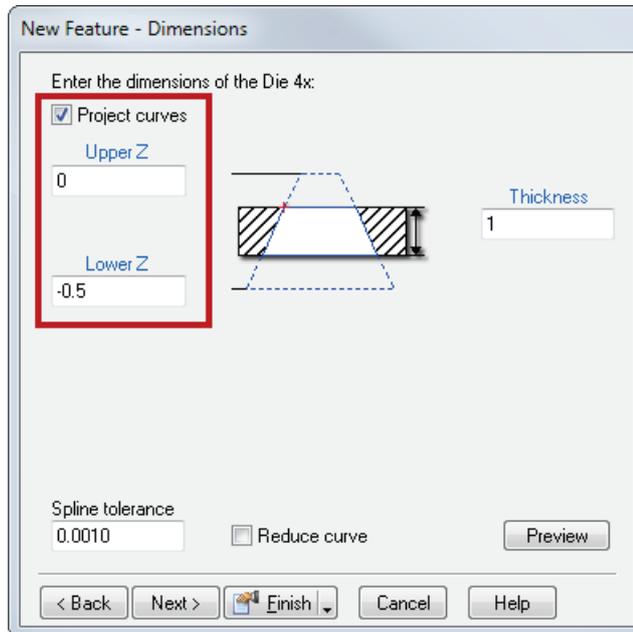
- You can project curves to different Z levels when creating Wire EDM features (see page 45).

Project curves

In FeatureCAM 2014 R2, you can project feature curves to different Z levels when using FeatureRECOGNITION to recognize a 4-Axis Wire feature from a solid.

This enables you to easily specify whether the feed and speed information is calculated from the feature height or the stock height.

There are new options on the **Dimensions** tab of the **Wire Feature Properties** dialog and the **Dimensions** page of the **New Feature** wizard for Wire EDM features:



Thickness — Enter the thickness of the feature, or click the blue label and select two points in the graphics window. The thickness is the depth below the feature curve.

Project curves — Select this option to specify the upper and lower Z levels of the feature:

- When deselected, feed and speed information is calculated based on the **Thickness** value.
- When selected, feed and speed information is calculated based on the difference between the **Upper Z** and **Lower Z** values, and the **Thickness** is ignored.

Upper Z — Enter the Z level of the top of the feature, or click the blue label and select a point in the graphics window.

Lower Z — Enter the Z level of the bottom of the feature, or click the blue label and select a point in the graphics window.



*To calculate feed and speed information based on the stock thickness, select the upper and lower stock boundaries as the **Upper Z** and the **Lower Z** values.*

Machine design improvements

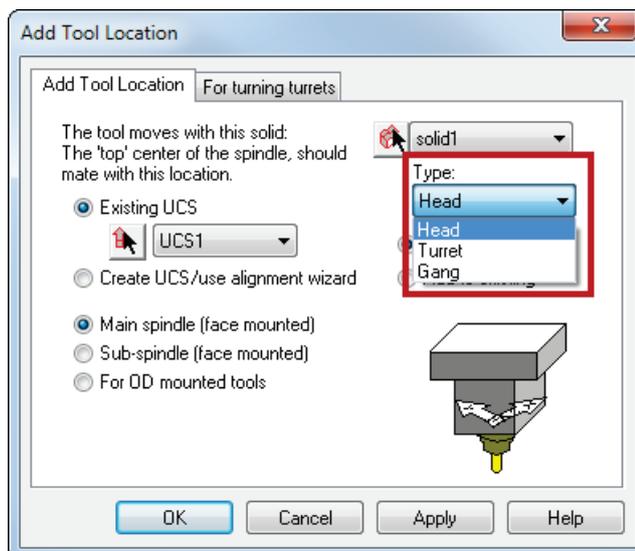
FeatureCAM 2014 R2 includes these Machine Design improvements:

- You can specify the locations of linear gang tools (see page 47).
- You can send Machine Design files (.md) when sending other files (see page 48).

Add Tool Location dialog improvements

In FeatureCAM 2014 R2, there are additional options in the **Add Tool Location** dialog to make machine design for gang tooling easier (where the tool block holds tools in a linear pattern).

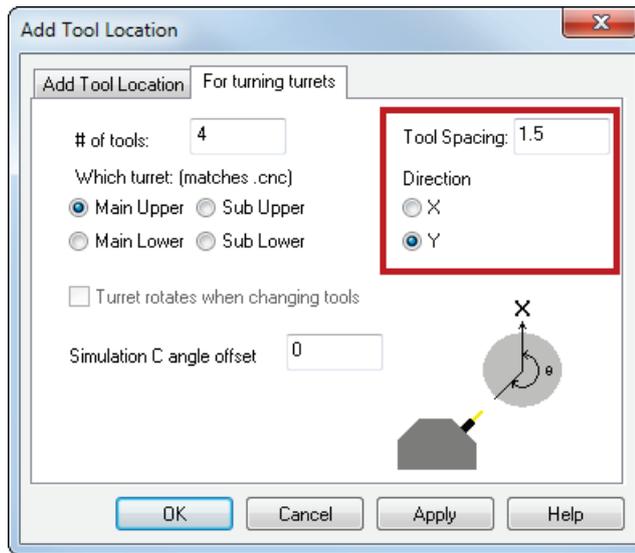
On the **Add Tool Location** tab, you can select the **Type**:



Select from:

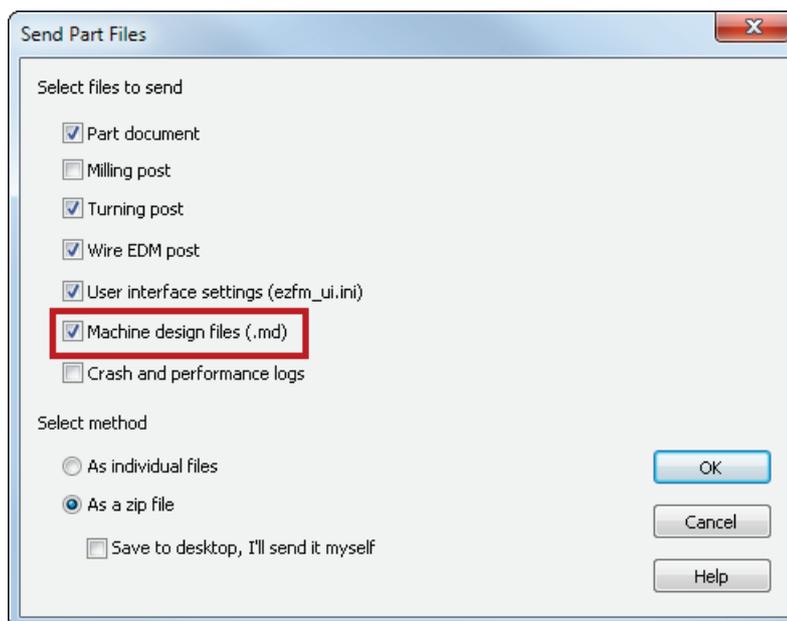
- **Head** — Add a single head tool.
- **Turret** — Add multiple tools rotated about the LCS of the selected solid.
- **Gang** — Add tools in a linear pattern.

Selecting a **Type of Gang** on the **Add Tool Location** tab enables you to enter the **Tool Spacing** and select the **Direction** on the **For turning turrets** tab:



Send Machine Design files

In FeatureCAM 2014 R2, there is an option in the **Send Part Files** dialog to send **Machine design files (.md)**:



This enables you to send the machine design files when sending other files.

This is useful when reporting issues, especially for 5-axis and turn-mill parts.

Machine design files (.md) is selected by default.

Import improvements

FeatureCAM 2014 R2 includes these Import improvements:

- You can import SolidWorks 2014 files (see page 49).
- **SolidWorks Hole Recognition** recognizes dowel holes in SolidWorks models, and machines them using a reamed hole operation (see page 49).
- You can import SolidEdge ST5 and ST6 files (see page 49).

SolidWorks 2014 files

In FeatureCAM 2014 R2, you can import SolidWorks 2014 files.



This is dependant on the version of the `swdocumentmgr.dll` file, not the FeatureCAM version.

Dowel holes

In FeatureCAM 2014 R2, **SolidWorks Hole Recognition** recognizes dowel holes in SolidWorks models, and machines them using a reamed hole operation.

To recognize a dowel hole:

- 1 On the **SolidWorks Hole Recognition** page of the **Import** wizard, select **Use SolidWorks to extract information about the hole types and dimensions**.
- 2 Click **Finish** to close the **Import** wizard.



Automatic Feature Recognition does not use a Ream operation for dowel holes imported from SolidWorks models.

SolidEdge ST5 and ST6 files

In FeatureCAM 2014 R2, you can import SolidEdge ST5 and ST6 files.

XBUILD improvements

FeatureCAM 2014 R2 includes this XBUILD improvement:

- The CNC error message displays the line of code which causes the error (see page 50).

Improved Error message

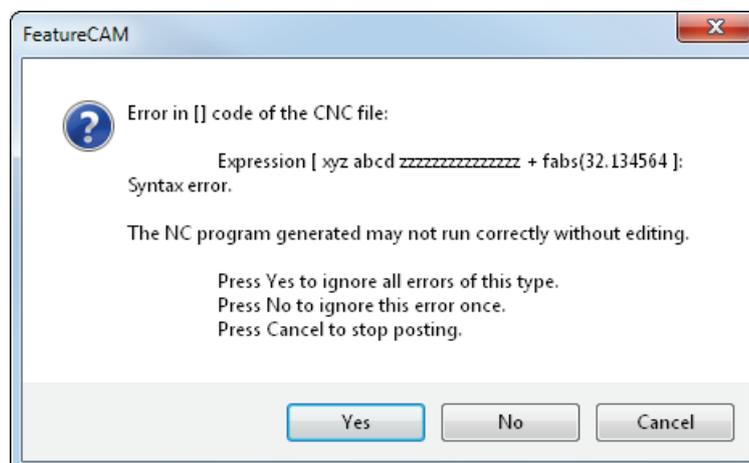
In FeatureCAM 2014 R2, the CNC error message displays the line of code which causes the error, so you can find and fix the problem more easily.

For example, if you enter the following code in the **Program Start** Format of a CNC file:

```
PROGRAM-START<EOB>
```

```
[ : xyz abcd zzzzzzzzzzzzzzzzzzz + fabs(32.134564 ) ]
```

The following error message is displayed when you test the CNC file:



Add-in Improvements

FeatureCAM 2014 R2 includes this Add-in improvement:

- In the Setup Sheet add-in, you can display whether cutter compensation is enabled for each operation (see page 51).

Setup Sheet Cutter Compensation

In FeatureCAM 2014 R2, in the Setup Sheet add-in, you can display whether cutter compensation is enabled for each operation.

Add the variable tag `{operation.cutter_comp}` to the Setup Sheet template to display whether cutter compensation is enabled for each operation:

Setup : Setup1					
Setup Origin	0.0000, 0.0000, 0.0000				
Feature Name	Feature Type	Operation	Tool Number	Tool Name	Cutter Comp.
rect_pock1	Rectangular Pocket	Rough Taper	1	endmill0500:reg	Y
rect_pock2	Rectangular Pocket	Rough Taper	1	endmill0500:reg	N
rect_pock1	Rectangular Pocket	Rough Taper	2	endmillM1000:reg	Y
rect_pock1	Rectangular Pocket	Finish Taper	2	endmillM1000:reg	N
rect_pock2	Rectangular Pocket	Rough Taper	2	endmillM1000:reg	N
rect_pock2	Rectangular Pocket	Finish Taper	2	endmillM1000:reg	N
rect_pock1	Rectangular Pocket	Finish	3	chamfer0312	Y



*Click **Help** in the **Setup Sheet Options** dialog to display the **Setup Sheet Help**.*

What's New in FeatureCAM 2014 R1

FeatureCAM 2014 R1 offers all of the original features of FeatureCAM 2013 R3, but with numerous improvements. The most significant improvements are detailed in the topics that follow.



FeatureCAM has three major enhancement releases per year (R1, R2, and R3) issued once every four months. This section describes what's new for FeatureCAM 2014 R1 and covers four months of development. To see a year's worth of development, please also see the What's New topics for the previous two releases.

- **What's New in FeatureCAM 2013 R3** (see page 85)
- **What's New in FeatureCAM 2013 R2** (see page 99)

User interface improvements

FeatureCAM 2014 R1 includes these user interface improvements:

- There is more flexibility with naming features (see page 53).
- You can now view and edit **Layers** (see page 53) in the **Part View**.
- There are additional **Part Documentation** (see page 56) fields.
- You can now send crash and performance logs (see page 58) by email.
- You can change the location of a Setup aligned with a revolved surface to the opposite end (see page 57).
- The more modern **Shaded Grey** (see page 59) user interface style is now the default.
- The tool display window (see page 59) is now more attractive thanks to anti-aliasing.

- Licensed product components are activated by default (see page 60).
- There are new license expiry warnings (see page 60).

Feature names

In FeatureCAM 2014 R1, you can use numbers at the start of feature names and include other characters such as spaces and %, for example **10mm hole pattern** or **pocket1 - 10% overload**.



You must ensure that your post supports the characters that you want to use.

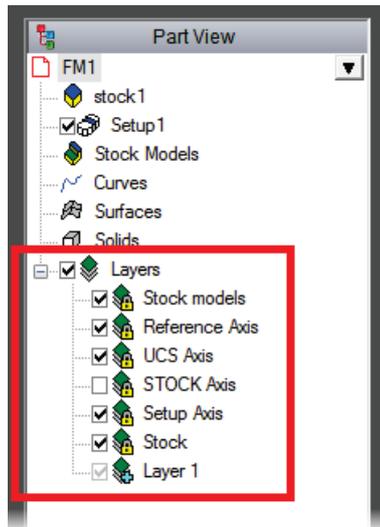
Characters you can use:

- , comma
- .
- " double quote mark
- / slash
- hyphen

ALT+ code characters, such as Ø, ¼, ½, and so on.

Layers in Part View

In FeatureCAM 2014 R1, you can see **Layers** in the **Part View**:



You can quickly show or hide layers using the check boxes.

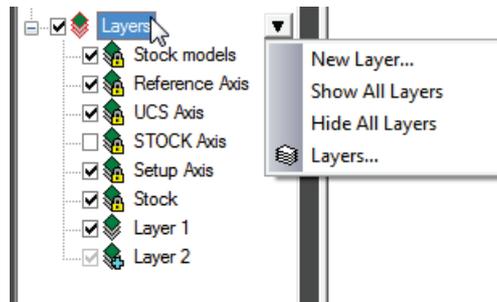
 **System layer** — You cannot edit a System layer, but you can show or hide it.

 **User layer** — Add your own layers to organize your work. To add a new layer, right-click on the Layers item and select New Layer from the context menu.

 **Current layer** — New items are added to the current layer. Double-click on a user layer to make it the current layer. You cannot hide the current layer.

Layers context menus

Right-click on the **Layers** item in the **Part View** to display a context menu:



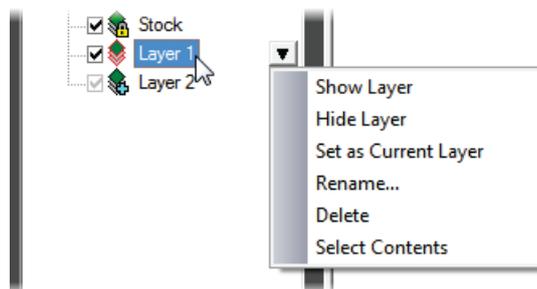
New Layer — Select this option to create a new user layer.

Show All Layers — Select this option to show the contents of all layers in the Graphics area.

Hide All Layers — Select this option to hide the contents of all layers in the Graphics area.

Layers — Select this option to open the **Layers** dialog.

Right-click on a layer name in the **Part View** to display a different context menu:



Show Layer — Select this option to show the contents of the layer in the Graphics area.

Hide Layer — Select this option to hide the contents of the layer in the Graphics area.

Set as Current Layer — Select this option to set the layer as the current layer. New items are added to the current layer.

Rename — Select this option to rename the layer.

Delete — Select this option to delete the layer.

Select Contents — Select this option to select the contents of the layer. This is useful, for example, if you want to move them to another layer.

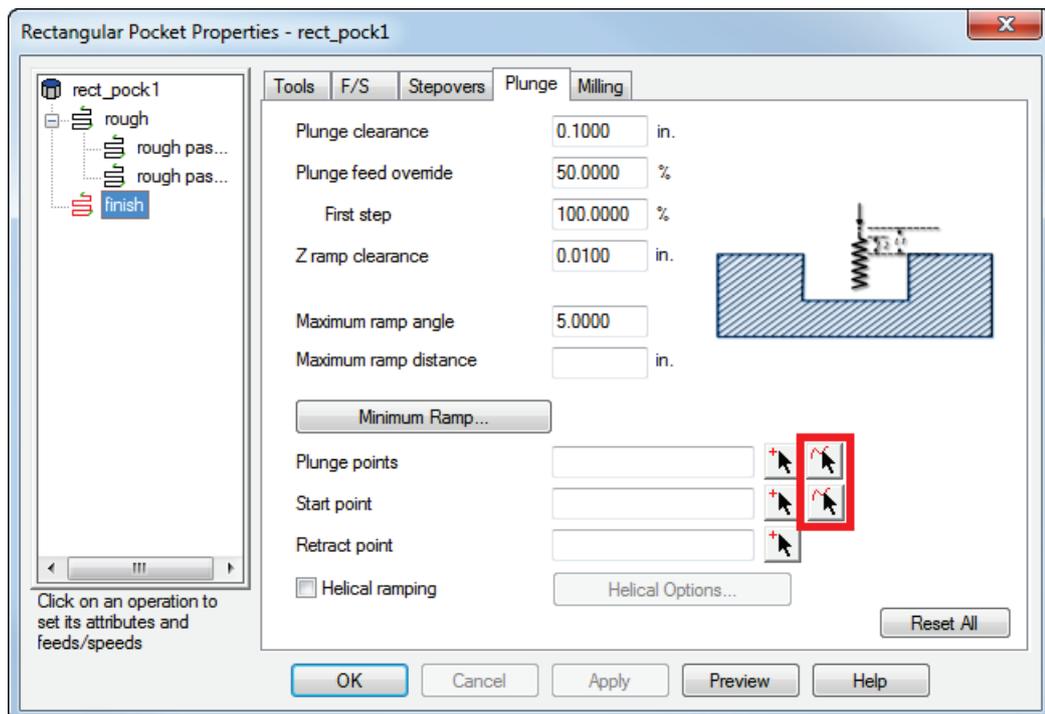


*The **Set as Current Layer**, **Rename**, and **Delete** options are unavailable for system layers.*

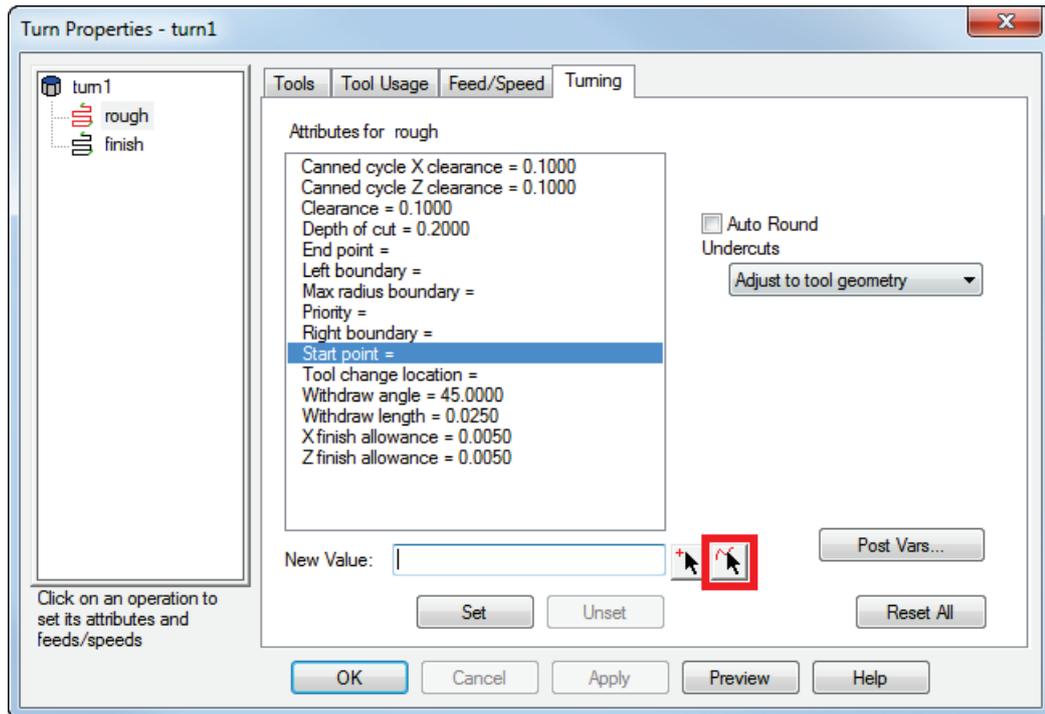
New Pick Curve buttons

In FeatureCAM 2014 R1, there are several new **Pick Curve**  buttons. This makes it quicker to set curves for the attributes. Previously you had to remember or copy-and-paste the curve name.

For milling, there are new **Pick Curve** buttons for the **Plunge points** and **Start point** options on the **Plunge** tab:

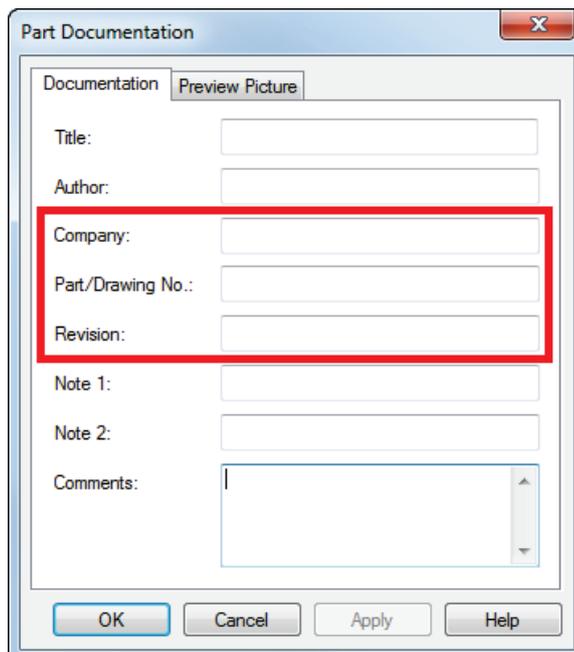


For turning, there are new **Pick Curve** buttons for the **Start point** and **End point** attributes on the **Turning** tab:



New Part Documentation fields

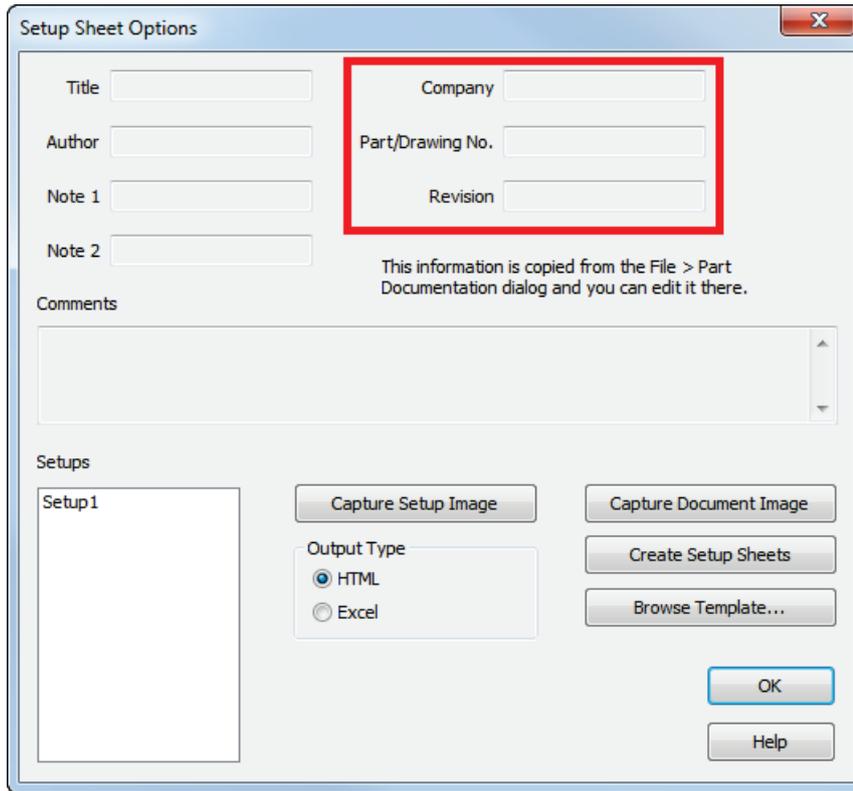
In FeatureCAM 2014 R1, there are three new fields in the **Part Documentation** dialog:



There are now dedicated fields for **Company**, **Part/Drawing No.**, and **Revision**. You can use the **Note 1**, **Note 2**, and **Comments** fields for other information not covered by these fields.

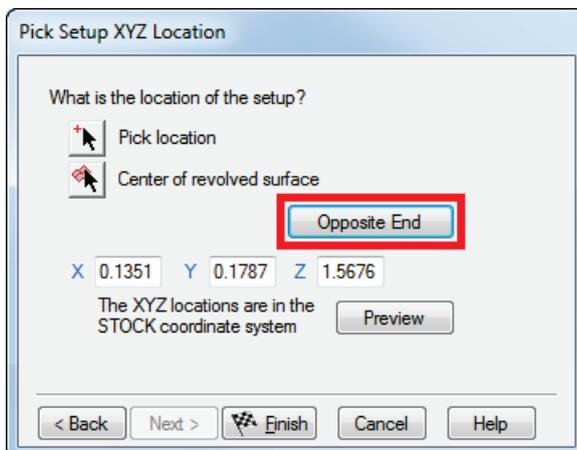


If you use the custom setup sheet add-in, these values are copied to the **Setup Sheet Options** dialog.



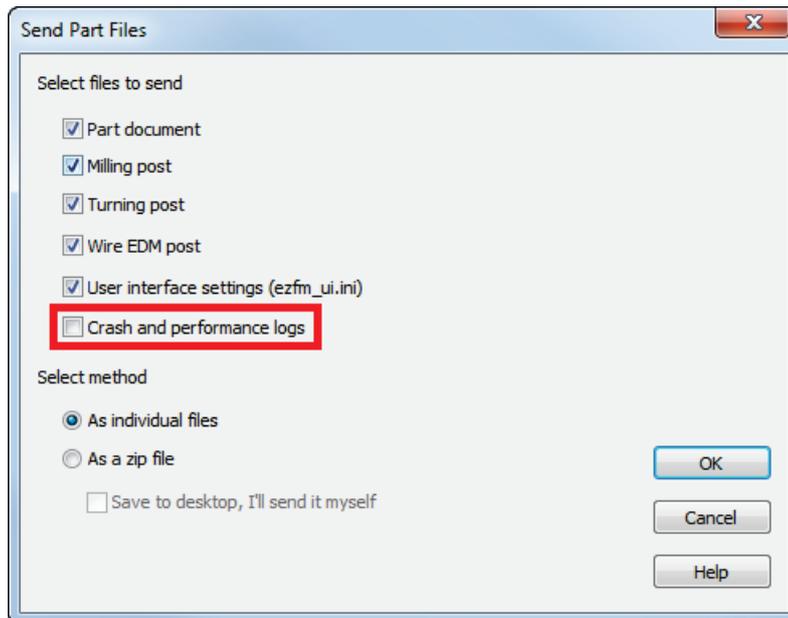
Setup at opposite end of cylinder

In FeatureCAM 2014 R1, you can change the location of a Setup aligned with a revolved surface to the opposite end:



Send crash logs

There is a new option in the **Send Part Files** dialog:



Select the **Crash and performance logs** option to create a separate **.zip** file of your system's crash and performance logs.

Size confirmation

If you are sending **.zip** files, a **Size Confirmation** warning dialog is displayed:

The average email server limits attachments to 10 MB.

The total size of the attachment(s) is: N (where **N** is the size of your attachment(s))

Would you like to continue sending?

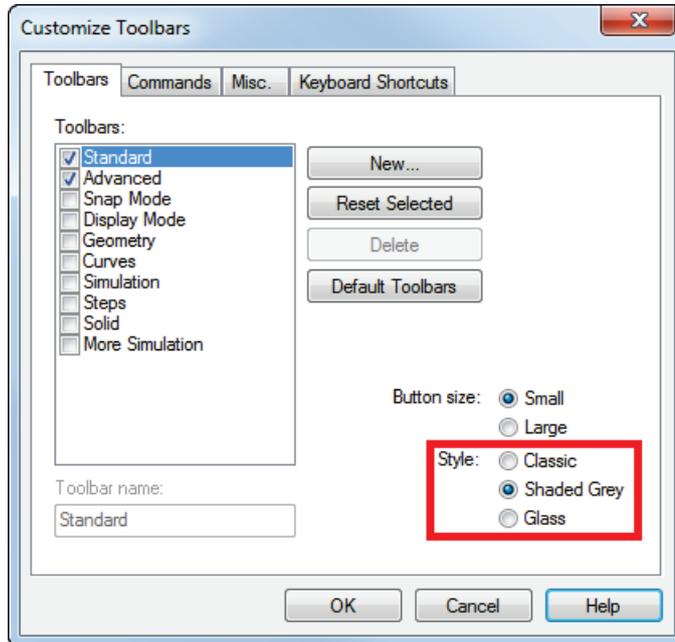
Click **Send** to attach the files to an email.

Click **Don't Send** to keep the files on your desktop.

Click **Cancel** to discard the files.

Shaded Grey default style

In FeatureCAM 2014 R1, **Shaded Grey** is the default user interface style. You can change the style back to the previous style, **Classic**, or the alternative style, **Glass**, in the **Customize Toolbars** dialog.

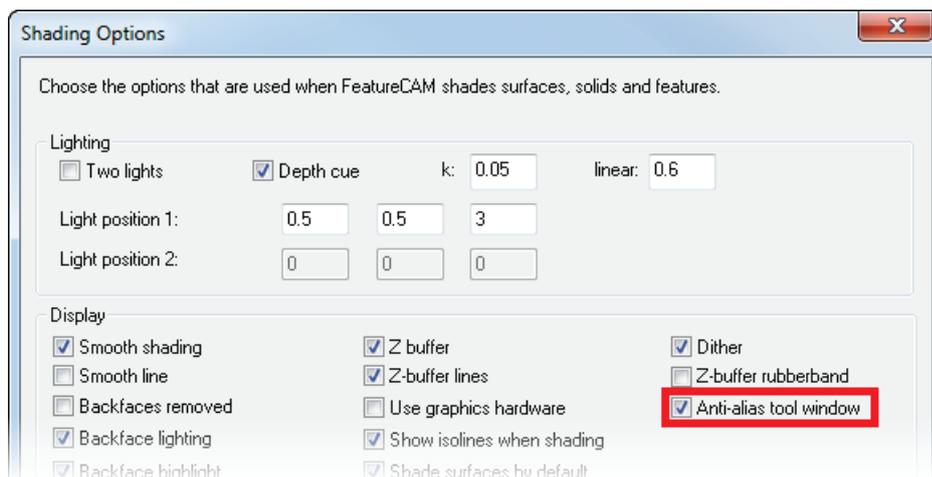


More attractive tool display window

The display of tools and holders in the **Tool Manager** and **Tool Properties** dialog is now more attractive thanks to the use of anti-aliasing in the tool display window.

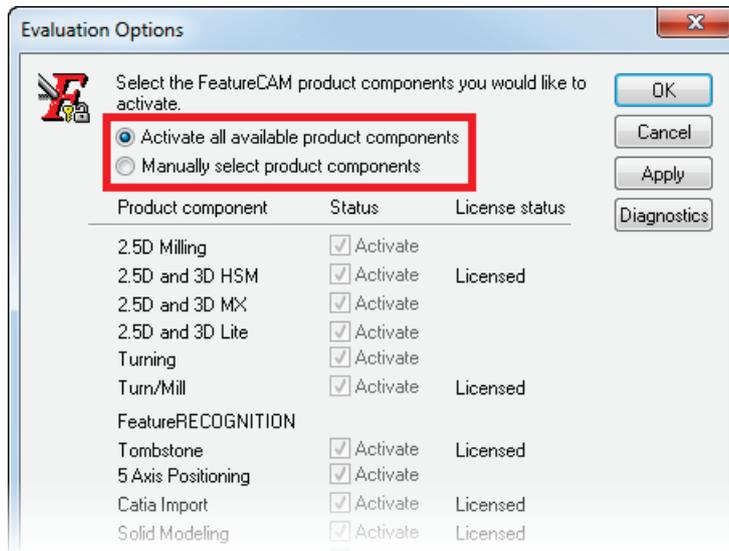


*If your graphics card or driver cannot support the anti-aliasing well, you can disable anti-aliasing in the tool window. There is a new **Anti-alias tool window** option in the **Shading Options** dialog. It is enabled by default.*



Licensed components activated by default

In FeatureCAM 2014 R1, the components that you have licensed are activated automatically. The new **Activate all available product components** option is selected by default:



If you want to evaluate components that you have not licensed, select the **Manually select product components** option and select the components individually.

License expiry warning

Warning messages are now displayed when your license is nearing expiry so that you have time to make arrangements to renew the license.

Import improvements

FeatureCAM 2014 R1 includes these import improvements:

- OpenDesign AutoCAD (see page 61) upgraded.
- Spatial R23 SP1 Interop (see page 61) integrated.
- Spatial 3D R23 SP2 Interop (see page 61) integrated.
- Import Using Exchange errors are logged (see page 61).
- Support for entry-level solid wireframe import (see page 61).

OpenDesign AutoCAD

Upgraded to the latest OpenDesign AutoCAD import library, version 3.09.0 (supporting DWG v2013). Includes multi-threaded loading.

Spatial R23 SP1 Interop integrated

Spatial R23 SP1 Interop is now integrated.

- The Pro/E Reader component supports import of the User Defined Attribute (UDA) and the Name Attribute from Creo 1.0.
- The Parasolid Direct Reader component supports v25 Parasolid.
- Analytical faces from the NX Reader are no longer split. Only non-analytical faces are split at their seam.

Spatial 3D R23 SP2 Interop integrated

Spatial 3D R23 SP2 Interop is now integrated.

- The CATIA V5 Reader/Writer component supports the import/export of CATIA V5 R23 parts and assemblies (Windows platforms only).
- The NX Direct Reader component supports version NX 8.5 for only BREP and Assembly.

Import Using Exchange logged

Information from the Import Using Exchange feature is now stored in FeatureCAM's log file.

Entry-level solid wireframe import

In FeatureCAM 2014 R1, users of the basic 2.5D module can import solid models as wireframe using **File > Import Using Exchange**.

2.5D Milling improvements

FeatureCAM 2014 R1 includes these 2.5D milling improvements:

- You can automatically avoid clamps (see page 62).
- You can do a Chamfer operation before a Finish operation (see page 65).
- You can use a solid as a milling tool holder (see page 66).
- Engraving fonts from onlinefonts.com (see page 66) are supported.

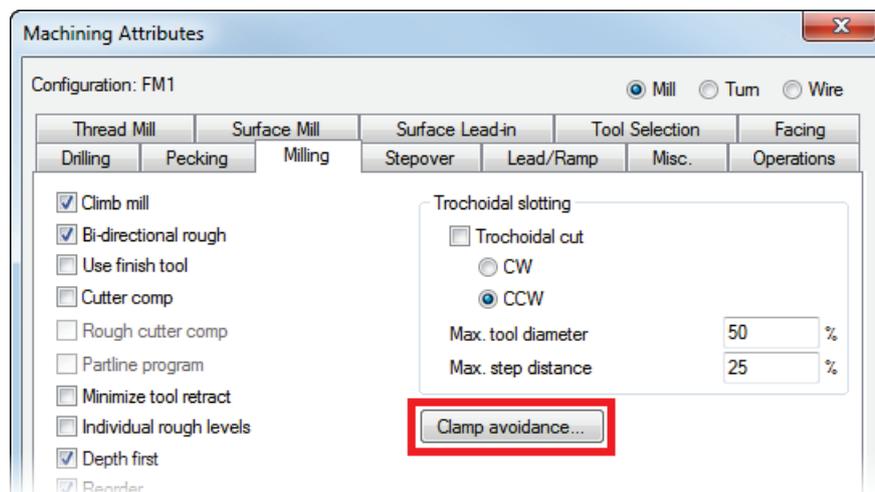
Automatic clamp avoidance

In FeatureCAM 2014 R1, you can automatically avoid solids that are set as clamps when using 2.5D NT-style toolpaths and 3D toolpaths.

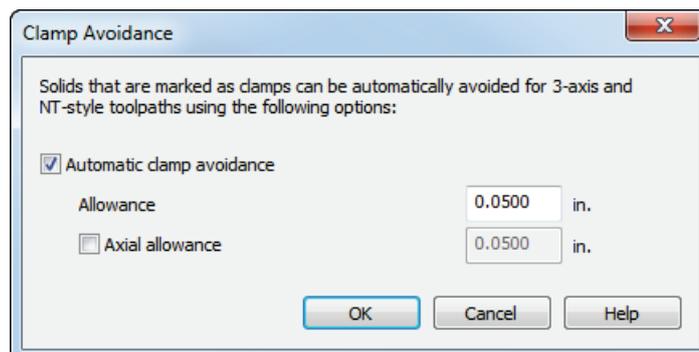


You must use NT-style toolpaths for both the Rough and Finish operations to use this option for a 2.5D part.

There is a new **Clamp avoidance** button on the **Milling** tab in **Machining Attributes**:



Click the **Clamp avoidance** button to open the **Clamp Avoidance** dialog:



Automatic clamp avoidance — Select this option to automatically avoid solids that are marked as a clamp. To mark a solid as a clamp, right-click its name in the **Part View** and select **Use Solids as Clamp** in the context menu.



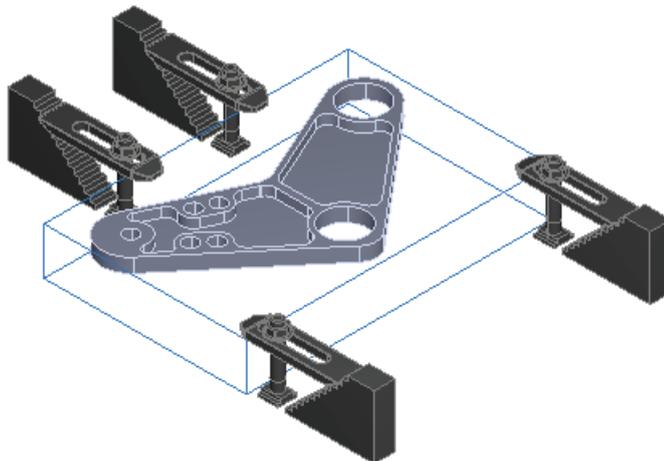
This option applies to 2.5D NT-style toolpaths and 3D toolpaths. You must use NT-style toolpaths for both the Rough and Finish operations to use this option for a 2.5D part.

Allowance — Enter the minimum distance that you want to leave around clamps.

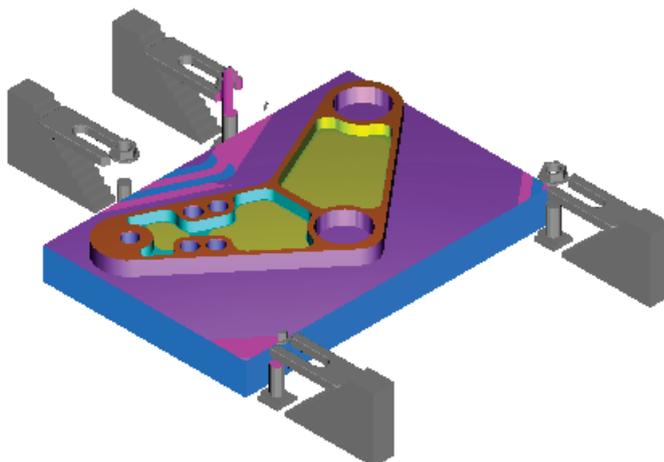
Axial allowance — To set separate axial and radial allowances, select this option and enter the minimum axial (XY) distance that you want to leave around clamps. If you enable **Axial allowance**, the **Allowance** value is applied to the radial (Z) distance only.

Example:

This example bracket part is held in place by four clamps:



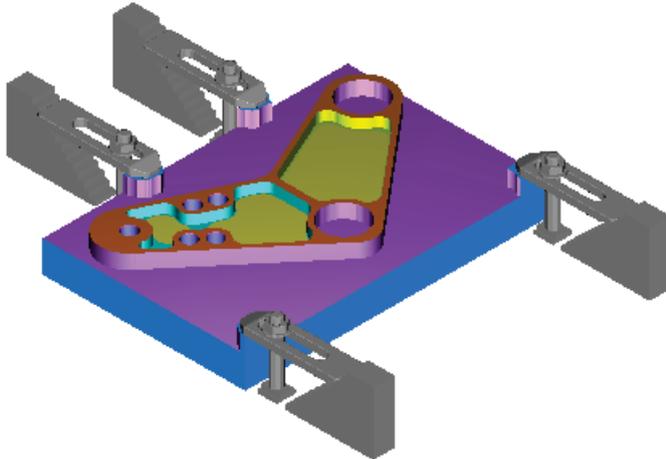
With **Automatic clamp avoidance** disabled, there is a collision with the clamps in the bright pink areas:



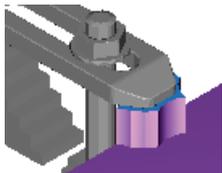
Here is the back clamp in more detail, the front of the clamp has been machined away:



With **Automatic clamp avoidance** enabled, the clamps are avoided:

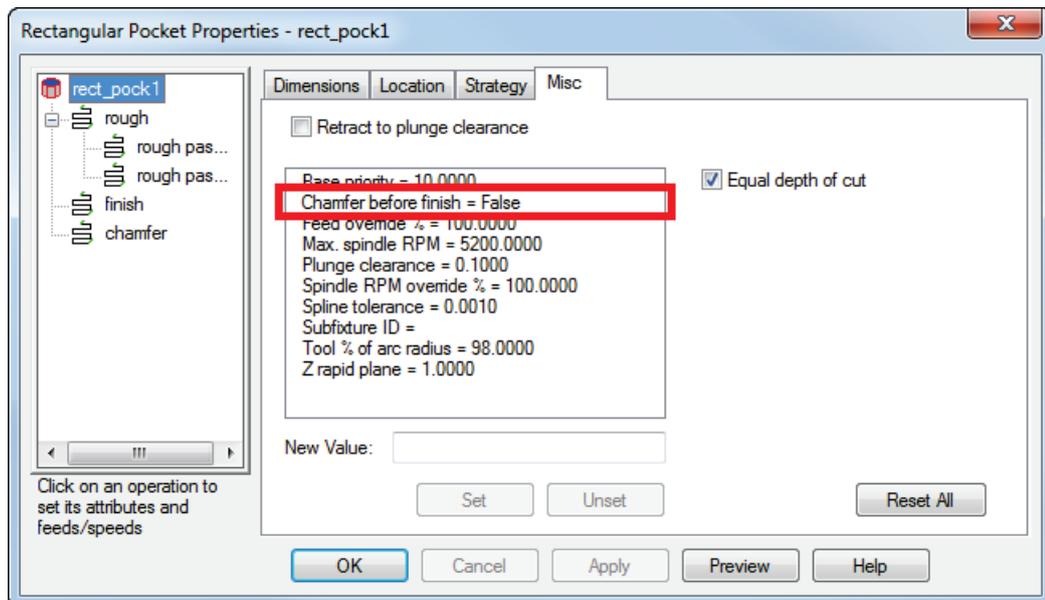


You can see the upstands of material, which have been left around the clamps:



Chamfer before Finish

In FeatureCAM 2014 R3, there is a new **Chamfer before finish** option on the **Misc** tab:



Enable this option to do the Chamfer operation before the Finish operation.

To enable **Chamfer before finish**:

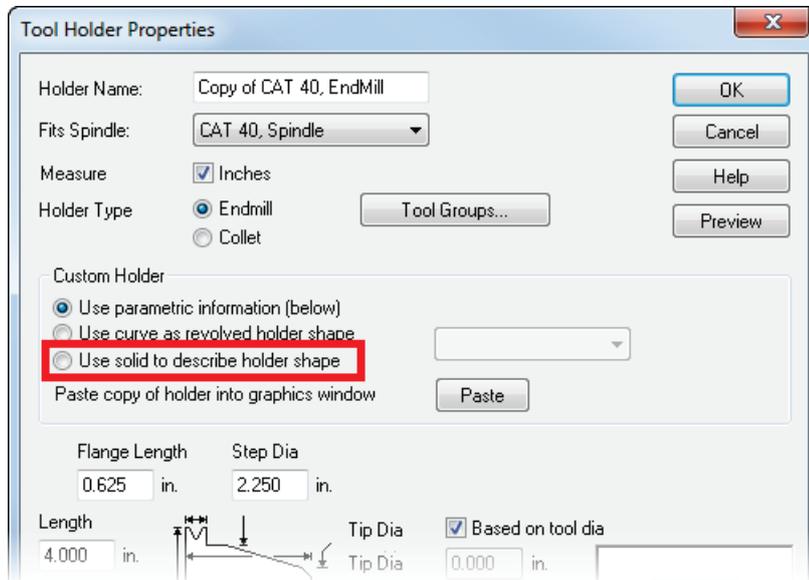
- 1 Select **Chamfer before finish** in the list of attributes.
- 2 Select **True** from the **New Value** menu.
- 3 Click the **Set** button to save the new value.

The order of operations is updated in the feature Properties dialog and in the **Op List**.

Solid toolholder

In FeatureCAM 2014 R1, you can use a solid to represent the tool holder for milling tools. This is useful for creating custom holders, where the holder cannot be described using a revolved curve.

There is a new option in the **Tool Holder Properties** dialog:



Select **Use solid to describe holder shape** and select the solid from the list.

OLF engraving fonts

In FeatureCAM 2014 R1, engraving fonts from our partner onlinedfonts.com are supported. FeatureCAM ships with an incomplete test font to show that OLF fonts are recognized. You can purchase more fonts from <http://onlinedfonts.com>.

Solid improvements (SOLID)

FeatureCAM 2014 R1 includes these solid improvements:

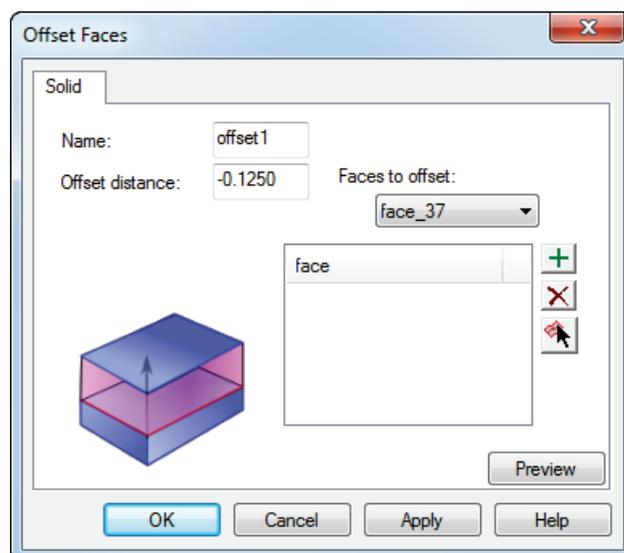
- You can now offset individual faces of a solid (see page 67).

Offset faces

In FeatureCAM 2014 R1, you can offset individual faces of a solid. There is a new **Offset Faces** solid constructor.

To offset a face or faces of a solid:

- 1 Open the **Offset Faces** dialog in one of these ways:



- In the **Solid Wizard**, select **Shape Modifiers**, then **Offset Faces**, and click **Next**.
 - On the **Solid** toolbar, click the **Offset Faces**  button in the **Modify solid** menu.
 - From the menu, select **Construct > Solid > Modifiers > Offset Faces**.
- 2 Optionally enter a **Name** for the solid, or leave the default name.
 - 3 Enter the **Offset distance**. A negative distance offsets the faces into the solid, a positive distance offsets the surfaces outwards.
 - 4 Select the name of the face you want to offset from the menu and click the **Add item**  button or click the **Pick surfaces**  button and select the surface in the Graphics window.
 - 5 Optionally click the **Preview** button to see the results of the current settings in the Graphics window.
 - 6 Click **OK** (or **Finish** if you're using the wizard).

Turn improvements (TURN)

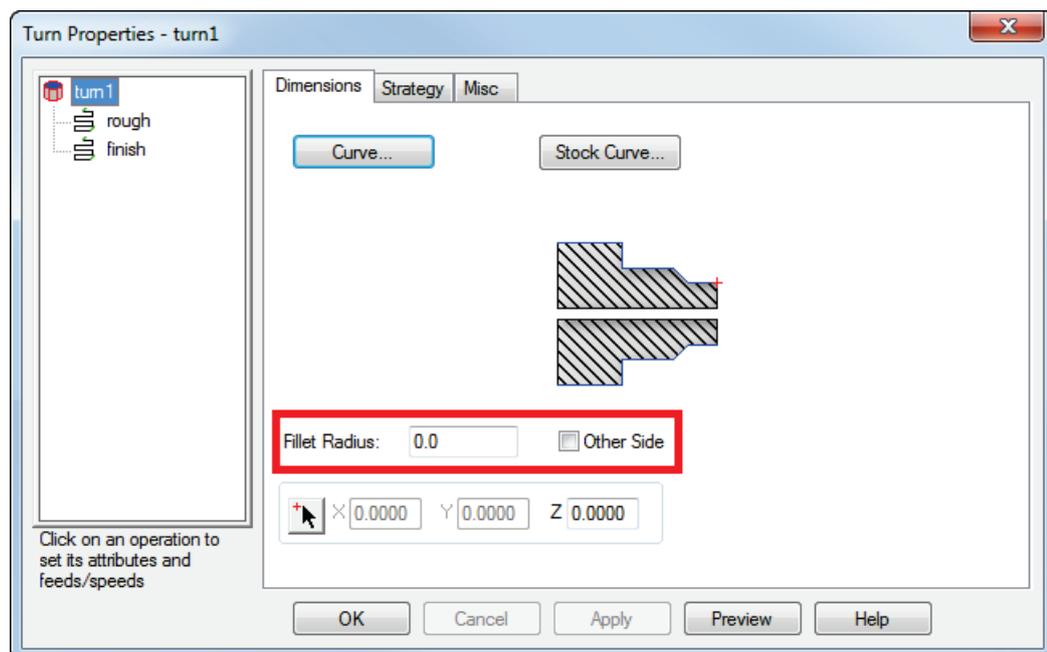
FeatureCAM 2014 R1 includes these turn improvements:

- You can now do automatic corner rounding (see page 68).
- There is now support for steady rest and tailstock (see page 70).

Automatic corner rounding

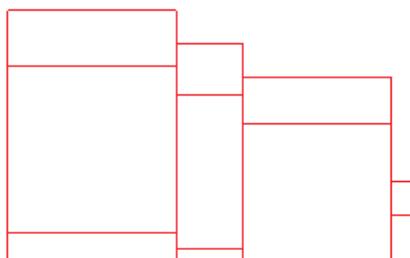
Programming turned parts is quicker and easier in FeatureCAM 2014 R1. You can automatically deburr internal or external sharp edges of a Turn or Bore feature, without having to edit the geometry, using the new **Fillet Radius** setting.

Enter a value to enable automatic filleting:

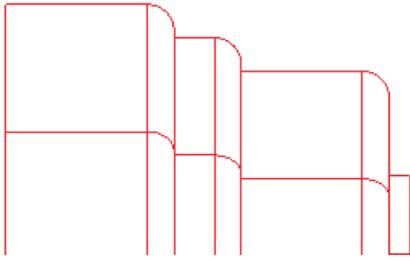


If you enter a **Fillet Radius** value and click Apply, the Graphics window is updated, so you can preview the fillet.

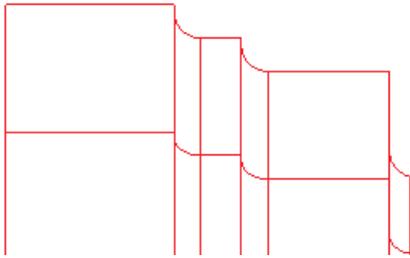
For example, no **Fillet Radius**:



With a **Fillet Radius** value applied:

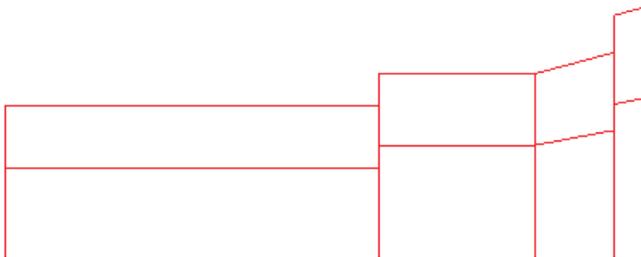


If you want to apply the fillet to internal instead of external corners, select the **Other Side** option:

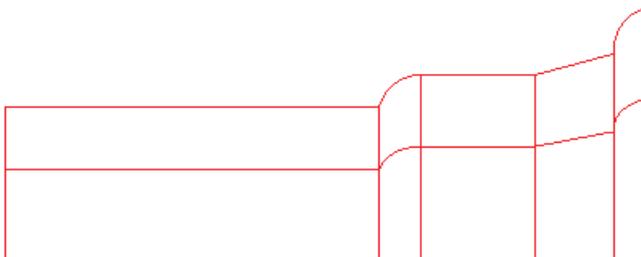


Automatic corner rounding works equally well for a Bore feature.

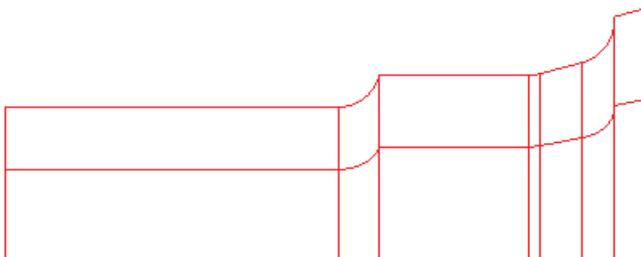
For example, no **Fillet Radius**:



With a **Fillet Radius** applied:



If you want to apply the fillet to external instead of internal corners, select the **Other Side** option:



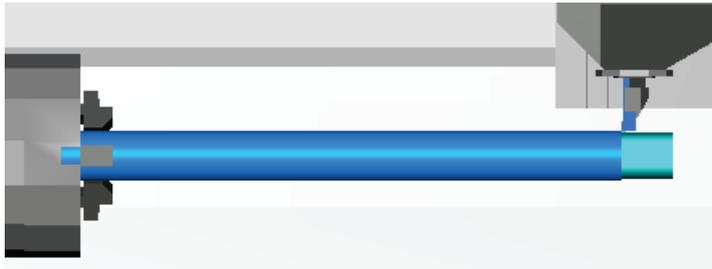
Steady rest and tailstock support

FeatureCAM 2014 R1 supports the programming and simulation of steady rests and tailstock. Collisions with steady rests and tailstock are detected.

This example part is 1.5 m long:

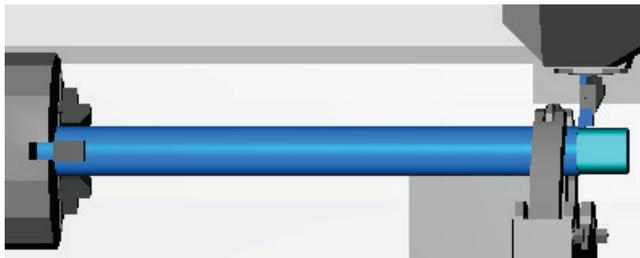


Looking at a machine simulation, you can see that there is a long length of the part sticking out of the jaws:



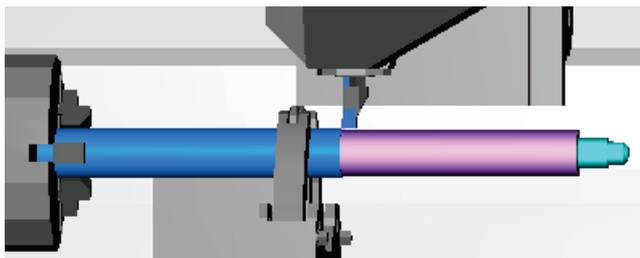
You can use a steady rest to support the part as you machine each feature. To use a steady rest, you create a **Part Handling** feature with a type of **Part support on** and a **Support type** of **Steadyrest**.

In the example, the steady rest is turned on before the features at the end of the part are created.

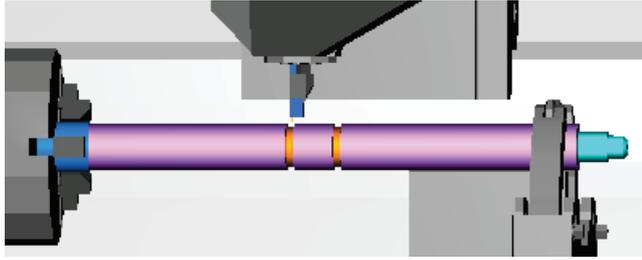


You can move the steady rest by creating an additional **Part support on** feature.

In the example, the steady rest is moved further down the part to support the Turn operation:

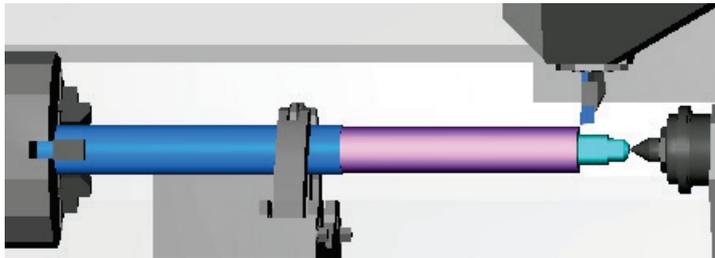


An additional **Part support on** feature is created to move the steady rest back to the original position, to cut the other Turn feature and the Groove features:



You can use a tailstock in a similar way. You create a **Part Handling** feature with a type of **Part support on** and a **Support type** of **Tailstock**.

In the example, the tailstock is used to support the end of the part while the steady rest is supporting the middle:



To send the steady rest or tailstock home, you create a **Part Handling** feature with a type of **Part support off**.

To enable a steady rest:

- 1 In the turning **New Feature** wizard, select the **Part Handling** feature and click **Next**.
- 2 Select **Part Support On** and enter a value for the **Grab distance**. This is the distance in Z where you want to use the steady rest.
- 3 Click **Next**.
- 4 For the **Support type**, select **Steadyrest**.
- 5 If your machine has multiple turrets, select the correct **Transfer turret**.
- 6 If your steady rest is fitted to a turret, click the **Turret control** button and set the correct **Index** position.
- 7 Click **Finish**.

A **part_support_on** feature is added to the **Part View**. You may need to drag the feature to a different position in the **Part View** and set a **Base priority** for the feature to set the correct order of features.

To disable a steady rest:

- 1 In the turning **New Feature** wizard, select the **Part Handling** feature and click **Next**.
- 2 Select **Part Support Off** and enter a value for the **Grab distance**. This is the distance in Z where you want to use the steady rest.
- 3 Click **Next**.
- 4 For the **Support type**, select **Steadyrest**.
- 5 If your machine has multiple turrets, select the correct **Transfer turret**.
- 6 If your steady rest is fitted to a turret, click the **Turret control** button and set the correct **Location** and **Index** position.
- 7 Click **Finish**.

A **part_support_off** feature is added to the **Part View**. You may need to drag the feature to a different position in the **Part View** and set a **Base priority** for the feature to set the correct order of features.

To enable a tailstock:

- 1 In the turning **New Feature** wizard, select the **Part Handling** feature and click **Next**.
- 2 Select **Part Support On** and enter a value for the **Grab distance**. This is the distance in Z where you want to use the steady rest.
- 3 Click **Next**.
- 4 For the **Support type**, select **Tailstock**.
- 5 If your machine has multiple turrets, select the correct **Transfer turret**.
- 6 If your tailstock is fitted to a turret, click the **Turret control** button and set the correct **Index** position.
- 7 Click **Finish**.

A **part_support_on** feature is added to the **Part View**. You may need to drag the feature to a different position in the **Part View** and set a **Base priority** for the feature to set the correct order of features.

To disable a tailstock:

- 1 In the turning **New Feature** wizard, select the **Part Handling** feature and click **Next**.
- 2 Select **Part Support Off** and enter a value for the **Grab distance**. This is the distance in Z where you want to use the steady rest.
- 3 Click **Next**.

- 4 For the **Support type**, select **Tailstock**.
- 5 If your machine has multiple turrets, select the correct **Transfer turret**.
- 6 If your tailstock is fitted to a turret, click the **Turret control** button and set the correct **Location** and **Index** position.
- 7 Click **Finish**.

A **part_support_off** feature is added to the **Part View**. You may need to drag the feature to a different position in the **Part View** and set a **Base priority** for the feature to set the correct order of features.

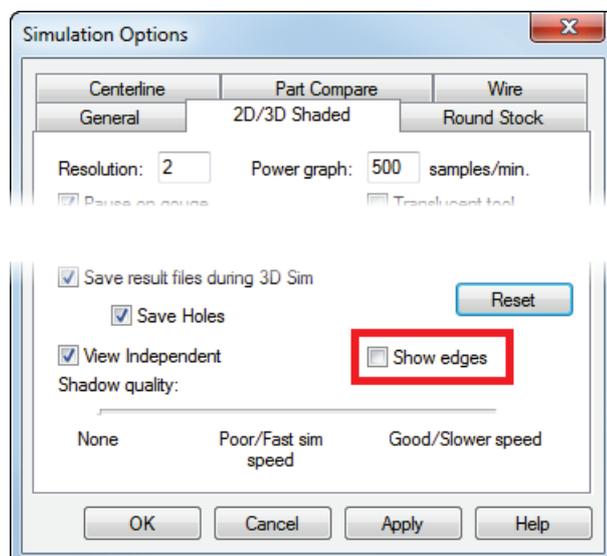
Machine simulation improvements (MSIM)

FeatureCAM 2014 R1 includes these machine simulation improvements:

- You can now show edges (see page 73) during machine simulation.
- Jaws, steady rest, and tailstock (see page 70) are now supported for turning machine simulation.
- For machines with two spindles, you can play the simulation once only (see page 75).

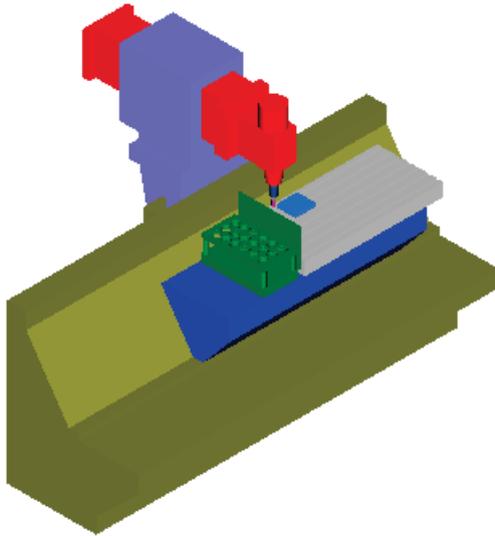
Show edges

In FeatureCAM 2014 R1, there is a new **Show edges** option for machine simulation:

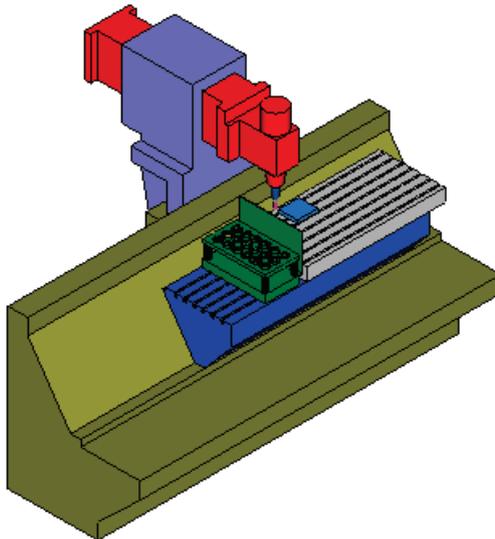


Select this option to use an outline for the edges of the machine. You can see the machine edges more clearly using this option.

This example shows the default behavior with **Show edges** deselected:

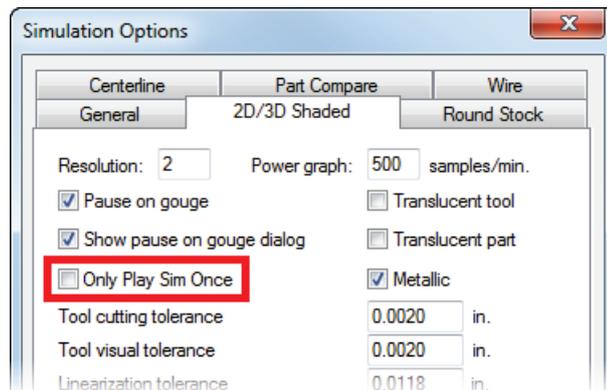


This is the same example with **Show edges** selected:



Play simulation once

In FeatureCAM 2014 R1, there is a new **Only Play Sim Once** option in **Simulation Options**:



For some machines with two spindles, the simulation plays through the simulation twice. If you want to see the part simulated only once, select this option.

Machine design improvements (MSIM)

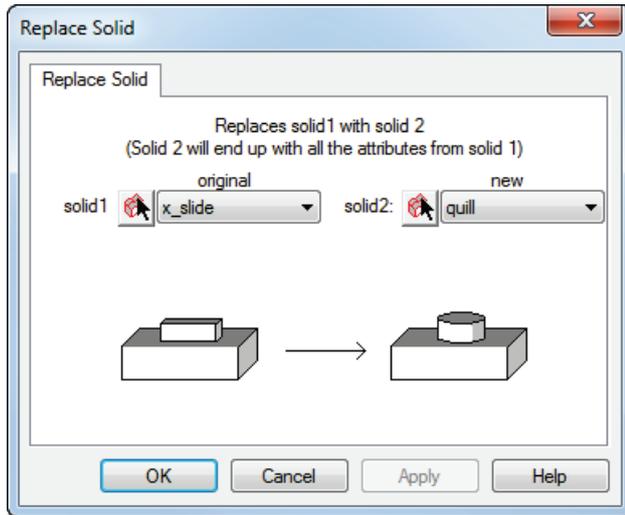
FeatureCAM 2014 R1 includes these machine simulation improvements:

- You can now replace a solid (see page 75) and keep the attributes of the original.
- It is now easier to define parent/child relationships (see page 76).
- It is now simpler to create milling machines (see page 77).
- You get feedback when a UCS is not set (see page 77).

Replace solid

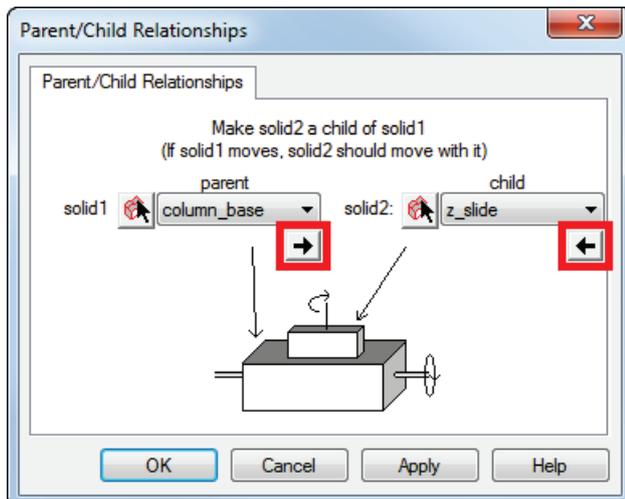
There is a new **Replace Solid** dialog for machine design in FeatureCAM 2014 R1. To open it, select **Machine Design > Replace Solid** from the menu.

Use the **Replace Solid** dialog to replace **solid1** with **solid2**, and keep the machine design attributes from **solid1**, such as parent/child relationships, movement, and so on.



Easier relationship definitions

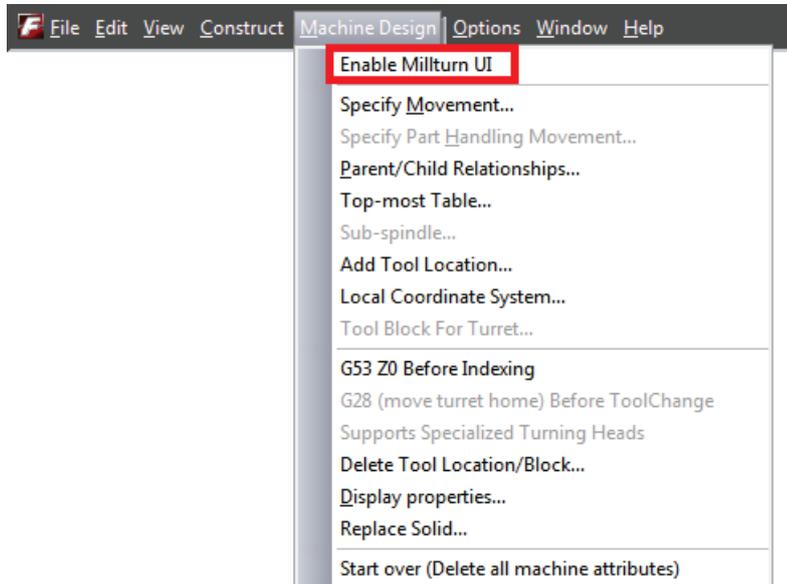
In FeatureCAM 2014 R1, there are two new arrow buttons in the **Parent/Child Relationships** dialog:



As you work up or down through the machine hierarchy, you can save time by moving the previously selected solid from the **parent** to the **child** selection or the **child** to the **parent** selection using the arrow buttons.

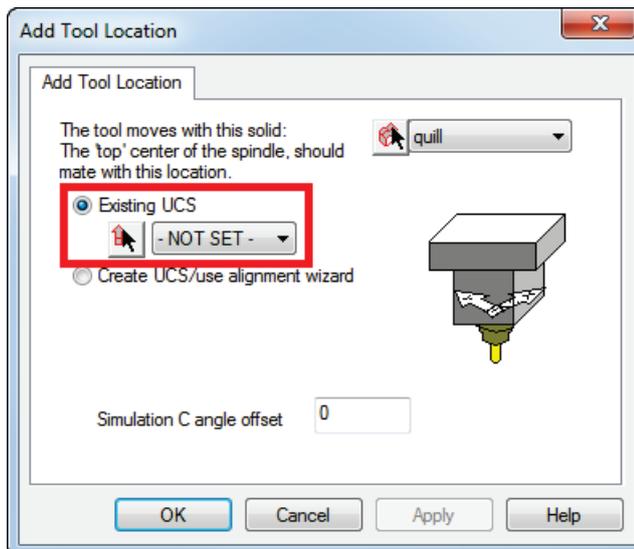
Simplified milling machine interface

Machine design now shows only the options relevant to a milling machine, by default in the user interface. The options relevant to turning machines are unavailable in the menu, until you select the new **Enable Millturn UI** menu option:



Improved user interface

In FeatureCAM 2014 R1, you get feedback when you have not yet set a UCS, for example:



Probing improvements

FeatureCAM 2014 R1 includes these improvements to the Probing add-in:

- You can now edit default values (see page 78).
- There are three new decision-making features (see page 79).
- Turn/mill probing simulation (see page 82) has been improved.

Edit default values

In FeatureCAM 2014 R1, default measurement attributes have been added to the **Probing Settings** dialog:

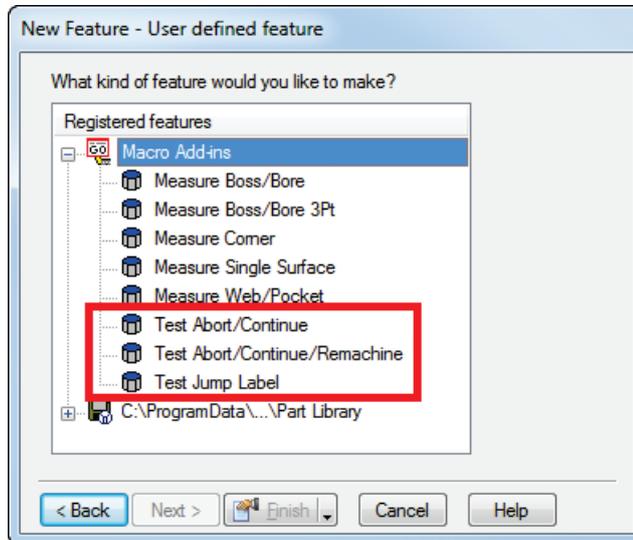
Probing Settings			
General Defaults			
Default Tool Name	<input type="text"/>	<input type="checkbox"/> Show Overtravel	
Measurement Defaults			
Diameter	<input type="text" value="2"/>	Clear Height	<input type="text" value="0.4"/>
Standoff	<input type="text" value="0.4"/>	Measure Height	<input type="text" value="0.4"/>
Overtravel	<input type="text" value="0.1"/>	Radius Angle A	<input type="text" value="0"/>
Tolerance Value	<input type="text" value="0.01"/>	Radius Angle B	<input type="text" value="120"/>
Nominal Value	<input type="text" value="2"/>	Radius Angle C	<input type="text" value="240"/>
Conditional Defaults			
Comparison Operator	<input type="text" value="LT"/>	Continue Label	<input type="text" value="1001"/>
Nominal Value Register	<input type="text" value="102"/>	Abort Label	<input type="text" value="9999"/>
Tolerance Value Register	<input type="text" value="103"/>	Abort Message	<input type="text" value="Error: Aborting"/>
Remachine Flag Register	<input type="text" value="101"/>	Remachine Failure Label	<input type="text" value="1002"/>
Temporary Register	<input type="text" value="104"/>	Remachine Failure Message	<input type="text" value="Error: Second Req"/>
Result Register	<input type="text" value="510"/>	Size Error Label	<input type="text" value="1003"/>
Label	<input type="text" value="1002"/>	Size Error Message	<input type="text" value="Error: Too Big"/>
OK Cancel Reset Defaults			

You can edit the default values of any of these attributes. Click the **Reset Defaults** button to return to the original values.

Decision-making features

It is easier to program probing in FeatureCAM 2014 R1, using decision-making features.

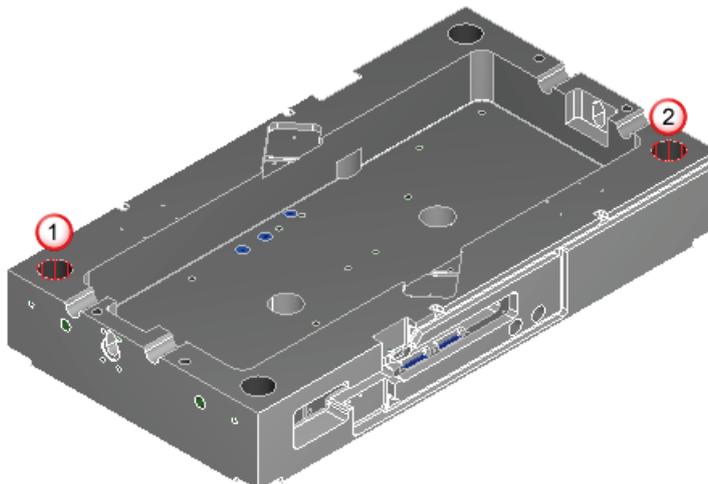
There are three decision-making features:



- **Test Abort/Continue** — Based on the results of your measurement, this feature type aborts the current milling process or continues as normal.
- **Test Abort/Continue/Remachine** — Based on the results of your measurement, this feature type aborts the current milling process, continues as normal, or remachines the feature.
- **Test Jump Label** — Places an N block number label in the NC code that the program can use, for example a start label, or a place in the code to jump to.

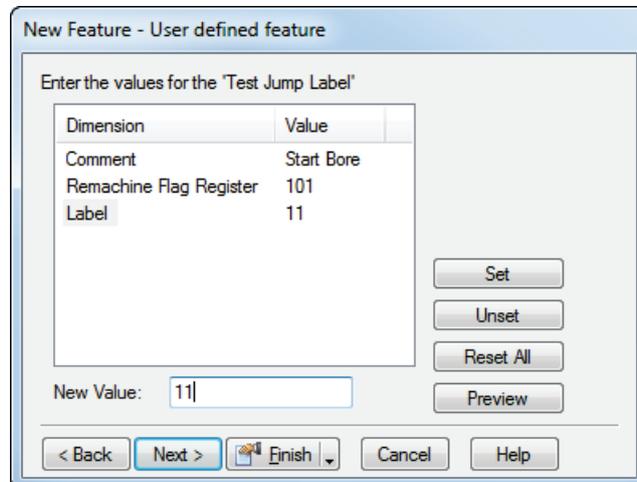
Decision-making example

This example part has many features.



After milling a Face feature, the first Bore feature ① is milled, then the second Bore feature ②, followed by the rest of the features. To avoid unnecessary machining time, you can probe Bore ① after it is cut and make a decision whether to continue, remachine it, or abort the program, depending on the results of the probe.

- 1 Create a **Test Jump Label** feature to start the process.



The NC code for this Test Jump Label feature reads:

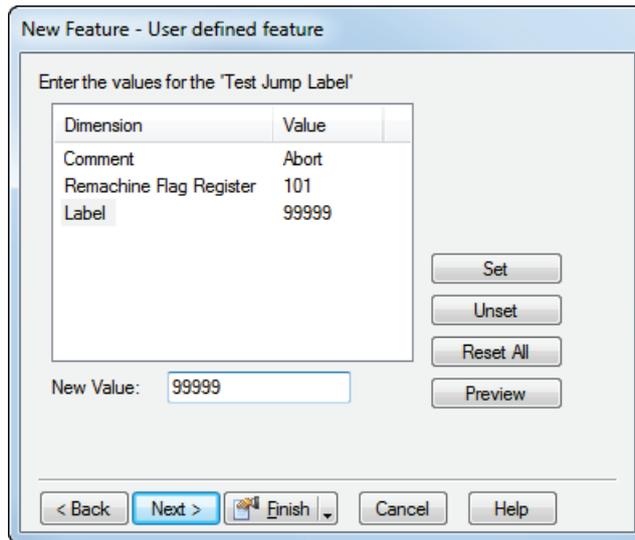
```
(Test Jump Label PROBEOPER TEST_JUMP_LABEL1)
(Start Bore)
#101=0
N11
```

- 2 You want the decision-making process to start after the Face feature, so set the **Base priority** for the Test Jump Label to **2**, and drag the feature to the correct place in the **Part View**.
- 3 Create a Measure Boss/Bore feature to probe Bore1 after it has been milled.
- 4 Create a Test Abort/Continue/Remachine feature to control what happens after the probing. For this example the following values are set:
 - Nominal Value 27.000**
 - Tolerance Value 0.050**
 - Continue Label 12**
 - Remachine Label 11** (the value that was set for the starting Test Jump Label)
 - Size Error 13**
- 5 Set the **Base priority** to **5** because you want this to be the 5th feature (after the Face feature, starting Test Jump Label, Bore1 feature, and Measure Boss/Bore feature).

The NC code for this Test Abort/Continue/Remachine feature reads:

```
( Test Abort/Continue/Remachine  PROBEOPER
TEST_ABORT_CONTINUE_REMACHINE1 )
(Remachining check conditional)
(Begin decision sequence)
#102=27  (nominal value)
#104=[#102-#510]  (signed difference of actual from
nominal)
#103=0.05 (tolerance value)
(Decision 1)
IF [ABS[#104]LT#103] GOTO 12  (within tolerance case)
(Decision 2)
IF [#104LT0]GOTO 13
(Decision 1 and 2 skipped)
[#101=#101+1]
IF [#101EQ1] GOTO 11 (run toolpath again)
DPRNT[Error: Second Required Remachining]
GOTO 99999
(Decision 2 result)
N13
DPRNT[Error: Too Big]
GOTO 99999
(Decision 1 result)
N12
N835 M5 M9
N840 G91 Z0
N845 M01
```

6 Create an Abort label at the end of the NC code:



7 Set its **Base priority** to a large number, such as **100**.

The NC code for this Test Jump Label feature reads:

```
( Test Jump Label PROBEOPER TEST_JUMP_LABEL3 )  
(Abort)  
#101=0  
N99999
```

Turn/mill probing simulation improved

The accuracy of turn/mill probing simulation has been improved by handling rotary indexing moves.

XBUILD improvements

FeatureCAM 2014 R1 includes these XBUILD improvements:

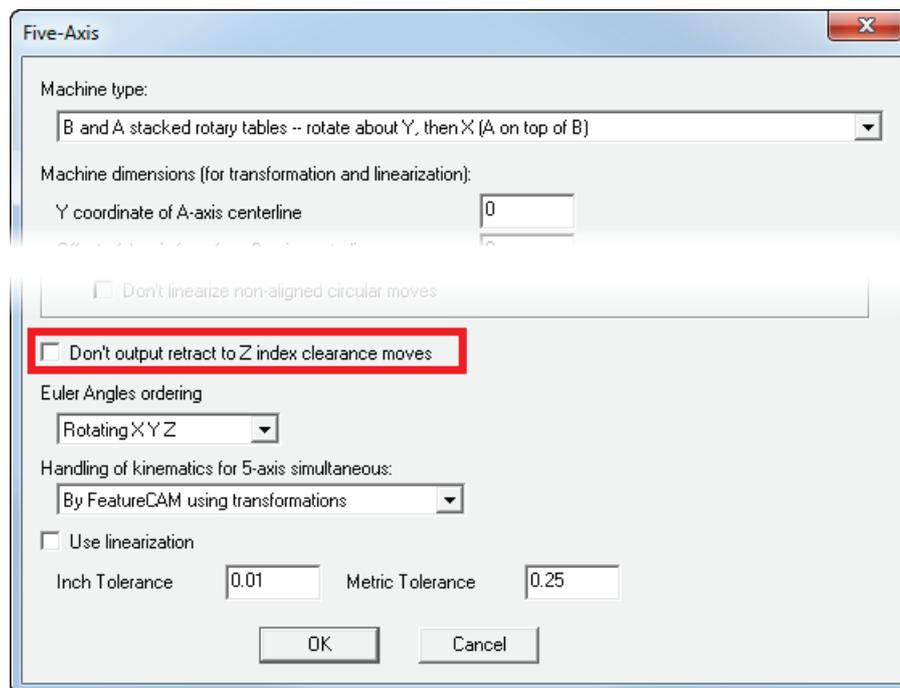
- There is a new `<INDEX-AXIS>` (see page 83) reserved word.
- There is a new **Don't output retract to Z index clearance moves** (see page 83) option for use with the Smith Bits APT post.
- You can now set the **Force segment start for each operation** (see page 84) option in XBUILD.

New reserved words

<code><INDEX-AXIS></code>	Returns the index axis (X, Y, or Z) for 4th-axis indexed parts, otherwise returns <code>None</code> .
---------------------------------	---

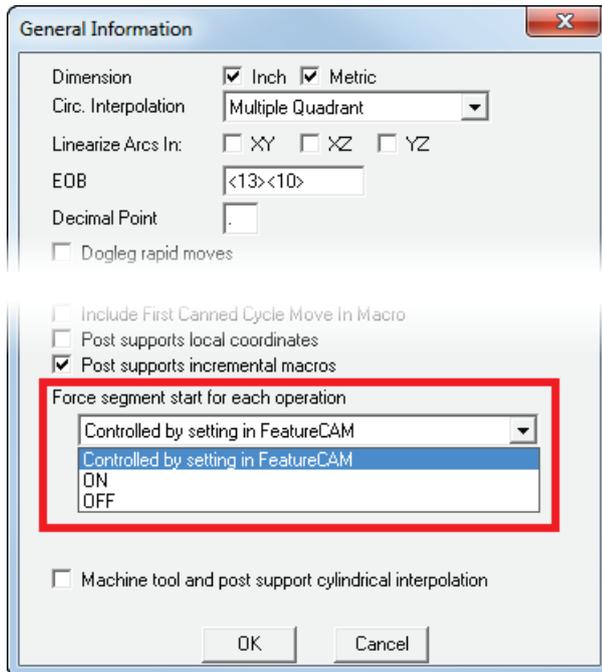
Z index clearance optional

There is a new **Don't output retract to Z index clearance moves** option in XBUILD. If you are using the Smith Bits APT post, you must select this option. This enables the correct APT code to be output for hole canned cycles that are not aligned with the Z-axis in RTCP (Rotational Tool Center Point).

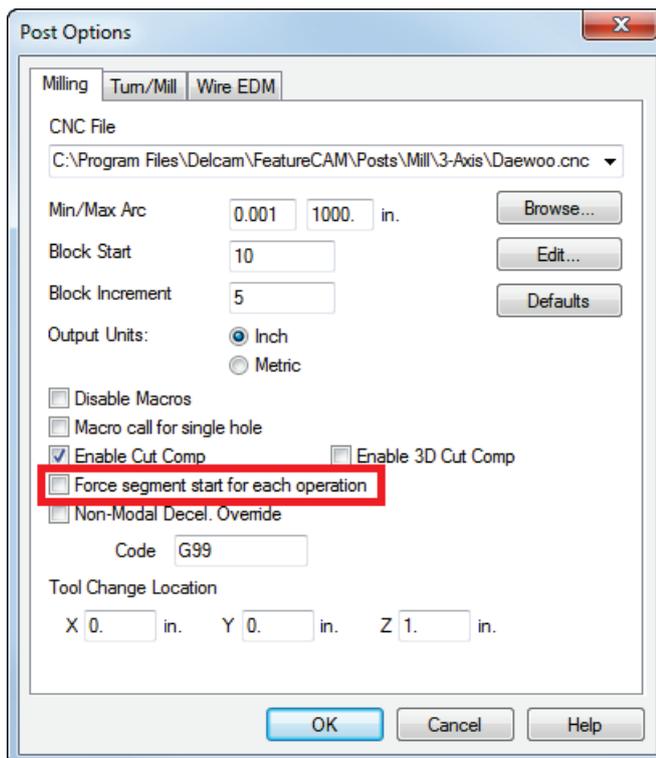


Force segment start option moved

In FeatureCAM 2014 R1, you can set the **Force segment start** for each operation option in XBUILD in the **General Information** dialog.

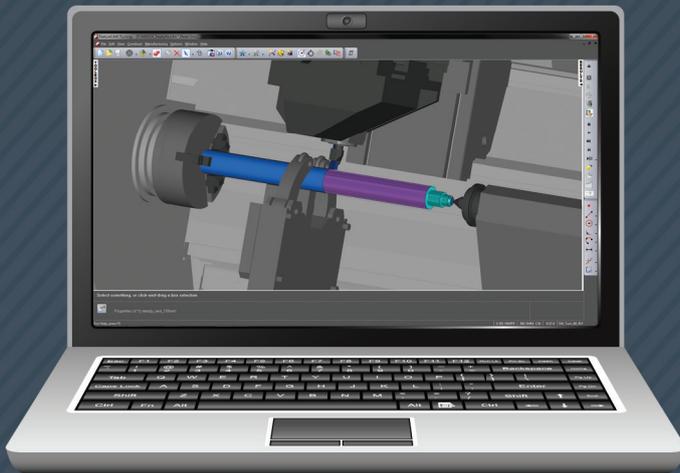


The previous and default behavior is to set this option in the **Post Options** dialog in FeatureCAM:



If you select **ON** or **OFF** in XBUILD, the status is displayed in the **Post Options** dialog.

FeatureCAM 2014 R3



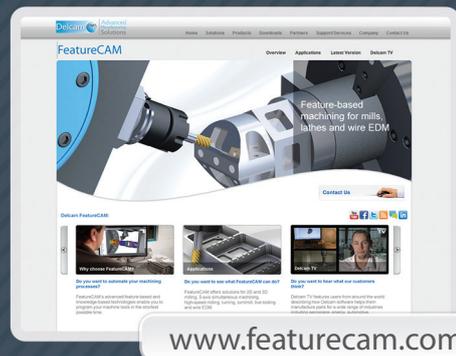
Feature-based CAM software
for mills, multi-tasking lathes
and wire EDMs



www.delcam.tv



www.delcam.tv/lz



www.featurecam.com



Powering your productivity

Delcam plc
Small Heath Business Park,
Birmingham,
B10 0HJ
United Kingdom
T: +44 (0) 121 766 5544
E: marketing@delcam.com

www.delcam.com



THE QUEEN'S AWARDS
FOR ENTERPRISE:
INNOVATION
2010



THE QUEEN'S AWARDS
FOR ENTERPRISE:
INTERNATIONAL TRADE
2011

To contact your local reseller,
visit www.delcam.com/resellers

