



Advanced  
Manufacturing  
Solutions

# FeatureCAM 2013

What's New

---

FeatureCAM 2013 R3

# What's New



Copyright © 1995-2013 Delcam plc. All rights reserved. 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: 2013 R3 Date: 22 April 2013 14:43

# Contents

## What's New in FeatureCAM 2013 R3 1

User interface improvements .....	1
Balloon help.....	1
Tooling improvements.....	2
Never select tool.....	3
2.5D: Rounding Mills — Set touch-off point at tip .....	3
New probing tool group .....	4
2.5D Milling improvements.....	6
NT toolpaths for finishing.....	6
3D improvements.....	8
Import surfaces (3D LITE) .....	8
Finish axial allowance (3D LITE) .....	8
Leave axial allowance (3D MX) .....	9
3D spiral out .....	9
Other important improvements .....	9
Part library: 64-bit format .....	10
Vertical mill/turn (TURNMILL, MTT, 5AP).....	10
Import improvements .....	11
SolidWorks: Manage associativity .....	12
Parasolid kernel upgrade to 24.1 .....	12
Turn/mill and 4th-axis improvements .....	13
Turn/mill & 4th-axis: Change index axis .....	13

## What's New in FeatureCAM 2013 R2 15

User interface improvements .....	15
Improved background gradient.....	16
Tool Manager improvements.....	18
Tool Properties dialog improvements .....	19
Color improvements .....	20
Plunge tab .....	22
Helical Options .....	23
Show/hide all in Setup .....	24
Simulation start point buttons .....	24
File send.....	25
Custom setup sheets.....	26
Network database improvements .....	28
SQL Server Express now available .....	28
Simulation improvements.....	30
Better 3D simulation performance .....	30
Round stock display improved.....	30

2.5D improvements.....	31
Automatically avoid fixtures and clamps.....	31
4-axis wrapping: Easier programming of barrel cams .....	33
3D improvements.....	33
All 3-axis finishing operations can use a stock model .....	33
Advanced stock model settings .....	34
Turn improvements.....	37
Side liftoff angle.....	38
Arc lead-in/out for Turn/Bore feature .....	40
XBUILD: New transfer reserved word .....	41
Turn/Mill improvements.....	41
OD location for features from curves reinstated .....	41
Import improvements .....	42
Export improvements.....	42
XBUILD improvements .....	42

## What's New in FeatureCAM 2013 R1 44

General improvements.....	44
New selection method .....	45
Easier feed and speed unit conversion .....	47
Use STL as a clamp for 3D simulation .....	48
Snap to STL .....	49
Lengthen Sim Speed slider .....	49
Probing: New options for Probing UDF .....	50
XBUILD: New reserved words.....	50
FeatureMILL 2.5D improvements .....	52
Drill/Mill: Continuous spiral for rough endmill.....	53
Global Toolpath edits and transforms.....	54
Zigzag toolpath for Face.....	54
FeatureMILL 3D improvements .....	56
Stock model operation.....	56
New mid-level 3D product .....	57
Override default start points (3D MX) .....	58
Bottom up for Z-level finish (3D MX) .....	61
Z-level rough area filter (3D HSM).....	63
Step cutting (3D HSM).....	66
Cusp control for Steep and Shallow strategy (3D HSM).....	70
5-axis improvements.....	71
Positive/negative position (5AP).....	72
Override default start points (5AS) .....	74
5-axis options for more strategies (5AS) .....	76
Smoothing distance (5AS).....	76
Tool axis stabilization (5AS) .....	79
Improved tool axis options (5AS).....	84
FeatureTURN improvements .....	85
Integrated part handling (TURN) .....	85
Control feed rate on approach moves (TURN).....	87
Canned cycle support for semi-finish (TURN) .....	90
STL curve of revolution recognition (TURN).....	91

FeatureTURNMILL improvements .....	92
B-axis rotary (TURNMILL) .....	92
Use Y axis with Points List pattern (TURNMILL) .....	94
Import wizard (TURNMILL).....	95
AFR for SolidWorks Holes (TURNMILL).....	95

## Index

97



# What's New in FeatureCAM 2013 R3

FeatureCAM 2013 R3 offers all of the original features of FeatureCAM 2013 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 2013 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 2013 R2** (see page 15)
- **What's New in FeatureCAM 2013 R1** (see page 44)

---

## User interface improvements

FeatureCAM 2013 R3 includes these user interface improvements:

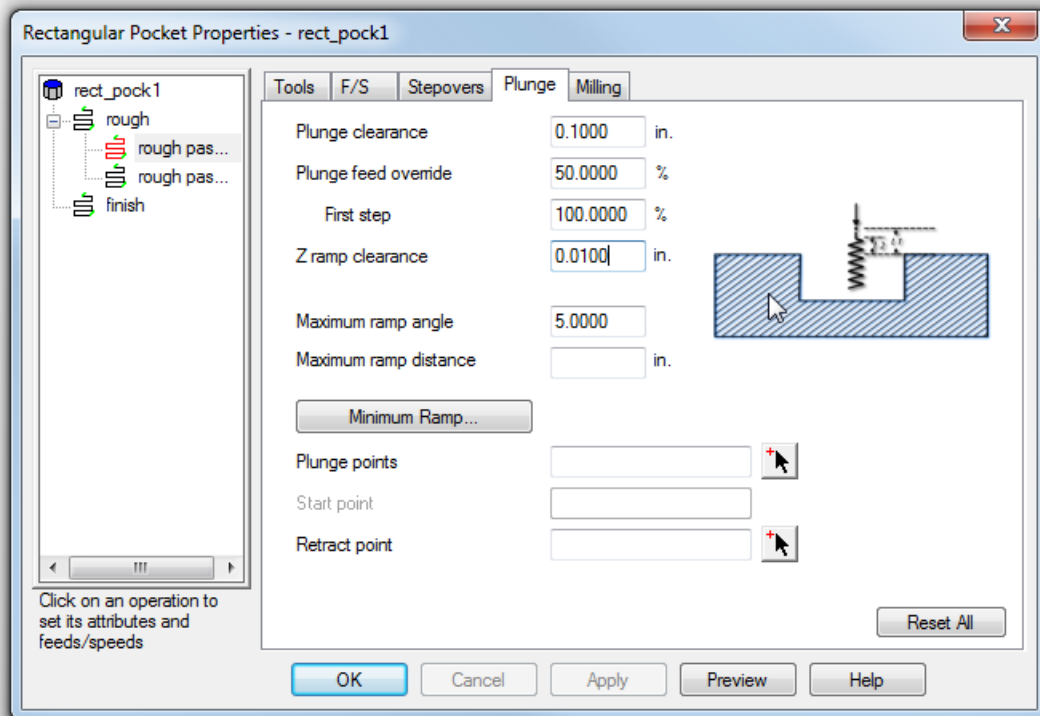
- Additional help information is available to you in the form of pop-up **Balloon help** (see page 1).

### Balloon help

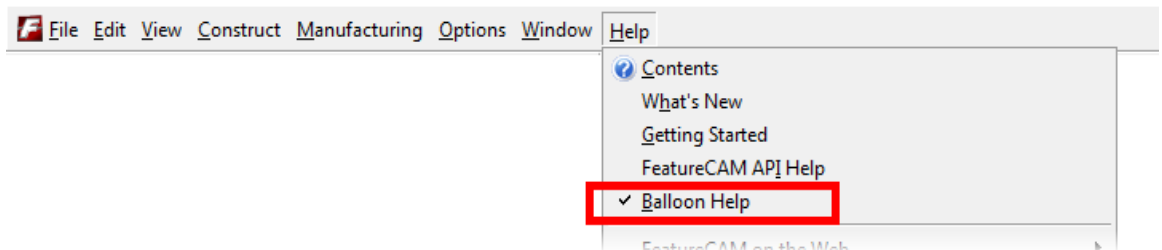
Starting with the **Plunge** tab attributes, balloon help will be rolled out across FeatureCAM attributes, making it easier and quicker for you to understand how a particular attribute works.



To display balloon help, hover your mouse pointer over the attribute image on the **Plunge** tab. A balloon help window is displayed containing further information, for example:



If you want to disable balloon help, you can deselect **Help > Balloon Help** from the menu.



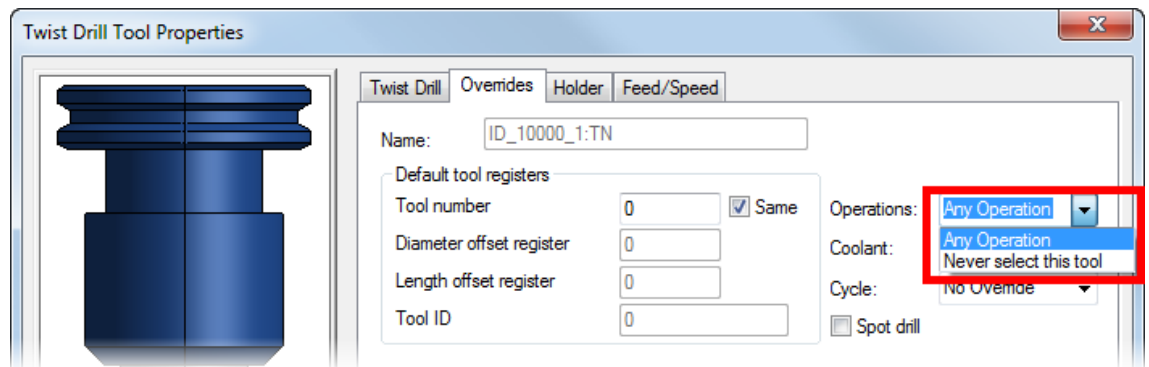
## Tooling improvements

FeatureCAM 2013 R3 includes these tooling improvements:

- The **Never select this tool** (see page 3) option is available for more milling tool types.
- You can now set the touch-off point for **Rounding Mill** (see page 3) tools at the tip of the tool.
- There is a new **Probing** (see page 4) tool group.

## Never select tool

In FeatureCAM 2013 R3, the **Never select this tool** option, that was previously available only for End Mill tools, has been added to other tool types, such as Twist Drill:

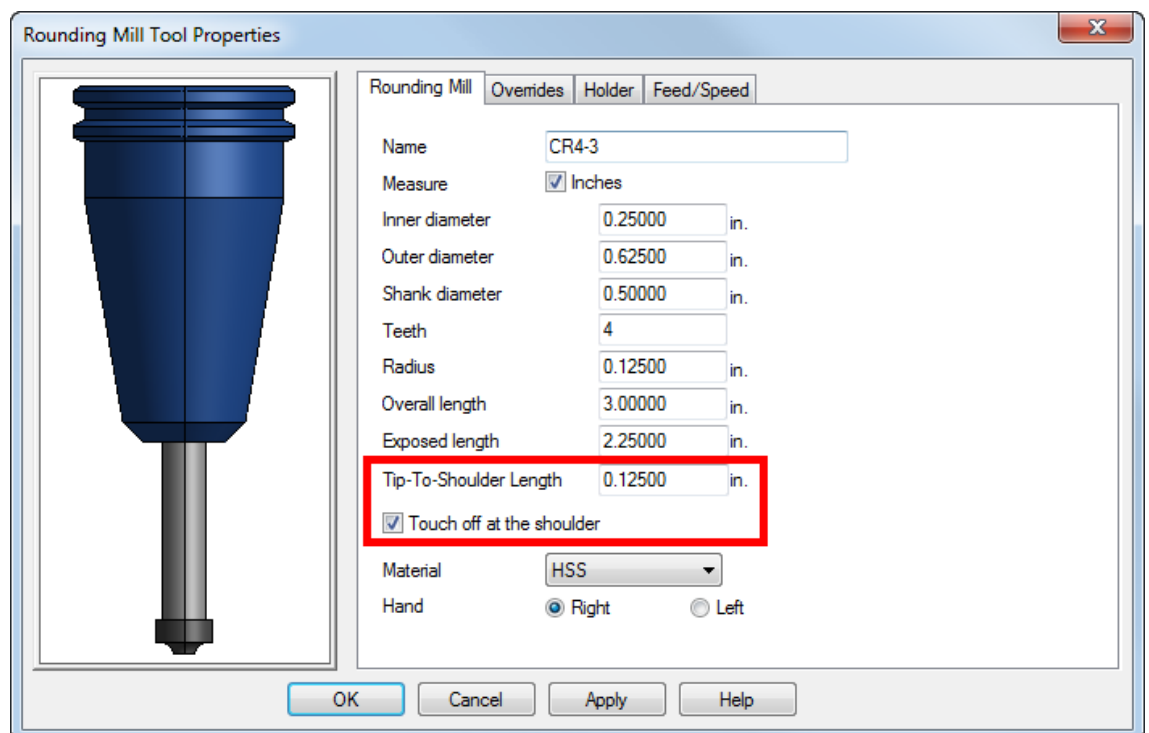


If you select **Never select this tool**, FeatureCAM does not select the tool automatically, but you can choose the tool manually.

## 2.5D: Rounding Mills — Set touch-off point at tip

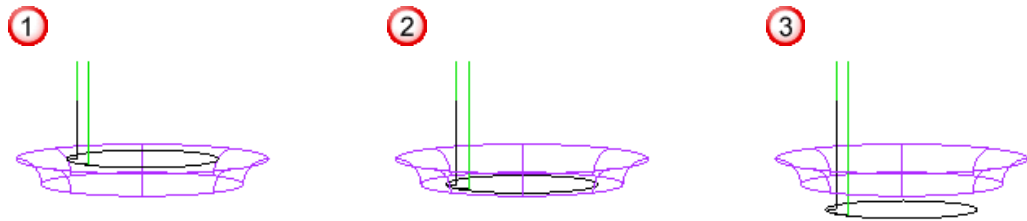
In FeatureCAM 2013 R3, you can set the touch-off point to be the tip of the tool instead of the shoulder.

There are two new options in the **Rounding Mill Tool Properties** dialog:



**Tip-To-Shoulder Length** — Enter the length between the shoulder and the tip of the tool. This length is the same as the arc **Radius** by default, but you can increase it.

**Touch off at the shoulder** — Rounding Mill tools touch off at the shoulder by default. Deselect this option to move the touch-off point to the tip of the tool.



① This example shows the default behavior with **Touch off at the shoulder** selected. The toolpath is at the top of the Round feature.

② This example shows **Touch off at the shoulder** deselected. The tool touches off at the tip. The toolpath is at the bottom of the Round feature.

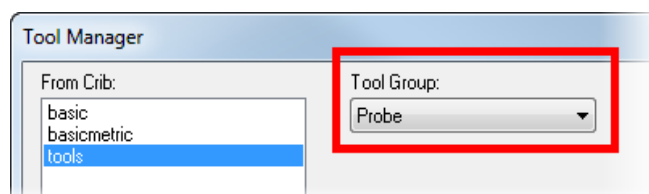
③ This example shows **Touch off at the shoulder** deselected and a **Tip-To-Shoulder Length** that is double the **Radius** value. The tool touches off at the tip. The toolpath is below the bottom of the Round feature.



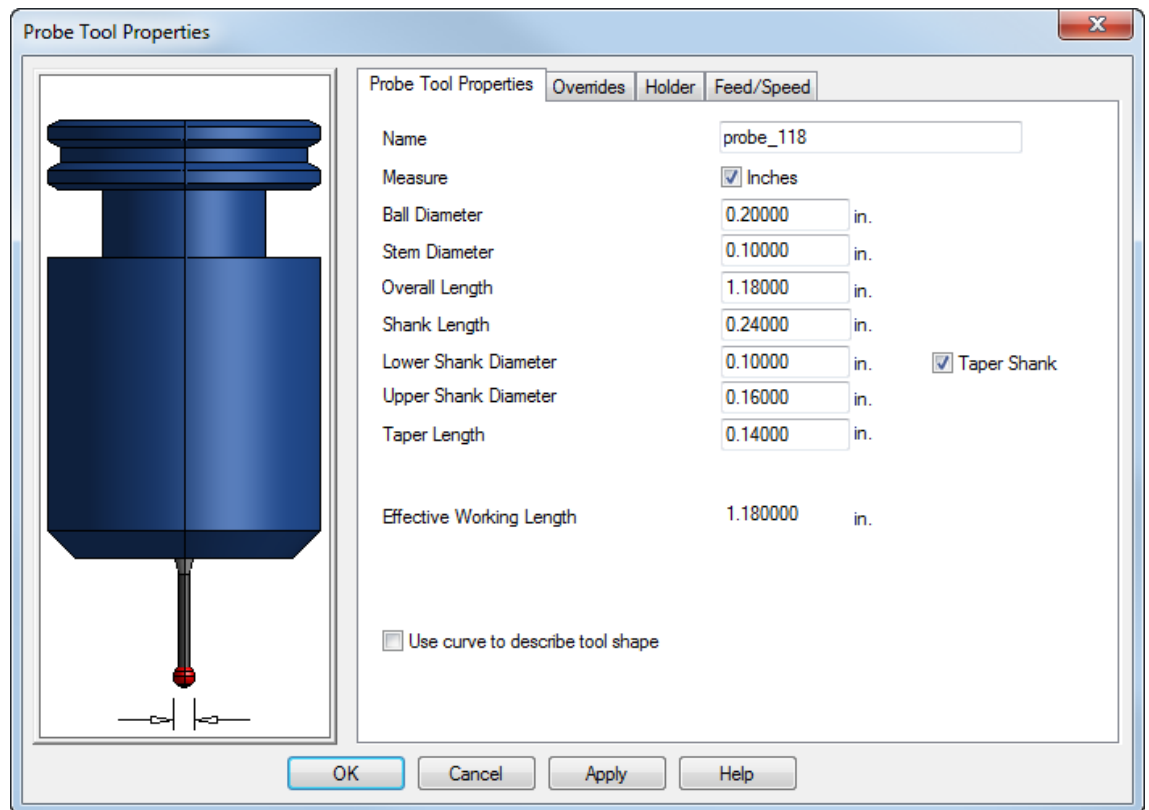
*When **Touch off at the shoulder** is selected, the **Exposed length** value does not include the tip. When it is deselected, the **Exposed length** value includes the tip length.*

## New probing tool group

There is a new **Probe** tool group in FeatureCAM 2013 R3:

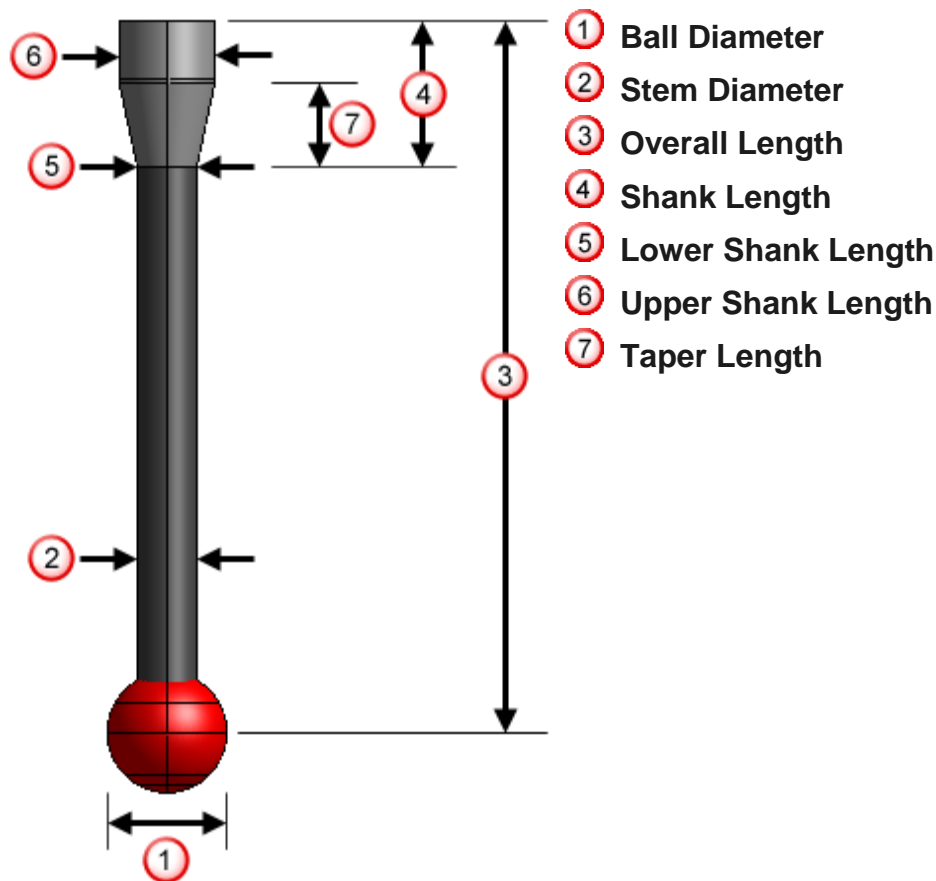


The **Probe Tool Properties** dialog works in a similar way to the other tool groups:



**Name** — Enter a name that identifies the tool. The name must be unique among all the tools in the crib.

**Measure** — This indicates the units that are used for reporting the tool's dimensions. Select **Inches** for inch units or deselect it for millimeters.



**Taper Shank** — Deselect this option to have a shank with a constant diameter and enter the **Shank Diameter** instead of **Lower Shank Length** and **Upper Shank Length**.

## 2.5D Milling improvements

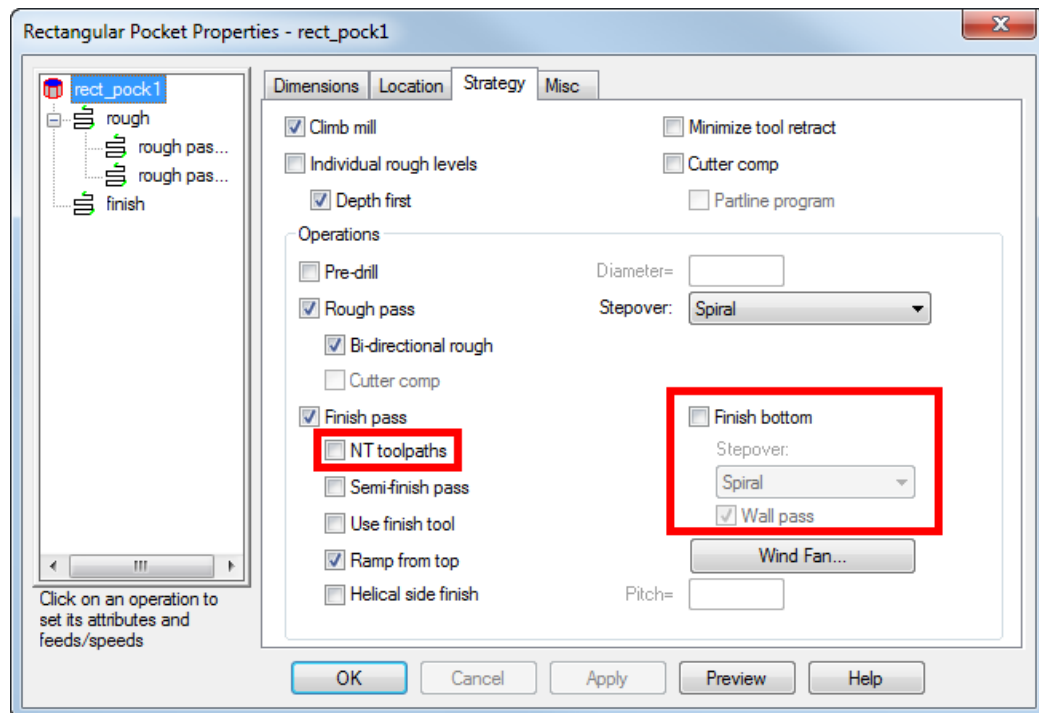
FeatureCAM 2013 R3 includes these 2.5D milling improvements:

- **New technology** (see page 6) (NT) toolpaths are available for finish operations.

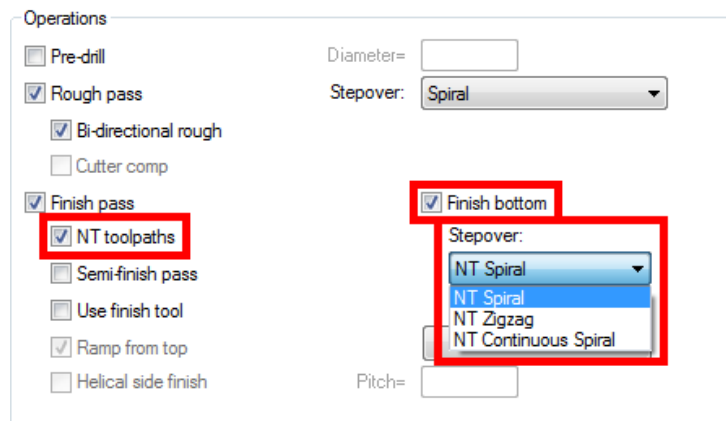
### NT toolpaths for finishing

In FeatureCAM 2013 R3, NT toolpaths are available for the Finish pass.

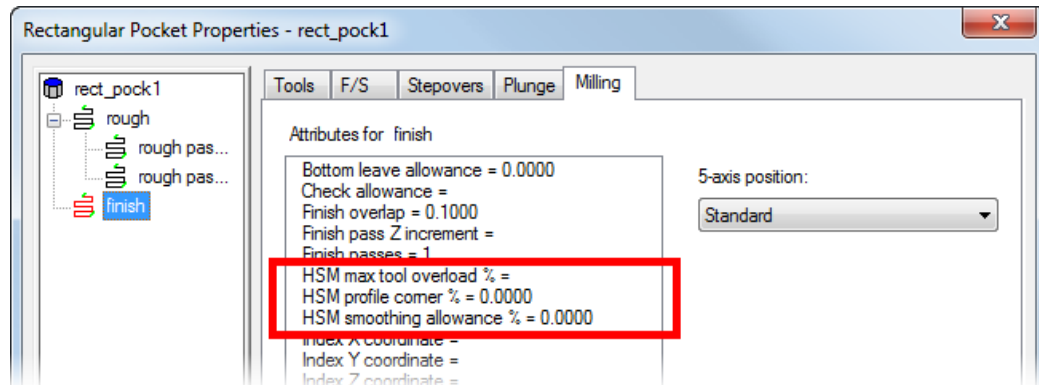
There is a new **NT toolpaths** option on the **Strategy** tab for the **Finish pass**. The **Finish bottom** and **Wall pass** options have moved.



Select **NT toolpaths** and **Finish bottom** to enable the **Stepover** menu, where you can select one of the NT toolpaths:



When using NT toolpaths for the Finish pass, the HSM attributes are available on the **Milling** tab, and work in a similar way to the Rough pass HSM attributes:



## 3D improvements

FeatureCAM 2013 R3 includes these 3D milling improvements:

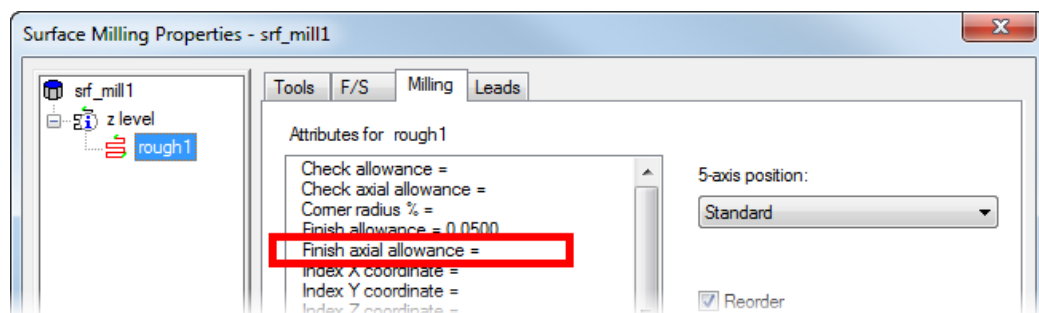
- Import surfaces (see page 8) (3D LITE)
- New **Finish axial allowance** (see page 8) attribute (3D LITE)
- New **Leave axial allowance** (see page 9) attribute (3D MX)
- 3D spiral outwards (see page 9) (3D HSM)

## Import surfaces (3D LITE)

You can now import 3D surfaces from **.dxf**, **.dwg**, and **.iges** files if you have the 3D LITE product.

## Finish axial allowance (3D LITE)

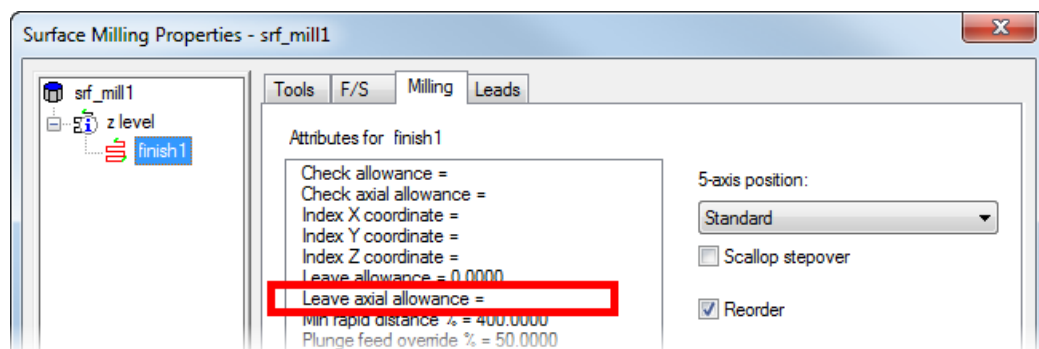
In FeatureCAM 2013 R3, you can set separate axial and radial finish allowance values for the Z-level rough strategy. There is a new **Finish axial allowance** attribute.



**Finish axial allowance** — Enter the amount of axial (XY) material to leave on a feature after the Rough pass. If you enter a value for **Finish axial allowance**, the value for **Finish allowance** is applied to radial (Z) material. If you leave **Finish axial allowance** blank, the value for **Finish allowance** is applied to axial and radial material.

## Leave axial allowance (3D MX)

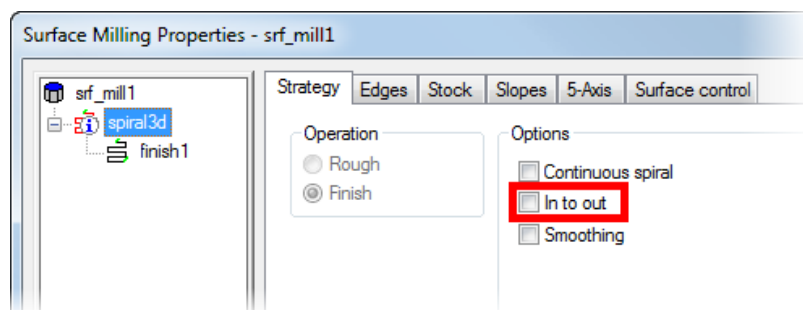
In FeatureCAM 2013 R3, you can set separate axial and radial leave allowance values for the Z-level finish strategy. There is a new **Leave axial allowance** attribute.



**Leave axial allowance** — Enter the amount of axial (XY) material to leave on a feature after the Finish pass. If you enter a value for **Leave axial allowance**, the value for **Leave allowance** is applied to radial (Z) material. If you leave **Leave axial allowance** blank, the value for **Leave allowance** is applied to axial and radial material.

## 3D spiral out

In FeatureCAM 2013 R3, you can make the 3D spiral strategy move outwards. Select the new **In to out** option on the **Strategy** tab:



*The **Continuous spiral** and **Smoothing** options have been moved to this tab.*

## Other important improvements

FeatureCAM 2013 R3 includes these other important improvements:

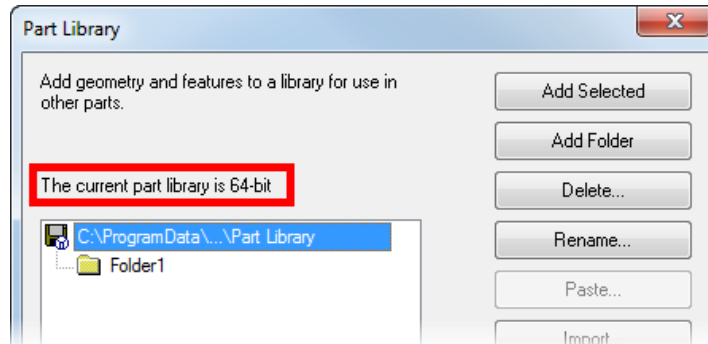


- The **Part Library** (see page 10) is available in 64-bit format.
- There is a new **Vertical mill/turn** (see page 10) document type.

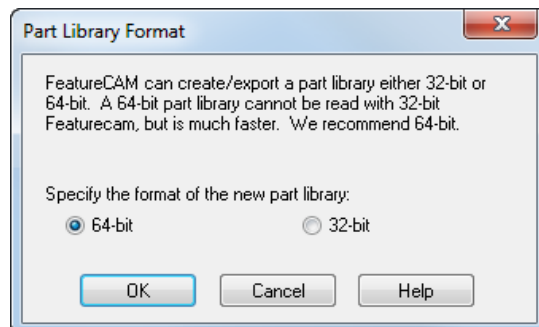
## Part library: 64-bit format

In FeatureCAM 2013 R3, you can create and export a 64-bit format part library.

The bit-ness of the current library is displayed in the **Part Library** dialog:



When you create or export a part library, the **Part Library Format** dialog is displayed, where you specify a 64-bit or 32-bit part library.

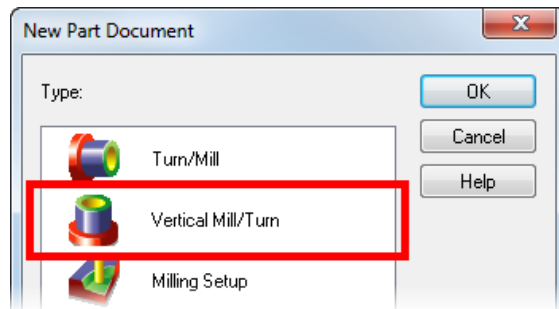


*A 32-bit version of FeatureCAM cannot read a 64-bit part library, but a 64-bit part library is faster and recommended.*

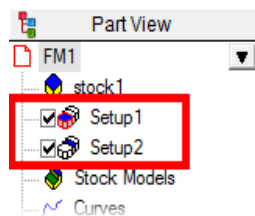
## Vertical mill/turn (TURNMILL, MTT, 5AP)

FeatureCAM 2013 R3 supports vertical mill/turn machines such as Okuma VTM-120YB, Mori Seiki SuperMILLER 400, Mori Seiki NMV5000 DBC, Mazak Integrex e-800V II (side spindle), Matsuura CUBLEX-25, and DMG FD Series.

There is a new **Vertical Mill/Turn** document type:



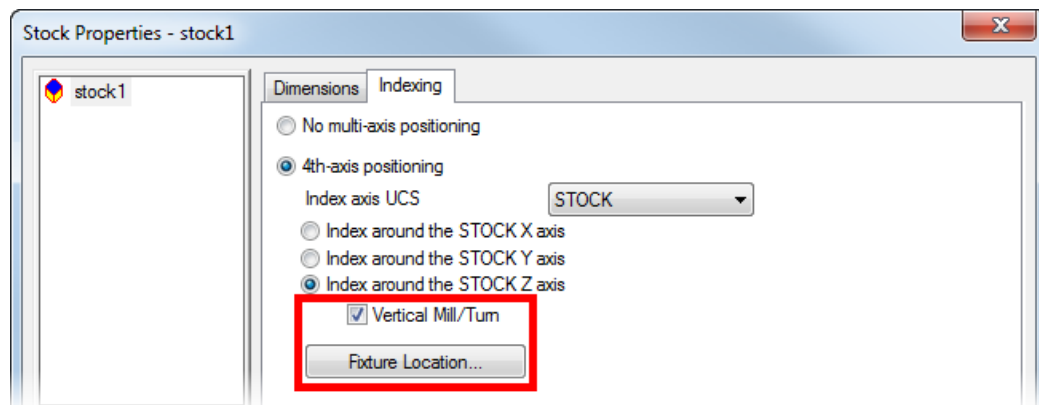
When you open a new Vertical Mill/Turn document, two Setups are created by default:



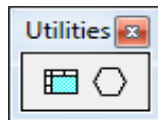
**Setup1** is a turning Setup

**Setup2** is a milling Setup

The new **Vertical Mill/Turn** option in **Stock Properties** is automatically selected for a Vertical Mill/Turn document, and the **Fixture Location** button is available for the milling part of the document:



There is a new [VerticalMillTurnAddin.bas](#) add-in available. Load and run the add-in to access these two buttons:



The **PostMillTurnSetups** button combines the two Setups into a single NC code file.

The **SimTurnMillingSetups** button combines the two Setups into one simulation for 3D or machine simulation.

---

## Import improvements

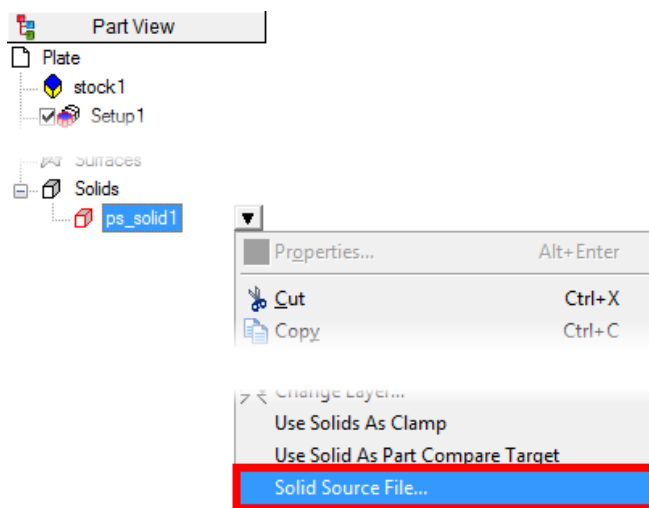
FeatureCAM 2013 R3 includes these import improvements:

- You can now manage associativity (see page 12) for SolidWorks files.
- The Parasolid kernel has been upgraded (see page 12) to version 24.1.

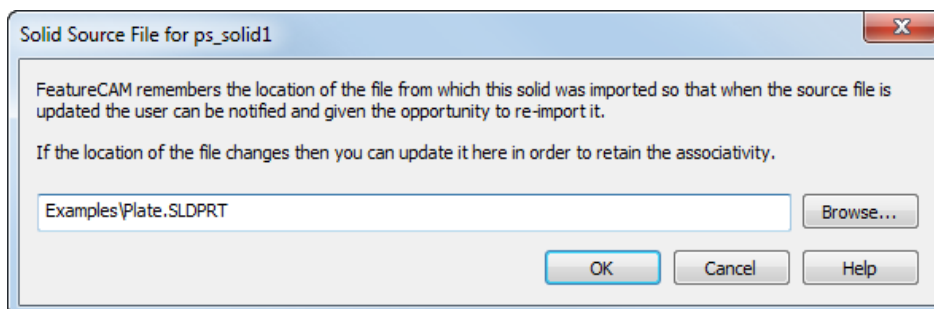
## SolidWorks: Manage associativity

In FeatureCAM 2013 R3, it is easier to manage the associativity with SolidWorks files. You can now change the **.sldprt** file name with which a FeatureCAM file is associated.

Right-click the name of the solid in the **Part View** and select **Solid Source File** from the context menu:



The **Solid Source File** dialog is displayed:



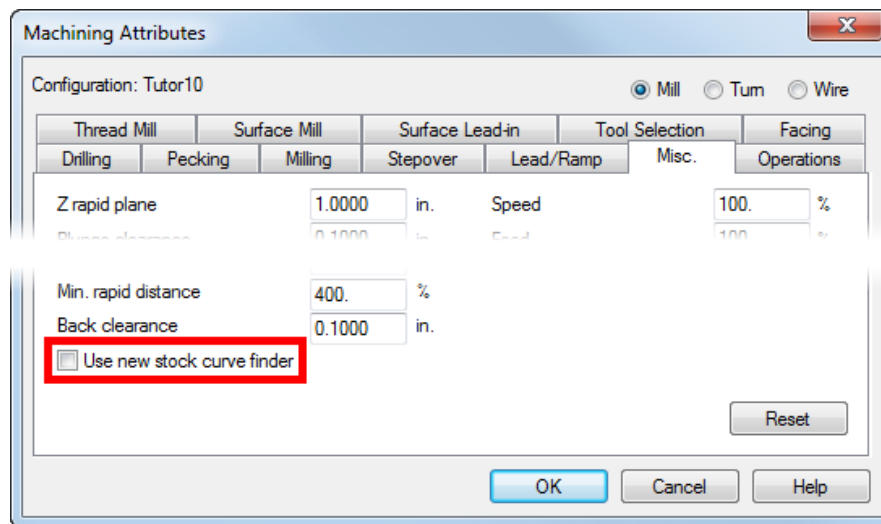
If the solid source **.sldprt** file is renamed or moved, you can set the new name and/or location here to keep the associativity. Click the **Browse** button to find the source file.

If associativity is kept, FeatureCAM displays a warning when the source file is updated, to ask if you also want to update the **.fm** file.

## Parasolid kernel upgrade to 24.1

This is a major upgrade of the Parasolid kernel for FeatureCAM, which fixes a number of bugs.

As a result of the upgrade, there is a new option in FeatureCAM.



In certain cases, the Parasolid kernel does not compute a Stock curve correctly. If you have problems trying to compute a Stock curve from a Parasolid file, try selecting the **Use new stock curve finder** option on the **Misc.** tab of **Machining Attributes**.



*This option is deselected by default because the alternative method is slower.*

---

## Turn/mill and 4th-axis improvements

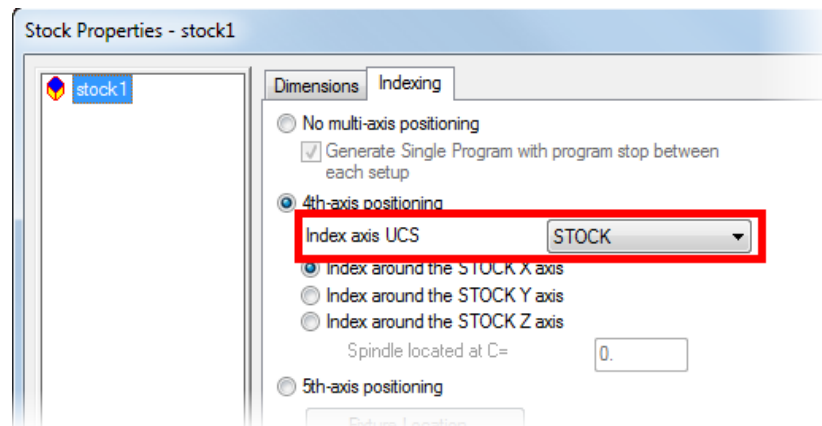
FeatureCAM 2013 R3 includes these Turn/mill improvements:

- You can now change the index axis (see page 13) for turn/mill and 4th-axis parts.

### Turn/mill & 4th-axis: Change index axis

In FeatureCAM 2013 R3, you can change the index axis for turn/mill and 4-axis positioning parts. You no longer need to use the STOCK index axis.

There is a new **Index axis UCS** option on the **Indexing** tab of the **Stock Properties** dialog:



If you create a new UCS, it is available for you to select in the **Index axis UCS** menu.

# What's New in FeatureCAM 2013 R2

FeatureCAM 2013 R2 offers all of the original features of FeatureCAM 2013 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 2013 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 2013 R1** (see page 44)
- **What's New in FeatureCAM 2012 R3**

---

## User interface improvements

FeatureCAM 2013 R2 has a more modern and attractive user interface:

- The background (see page 16) is improved and matches your choice of interface style.
- The **Tool Manager** (see page 18) and **Tool Properties** (see page 19) dialogs have many improvements.
- There have been various color improvements (see page 20).

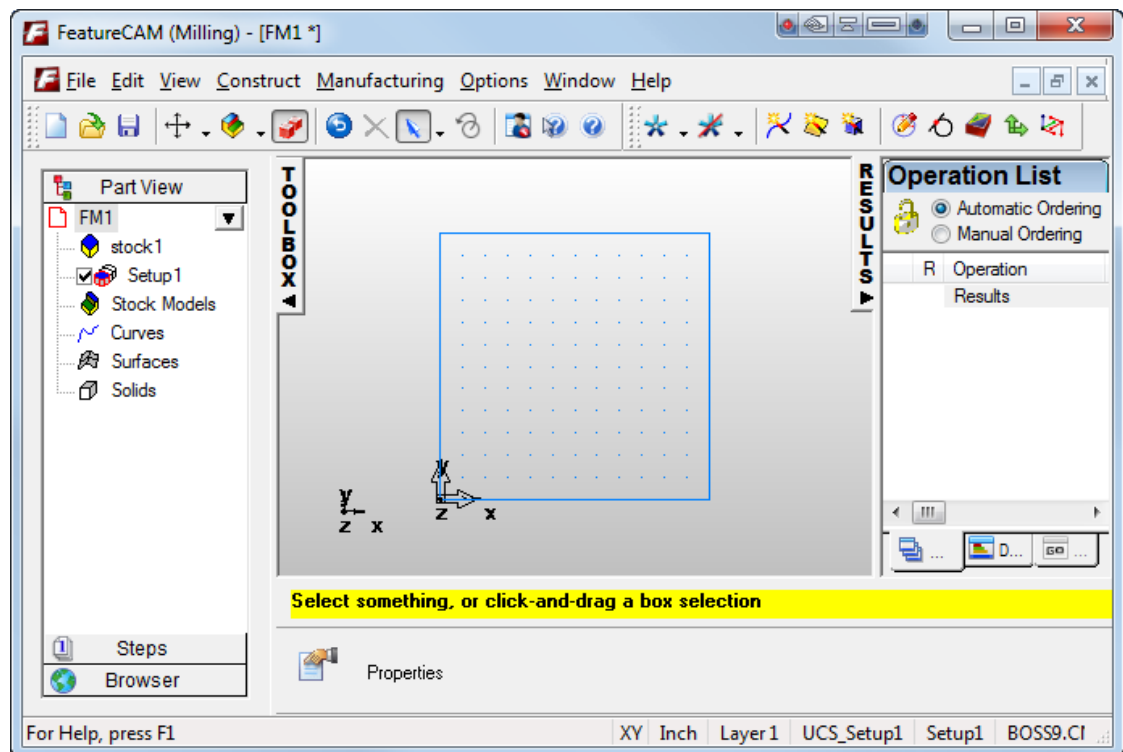
Other user interface improvements include:

- A new **Plunge** (see page 22) tab grouping all attributes relating to plunging.
- The machining attributes relating to helical ramping are now in a new dedicated **Helical Ramp Options** (see page 23) dialog.

- You can now easily show or hide all features (see page 24) in a Setup.
- There are new buttons available relating to simulation starting points (see page 24).
- You can now zip multiple files (see page 25) relating to the current document, ready to email.

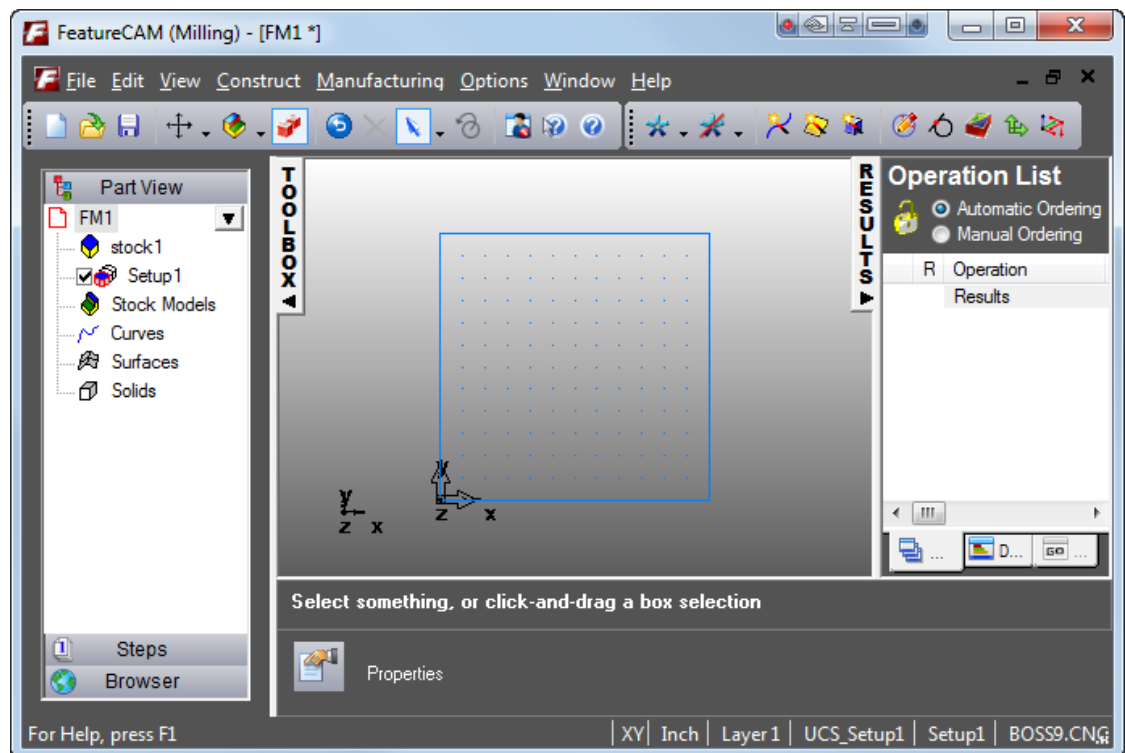
## Improved background gradient

The default background gradient has been improved:

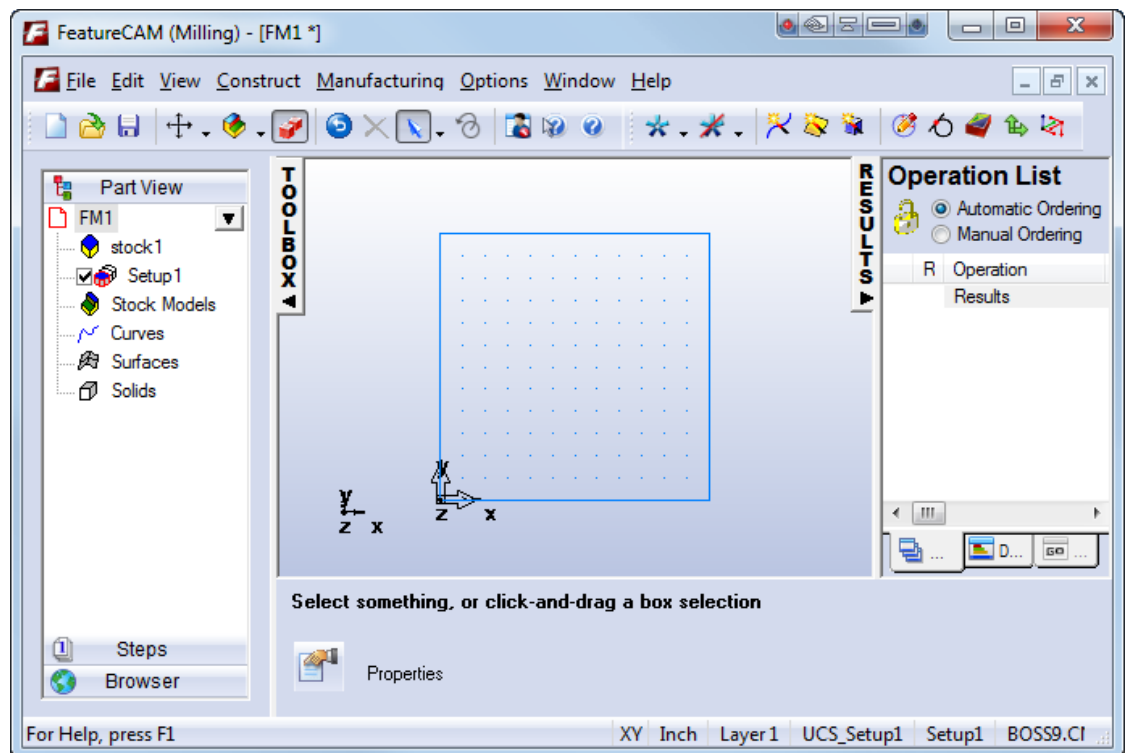


If you change the **Style** of the interface in the **Customize Toolbars** dialog, FeatureCAM asks you if you want to change the background color scheme to one that matches:

## Shaded Grey:



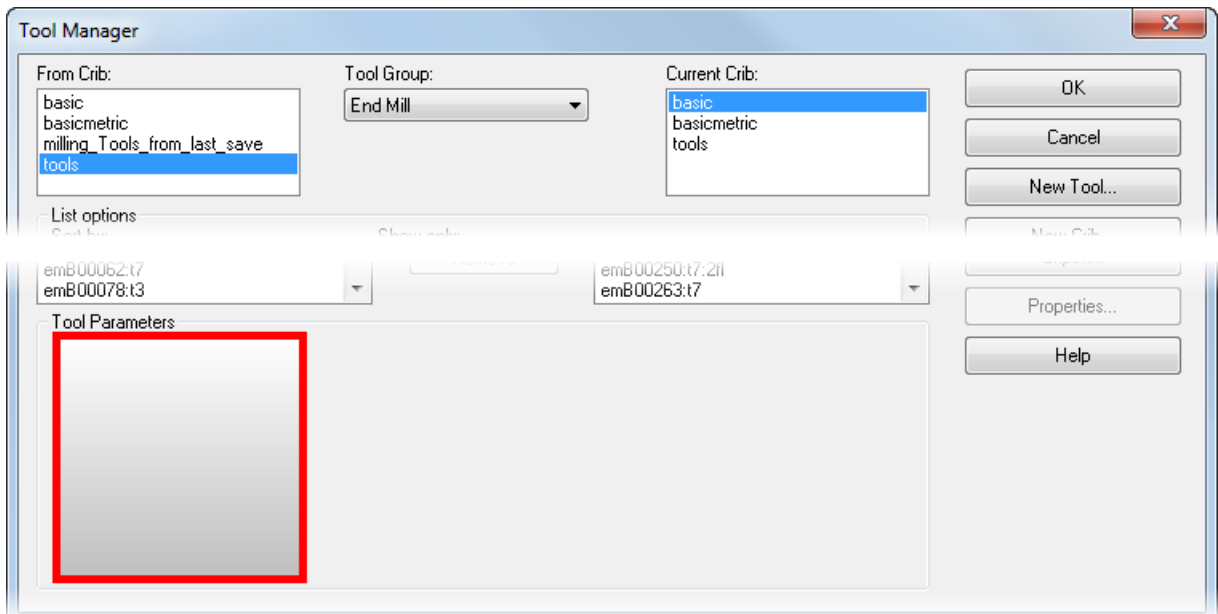
## Glass:



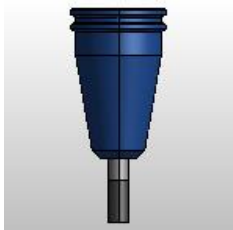


## Tool Manager improvements

The **Tool Manager** preview window background now matches the background (see page 16) of the graphics area:



When you select a tool, the preview image is now much more detailed. It shows the holder and the shank:



If you move your mouse pointer over the image, the tool is automatically centered:



*To turn off this setting, right-click on the image and deselect **Automatic Center Tool** from the context menu.*

You can also display a representation of the flutes on end mill tools. To do this, right-click in the preview window and select **Show Flutes on End Mills** in the context menu. The display represents the number of flutes and the handedness of the tool, for example:

2 flutes,  
right-handed



4 flutes,  
right-handed



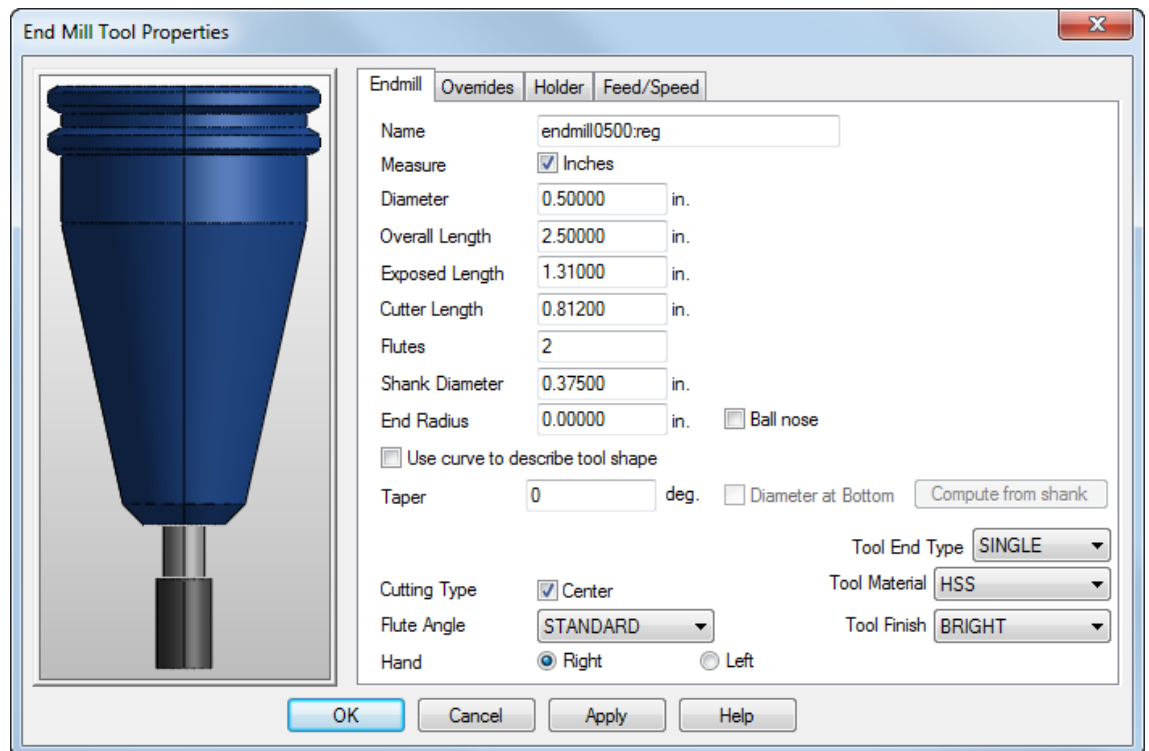
4 flutes,  
left-handed



## Tool Properties dialog improvements

The **Tool Properties** dialog has been improved in FeatureCAM 2013 R2.

The tool preview window, now on the left of the dialog, is much larger and separate from the tabs so that it is always visible:

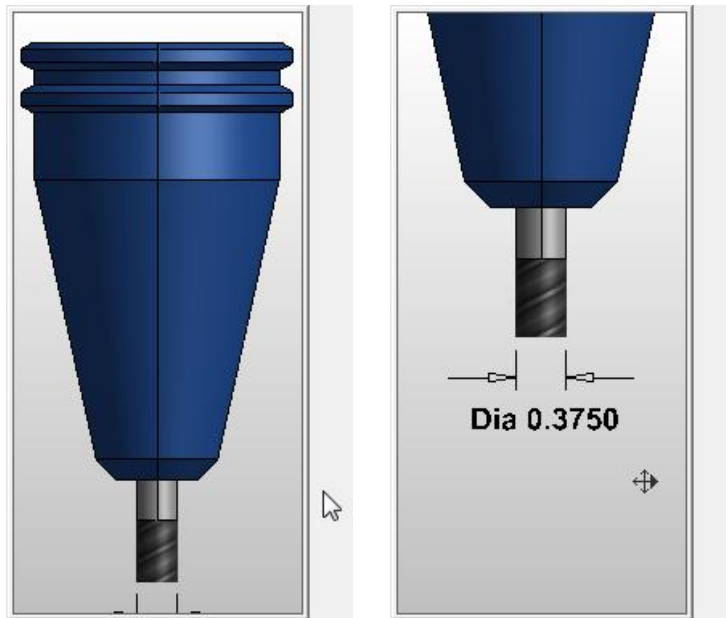


The tool holder is now shown in the preview window, by default. To hide the tool holder, right-click in the preview window and deselect **Show Tool Holder** in the context menu.

The tool shank is also now shown.

The tool, holder, and shank have a dark outline to make them clearer.

You can automatically center the tool in the preview window by moving your mouse pointer over the image:



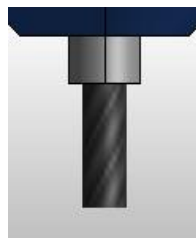
*To turn off this setting, right-click on the image and deselect **Automatic Center Tool** from the context menu.*

You can also display a representation of the flutes on end mill tools. To do this, right-click in the preview window and select **Show Flutes on End Mills** in the context menu. The display represents the number of flutes and the handedness of the tool, for example:

2 flutes,  
right-handed



4 flutes,  
right-handed



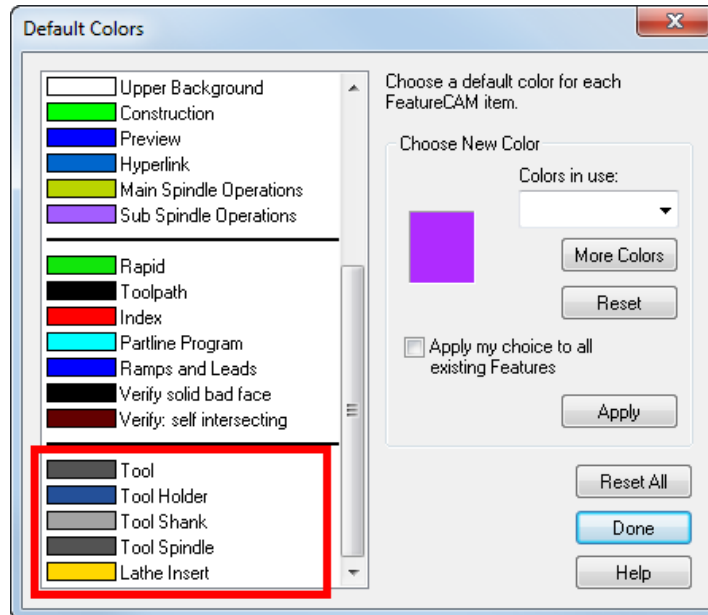
4 flutes,  
left-handed



## Color improvements

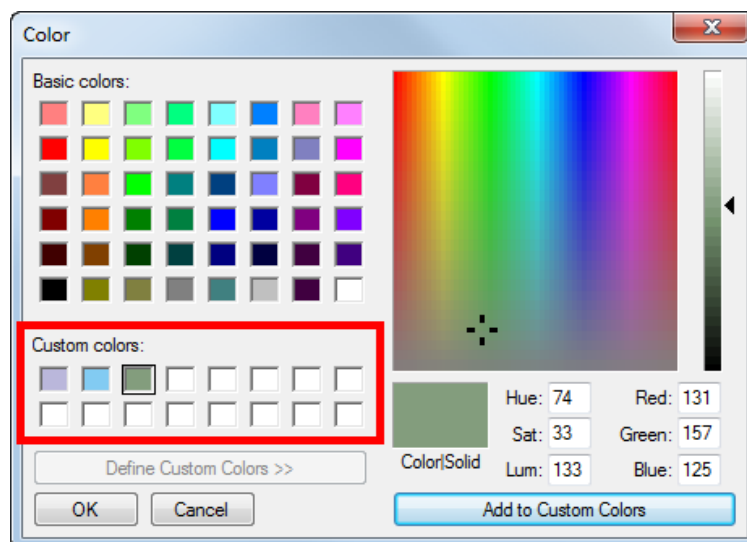
In FeatureCAM 2013 R2, there are new color controls throughout, to make the software more appealing.

The default **Feature** and **Sub Spindle Operations** colors have been improved and you can now change the default **Tool** colors:



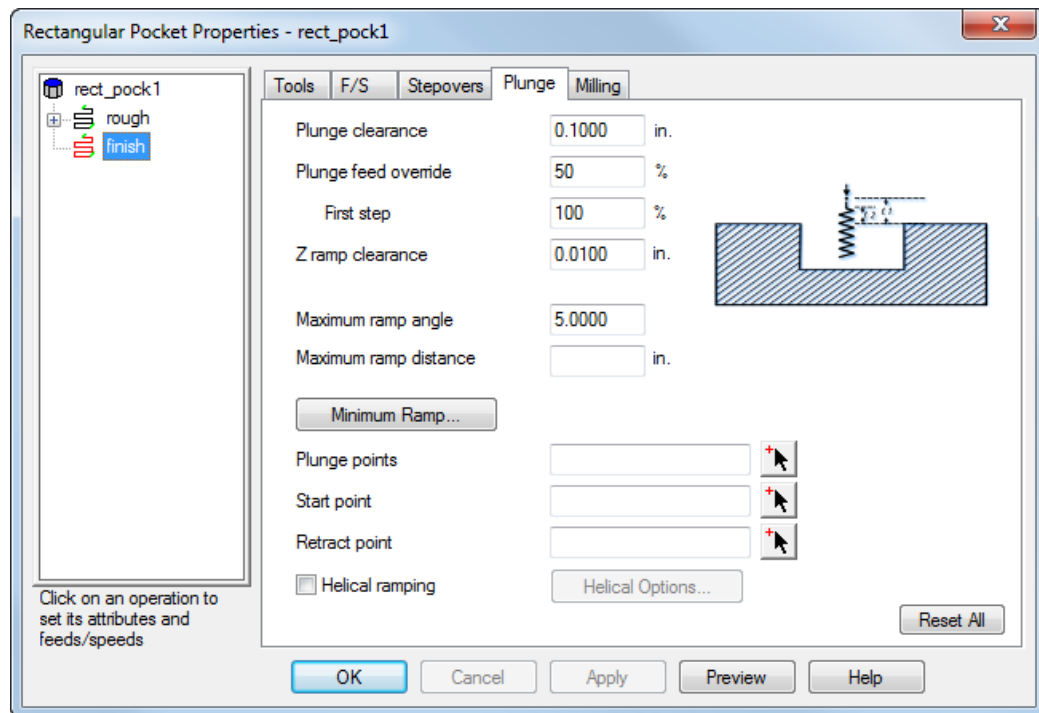
There is a new **Reset All** button in the **Default Colors** dialog to enable you to reset the default color of all items at the same time.

Any **Custom colors** that you define are saved in the settings file and are available next time you use FeatureCAM.

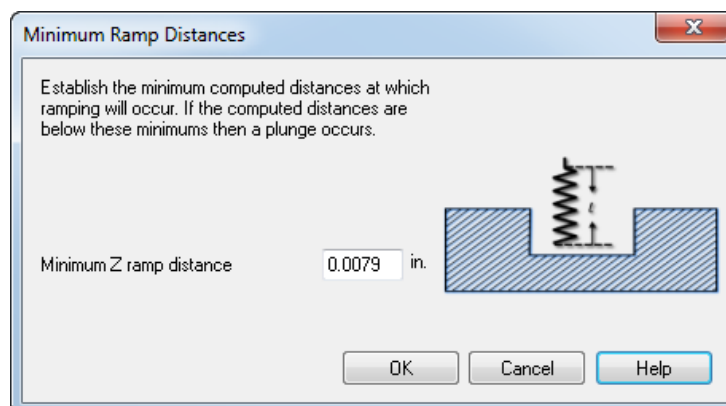


## Plunge tab

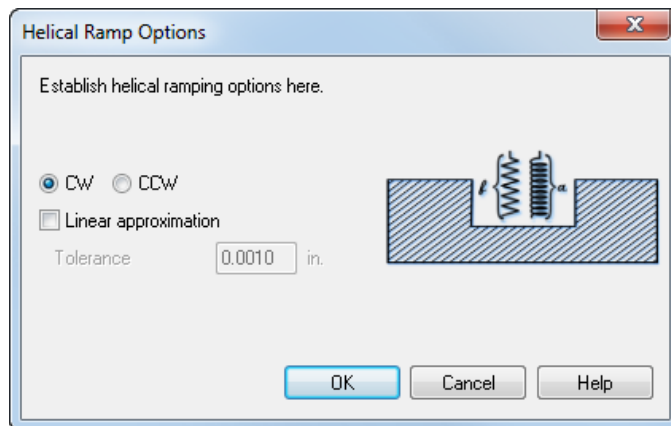
In FeatureCAM 2013 R2, the attributes relating to plunging are conveniently located together on a new **Plunge** tab:



Click the **Minimum Ramp** button to open the new **Minimum Ramp Distances** dialog:

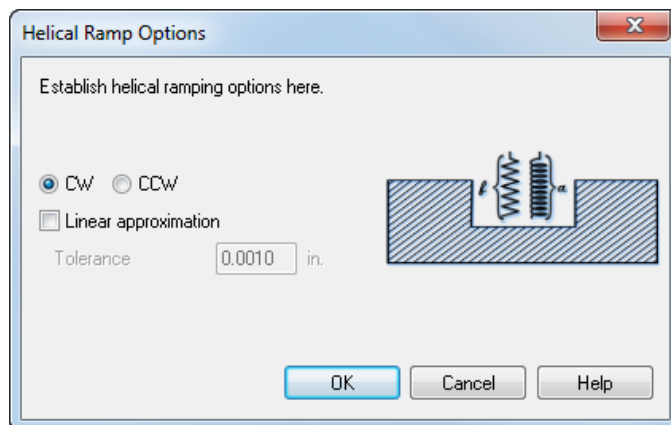


When using **Helical** ramping, click the **Helical Options** button to open the **Helical Ramp Options** dialog:

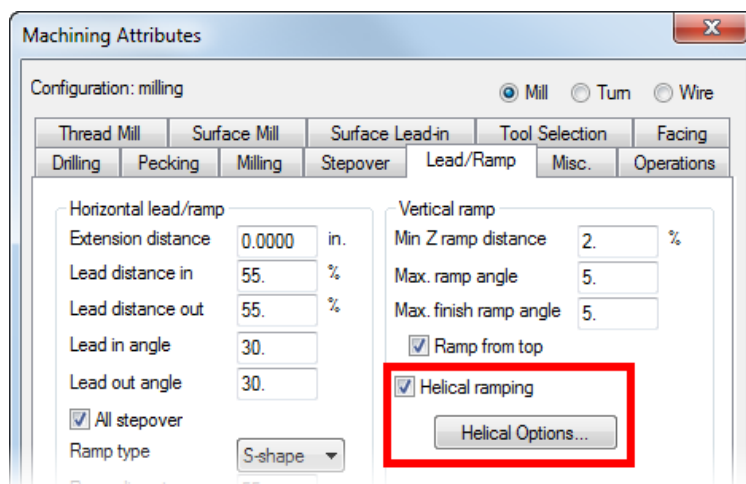


## Helical Options

The machining attributes relating to helical ramping are now in a new dedicated **Helical Ramp Options** dialog:



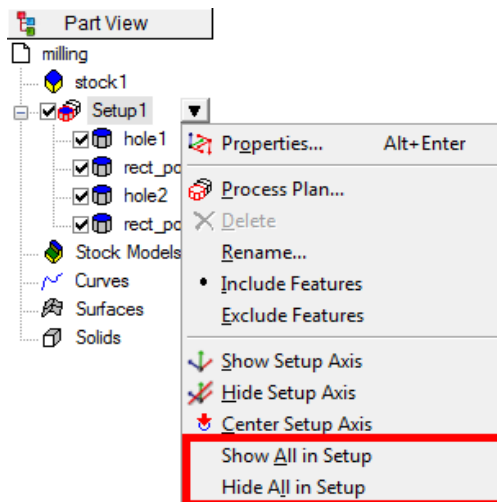
To open this dialog, select **Helical ramping** and click the **Helical Options** button on the **Lead/Ramp** tab:



## Show/hide all in Setup



In FeatureCAM 2013 R2, you can easily show or hide all features in a Setup. There are two new options in the context menu for a Setup in the **Part View**:

- **Show All in Setup**
- **Hide All in Setup**



## Simulation start point buttons

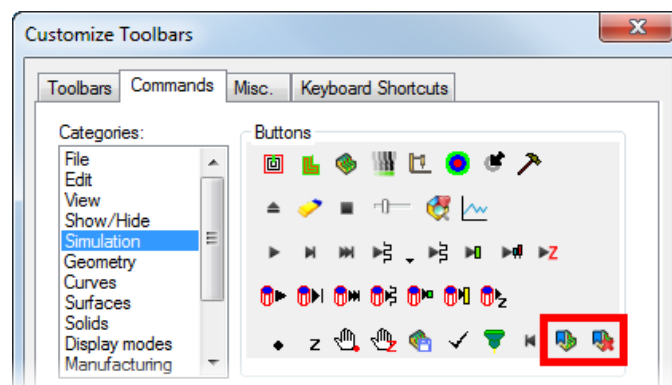
In FeatureCAM 2013 R2, there are two new buttons available:

-  **Set sim results as starting point**
-  **Clear starting point**

To add these buttons to a toolbar:

- 1 Select **View > Toolbars** from the menu.  
The **Customize Toolbars** dialog is displayed.
- 2 On the **Commands** tab, select **Simulation** in the **Categories** list.

The new buttons are available:

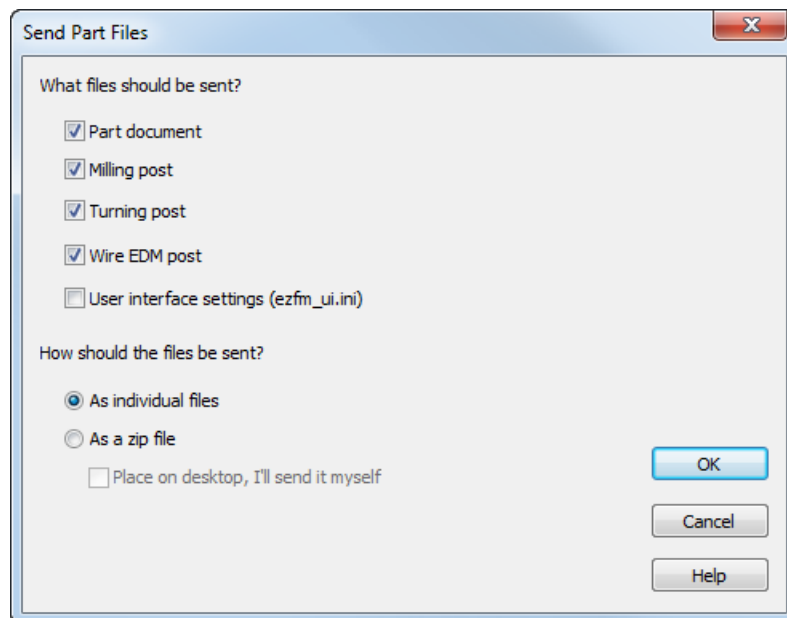


- 3 Drag the icon you want onto the toolbar you want to add it to.

## File send

In FeatureCAM 2013 R2, you can zip multiple files relating to the current document, ready to email.

Open the new **Send Part Files** dialog by selecting **File > Send** from the menu.



Select the files you want to send from:

**Part document** — The current [.fm](#) file.

**Milling post** — The current Milling [.cnc](#) file.

**Turning post** — The current Turning [.cnc](#) file.

**Wire EDM post** — The current Wire EDM [.cnc](#) file.

**User interface settings** — The [.ini](#) file containing your user interface preferences.

Select how you want to send the files:

**As individual files** — The individual files are attached to a new email.

**As a zip file** — The files are zipped and the [.zip](#) file is attached to a new email.

**Place on desktop** — The files are zipped and the [.zip](#) file is saved to your desktop.



## Custom setup sheets

In FeatureCAM 2013 R3, you can create custom setup sheets for milling, turning, and turn/mill parts using a new add-in.


The add-in extracts information from the **Part Documentation** dialog and enables you to take images for each Setup.


You can use setup sheets to give information to the machine operator about the manufacturing, tooling, and toolpaths of a part.

To load and run the add-in:

- 1 Select **Options > Add-Ins** from the menu.

The **Macro Add-ins** dialog is displayed.

- 2 Click the **Browse**  button and browse to the [SetupSheet.dll](#) file. If you installed FeatureCAM in the default location, the file is at [C:\Program Files\Delcam\FeatureCAM\Addins\SetupSheet\SetupSheet.dll](#).
- 3 In the **Macro Add-ins** dialog, in the **Add-In Files** list, ensure that the check box to the left of the [SetupSheet.dll](#) file address is selected.
- 4 Click **OK**.

The **Utilities** toolbar is displayed, containing the **SetupSheet**  button.

- 5 Click the **SetupSheet** button to run the add-in.

The **Setup Sheet Options** dialog is displayed:

The **Title**, **Author**, **Note 1**, **Note 2**, and **Comments** values are copied from the **Documentation** tab of the **Part Documentation** dialog and you can edit them there.

You can use this dialog to capture an image for each Setup in the current document, as well as an image to represent the whole document.

To capture an image for a Setup, select the Setup name in the **Setups** list and click the **Capture Setup Image** button to capture the current contents of the Graphics window.



*First run a simulation only for the Setup you want to capture by deselecting other Setups in the **Part View**; then adjust the view to show a good orientation of the Setup, and open the **Setup Sheet Options** dialog to capture the image in the Graphics window.*

You must use a template to create the setup sheets. Click the **Browse Template** button to find and set the template you want to use. There is a template, [SetupSheetTemplate.html](#), in the [Addins\SetupSheet](#) folder.

To create the setup sheets, click the **Create Setup Sheets** button. The part is simulated to generate toolpaths and the setup sheet is displayed in your web browser. You can save the HTML file from your browser.

To open the setup sheet in the FeatureCAM Browser, select **File > Open** from the menu and browse to the setup sheet HTML file.

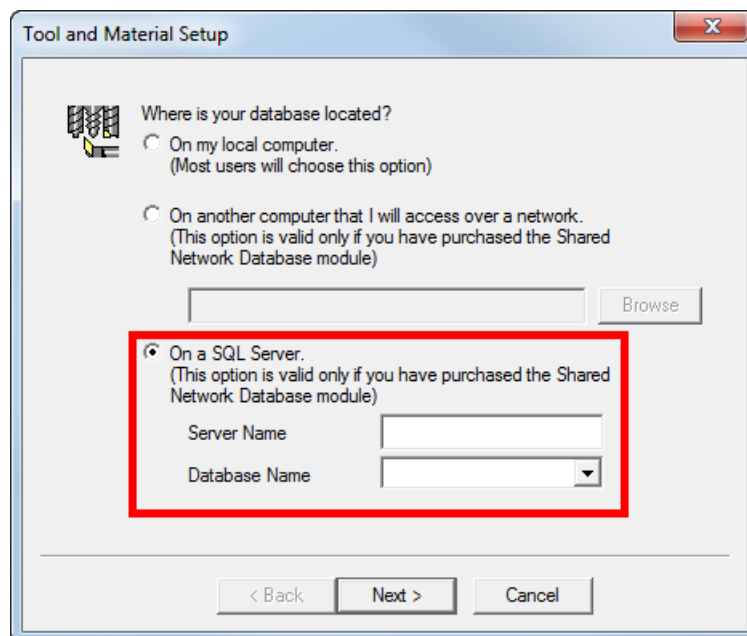
## Network database improvements

FeatureCAM 2013 R3 includes this network database improvement:

- You can now use Microsoft SQL Server Express (see page 28) for your network database.

### SQL Server Express now available

In FeatureCAM 2013 R3 you can use SQL Server for your network database. This gives better performance and reliability for your network database. There is a new option in the **Tool and Material Setup** dialog:



Microsoft SQL Server 2012 Express is a free edition of SQL Server that you may use to host your Delcam tools and materials database used by FeatureCAM. It is available in 32-bit and 64-bit editions and in these languages:

- |                         |                       |
|-------------------------|-----------------------|
| ▪ Chinese (simplified)  | ▪ Japanese            |
| ▪ Chinese (traditional) | ▪ Korean              |
| ▪ English               | ▪ Portuguese (Brazil) |
| ▪ French                | ▪ Russian             |
| ▪ German                | ▪ Spanish             |
| ▪ Italian               |                       |

When using Microsoft SQL Express to host your FeatureCAM tools and materials you need a server computer to host the SQL Server software and the tools/materials database. This server computer should be a different computer from the client workstations where users are running FeatureCAM.



*The database server can host multiple databases. So your server can contain several different FeatureCAM tools and materials databases, each from different versions of FeatureCAM, different departments in your organization, or for whatever other reasons you require. Generally this isn't be necessary, but it is possible if you need to do this.*

## Installation

- 1 Pick a computer at your facility that you wish to host the FeatureCAM tools and materials database. We'll call this the *database server*. The database server must meet these criteria:
    - It must be running Microsoft Windows.
    - It must be connected to your network.
    - It must be able to run Microsoft SQL Server Express 2012.
    - You can see system requirements at <http://msdn.microsoft.com/en-us/library/ms143506.aspx> (<http://msdn.microsoft.com/en-us/library/ms143506.aspx>)
  - 2 Find out the hostname of the database server.
  - 3 Download either the **Express (Database Only)** or **Express with Tools** from <http://www.microsoft.com/betaexperience/pd/SQLEXPCTAV2/en-us/default.aspx> (<http://www.microsoft.com/betaexperience/pd/SQLEXPCTAV2/en-us/default.aspx>). If your **database server** is a 32-bit Windows, then you'll need to download the 32-bit version of SQL Server Express. If 64-bit, then download the 64-bit version instead.
- 
- The bitness of your download must match the bitness of the database server computer, and has nothing to do with the bitness of your client workstations. You may download the language of your choice.*
- 4 Install Microsoft SQL Server Express on your database server.
  - 5 From any client workstation using FeatureCAM 2013 R3, find **INITDB** in the **Start** menu and run it.
  - 6 In the **Tool and Material Setup** dialog, select **On a SQL Server** and enter the hostname of the database server computer as the **Server Name**.

- 7 After entering the **Server Name**, you can select an existing database on the database server in the **Database Name** menu, or create a new one.
- 8 The first time you run of INITDB you can do one of the following:
  - Initialize a new database with all the default tools.
  - Import tools from another database elsewhere on your network
  - Upgrade an existing database to the latest version of FeatureCAM and load it with all of the default tooling.Subsequent client installations default to this choice.

---

## Simulation improvements

FeatureCAM 2013 R2 includes these simulation improvements:

- 3D simulation (see page 30) is faster.
- Round stock (see page 30) is displayed more accurately.

### Better 3D simulation performance

3D simulation performance has been improved in FeatureCAM 2013 R2.

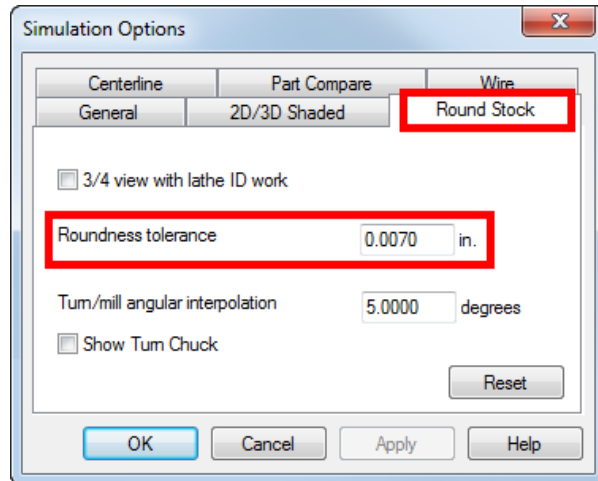
The results of the changes made are very dependent on the simulation, and also depend to some extent on the graphics card being used. However, as a general rule for simple simulations being rendered after every cut, the overall speed (including cutting as well as rendering) has been improved by 10% to 15%.

For very large tools, performance was often poor because the entire tool, often consisting of thousands of facets, was being simulated, when only a small portion of it affected the stock. The changes in this version mean that 3D simulation will often recognize these situations and only carry out simulation with the part of the tool that is close to the stock. This can give big performance improvements in some situations, and often gives improved quality for multi-axis simulations.

### Round stock display improved

In FeatureCAM 2013 R2, you can make the simulation of round stock (used by turning and indexing parts) more accurate.

There is a new **Round Stock** tab in **Simulation Options** which includes the new **Roundness tolerance** attribute.



**Roundness tolerance** — Reduce the tolerance to make round stock appear rounder and less faceted.

Higher **Roundness tolerance**:



Lower **Roundness tolerance**:



---

## 2.5D improvements

FeatureCAM 2013 R2 includes these 2.5D improvements:

- You can automatically avoid fixtures and clamps (see page 31).
- With 4-axis wrapping, it is easier to program barrel cams (see page 33).

### Automatically avoid fixtures and clamps

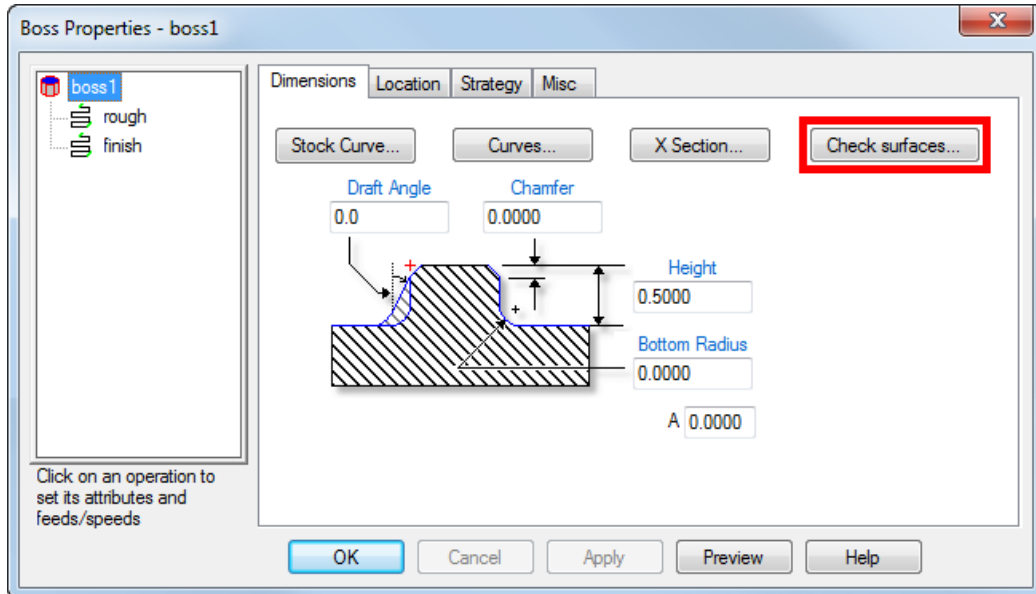
In FeatureCAM 2013 R2, you can automatically clip or trim 2.5D NT roughing toolpaths against fixtures and clamps.

These features are supported:

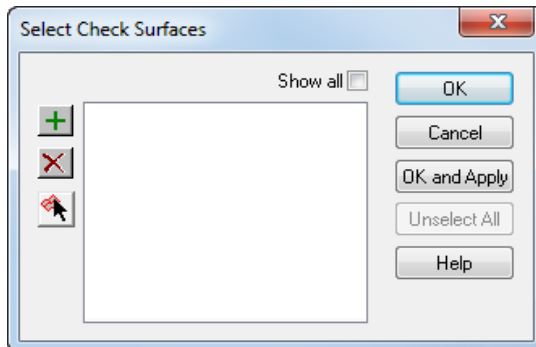
- Boss
- Pocket
- Side

- Face features created with feature recognition (if the plane of the face intersects the clamp solids)

When you select an NT roughing toolpath as the **Stepover type** on the **Strategy** tab for one of these features, a new **Check surfaces** button is available on the **Dimensions** tab:



Click the **Check surfaces** button to open the **Select Check Surfaces** dialog:



Use this dialog to select surfaces you want to use to limit machining.

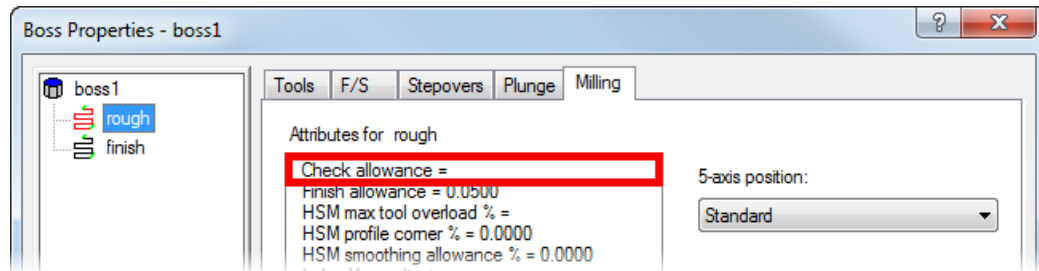
Check surfaces are surfaces that denote areas that you do not want to machine. Select surfaces that are more horizontal than vertical. A check surface acts as a boundary up to which milling occurs.



*If you select a vertical check surface the milling may resume on the other side of it if the surface to be milled extends beyond the check surface.*

- 1 Select the surface(s) in the list box or click **Pick** and select a surface with the mouse. To pick additional surfaces, click **Pick** again before selecting each additional surface.
- 2 Click **OK** to return to the **Feature Properties** dialog.

Click **OK** and **Apply** to apply your surface selection to the feature and return to the **Feature Properties** dialog. Optionally enter the minimum distance that you want to leave around the check surface(s), using the new **Check allowance** attribute on the **Milling** tab for the rough operation:



## 4-axis wrapping: Easier programming of barrel cams

In FeatureCAM 2013 R2, it is easier to program barrel cam features. This particular feature uses 4-axis milling and takes a centerline of the cam slot, along with the diameter of the follower, and produces a toolpath with a tool of a given diameter that may be smaller than the follower. When wrapped around the X axis, small Y moves are used in conjunction with adjustments to A in order to get the walls of the slot to be correct.

---

## 3D improvements

FeatureCAM 2013 R3 has these 3D improvements:

- All 3-axis finishing operations (see page 33) can now use a stock model.
- There are now advanced stock model settings (see page 34).

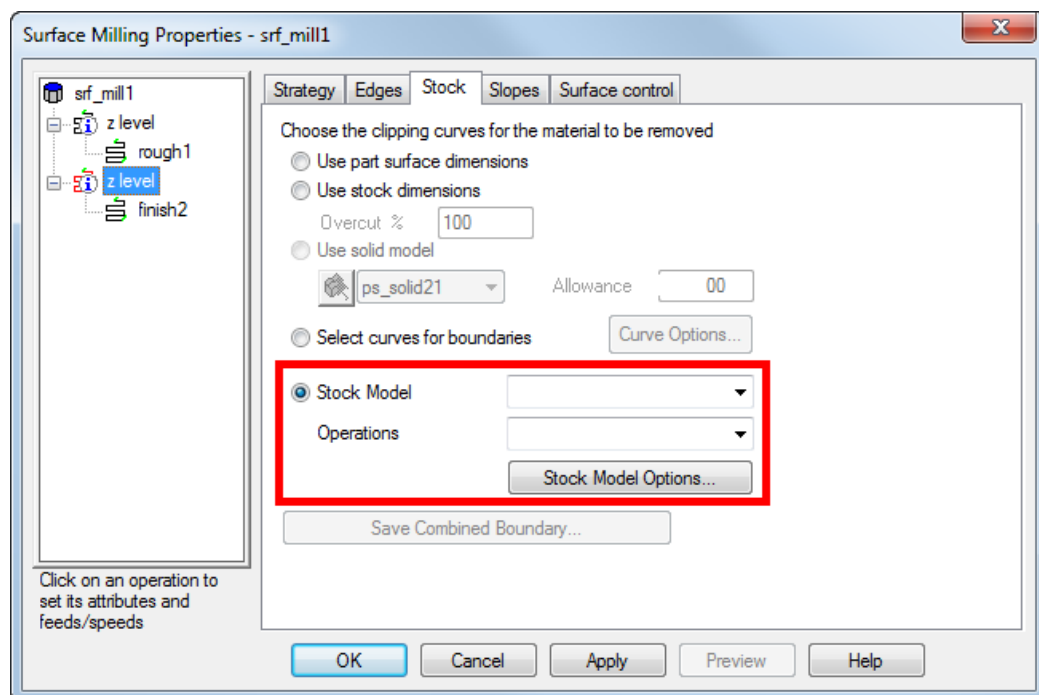
## All 3-axis finishing operations can use a stock model

In FeatureCAM 2013 R2, you can use a stock model with all 3-axis finishing operations:



- Parallel finish
- Z-level finish
- Isoline finish
- 2D spiral
- 3D spiral
- Radial
- Flowline
- Between 2 curves
- Horizontal + vertical
- Corner remachining
- Pencil
- Swarf
- Steep and shallow

The **Stock Model** option is now available on the **Stock** tab for these operations:



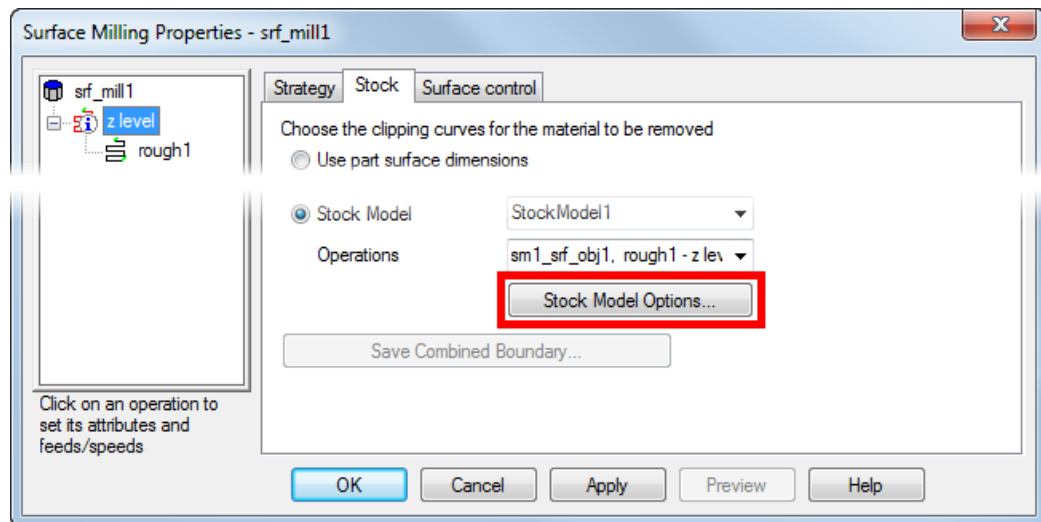
*Stock models are not yet available for 4-axis and 5-axis operations.*

Use the new **Stock Model Options** (see page 34) button to access advanced stock model engagement options.

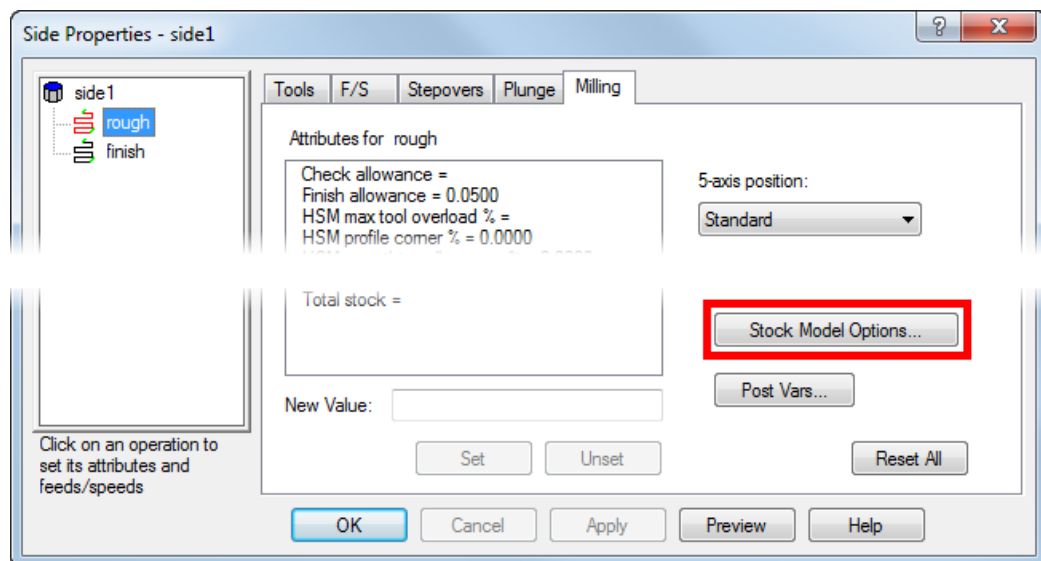
## Advanced stock model settings

In FeatureCAM 2013 R2 there is a new **Stock Model Settings** dialog to control roughing and finishing operations that use a stock model.

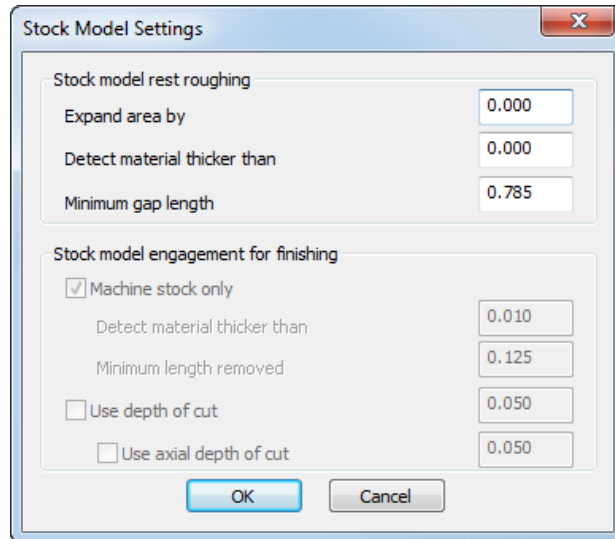
Access the **Stock Model Settings** dialog by clicking the new **Stock Model Options** button on the Stock tab, for 3D strategies:



And the **Milling** tab for 2.5D NT toolpath operations:



The **Stock Model Settings** dialog contains several attributes for roughing and finishing:



### Stock model rest roughing

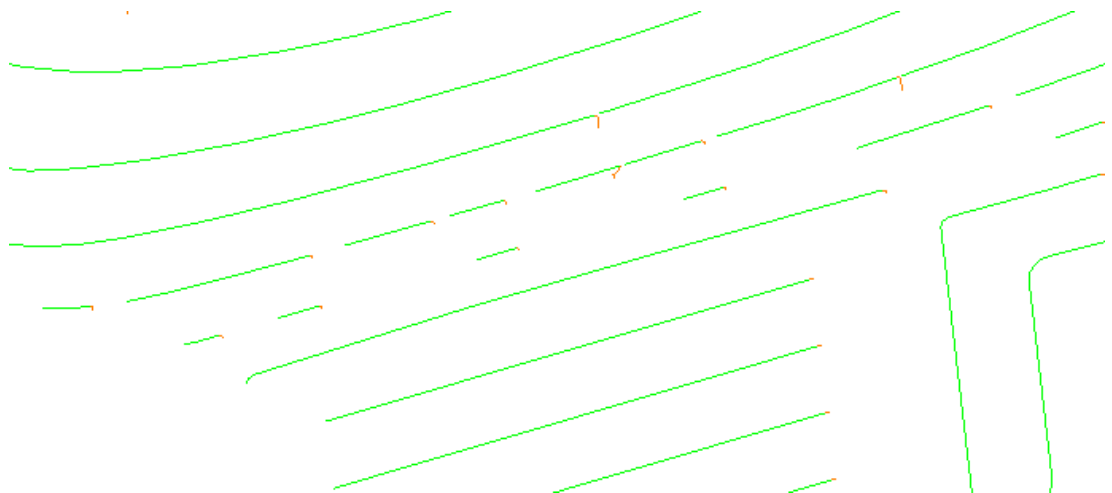
These attributes apply to the rough operation.

**Expand area by** — Enter a value to expand rest areas by the specified distance (measured along the surface). A negative value reduces the size of rest areas.

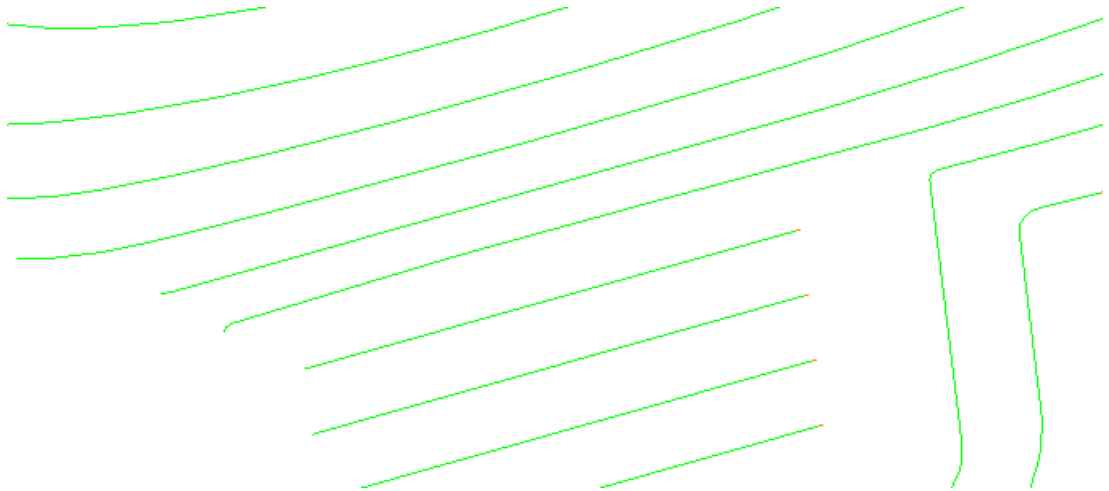
**Detect material thicker than** — Enter a threshold value. FeatureCAM ignores rest material that is thinner than the specified threshold.

**Minimum gap length** — Enter the gap length, which controls fragmentation by replacing gaps shorter than this distance with a toolpath segment. A large value reduces fragmentation, but increases the length of the toolpath that isn't actually cutting material. A small value produces shorter toolpaths but increases the number of toolpath lifts.

**A Minimum gap length of 0:**



A **Minimum gap length** of the default value:



### Stock model engagement for finishing

These attributes apply to the finish operation. They reduce tool wear, improve surface finish, and stop the tool engaging with the material excessively.

**Machine stock only** — Select this option to limit the toolpaths to areas where a minimum amount of stock is being removed from the stock model.

**Detect material thicker than** — Enter a threshold value. FeatureCAM ignores rest material that is thinner than the specified threshold.

**Minimum length removed** — Enter a threshold value. FeatureCAM ignores toolpath portions that are shorter than this.

**Use depth of cut** — Select this option to limit the depth of cut to the value you enter.

---

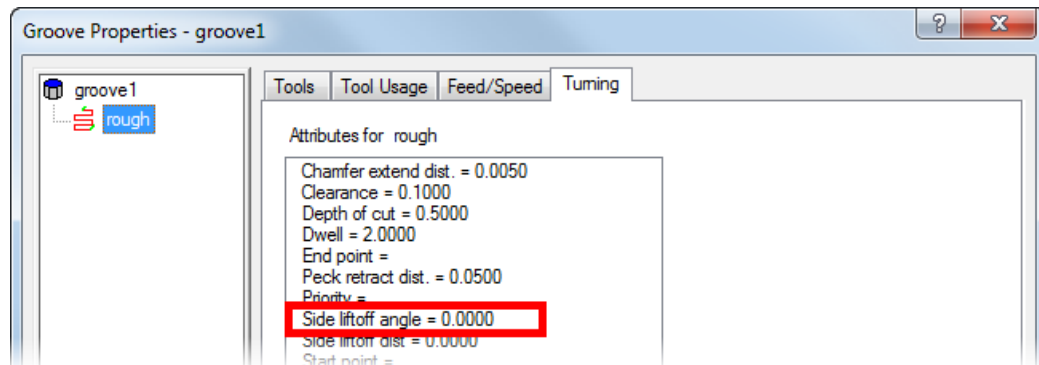
## Turn improvements

FeatureCAM 2013 R2 includes these turn improvements:

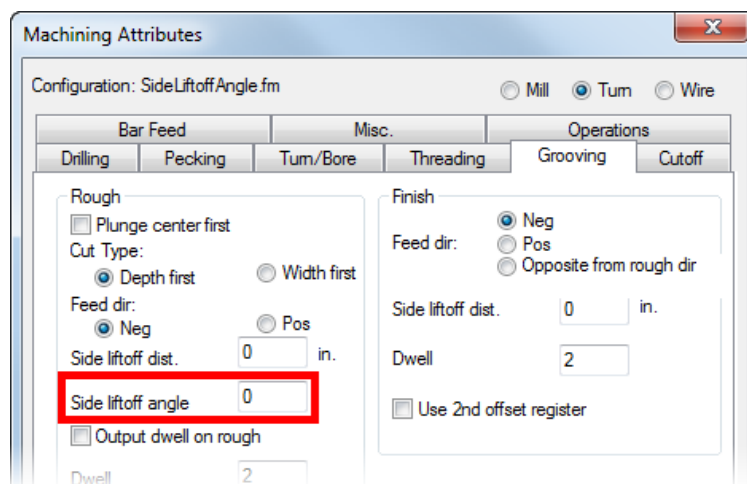
- You can control the liftoff angle with a new attribute (see page 38).
- You can use an arc to lead in and out (see page 40) for the finish pass of a Turn or Bore feature.
- There is a new reserved word (see page 41) in XBUILD for a Transfer feature.

## Side liftoff angle

In FeatureCAM 2013 R2, there is a new **Side liftoff angle** attribute for turned Groove features, on the **Turning** tab:



You can also set its default value for the current document on the **Grooving** tab of **Machining Attributes**:



This attribute enables you to lift the tool off the part after each plunge cut, at the angle you specify.



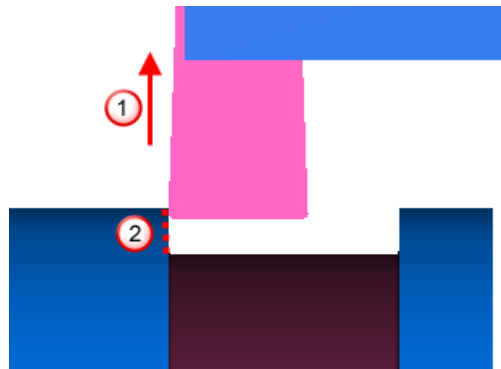
*You must also set the **Side liftoff dist.** attribute.*

### Example

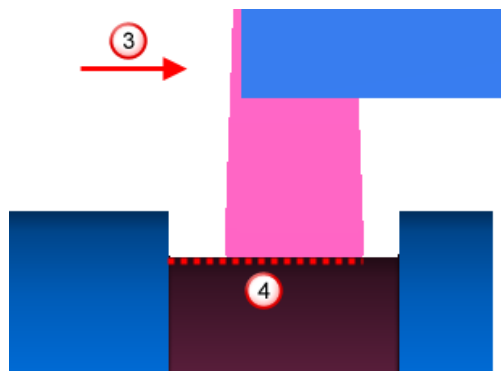
This part has a Groove feature, shown in pink:



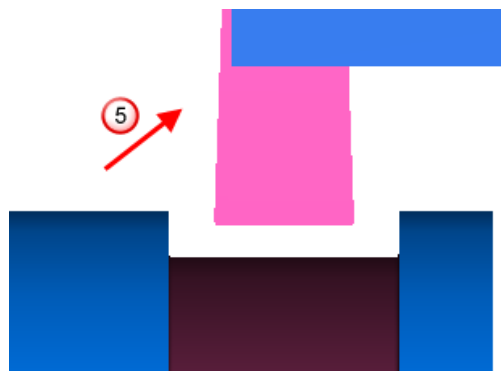
The default behavior is for the tool to lift off the part at 90°, shown by ①, after each plunge cut. This results in tool contact with the uncut material, at ②, when the tool is retracting at a rapid feed rate along the X axis:



You can avoid this by using the **Side liftoff dist.** attribute, to move the tool back along the Z axis ③, before lifting off. However this results in the tool rubbing along the remaining material along the Z axis ④.



Use the **Side liftoff angle** attribute, in conjunction with the **Side liftoff dist.** attribute to lift the tool off at the specified angle for the specified distance ⑤, and avoid rubbing the tool along remaining material in any direction:

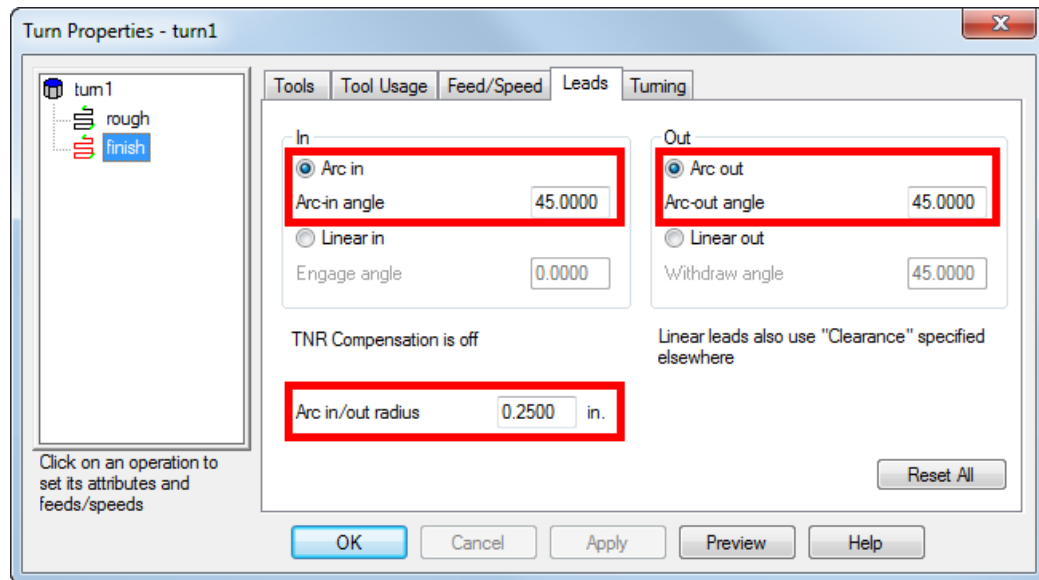


**Side liftoff dist** and **Side liftoff angle** are ignored for the retract move at the end of the first plunge. The liftoff move is performed at the plunge feed rate. If the groove is a round-bottomed groove, then liftoff is not used, even when specified.

## Arc lead-in/out for Turn/Bore feature

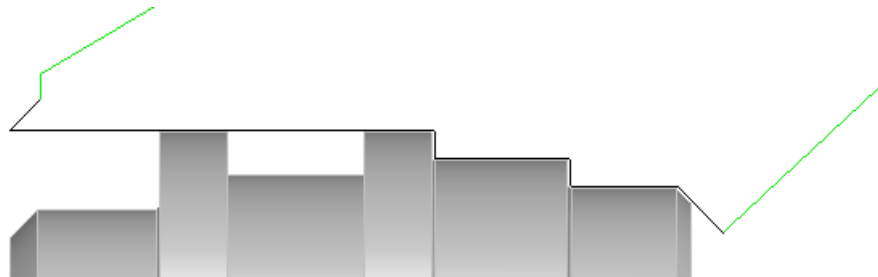
In FeatureCAM 2013 R2, you can use an arc to lead in and out for the finish pass of a Turn or Bore feature. This enables you to finish large parts in sections.

There is a new **Leads** tab for the finish operation of Turn and Bore features, with the new **Arc in** and **Arc out** options:

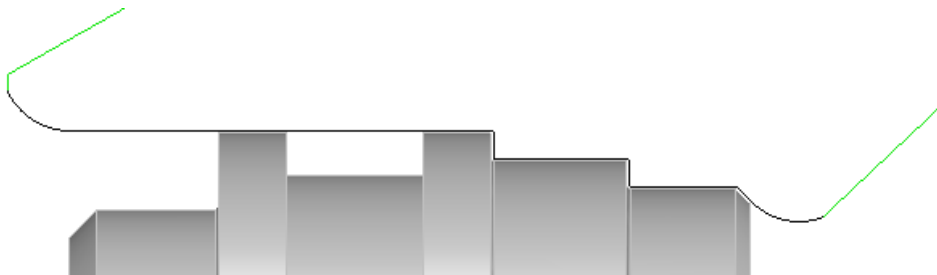


*The linear lead options that were previously on the **Turning** tab are now here.*

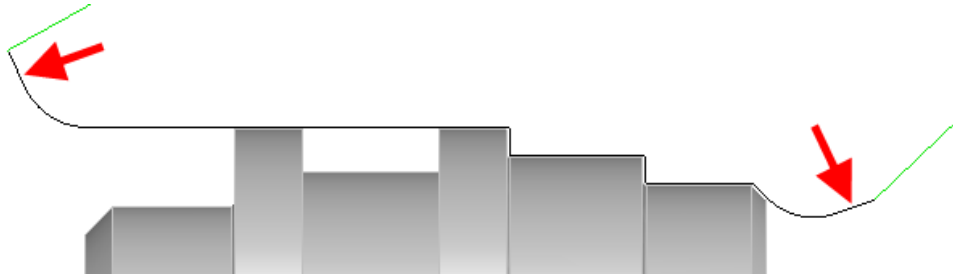
This example uses **Linear in** and **Linear out**:



This example uses **Arc in** and **Arc out**:



When using **Arc in** and **Arc out** with **TNR Comp** enabled, small linear moves are added to the toolpath before the arc in and after the arc out. Compensation is turned on during the linear segments:



## XBUILD: New transfer reserved word

There is a new reserved word in XBUILD:

`<Z-XFER-COORD>` is the Z coordinate of the Transfer feature in setup coordinates. This takes into account previous bar pulls.



*For a Cutoff feature with transfer, there is no concept of feature location (because the only location is the cutoff location). In this case the `<Z-XFER-COORD>` is assumed to be the location of the work piece coordinate system, plus any previous bar pulls.*

---

## Turn/Mill improvements

FeatureCAM 2013 R3 includes this turn/mill improvement:

- The OD location (see page 41) for features from curves has been reinstated.

### OD location for features from curves reinstated

In FeatureCAM 2013 R2, the OD option has been reinstated on the **Location** tab for turn/mill features from curves.

This capability was removed from the software long ago, but has been missed. Specifically the ability to make a pattern of features has been restored, where the object of the pattern is located in the XY plane as opposed to *in-place*, because many users preferred this original approach.



---

## Import improvements

- Spatial's InterOp R23 is integrated:
  - Pro/E Reader adds support for Creo 2.
  - UG NX Reader now supports Unigraphics NX 8.0 version files
  - CATIA V5-6 R2013 (R22)

---


## Export improvements


FeatureCAM 2013 R3 has support for export to CAMplete TruePath.

---

## XBUILD improvements

There are two new ways to manipulate strings in XBUILD:

Operator	Function	Example	Result
<code>InStr([start, ]string1,string2[,compare] )</code>	Returns the position of the first occurrence of <code>string2</code> within <code>string1</code> .  <i>String indexes are 0-based. The return value indicating "string not found" is -1.</i>	<code>[instr ("abcdef", "def")]</code>	Returns 3.0

<pre>InStrRev (string1, string2[, start[, compare]])</pre>	<p>Returns the position of the first occurrence of <code>string2</code> within <code>string1</code>. The search begins from the end of the string, but the position returned counts from the beginning of the string.</p> <p> <i>String indexes are 0-based. The return value indicating "string not found" is -1.</i></p>	<pre>[instrrev("defabcdef", "def")]</pre>	<p>Returns 6.0</p>
--	---	---	--------------------

You can use these new operators, for example, to extract parts of tool names.

# What's New in FeatureCAM 2013 R1

FeatureCAM 2013 R1 offers all of the original features of FeatureCAM 2012 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.*

---


## General improvements

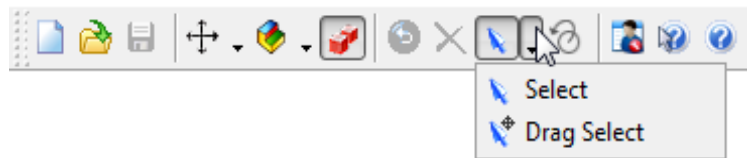
These are the most significant general improvements in FeatureCAM 2013 R1:

- There is a new selection method. **Drag Select** (see page 45) enables you to select multiple adjacent entities by dragging the mouse pointer across the.
- It is now easier to convert feed and speed (see page 47) values.
- You can now snap to STL models (see page 49) and also use them as clamps (see page 48).
- Controlling simulation speed is now easier because you can lengthen the simulation speed slider (see page 49) on the Simulation toolbar.
- You can do global edits and transforms (see page 54) on a Toolpath feature.
- FeatureCAM now supports the import of Spatial's 3D InterOp R22 SP2, NX 8 files.
- There are new options for the Probing add-in (see page 50).
- There are new reserved words (see page 50) in XBUILD.

## New selection method

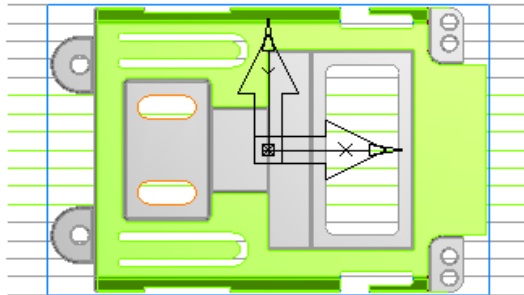
There is a new **Drag Select** option in FeatureCAM 2013 R1. This method enables you to select multiple adjacent entities by dragging the mouse pointer across them.

To enter Drag Select mode, click the **Drag Select**  button in the **Select Menu** in the **Standard** toolbar.



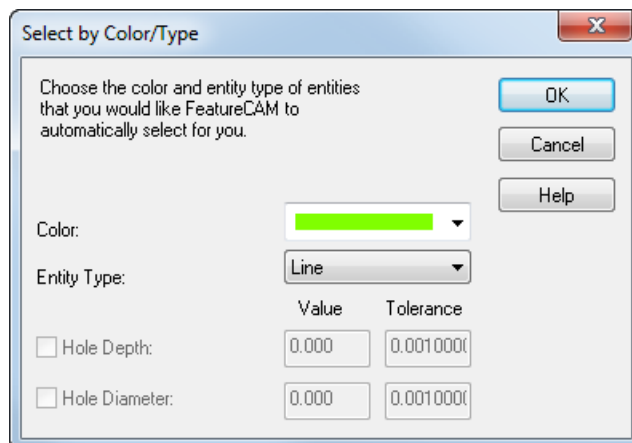
### Drag Select example

This example part contains different colored faces and geometry:



The existing selection methods do not work well for this part.

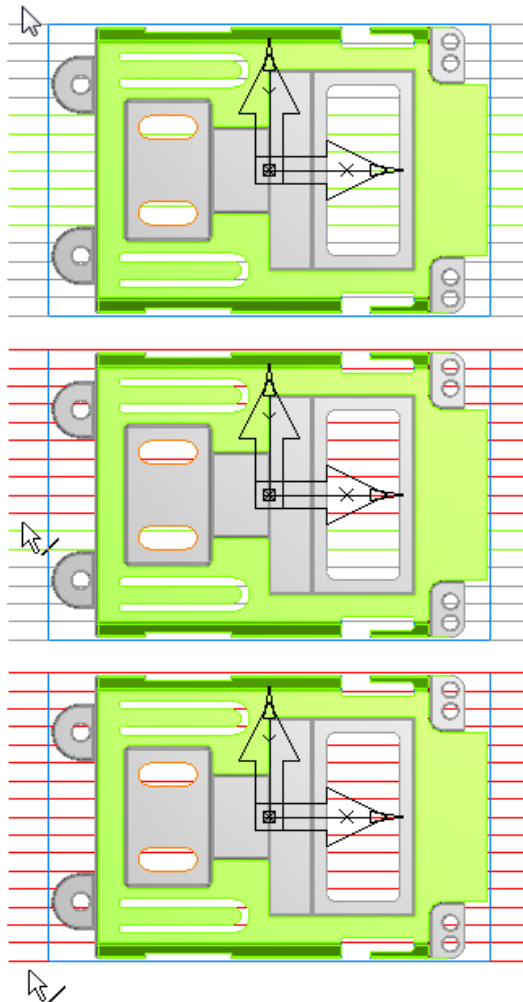
For example, if you use the **Select by Color/Type** dialog, you can select either the green lines or the gray lines, but not both at the same time:



If you use a box selection over the lines in a front view, you can't select only the lines without selecting some of the surfaces too:



Using the new **Drag Select** option, you can drag the mouse pointer over the lines to the left of the stock to select them all:



**Drag Select** can save time when selecting adjacent surfaces. With the standard **Select** mode, to select adjacent surfaces, you must use a box-select or **Ctrl+click** each surface. With **Drag Select**, just move the mouse pointer over adjacent surfaces. You can hold down the **Ctrl** key with **Drag Select** to add non-adjacent surfaces to the selection, or after you have released the mouse button, for example to change the view of the part.

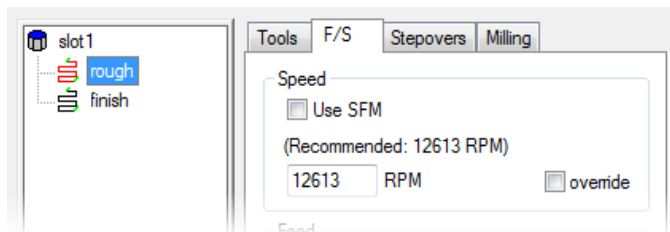
## Easier feed and speed unit conversion

In FeatureCAM 2013 R1, when you override the feed or speed value, then change units, FeatureCAM converts the value you entered to the new units. This works for milling and turning parts.

Previously, when you entered an override value, and changed units, FeatureCAM returned to the default value for the new units.

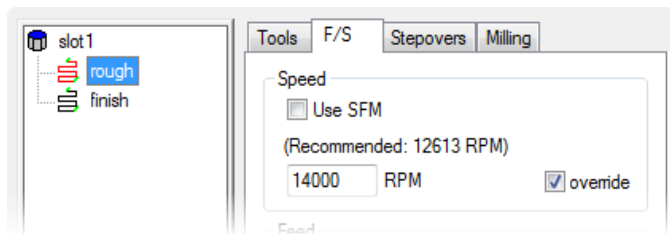
### Example

- 1 Create a Slot feature and view the **F/S** tab in the **Slot Properties** dialog:

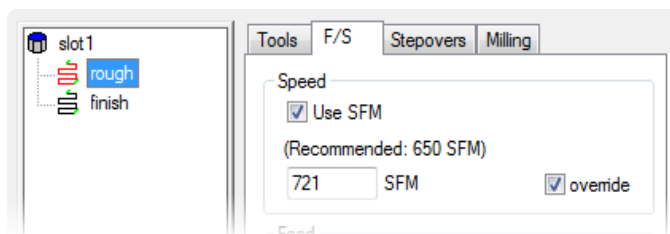


The recommended speed is entered by default.

- 2 Enter a different **Speed** value. The **override** box is checked automatically.



- 3 Change the **Speed** units:




FeatureCAM remembers the override value entered and converts it to the new units. Previously, FeatureCAM returned to the recommended value when you switched units.

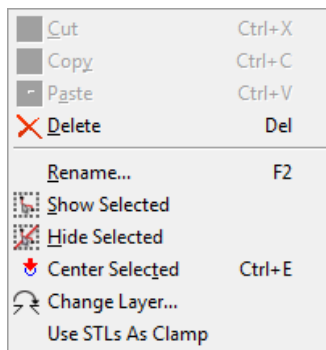
The **Feed** value works in a similar way.

## Use STL as a clamp for 3D simulation

In FeatureCAM 2013 R3, you can use STL files as a clamps for 3D simulation.

To use STL files as clamps:

- 1 Select the STLs that you want to use in one of these ways:
  - Select the face of an STL (or multiple-select the faces of more than one STL) in the Graphics window. If the STLs are not already shaded, click the **Shade**  button to shade them first.
  - Select the names of the STLs in the Part View.
- 2 Right-click on the selection.
- 3 Select **Use STLs As Clamp** from the context menu.



To see if an STL is used as a clamp or not, right-click it and look at the context menu. If there is a check mark next to the **Use STLs As Clamp** menu item, then that STL is currently used as a clamp:

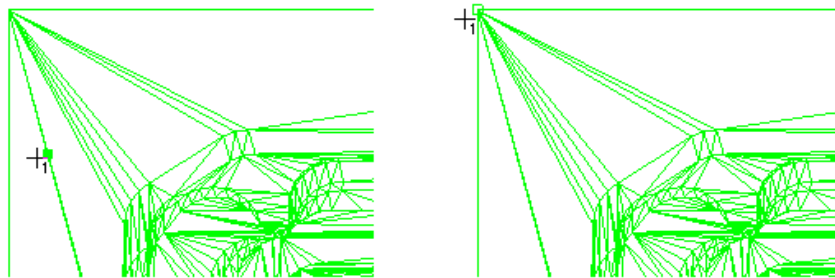
✓ Use STLs As Clamp

If there is no check mark, the STL is not currently used as a clamp.

To unset a solid as a clamp, click the **Use STLs As Clamp** menu item with the check mark. The check mark is removed and the solid is no longer a clamp.

## Snap to STL

In FeatureCAM 2013 R1, you can snap to points on an STL model using **Snap to Object** and **Snap to End Point**:



## Lengthen Sim Speed slider

In FeatureCAM 2013 R1, you can increase the length of the simulation speed slider on the **Simulation** toolbar, to make it easier to control the speed.

To do this:

- 1 Ensure the **Simulation** toolbar is displayed.
- 2 Select **View > Toolbars** from the menu. The **Customize Toolbars** dialog is displayed, but you don't need to do anything in the dialog.
- 3 Click on the **Sim Speed** slider in the **Simulation** toolbar and it displays a border.
- 4 Hover over the border at the right of the button and the mouse pointer changes to a double arrow icon.
- 5 Drag the right border out as far as you want.
- 6 Click **OK** to save your changes and close the **Customize Toolbars** dialog.

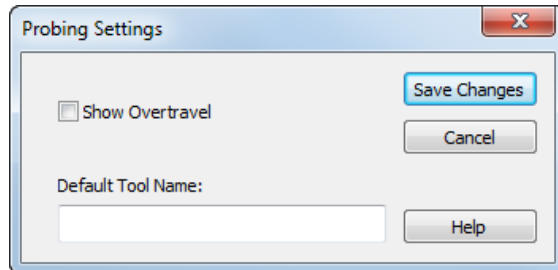




## Probing: New options for Probing UDF

In FeatureCAM 2013 R1, there are new options that you can set easily when you start the Probing Add-in.

After starting the Probing Add-in, the **Probing Settings** dialog is displayed:



**Show Overtravel** — Select this option to show a representation of the overtravel in the preview in the graphics window and during simulation.

**Default Tool Name** — By default, the add-in looks for a lollipop tool, but you can override that here. Enter a tool name.

## XBUILD: New reserved words

### 5-Axis:

<TOUCHOFF-UCS-X>	This is the position in X of the touch-off UCS in relation to the Stock Axis.
<TOUCHOFF-UCS-Y>	This is the position in Y of the touch-off UCS in relation to the Stock Axis.
<TOUCHOFF-UCS-Z>	This is the position in Z of the touch-off UCS in relation to the Stock Axis.
<UCS-XAXIS-X>	The X coordinate of the X axis vector of the current UCS.
<UCS-XAXIS-Y>	The Y coordinate of the X axis vector of the current UCS.
<UCS-XAXIS-Z>	The Z coordinate of the X axis vector of the current UCS.
<UCS-YAXIS-X>	The X coordinate of the Y axis vector of the current UCS.
<UCS-YAXIS-Y>	The Y coordinate of the Y axis vector of the current UCS.

<UCS-YAXIS-Z>	The Z coordinate of the Y axis vector of the current UCS.
<UCS-ZAXIS-X>	The X coordinate of the Z axis vector of the current UCS.
<UCS-ZAXIS-Y>	The Y coordinate of the Z axis vector of the current UCS.
<UCS-ZAXIS-Z>	The Z coordinate of the Z axis vector of the current UCS.

### Turn Part Handling feature:

<FEED-DIST>	This is the <b>Feed distance</b> value from a Part Handling feature in FeatureCAM.
<GRAB-DIST>	This is the <b>Grab distance</b> value from a Part Handling feature in FeatureCAM.
<SUB-FEED>	This is the <b>Subspindle feed rate</b> value from a Part Handling feature in FeatureCAM.
<PULL-DIST>	This is the <b>Pull distance</b> value from a Part Handling feature in FeatureCAM.
<CUTOFF-TRT>	This indicates which turret has the cutoff operation. The values are empty (no cutoff), main upper, main lower, sub upper, sub lower.
<ALREADY-SUP>	<b>TRUE</b> if <b>Already Supported</b> is enabled for a Part Handling feature in FeatureCAM.
<LEAVE-SUP>	<b>TRUE</b> if <b>Leave Supported</b> is enabled for a Part Handling feature in FeatureCAM.
<TURRET1-LOC>	This indicates the <b>Location</b> of turret 1 while the sub-spindle is moving, taken from the <b>Transfer Turret Control</b> dialog in FeatureCAM. The values are <b>None</b> , <b>Home</b> , <b>Escape</b> , or <b>Cutoff Location</b> .
<TURRET2-LOC>	This indicates the <b>Location</b> of turret 2 while the sub-spindle is moving, taken from the <b>Transfer Turret Control</b> dialog in FeatureCAM. The values are <b>None</b> , <b>Home</b> , <b>Escape</b> , or <b>Cutoff Location</b> .
<TURRET3-LOC>	This indicates the <b>Location</b> of turret 3 while the sub-spindle is moving, taken from the <b>Transfer Turret Control</b> dialog in FeatureCAM. The values are <b>None</b> , <b>Home</b> , <b>Escape</b> , or <b>Cutoff Location</b> .

<TURRET4-LOC>	This indicates the <b>Location</b> of turret 4 while the sub-spindle is moving, taken from the <b>Transfer Turret Control</b> dialog in FeatureCAM. The values are <b>None</b> , <b>Home</b> , <b>Escape</b> , or <b>Cutoff Location</b> .
<PART-CATCH>	<b>TRUE</b> if the <b>Part Catcher</b> option is enabled for a Part Handling feature in FeatureCAM.
<PUSH-PRESS>	<b>TRUE</b> if <b>Push/Press</b> is enabled for a Part Handling feature in FeatureCAM.
<OPEN-DWELL>	This is the dwell time after opening the spindle, taken from the <b>Open dwell</b> attribute value for a Part Handling feature in FeatureCAM.
<CLOSE-DWELL>	This is the dwell time after closing the spindle, taken from the <b>Close dwell</b> attribute value for a Part Handling feature in FeatureCAM.
<GRAB-TYPE>	Returns the <b>Spindle Action</b> value from a Part Handling feature in FeatureCAM.
<MAIN-ANGLE>	This is the <b>Main Angle</b> value from a Part Handling feature in FeatureCAM.
<SUB-ANGLE>	This is the <b>Sub Angle</b> value from a Part Handling feature in FeatureCAM.
<P-PULL-DIST>	This is the distance that the part was previously pulled out of the main spindle, before the current operation.

---

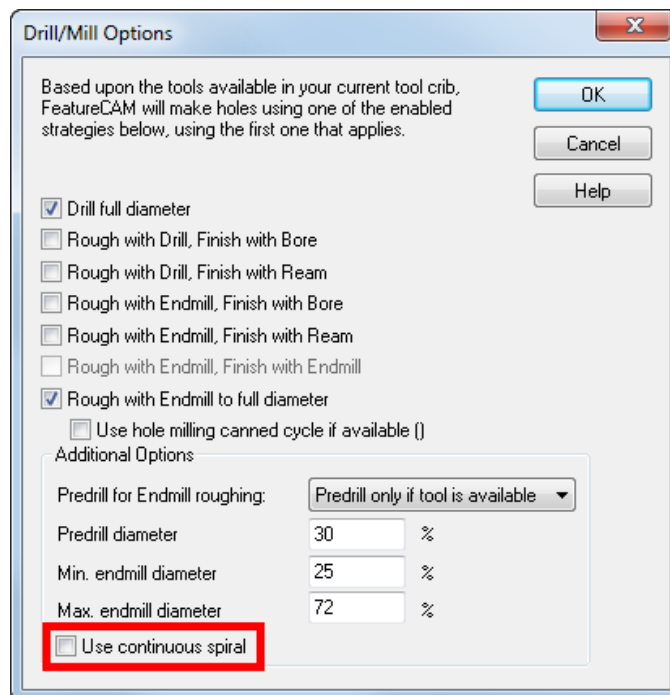
## FeatureMILL 2.5D improvements

These are the most significant 2.5D improvements in FeatureCAM 2013 R1:

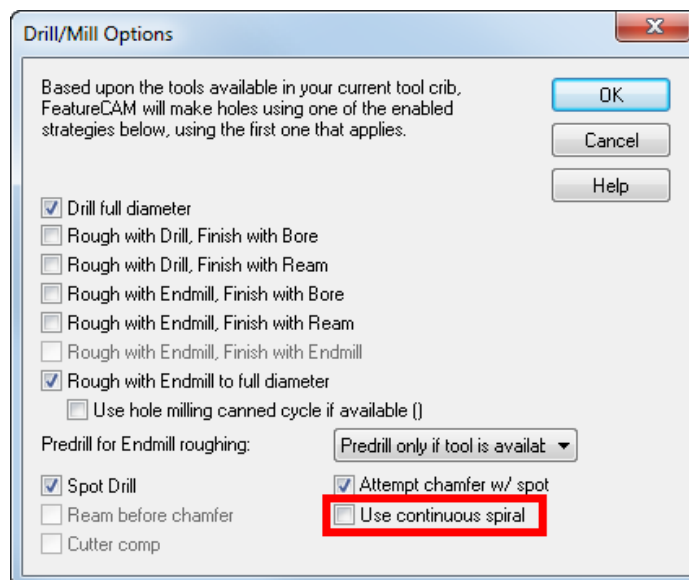
- You can now use continuous spiral toolpaths with drill/mill (see page 53) rough endmill operations.
- You can edit multiple segments (see page 54) in Toolpath features and transform them.
- You can now select the **Stepover type** (see page 54) (including Zigzag), for a Face feature created with Interactive Feature Recognition.

## Drill/Mill: Continuous spiral for rough endmill

In FeatureCAM 2013 R1, there is a new **Use continuous spiral** option in the **Drill/Mill Options** dialog in **Machining Attributes**:



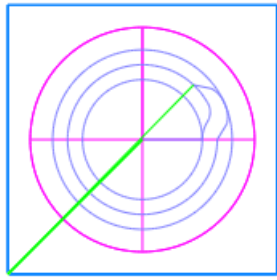
And at feature-level, accessed from the **Strategy** tab:



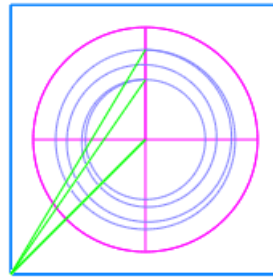
*You must select one of the **Rough with Endmill...** options to access the new option.*

The option uses the **NT Continuous Spiral** toolpath technology to eliminate nearly all stepovers.

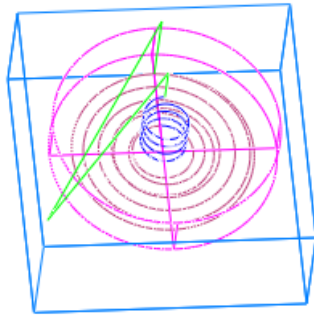
**Use continuous spiral**  
deselected



**Use continuous spiral**  
selected



You can use **Helical ramping** with **Use continuous spiral**, for example:



Traditional spiral toolpaths can produce spikes in tool load, another advantage of NT continuous spiral is that the tool load increases gradually.

## Global Toolpath edits and transforms

In FeatureCAM 2013 R1, you can edit multiple segments of a Toolpath feature on the **Toolpaths** tab.

To do this, **CTRL+select** or **SHIFT+select** the segments you want to edit and right-click the selection to view the context menu. You can set the coolant and feed rates for the selected segments.

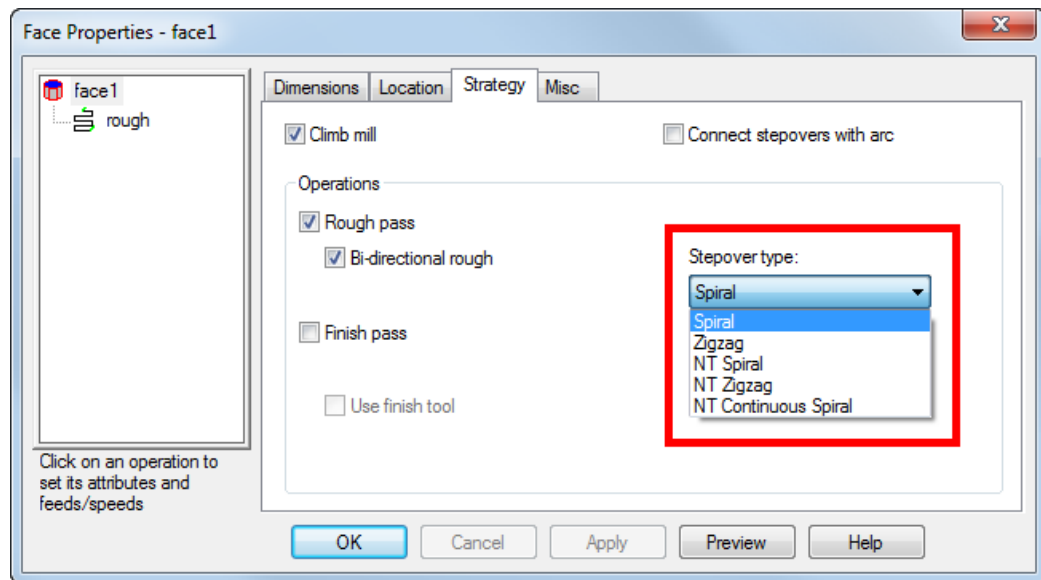
You can also transform a Toolpath feature in the same way as other features. Select **Edit > Transform** from the menu or click the **Transform** button on the **Standard** toolbar.

## Zigzag toolpath for Face

You can now use a Zigzag toolpath for a Face feature.

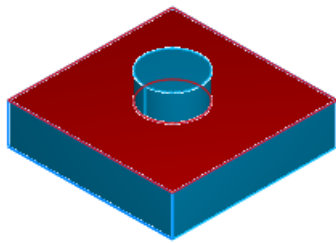
To do this, you must create a Face feature using **Extract with FeatureRECOGNITION**.

On the **Strategy** tab, you can select the **Stepover type** for the **Rough pass**:



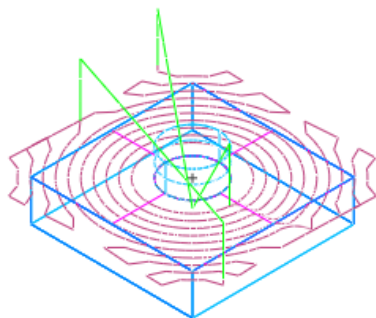
### Example

This part has a boss on the top face:

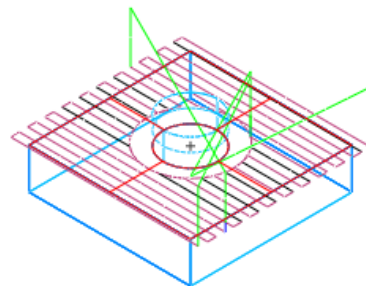


You can avoid the boss by creating the Face feature using Interactive Feature Recognition.

The default toolpath type is Spiral:



This is the same example with **NT Zigzag** selected as the rough Stepover type:



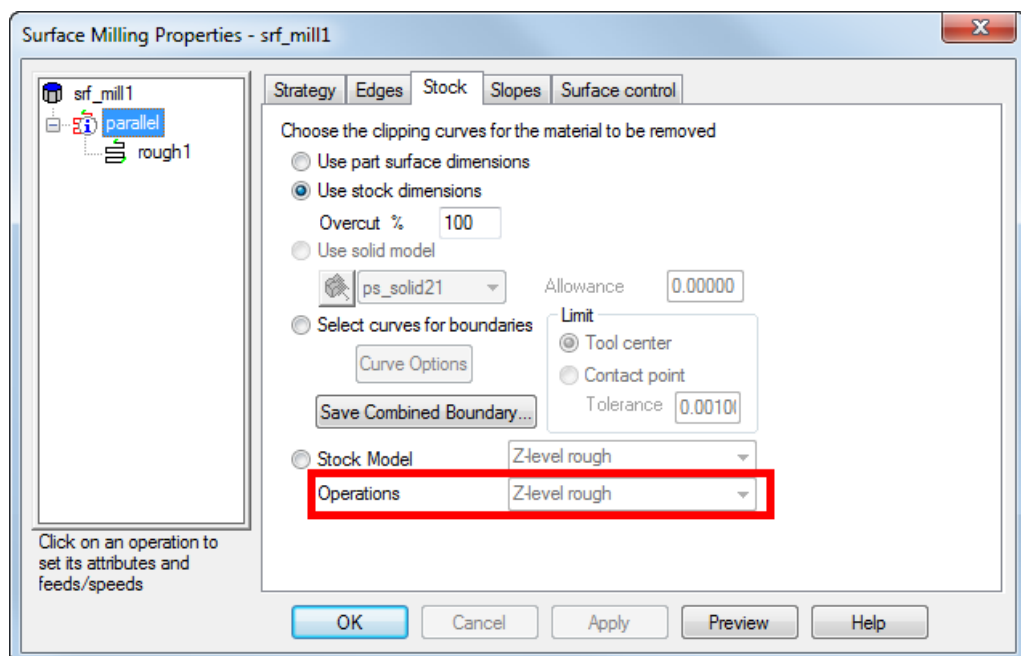
# FeatureMILL 3D improvements

These are the most significant 3D improvements in FeatureCAM 2013 R1:

- There is a new mid-level 3D product: **3D MX** (see page 57) and the top level 3D product becomes **3D HSM**.
- You can select a specific operation when using a stock model for 3D roughing.
- There is a new **Bottom up** (see page 61) attribute for Z-level finishing (requires 3D MX).
- You can override the default start points (see page 58) for certain strategies using a curve (requires 3D MX).
- The new **Area Removal** options enable you to remove small toolpath segments when using Z-level roughing and finishing (see page 63) and to prevent tool damage to non-center cutting tools (requires 3D HSM).
- Cusp control (see page 70) is now available for the Steep and Shallow strategy (requires 3D HSM).
- There is a new **Step cutting** (see page 66) option that reduces terracing for Z-level roughing toolpaths (requires 3D HSM).

## Stock model operation

In FeatureCAM 2013 R1, you can select a specific operation when using a stock model for 3D roughing.



## New mid-level 3D product

There is a new mid-level 3D product available in FeatureCAM 2013 R1: **3D MX**.

The top level, traditional 3D product is now called **3D HSM** and includes all 3D functionality and Feature Recognition.

The new 3D MX product offers you a 3D product midway between 3D Lite and 3D HSM. With 3D MX, you can machine multiple surfaces, and it includes Feature Recognition. You can also use the Z-level finish strategy.

The main differences between the three 3D products are:

3D Lite	3D MX	3D HSM
<ul style="list-style-type: none"><li>▪ Single surface</li><li>▪ Z-level rough</li><li>▪ Parallel rough</li><li>▪ Parallel finish</li><li>▪ Isoline</li><li>▪ 2D spiral</li></ul>	Everything in <b>3D Lite</b> , plus: <ul style="list-style-type: none"><li>▪ Multiple surfaces</li><li>▪ Feature Recognition</li><li>▪ Z-level finish</li><li>▪ Radial</li><li>▪ Flowline</li><li>▪ Four-axis</li><li>▪ Horizontal + vertical</li><li>▪ Between curves</li><li>▪ Swarf</li></ul>	Everything in <b>3D MX</b> , plus: <ul style="list-style-type: none"><li>▪ Plunge roughing</li><li>▪ Pencil</li><li>▪ Remachine</li><li>▪ Steep and shallow</li><li>▪ 3D spiral</li></ul>



*You need **3D MX** or **3D HSM** to use **5-axis Simultaneous**.*



## Override default start points (3D MX)

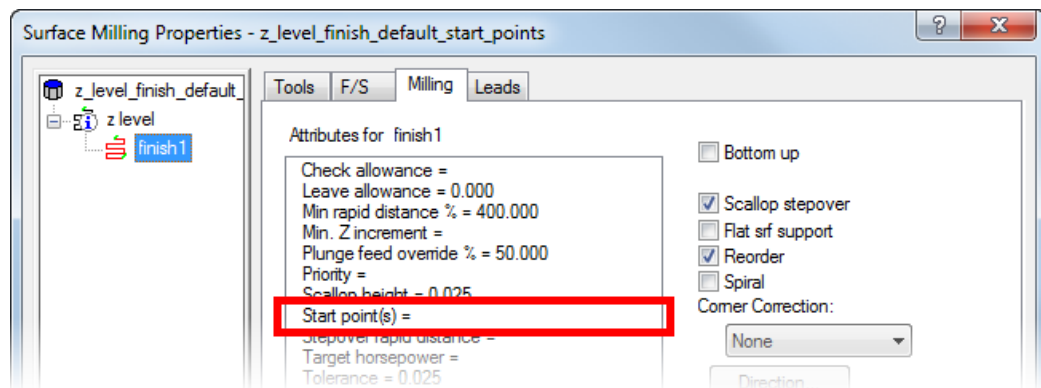
In FeatureCAM 2013 R1, you can override the default start point(s) for these 3D finishing strategy toolpaths, using a curve:

- Z-level finish (3D MX)
- Between 2 curves (3D MX)
- 3-axis swarf (3D MX)
- 3D spiral (3D HSM)

The start point is moved to the projected intersection of the curve and the toolpath.

You can use a curve with multiple segments to set multiple start points for a toolpath, and FeatureCAM uses alternating segments to set the start points.

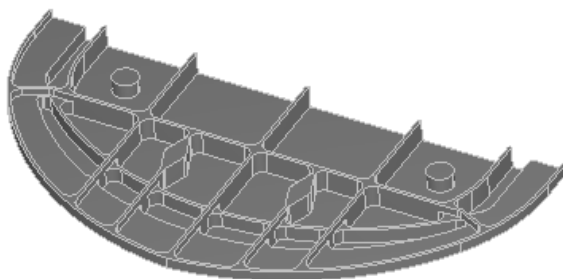
There is a new **Start point(s)** option on the **Milling** tab, where you enter the name of the curve:



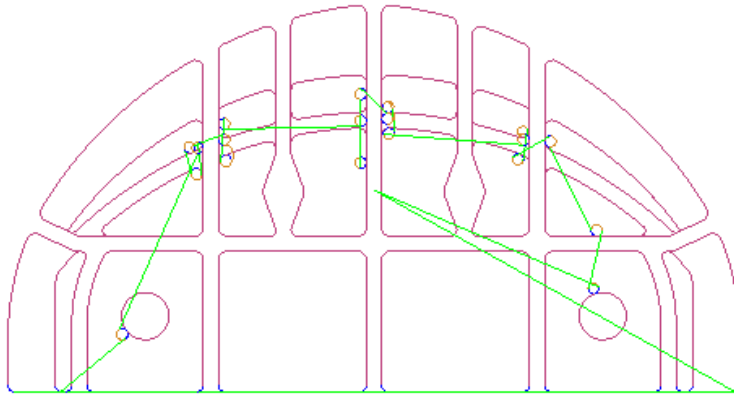
*This attribute applies only to closed toolpaths.*

### Multiple start points example

This example aerospace part has several open and closed areas:

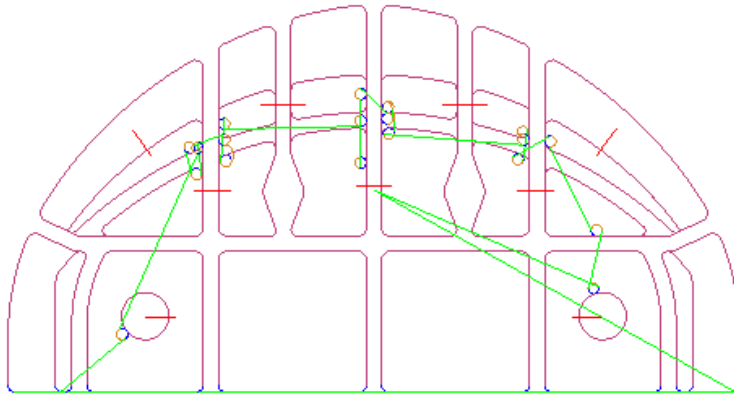


This centerline simulation of a Z-level finish toolpath for this part, shows the locations of the default start points:




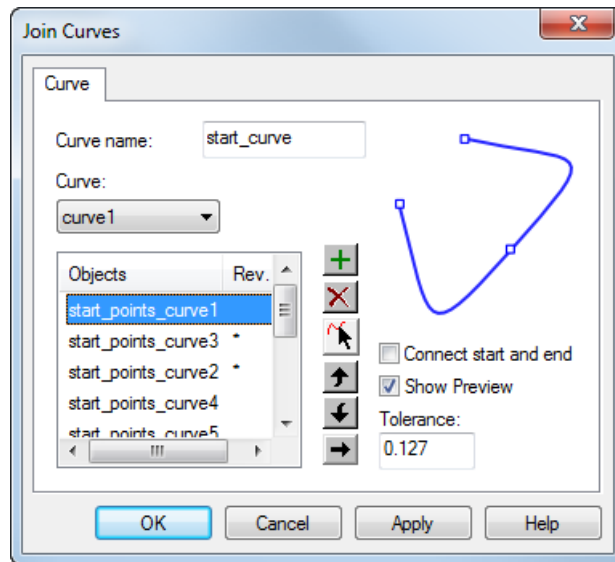
To move the default start points, first create curves that intersect the toolpath where you want the new start points to be.

In the example, the curves are shown in red:

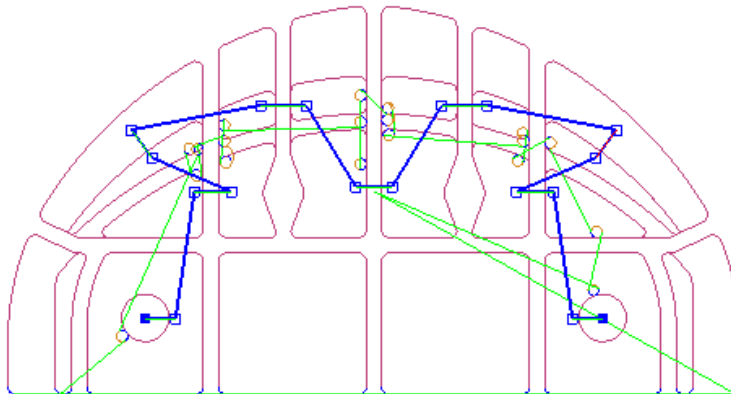




You must use a single curve per toolpath. You can use the **Join** constructor to create one curve from multiple curves. To open the **Join Curve** dialog, select **Construct > Curve > From Curve > Join** from the menu.


Select the curves and click the **Add**  button to add the curves to the list in the **Join Curves** dialog, for example:



Select the **Show Preview** option to preview the joined curves, for example:



If the curves are not listed in the correct order, select the curve you want to move in the list and use the **Move item up**  and **Move item down**  buttons.

You can also reverse a curve's direction using the **Reverse selected curve**  button.

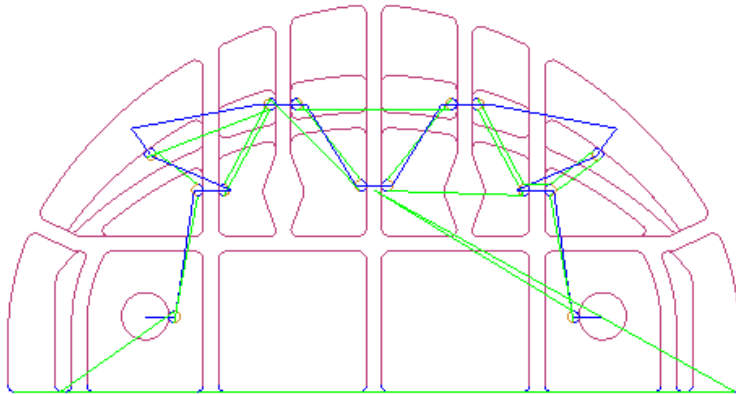
Enter a **Curve name** for the joined curve.



*Copy the **Curve name** to the clipboard so that you can paste it into the **Start point(s)** option.*

On the **Milling** tab for the finishing strategy, select the **Start point(s)** attribute, enter the name of the curve you created and click the **Set** button to save it.

Another centerline simulation shows that the start points have moved to the intersections of the joined curve with the toolpath, and the toolpath is more symmetrical:

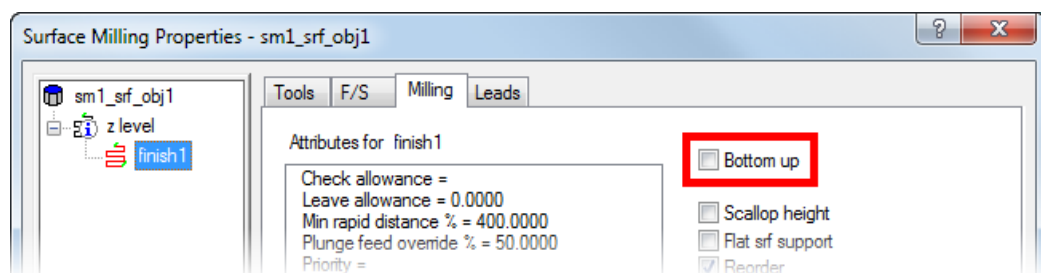


## Bottom up for Z-level finish (3D MX)

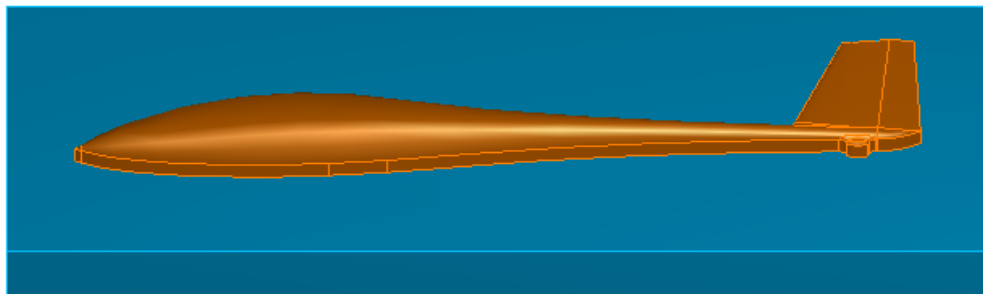
There is a new option for Z-level finishing in FeatureCAM 2013 R1.

On the **Milling** tab of the **Surface Milling Properties** dialog, select the new **Bottom up** option to machine the Surface feature from the bottom upwards.

Deselect the option to machine the feature from the top downwards.

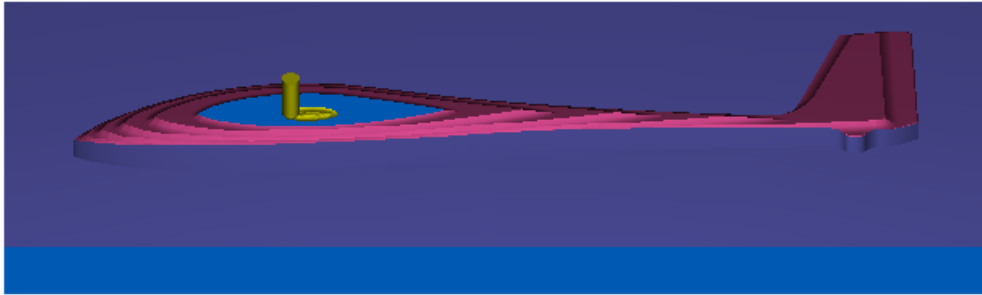


For example, you can use **Bottom up** to cut this glider mold part:



After a Z-level rough and a 2.5D finish, a Z-level finish strategy is used, with **Scallop height** to control the shape of the toolpath.

This is a 3D simulation with **Bottom up** deselected:

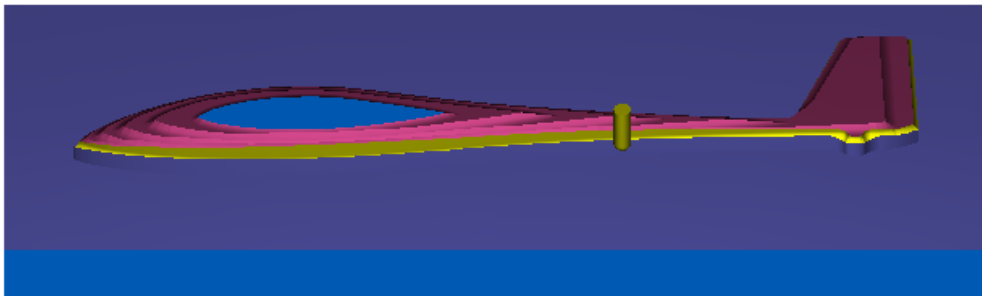


This is the previous and default behavior.

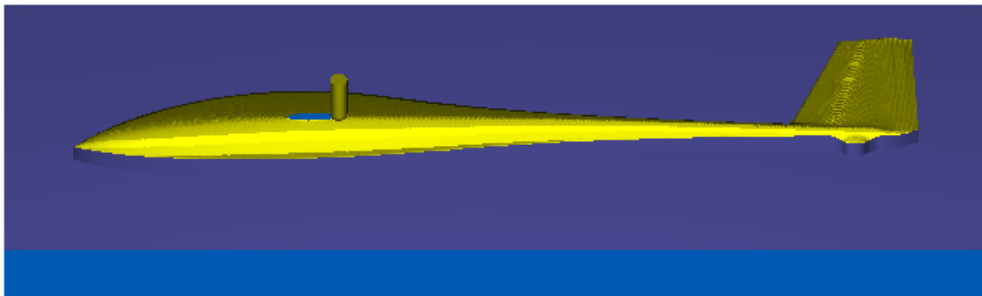
The Ball End Mill tool (shown in yellow) starts at the top in the middle cutting outwards and so is cutting with the center of the tool:



This is the result with **Bottom up** selected:



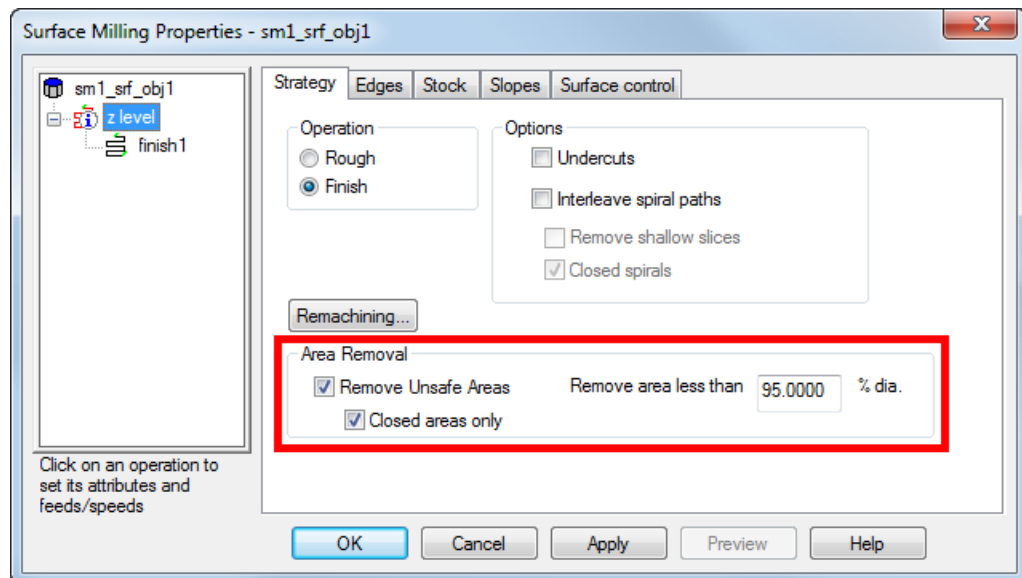
The tool starts at the bottom outside of the slope and mills upwards and inwards towards the center so that the side of the tool is used, which has a better cutting surface.



## Z-level rough area filter (3D HSM)

In FeatureCAM 2013 R1, you can remove small toolpath segments when using the Z-level roughing and finishing strategies, to prevent tool damage when using non-center cutting tools.

There is a new **Area Removal** section on the **Strategy** tab of the **Surface Milling Properties** dialog:



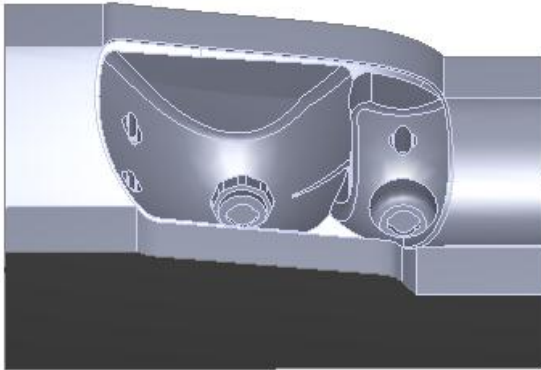
**Remove unsafe areas** — This removes small toolpath segments to prevent tool damage when using non-center cutting tools. When machining into small pockets, removing small segments stops the central, non-cutting underside of the tool from hitting non-machinable material. Unsafe segment removal filters out the machining of confined areas with small movement of the cutting tool.

**Remove area less than** — This removes segments that are smaller than the entered percentage of the tool's diameter, unless they surround a Boss feature.

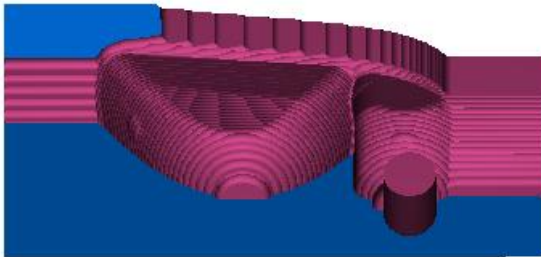
**Closed areas only** — Select this option to remove segments, in enclosed areas, that are smaller than the **Threshold** value.

### Z-level rough area removal example

This headlamp mold has a couple of cavities that work their way down into very small pockets:



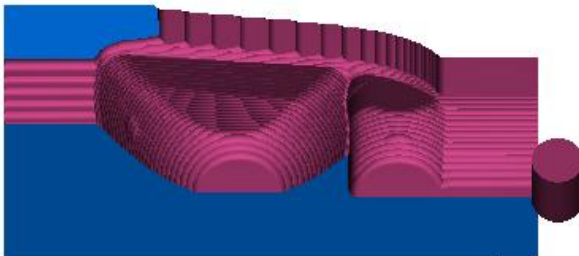
This is a cutaway view of a 3D simulation with **Remove unsafe areas** deselected:



This is the previous and default behavior.

The bottom of the pockets are barely bigger than the tool diameter, and the tool hardly moves in these areas. The non-center cutting region of the tool can't cut the material at the very bottom of the pockets.

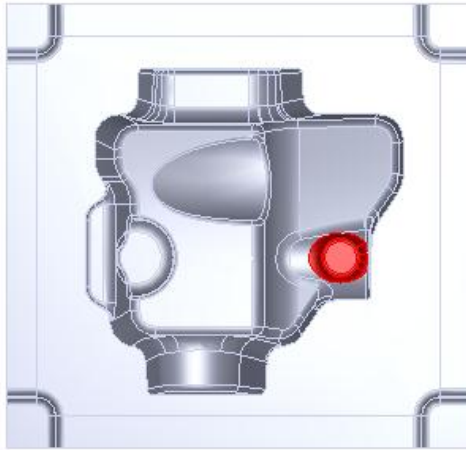
This is the simulation with **Remove unsafe areas** selected and **Remove area less than** set to **80 %** of the tool diameter:



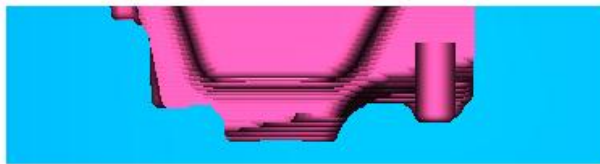
The two smaller pocket regions are removed from the toolpath. This protects the tool and prevents damage.

### Z-level finish area removal example

This cavity part has a small pocket (shown in red):



After using a Z-level rough on the part, this is a cutaway front view of a 3D simulation of a Z-level semi-finish with **Remove unsafe areas** deselected:



This is the previous and default behavior.

The bottom of the pocket is barely bigger than the tool diameter, and the tool hardly moves in this areas. The non-center cutting region of the tool can't cut the material at the very bottom of the pocket.

This is the simulation with **Remove unsafe areas** selected and **Remove area less than** set to **80 %** of the tool diameter:

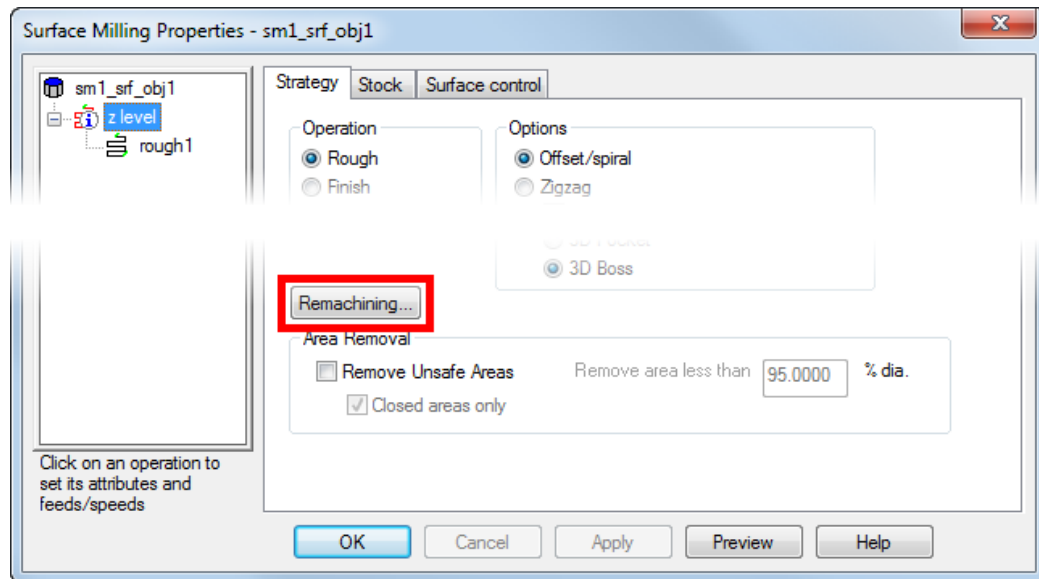


The small pocket region is removed from the toolpath. This protects the tool and prevents damage.

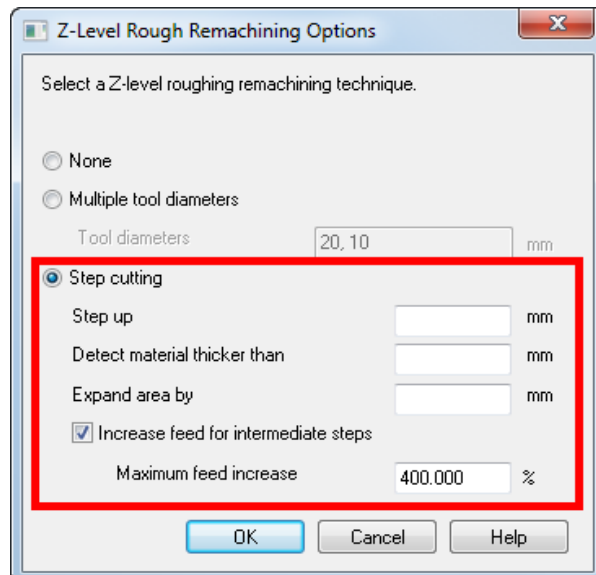


## Step cutting (3D HSM)

You can now reduce terracing when creating Z-level roughing toolpaths with large stepdowns, using the new **Step Cutting** option. There is a new **Remachining** button on the **Strategy** tab:



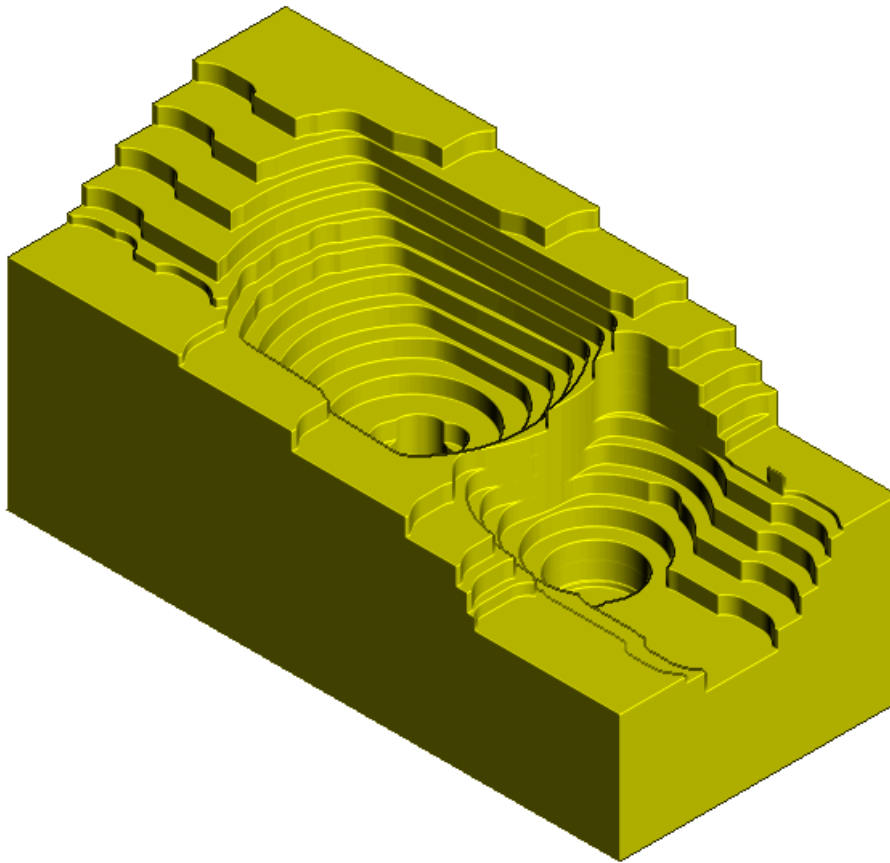
Click the **Remachining** button to open the **Z-Level Rough Remachining Options** dialog.



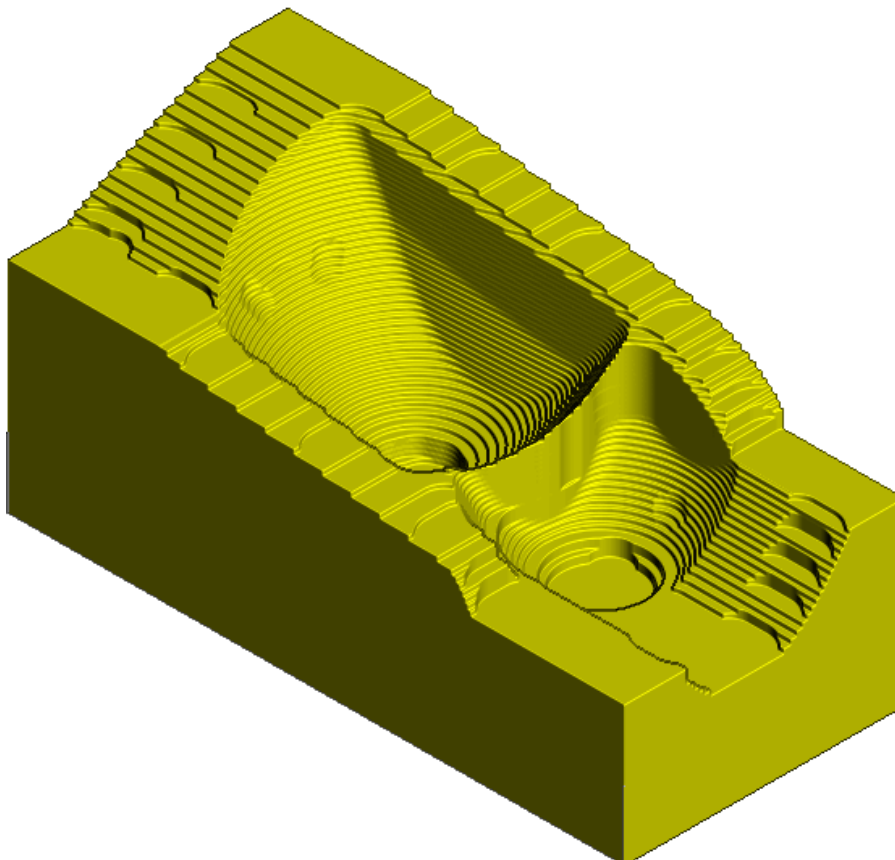
Select **Step cutting** to enable step cutting.

This option machines the terraces that would otherwise remain in a Z-level rough toolpath, with the same tool in the same toolpath. The terraces are machined from bottom to top. FeatureCAM increases the feed rate of the intermediate slices as the depth of cut gets progressively smaller, which enables a constant volume removal rate.

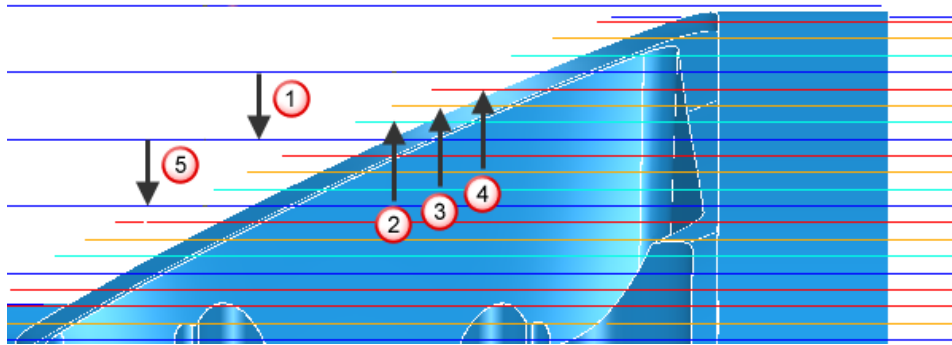
A Z-level rough toolpath with no remachining:



The same toolpath with step cutting:



The main slices extend over the whole part, but the step-cutting slices extend only as far as they are needed to remove terracing.

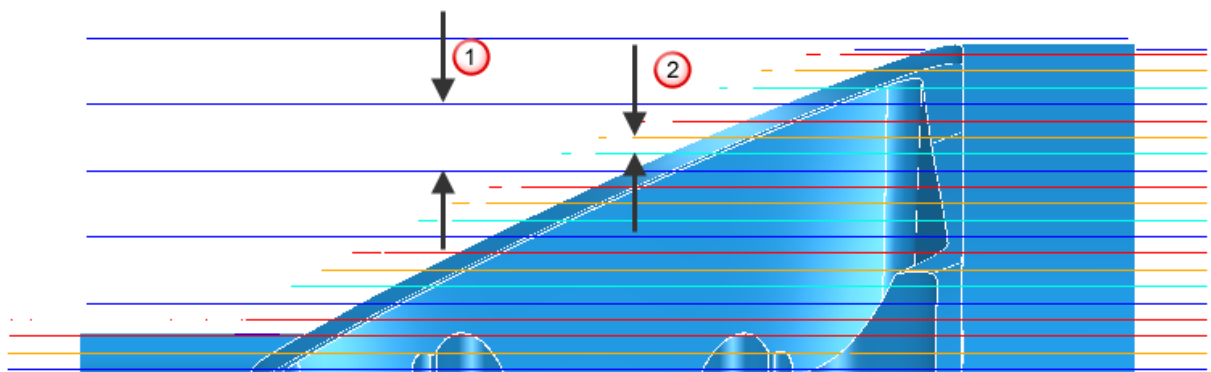


- ① The first pass is a main slice.
- ② The next pass is the lowest step-cutting slice.
- ③ The next pass is the next to lowest step cutting slice and so on up the part (shown by ④) until the previous main slice is reached.
- ⑤ The next main slice.

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

The attributes available to control step cutting are:

**Stepup** — Enter the distance between intermediate cutting levels.

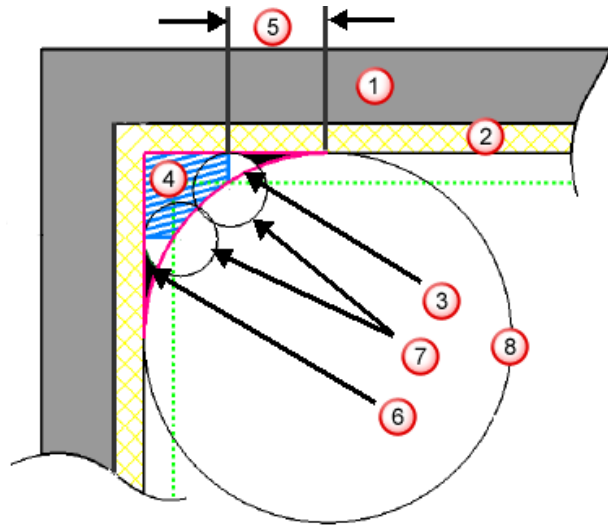


- ① Stepdown of the main slice.
- ② Stepup of the intermediate slice.

The main slice clears the majority of the material. The intermediate slices remove the terraces remaining after the main slice.

**Detect material thicker than** — Enter a threshold value. FeatureCAM ignores rest material that is thinner than the specified threshold.

**Expand area by** — Enter the distance by which to expand the rest areas, measured along the surface. Use with the **Detect material thicker than** value to reduce the areas to be machined to the details (for examples, corners), and then to offset these areas slightly to ensure that all the detail (for example, on the corners) is machined.



- ① — model
- ② — thickness
- ③ — true rest material (outlined in pink)
- ④ — actual rest material detected (blue hatched area)
- ⑤ — amount you need to expand the area by to include all the rest material
- ⑥ — undetected material (black area)
- ⑦ — tool
- ⑧ — reference tool

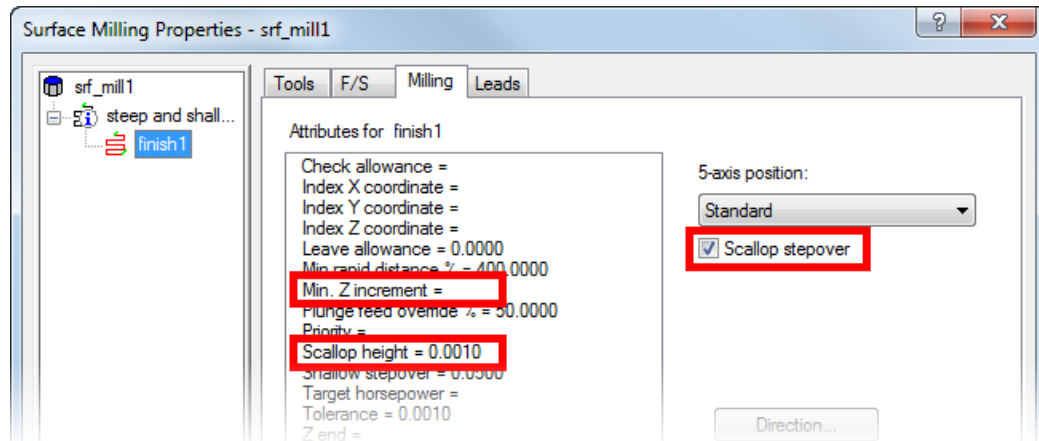
Use the **Expand area by** option to increase the rest area (the blue hatched area) and eliminate the undetected area (the black area).

**Increase feed on intermediate steps** — Select to increase the **Feed rate** of the intermediate slices. Because the stepdown of the intermediate slices is less than that of the main slice, you can increase the feed rate of the intermediate slices while maintaining the tool load. Each intermediate slice can have an increasing feed rate as the depth of cut gets progressively smaller.

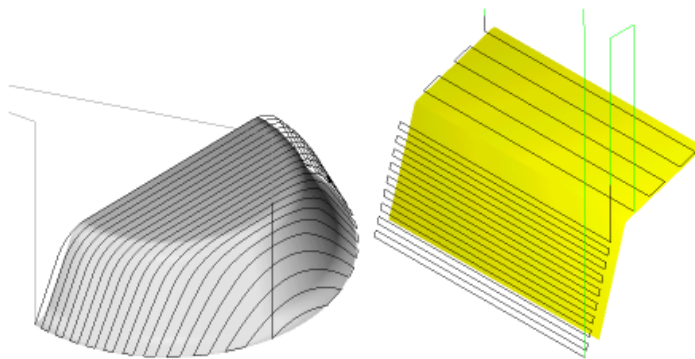
**Maximum feed increase (%)** — This is the maximum allowable cutting feed rate for the intermediate slices as a percentage of the normal feed rate. This value must be larger than 100%. A value of **300** means the cutting feed rate of the intermediate slices can be up to three times faster than for the main slices.

## Cusp control for Steep and Shallow strategy (3D HSM)

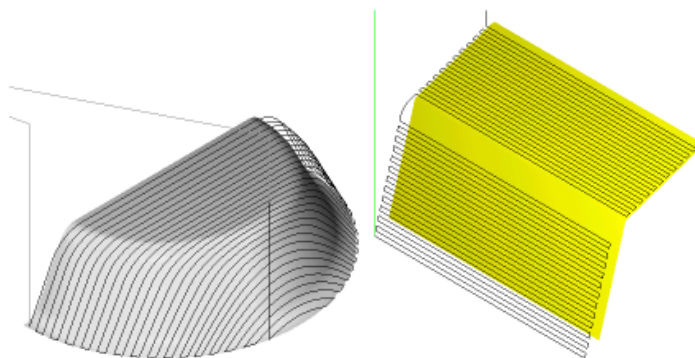
You can now control the Z-level (steep) portion of the Steep and Shallow strategy with the **Scallop stepover** option. Enable **Scallop stepover** to access the **Min. Z increment** and **Scallop height** attributes.



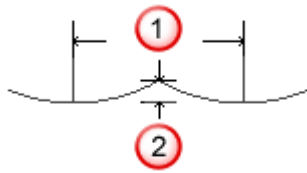
With **Scallop stepover** enabled, spacing of the toolpaths is calculated along the surfaces to give a uniform surface finish. These are surfaces cut without using **Scallop stepover**:



This image shows the same surfaces cut with **Scallop stepover**:



**Scallop height** — This is the absolute scallop height between passes for isoline milling, projection milling finishing passes, and Z-level finishing. This distance is measured along the surface and represents the maximum cusp height between neighboring passes as shown here.



- ① Stepover
- ② Scallop height

**Min Z increment** — This attribute is used with **Scallop height** for Z-level finish. The **Scallop height** attribute specifies the the maximum cusp height between neighboring passes. However, if the calculated value is smaller than the **Min. Z increment**, it is set to **Min. Z increment**.

---

## 5-axis improvements

These are the most significant 5-axis improvements in FeatureCAM 2013 R1.

5-axis positioning (5AP):

- You can now specify a positive or negative orientation (see page 72) for 5-axis positioning.

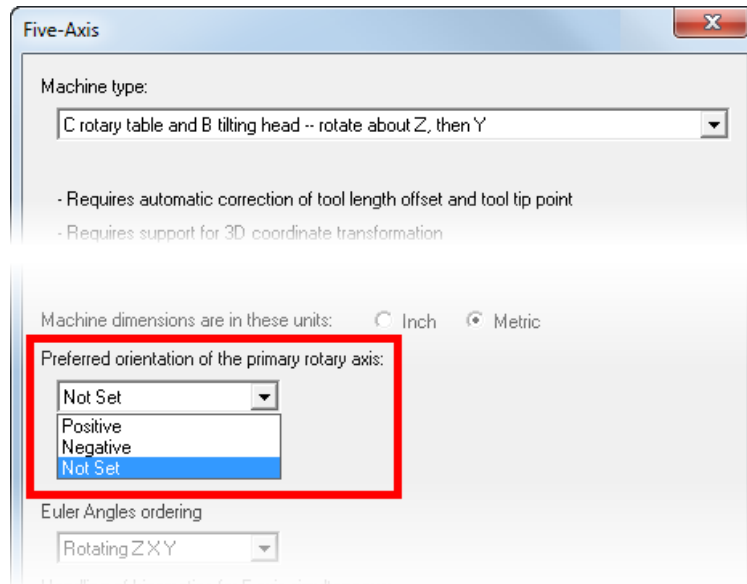
5-axis simultaneous (5AS):

- You can override the default start points (see page 74) for 5-axis toolpaths.
- Tool axis options (see page 76) have been added to Pencil, Corner Remachine, and Between two curves strategies.
- A new **Smoothing distance** (see page 76) option eliminates sudden tool changes.
- You can now smooth the Elevation and Azimuth axes independently with tool axis stabilization (see page 79).
- Existing tool axis options (see page 84) have been improved and you have more control when using **Lead and Lean**.

## Positive/negative position (5AP)

In FeatureCAM 2013 R1, you can specify positive or negative orientation for 5-axis positioning.

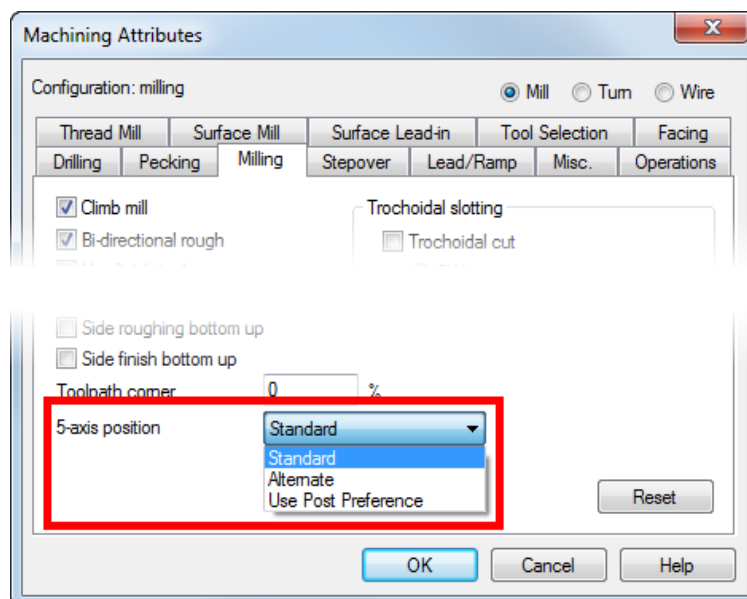
There is a new option in the **Five-Axis** dialog in XBUILD:



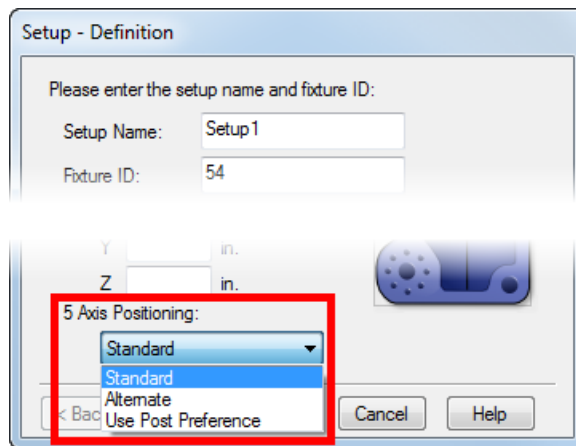
You can set **Positive** or **Negative** as the **Preferred orientation of the primary rotary axis**.

In FeatureCAM, there is now a **5-axis position** menu. As well as **Standard** and **Alternate** (previous behavior), you can now select **Use Post Preference**, to use the orientation that you set in XBUILD.

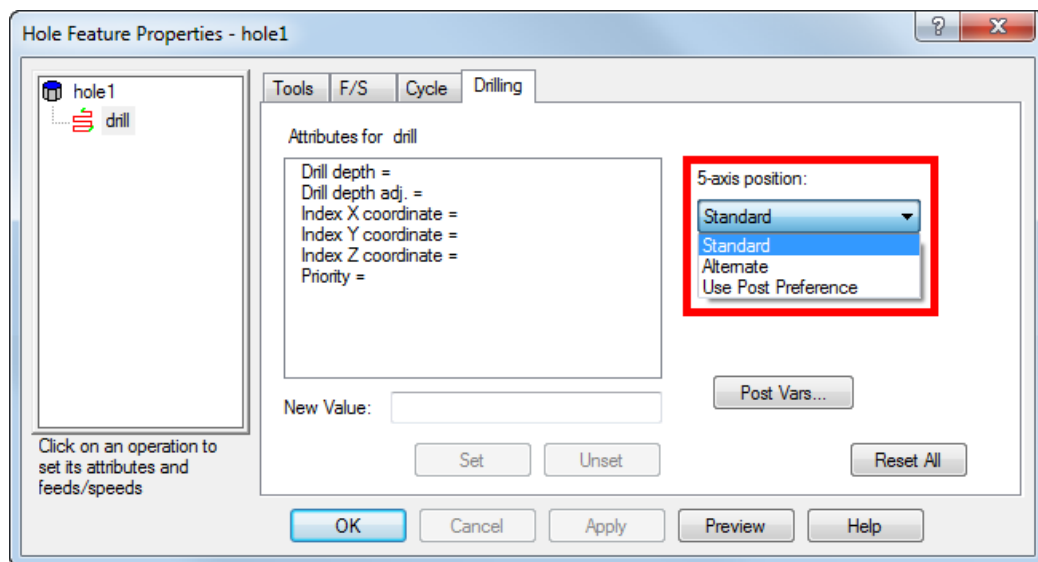
This menu is available in the **Machining Attributes** dialog:



The **Setup** dialog:



And the **Feature Properties** dialog:



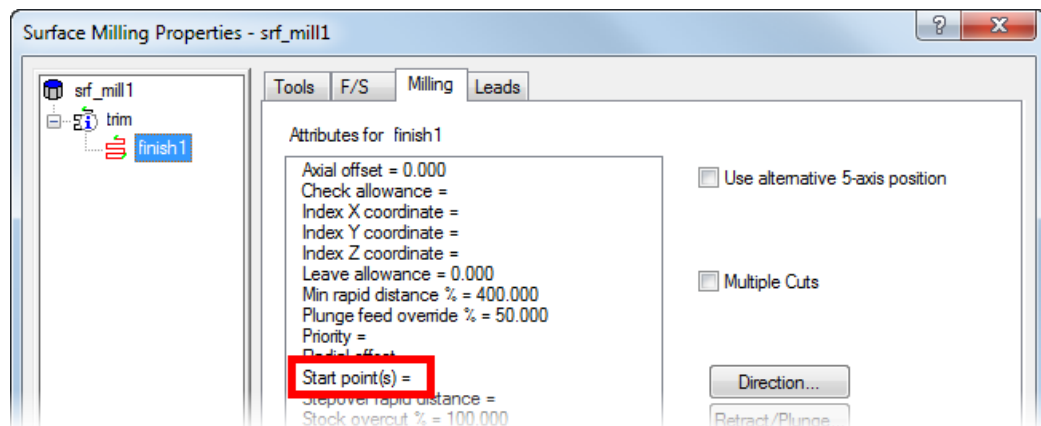


## Override default start points (5AS)

In FeatureCAM 2013 R1, you can override the default start point(s) for 5-axis trim and 5-axis swarf toolpaths, using a curve.

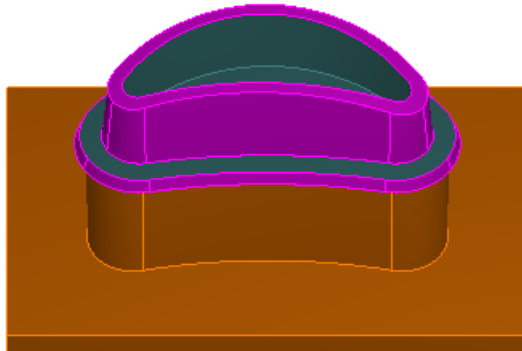
The start point is moved to the projected intersection of the curve and the toolpath. You can use a curve with multiple segments to set multiple start points for a toolpath, and FeatureCAM uses alternating segments to set the start points.

There is a new **Start point(s)** option on the **Milling** tab, where you enter the name of the curve:

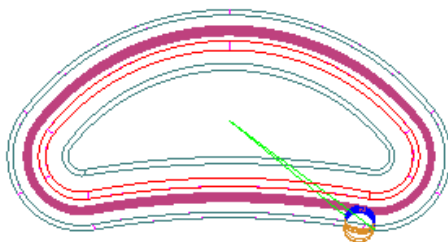


### 5-axis start points example

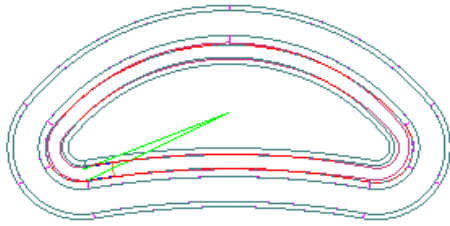
This example uses a 5-axis swarf and two 5-axis trim strategies.



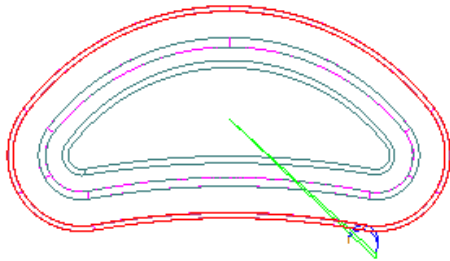
Viewing a centerline simulation from the top, you can see that the default start point for the 5-axis swarf operation is at the lower right of the part.



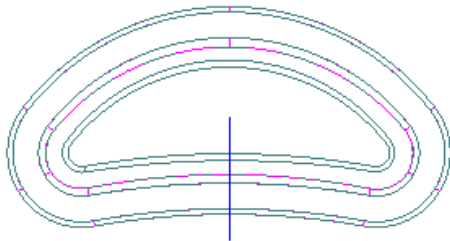
The default start point for the upper trim operation is at the lower left:



And the default start point for the lower trim operation is at the lower right:

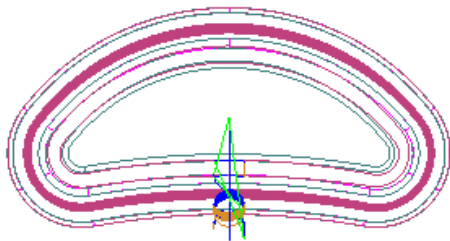


To override the default start points, you must create a curve that intersect the toolpaths where you want the new start points to be. In this example, the curve is shown in blue:



On the **Milling** tab for each strategy, select the **Start point(s)** attribute, enter the name of the curve you created and click the **Set** button to save it.

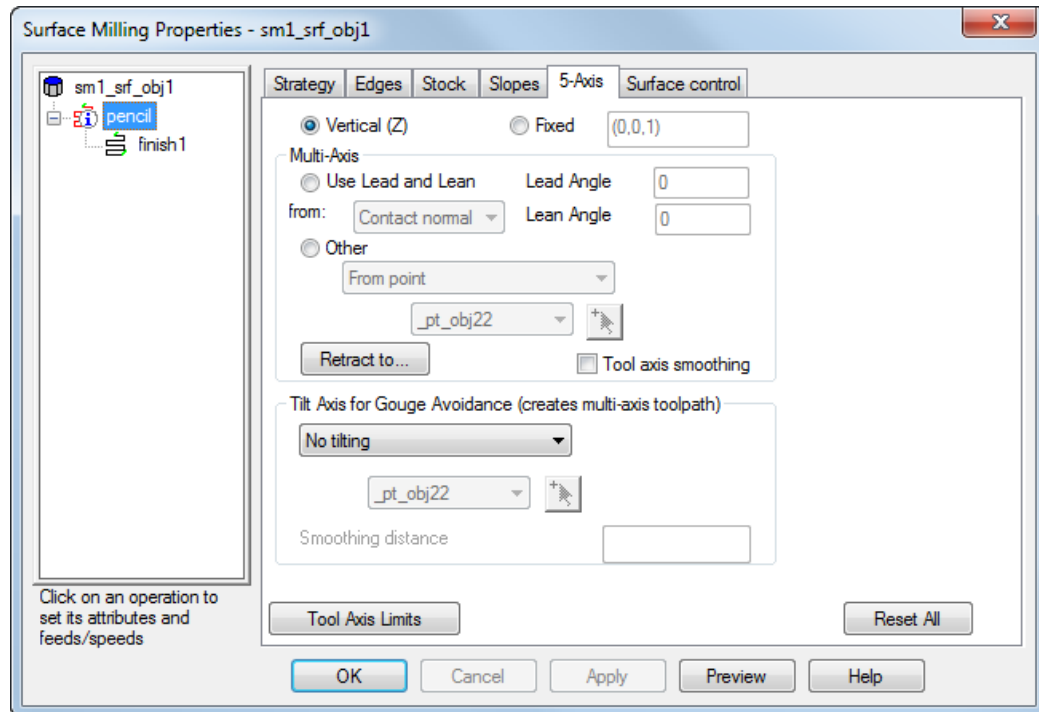
A centerline simulation of all three operations shows that the start points are now aligned along the curve:



## 5-axis options for more strategies (5AS)

In FeatureCAM 2013 R1, there are tool axis options for Pencil, Corner Remachine, and Between two curves strategies.

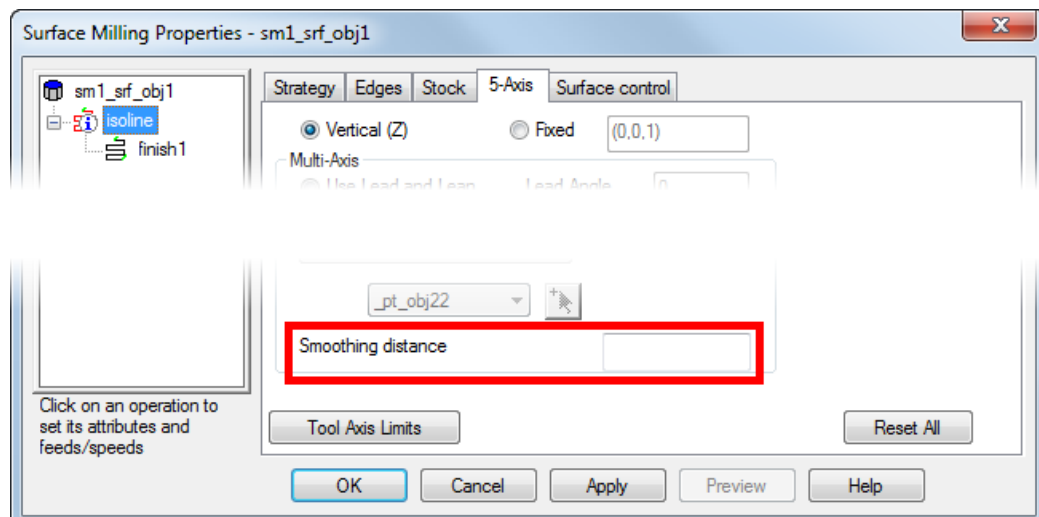
The **5-Axis** tab is now available for these strategies:



Previously, this tab was available only for Z-level finish, Parallel finish, 3D Spiral, Swarf, Isoline, Flowline, Horizontal + Vertical, and 5-axis trim strategies.

## Smoothing distance (5AS)

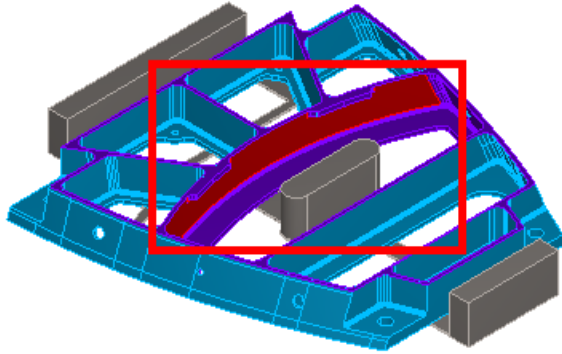
In FeatureCAM 2013 R1, there is a new **Smoothing distance** attribute:



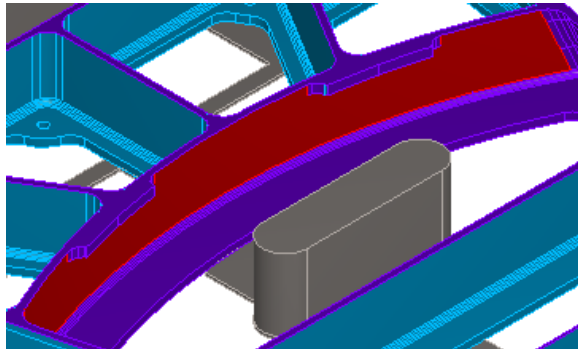
**Smoothing distance** eliminates sudden tool axis changes.

#### Smoothing distance example

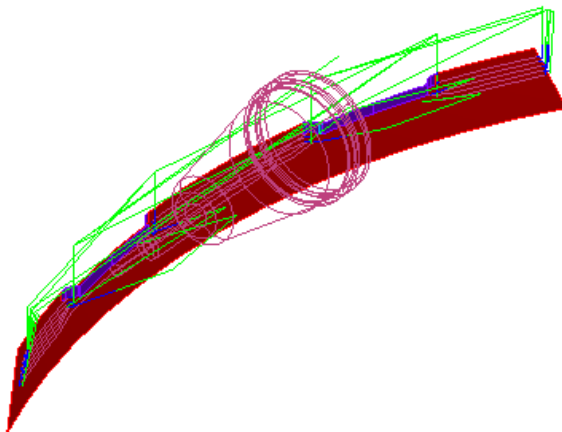
This example part has a vertical surface, shown in red, with two tabs at the top:



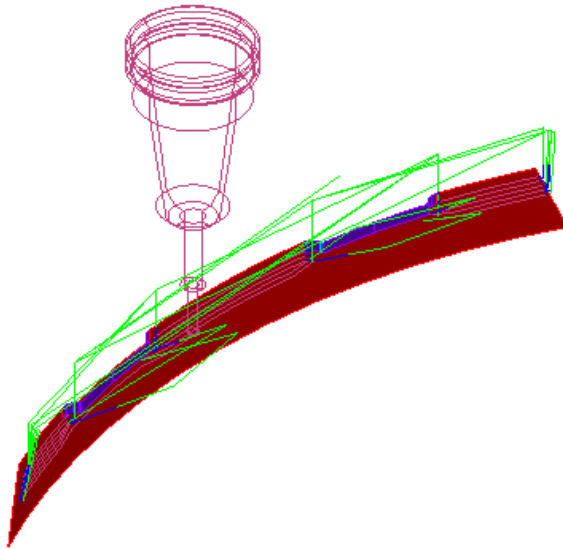
This is the surface in more detail:



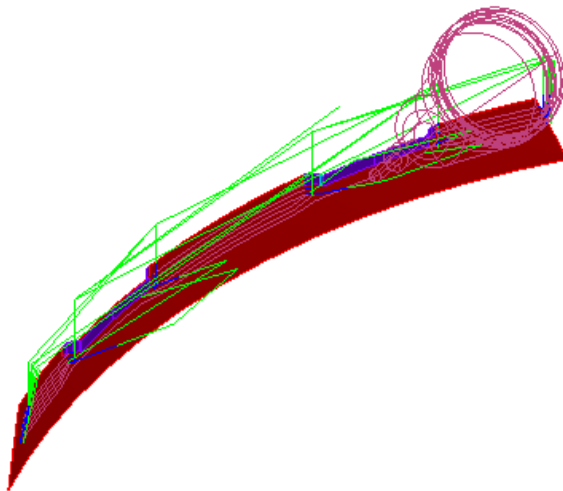
Using an Isoline strategy, with **Tool axis smoothing** enabled and a **Tilt axis** of **Lean then lead** with no **Smoothing distance** value, the tool has sudden changes in direction. For example, it machines the area below the first tab in a tilt position:



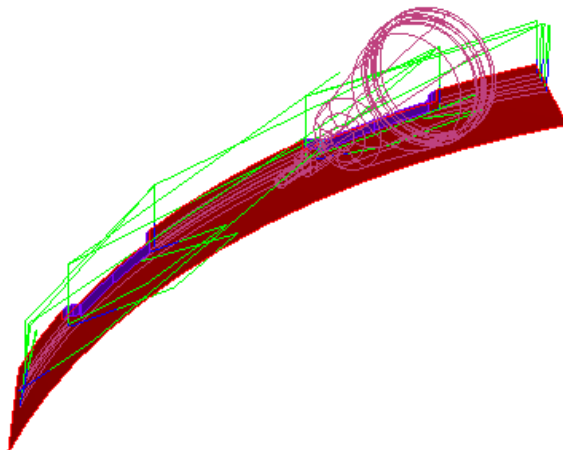
Then returns to a vertical position between the tabs:



Then returns to the tilt position to machine the area below the second tab:



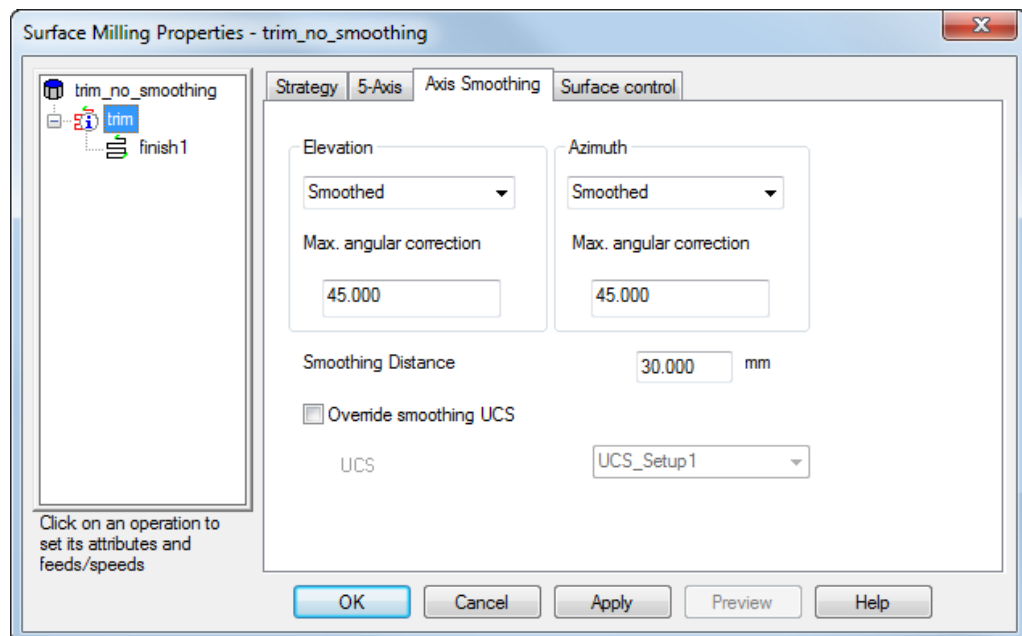
With a **Smoothing distance** value entered that is long enough to cover the area between the two tabs, the tool stays tilted when machining between the tabs:



## Tool axis stabilization (5AS)

In FeatureCAM 2013 R1 you can smooth the Elevation and Azimuth axes independently. Stepped stabilization smoothing locks the rotary axis movement by maintaining a fixed tool axis as much as possible within an angular range. This toolpath results in a series of 3+2-axis segments with 5-axis transitions between them. The transitions can be done on the surface or with a lift.

There is a new **Axis Smoothing** tab:



*You must select the **Tool axis smoothing** option on the **5-Axis** tab and click the **Apply** button to display the **Axis Smoothing** tab.*

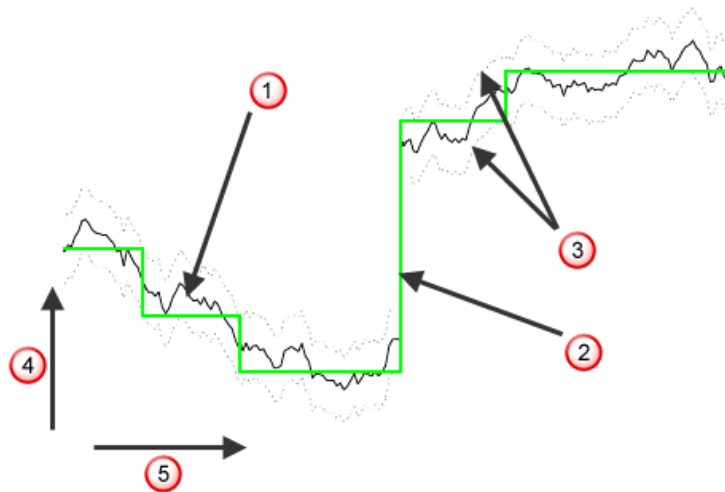
**Elevation** — Select how to smooth the elevation angle of the tool axis. An elevation of  $\pm 90^\circ$  aligns the tool with the Z axis, and an elevation of  $0^\circ$  means the tool is in the XY plane.

**None** — Select this option to have no smoothing. The tool axis orientation moves as and when it needs to move.

**Smoothed** — Select this option to enable the tool axis angle to change smoothly over the **Smoothing distance**. The change in angle will not be more than the **Maximum angular correction** unless the angle of the unsmoothed toolpath varies by more than the **Maximum angular correction** in less than **Smoothing distance**. In such regions, the angle may change by more than the **Maximum angular correction** to give a smooth result.

**Stepped on surface** — Select this option to enable the tool axis angle to change by up to the **Maximum angular correction** to form steps of constant value. To avoid sharp tool axis movements, a smooth transition between steps is made starting at the **Smoothing distance** away from the possible ends of adjacent steps. The tool always remains in contact with the surface.

**Stepped with links** — Similar to **Stepped on surface**, select this option to enable the tool axis angle to change by up to **Maximum angular correction** to form steps of constant value. In this case, the toolpath segments are subdivided at the end of each step, and link moves are inserted so the tool axis changes when the tool is not in contact with the model. This means that tool axis angle of each toolpath segment is constant.



- ① Original toolpath, no smoothing
- ② Smoothed toolpath
- ③ Maximum angular correction limits
- ④ Azimuth or elevation
- ⑤ Toolpath distance

**Maximum angular correction** — Enter the maximum angle the smoothed axis may deviate from the initial in the elevation direction. Changes in angle will not exceed this value unless the angle of the initial toolpath varies by more than the **Maximum angular correction** in less than the **Smoothing distance**. In such regions, the angle may change by more than the **Maximum angular correction** to give a smooth results.

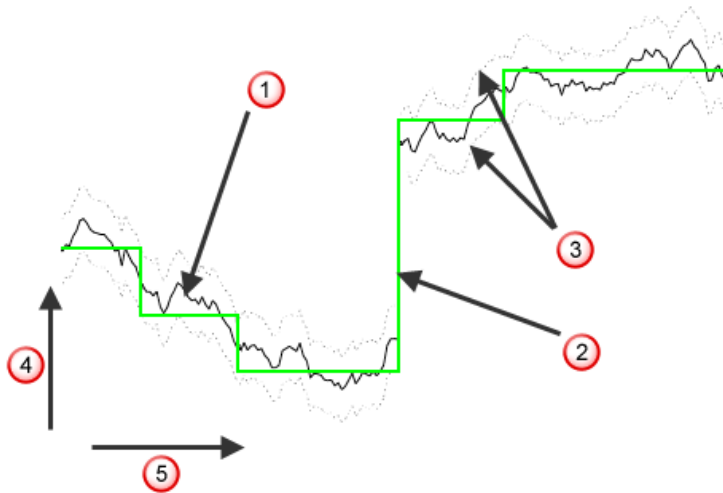
**Azimuth** — Select how to smooth the azimuth angle of the tool axis. Azimuth is the angle between the +X axis and the projection of the tool in the XY plane.

**None** — Select this option to have no smoothing. The tool axis orientation moves as and when it needs to move.

**Smoothed** — Select this option to enable the tool axis angle to change smoothly over the **Smoothing distance**. The change in angle will not be more than the **Maximum angular correction** unless the angle of the unsmoothed toolpath varies by more than the **Maximum angular correction** in less than **Smoothing distance**. In such regions, the angle may change by more than the **Maximum angular correction** to give a smooth result.

**Stepped on surface** — Select this option to enable the tool axis angle to change by up to the **Maximum angular correction** to form steps of constant value. To avoid sharp tool axis movements, a smooth transition between steps is made starting at the **Smoothing distance** away from the possible ends of adjacent steps. The tool always remains in contact with the surface.

**Stepped with links** — Similar to **Stepped on surface**, select this option to enable the tool axis angle to change by up to **Maximum angular correction** to form steps of constant value. In this case, the toolpath segments are subdivided at the end of each step, and link moves are inserted so the tool axis changes when the tool is not in contact with the model. This means that tool axis angle of each toolpath segment is constant.

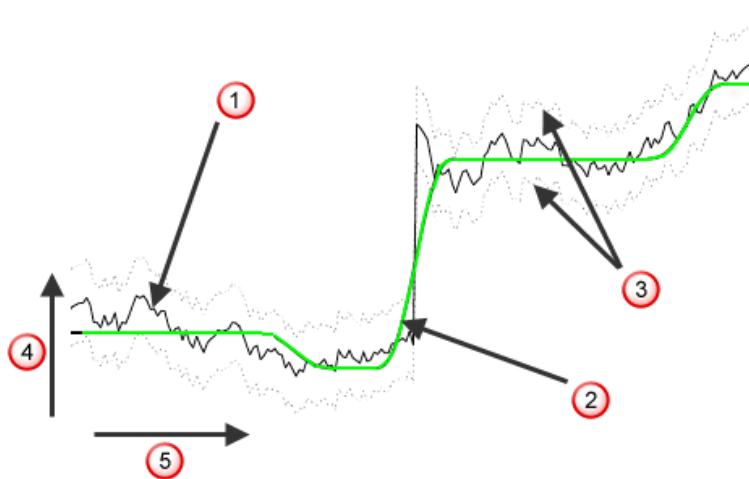


- ① Original toolpath, no smoothing
- ② Smoothed toolpath
- ③ Maximum angular correction limits
- ④ Azimuth or elevation
- ⑤ Toolpath distance



**Maximum angular correction** — Enter the maximum angle the smoothed axis may deviate from the initial in the azimuth direction. Changes in angle will not exceed this value unless the angle of the initial toolpath varies by more than the **Maximum angular correction** in less than the **Smoothing distance**. In such regions, the angle may change by more than the **Maximum angular correction** to give a smooth results.

**Smoothing distance** — Enter the distance over which to smooth the tool axis movement. When using **Stepped on Surface**, or with sudden changes in direction in the original toolpath (such as a right-angled corner), you get rapid changes of orientation of the tool axis which leaves dwell marks. To prevent this, the **Smoothing distance** blends the change in orientation and gives a much improved surface finish.



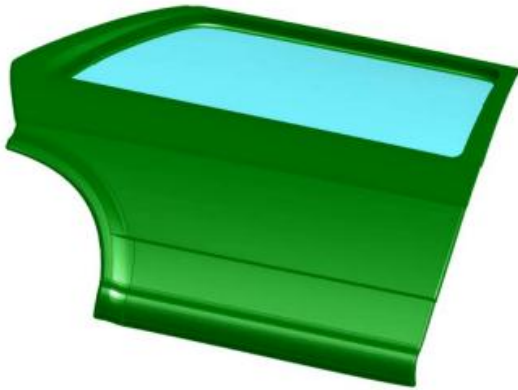
- ① Original toolpath, no smoothing
- ② Smoothed toolpath
- ③ Maximum angular correction limits
- ④ Azimuth or elevation
- ⑤ Toolpath distance

**Override smoothing UCS** — Select this option to use a different UCS to the UCS used to generate the toolpath to define elevation and azimuth for smoothing.

**UCS** — Select the UCS to use when smoothing.

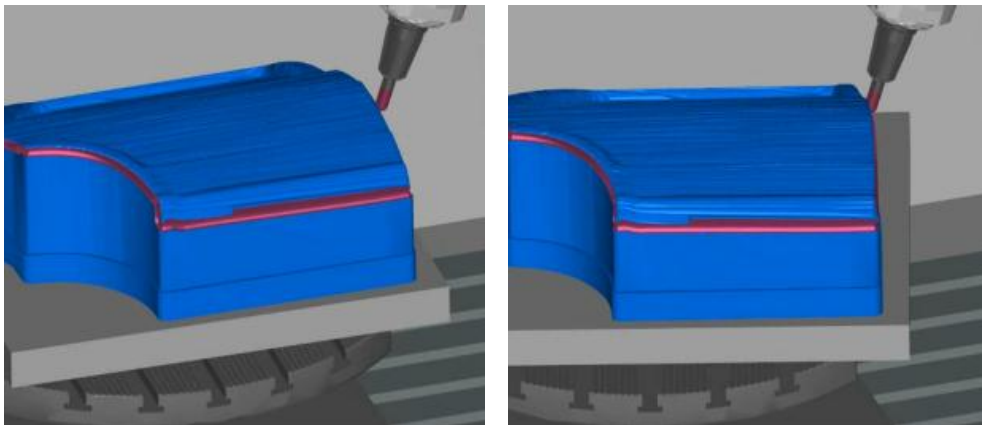
### Stepped on surface example

This car door part uses a trimming toolpath around the edge:

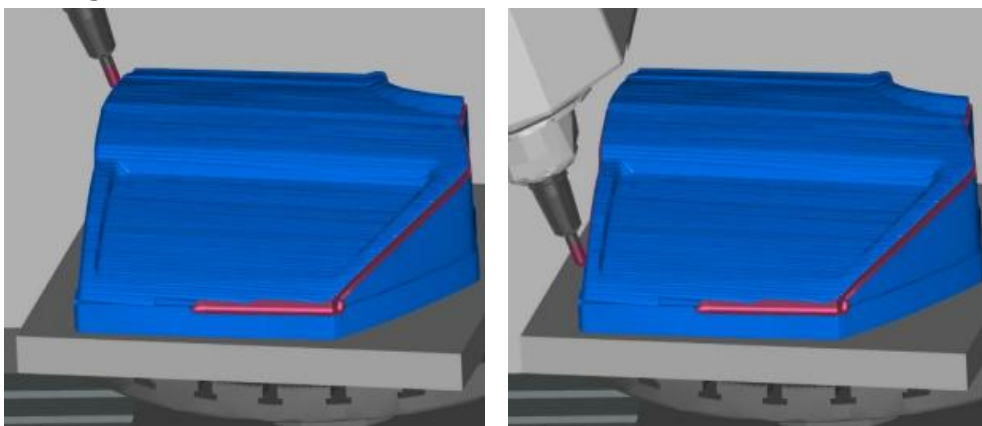


With no smoothing, the rotary table has a rocking motion and sudden direction changes can leave witness marks.

In some areas of the model, it is impossible to avoid the rotation, such as when the tool makes its way around the corners and the wheel arch. However, in areas such as on this long edge of the door, where the machine table is rocking backwards and forwards, that you can avoid this using the **Stepped on surface** options:

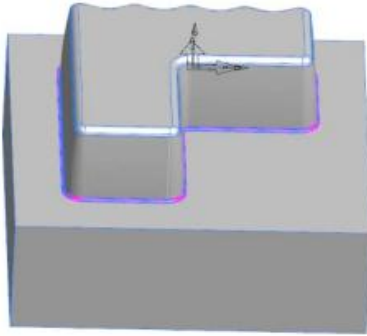


With **Stepped on surface** enabled for both **Elevation** and **Azimuth**, with a **Max. angular correction** of **80**, the table does not rotate while cutting this area:



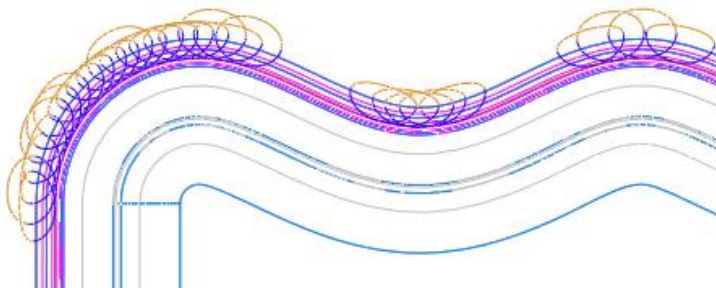
## Stepped with links example

This example part uses the Pencil strategy to cut the bottom fillet:



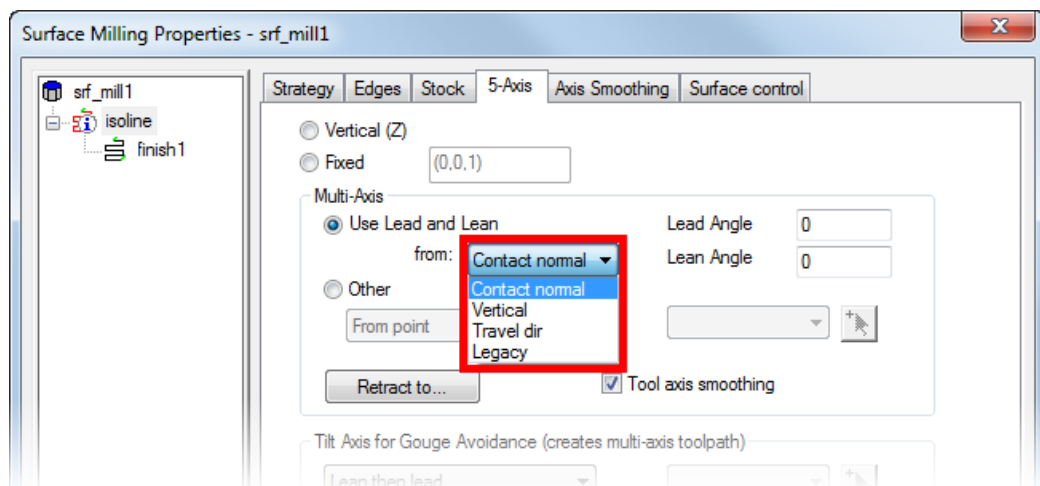
Enabling **Stepped with links** for both **Elevation** and **Azimuth** reduces the tool movement.

A centerline simulation shows the links that FeatureCAM inserts links so that it can change direction while not in contact with the surface:



## Improved tool axis options (5AS)

The tool axis options on the **5-Axis** tab have been improved:



For **Use Lead and Lean**, you can now select where the lead and lean angle is measured from.

- **Contact normal** — Select this option to lead and lean from the surface normal at the contact point.
- **Vertical** — Select this option to lead and lean from the Setup's Z direction.
- **Travel dir** — Select this option to lead and lean from the perpendicular to the direction of movement
- **Legacy** — Select this option to use the lead and lean style used in FeatureCAM 2012 R3.

For **Other**, the **From point**, **To point**, **From line**, **To line**, **From curve**, and **To curve** options are now more accurate.



*The old options are still available as **From point (old-style)**, **To point (old-style)**, **From line (old-style)**, **From curve (old-style)**, and **To curve (old-style)**.*

---

## FeatureTURN improvements

These are the most significant Turn improvements in FeatureCAM 2013 R1:

- Part transfers are now fully integrated into FeatureCAM with a new **Part Handling** (see page 85) feature.
- You can control the feed rate (see page 87) on approach moves.
- You can now recognize a turned profile (see page 91) from an **.stl** file.
- Canned cycles (see page 90) are now supported for a semi-finish pass.

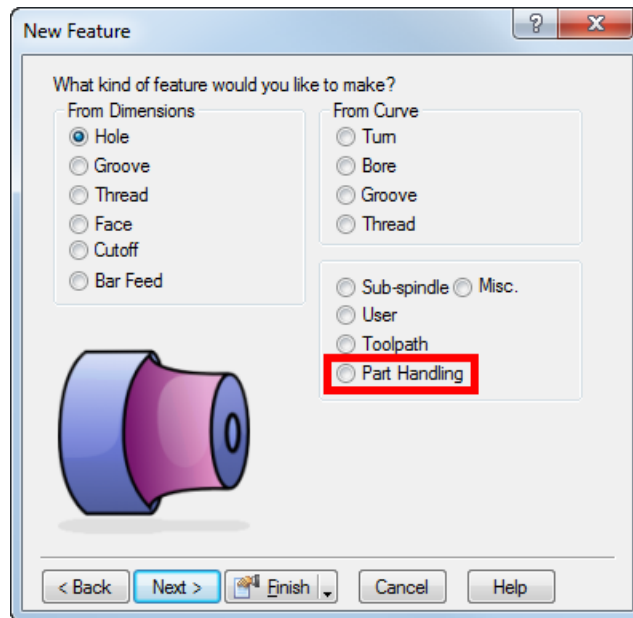
### Integrated part handling (TURN)

Part transfers are fully integrated into FeatureCAM 2013 R1. These transfer types are available:

- |                         |                    |
|-------------------------|--------------------|
| ▪ Slug transfer         | ▪ Part support on  |
| ▪ Reverse slug transfer | ▪ Part support off |
| ▪ Bar pull              |                    |

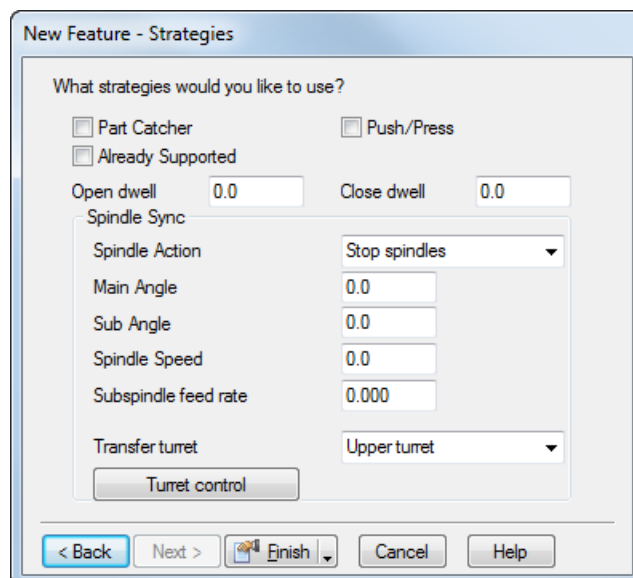
Post and simulation are supported.

There is a new **Part Handling** feature in the turn **New Feature** wizard:



Select **Part Handling** and click **Next** to open the **New Feature - Dimensions** page:

Select the type of Part Handling feature you want and enter the dimensions. Click **Next** to open the **New Feature - Strategies** page:



Set the attributes for the Part Handling feature, for example:

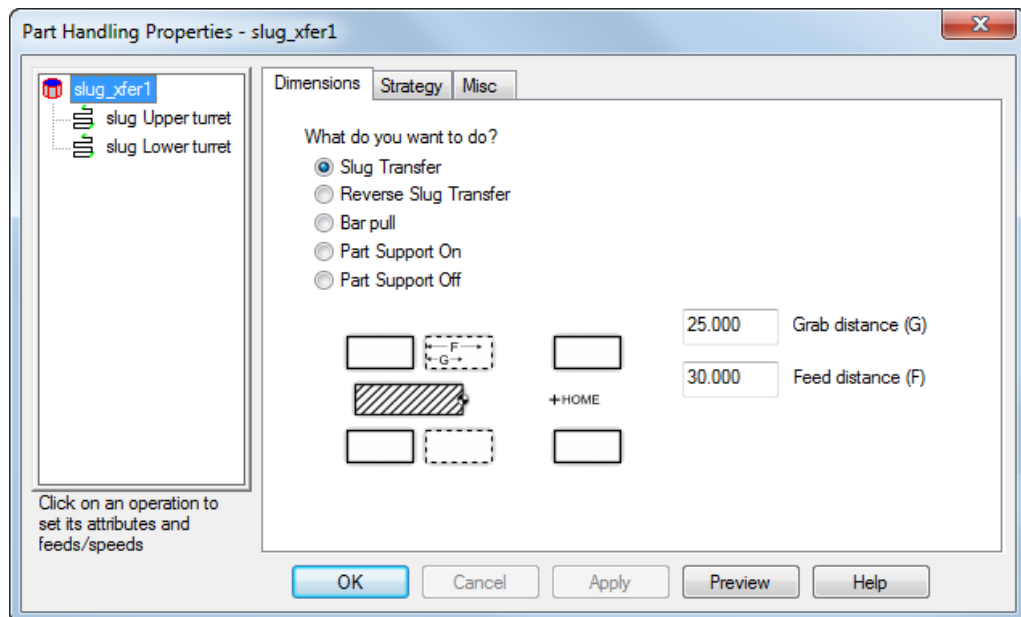
**Part catcher** — Select this option if you want to instigate the part catcher.

**Already Supported** — Select this option to indicate that the part is already supported.

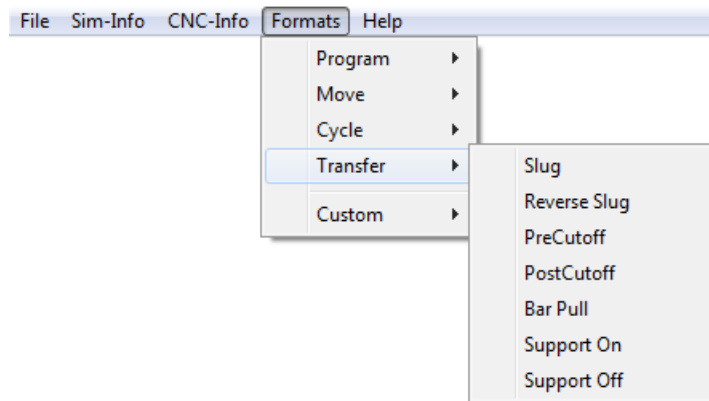
**Sub Angle** — Enter the angle that you want the sub-spindle to rotate to before it moves to collect the part.

Click **Finish** to save the Part Handling feature.

In the same way as for other features, the Part Handling feature name is displayed in the **Part View** and on the **Op List** tab, and you can edit the feature in the **Part Handling Properties** dialog:



There is also a new **Transfer** format type in the XBUILD menu:

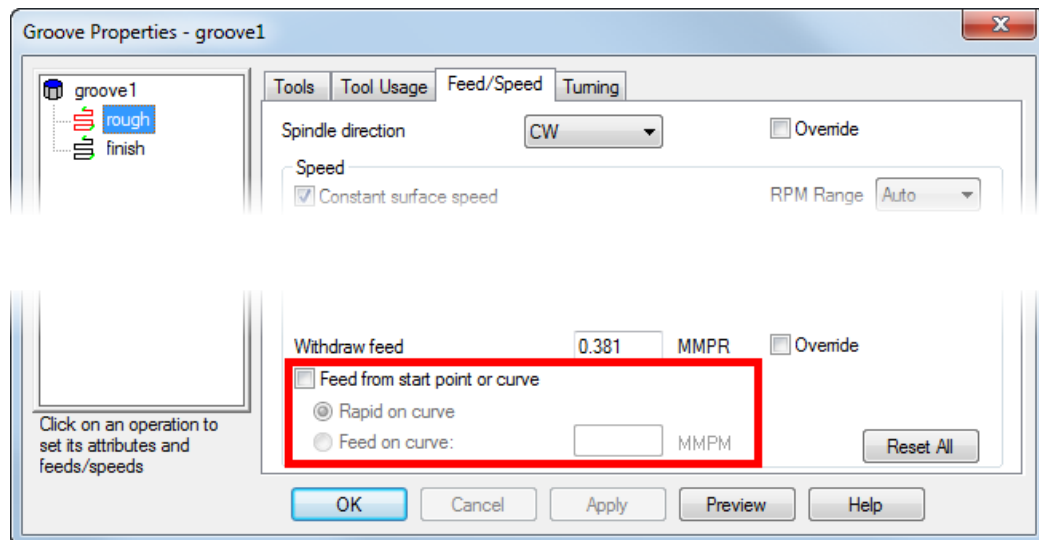


## Control feed rate on approach moves (TURN)

In FeatureCAM 2013 R1, you can set a start curve as a feed move for these operations:

- Turn
- Bore
- Groove
- Thread
- Face
- Drill
- Cutoff

There is a new **Feed from start point or curve** option on the **Feed/Speed** tab at feature level:



**Feed from start point or curve** — Select this option to use a feed move from the **Start point** or the end of the **Start curve** set on the **Turning** tab, to the beginning of the toolpath:

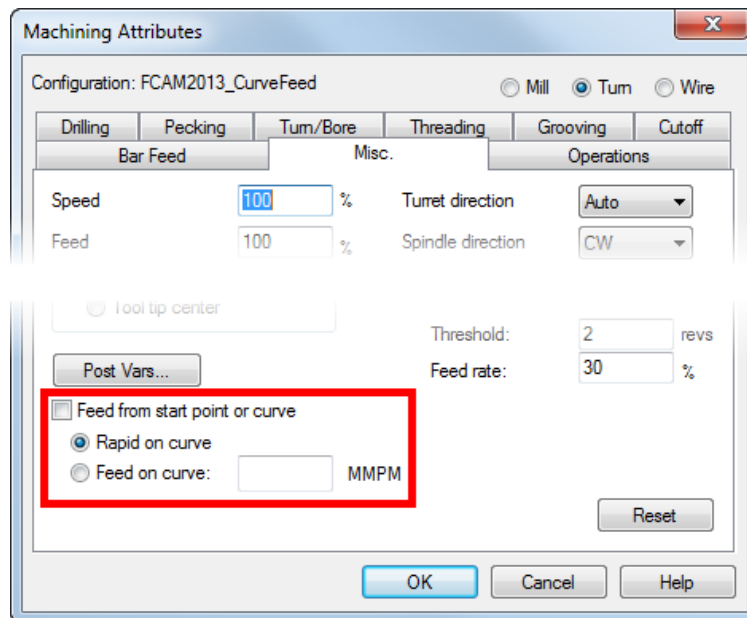
- **Rapid on curve** — Select this option if you are using a Start curve and want to use a rapid move along it.



*Selecting **Feed from start point or curve** and **Rapid on curve**, is the equivalent of selecting the **Feed from start** option on the **Turning** tab in previous versions of FeatureCAM.*

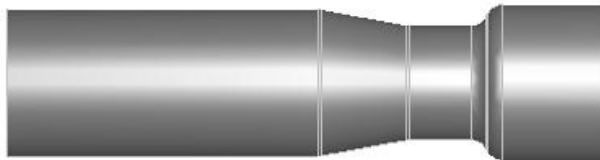
- **Feed on curve** — Select this option if you are using a Start curve and want to use a feed move along it. Enter the feed rate value(s). You can set the feed move as a single value, or use a comma separator to enter multiple values, for example **200, 100, 50**. If you enter multiple values, the feeds are applied to the curve segments in reverse order. So with these values, the last segment has a feed rate of **50**, the second last has a feed rate of **100** and any remaining segments have a feed rate of **200**. If you do not enter any feed rate values, FeatureCAM uses the default feed rate.

You can also set the default value for this option in **Machining Attributes**, on the **Misc.** tab:



### Start curve as feed move example

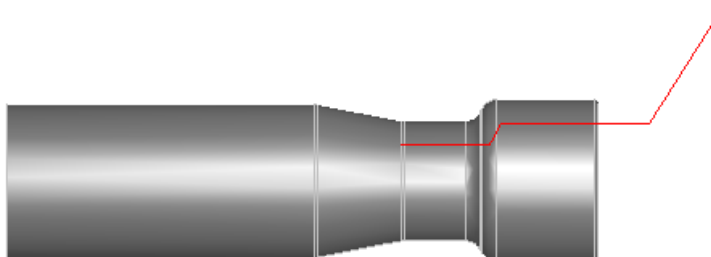
This example turned part has a Turn, Hole, and Groove feature:



A 3D simulation with 3/4 view shows that the tool has to pass through a narrow channel to access the Groove feature:

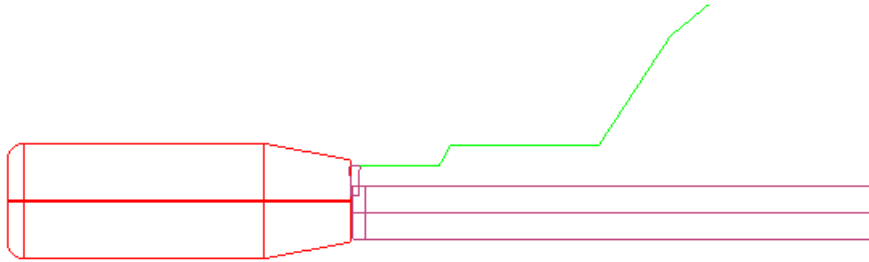


To control the movement of the tool through that narrow channel, you can create curves and set them as **Start point** and **End point** on the **Turning** tab.



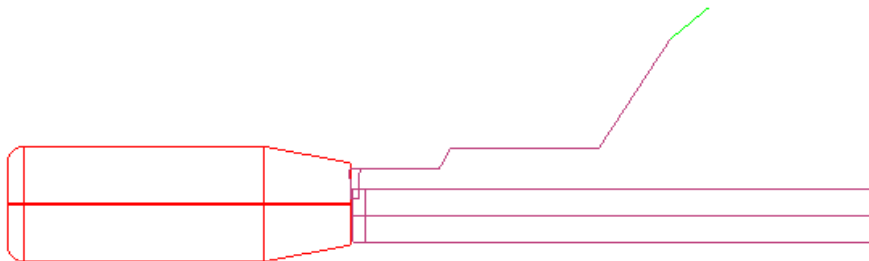


A Centerline simulation of just the Groove feature, shows the approach move in green, which is a rapid move:



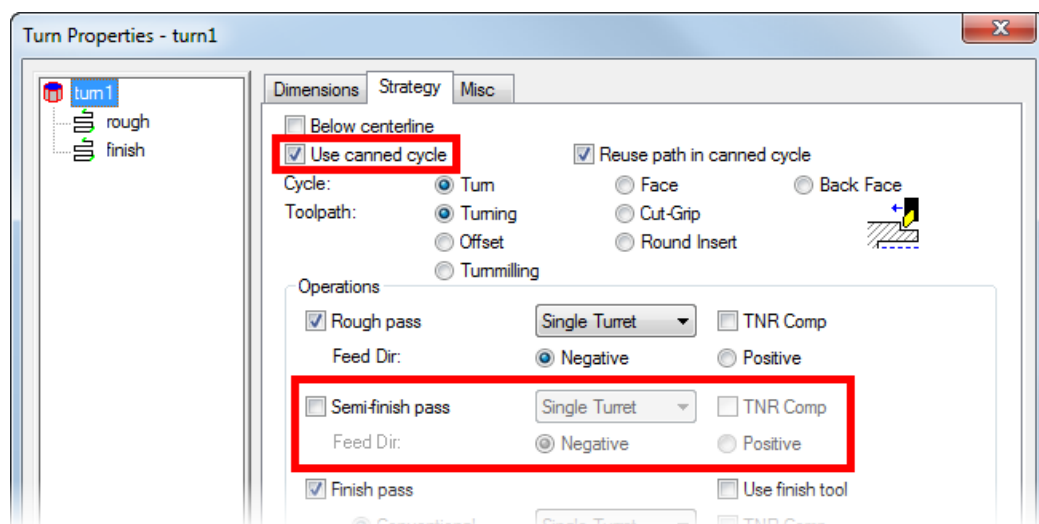
On the **Feed/Speed** tab, select **Feed from start point or curve** and **Feed on curve**. Enter the feed value(s) and click **Apply**.

The Centerline simulation now shows the approach move in purple, a feed move:



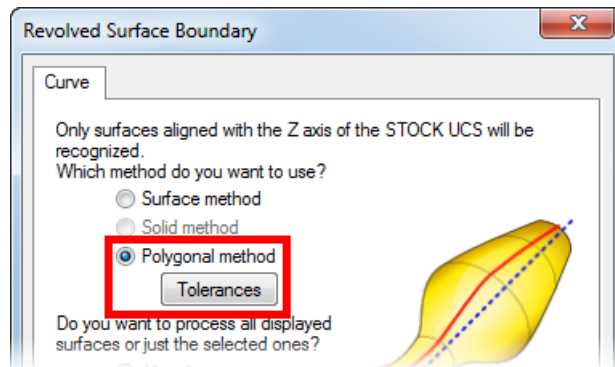
## Canned cycle support for semi-finish (TURN)

In FeatureCAM 2013 R1, you can use a finishing canned cycle with a semi-finish pass. The **Semi-finish pass** option is now available when the **Use canned cycle** option is selected, on the **Strategy** tab:



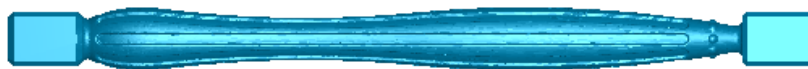
## STL curve of revolution recognition (TURN)

In FeatureCAM 2013 R1, you can recognize a turned profile from an .stl file using the **Polygonal method**:



The **Polygonal method** uses the simulation engine to triangulate the file and create a profile curve, based on the **Tolerances** set.

This example balustrade part is an imported .stl file.



With surface shading turned off, you can see the triangles that make up the .stl file:



To create the turned profile, select **Construct > Curve > From Surface > Revolved Surface Boundary** from the menu.

- 1 In the **Revolved Surface Boundary** dialog, select **Polygonal method**.
- 2 Optionally click the **Tolerances** button to change the default **Triangle tolerance** and **Arcline approx. tolerance** values.
- 3 Click the **Preview** button to see the boundary highlighted in dark blue in the Graphics window:



FeatureCAM has found the correct profile height of the revolved square ends of the balustrade.

- 4 Click **OK** to create the curve.

You can use this curve to create a Turn feature.

# FeatureTURNMILL improvements

These are the most significant Turn/Mill improvements in FeatureCAM 2013 R1:

- You can now set the tool used on a rotary Surface Milling feature to a fixed B angle (see page 92).
- FeatureCAM now recognizes Hole features (see page 95) from imported SolidWorks parts.
- The Import wizard is now available when importing turn/mill solids.
- You can now use the Y axis with a Points List pattern (see page 94).

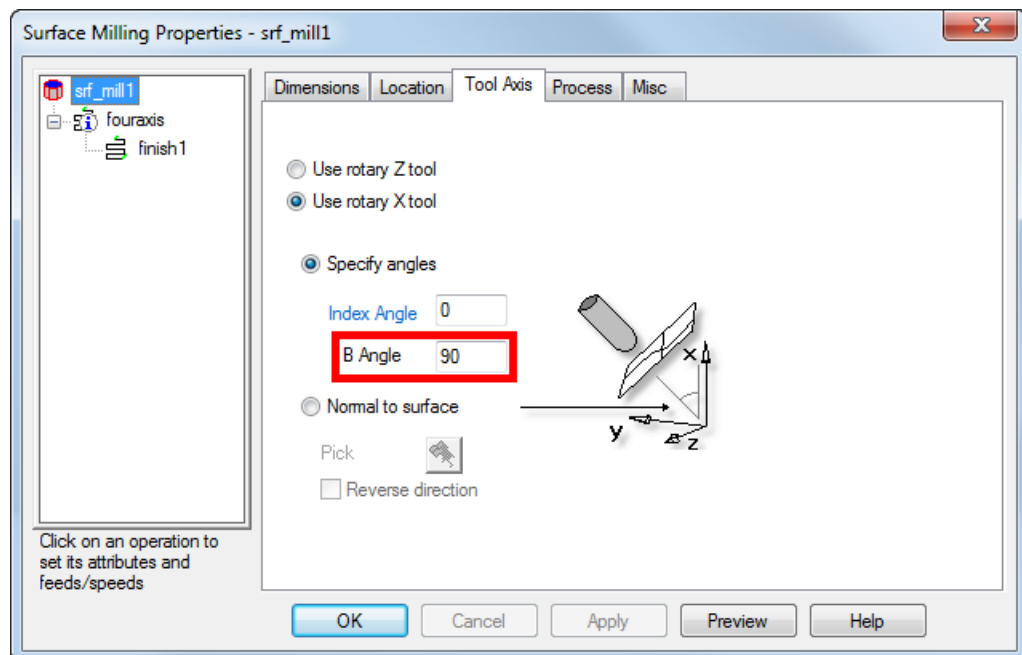
## B-axis rotary (TURNMILL)

In FeatureCAM 2013 R1, there is a new method in B-axis turn/mill that enables the tool used on a rotary Surface Milling feature to be set at a fixed B angle.



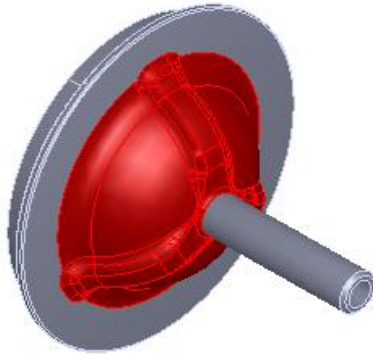
*To use this method you need the 3D, Turn/Mill and Advanced Turn/Mill components. You do not need the 5-axis Simultaneous component.*

Set the **B Angle** on the **Tool Axis** tab:



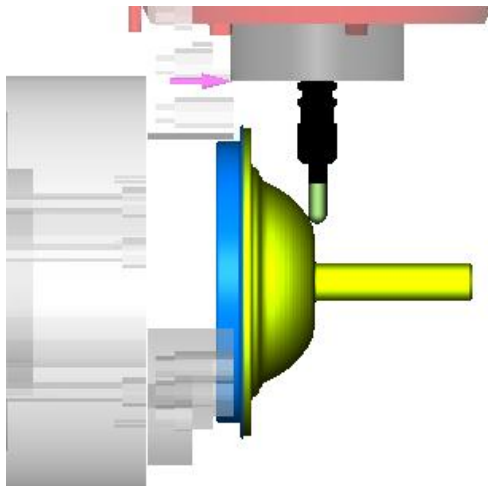
### Turn/Mill B-axis rotary example

This example cast part has a 3D Surface feature, shown in red:

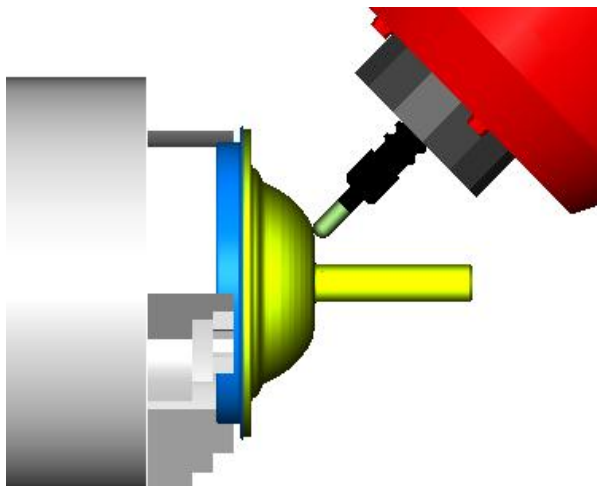


The shaft on the front of the part makes it difficult to machine with the default angle.

Using a 4-axis rotary strategy with the default angle, the head of the machine collides with the chuck, at the point marked with a pink arrow:

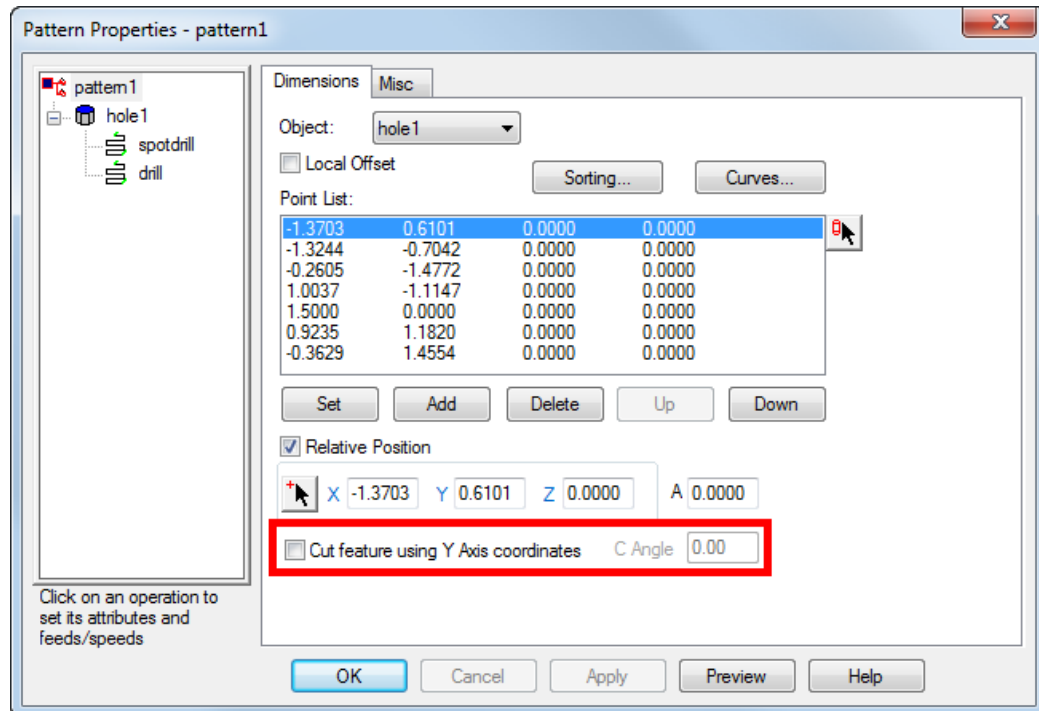


Using a fixed B-angle tool axis of 45 degrees, the tool tilts over and avoids any collisions:

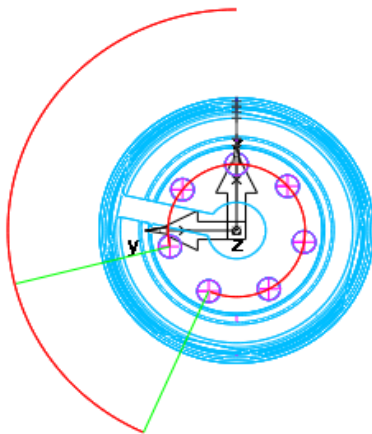


## Use Y axis with Points List pattern (TURNMILL)

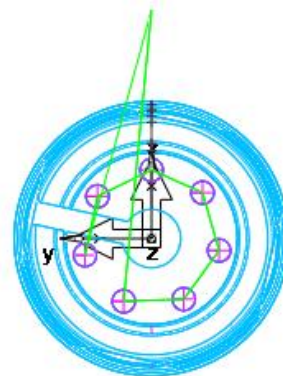
In FeatureCAM 2013 R1, there is a new **Cut feature using Y Axis coordinates** option when cutting a point list pattern on the Z face of a turn/mill part:



Previously, you could only rotate around the C axis, for example:



Now you can make use of your turn/mill machine's ability to use the Y-axis.



Select **Cut feature using Y Axis coordinates** and optionally enter a **C Angle**.

## Import wizard (TURNMILL)

In FeatureCAM 2013 R1, the Import wizard is available when importing turn/mill solids. In previous versions of FeatureCAM, the only alignment option was to pick a surface of revolution. Now there are many more options, similar to the import wizard for milling parts.

The **Import Results** dialog now displays when you import a solid and you have the option to use the import wizard to align the part.

## AFR for SolidWorks Holes (TURNMILL)

In FeatureCAM 2013 R1, you can recognize Turn/Mill Hole features from an imported SolidWorks part. FeatureCAM recognizes both holes on the face and around the index axis in one setup. It also recognizes patterns.



# Index

▪

.sldprt • 12  
.stl files • 48, 49, 91

<

<Z-XFER-COORD> • 41

## 2

2.5D improvements • 31

## 3

3D MX • 57  
3D products • 57  
3D simulation performance • 30  
3D spiral out • 9

## 5

5-axis position (5AP) • 72  
5-Axis tab (5AS) • 76

## 6

64-bit part library • 10

## A

Arc in • 40  
Arc leads • 40

Arc out • 40  
Arc-in angle • 40  
Arc-out angle • 40  
Area removal (3D HSM) • 63  
Associativity • 12  
Automatic Center Tool • 18  
Avoid clamps • 31  
Avoid fixtures • 31  
Axis smoothing (5AS) • 79  
Azimuth (5AS) • 79

## B

Background gradient • 16  
Balloon help • 1  
Bar pull (TURN) • 85  
Barrel cams • 33  
Bottom up • 61

## C

CATIA V5-6 R2013 (R22) • 42  
Check allowance (2.5D) • 31  
Check surfaces (2.5D) • 31  
Clamps • 31  
Closed areas only (3D HSM) • 63  
Contact normal (5AS) • 84  
Continuous spiral (Drill/Mill) • 53  
Convert units (feed, speed) • 47  
Creo 2 • 42  
Cusp control (3D HSM) • 70  
Custom colors • 20



## D

Detect material thicker than • 34  
DMG FD Series • 10  
Drag select • 45

## E

Elevation (5AS) • 79  
Expand area by • 34

## F

Feature color • 20  
Feed distance (TURN) • 85  
Feed from start point or curve (TURN) • 87  
Feed on curve (TURN) • 87  
File send • 25  
Finish axial allowance (3D LITE) • 8  
Finish bottom • 6  
Finish NT toolpaths • 6  
First step • 22  
Fixtures • 31  
Flutes • 19

## G

Glass style • 16  
Grab distance (TURN) • 85  
Graphics area • 16

## H

Hand • 19  
Handedness • 19  
Helical options • 22, 23  
Helical ramping • 22, 23  
Hide all in Setup • 24  
Hide features • 24

## I

In to out, 3D spiral (3D HSM) • 9  
Index axis UCS • 13  
InStr • 42  
InStrRev • 42  
InterOp R23 • 42

## L

Leave axial allowance (3D MX) • 9  
Left-hand tool • 19  
Linear approximation • 22, 23

## M

Machine stock only • 34  
Main angle (TURN) • 85  
Matsuura CUBLEX-25 • 10  
Maximum angular correction (5AS) • 79  
Maximum ramp angle • 22  
Maximum ramp distance • 22  
Mazak Integrex e-800V II • 10  
Min. Z increment (3D HSM) • 70  
Minimum gap distance • 34  
Minimum length removed • 34  
Minimum ramp • 22  
Minimum ramp distance • 22  
Minimum Z ramp distance • 22  
Mori Seiki NMV5000 • 10  
Mori Seiki SuperMILLER 400 • 10  
Multiple start points • 58

## N

Network database improvements • 28  
Never select this tool • 3  
NT toolpaths for finishing • 6

## O

OD location (TURNMILL) • 41  
Okuma VTM-120YB • 10  
Override (feed, speed) • 47  
Override smoothing UCS • 79  
Override start points • 58, 74

## P

Parasolid kernel 24.1 • 12  
Part catcher (TURN) • 85  
Part handling (TURN) • 85  
Part library • 10  
Part library format • 10  
Part support (TURN) • 85  
Plunge clearance • 22

- Plunge feed override • 22
- Plunge points • 22
- Plunge tab • 22
- Polygonal method • 91
- Positive/negative position (5AP) • 72
- Post processor • 50
- PostCutoff (TURN) • 85
- PreCutoff (TURN) • 85
- Preferred orientation (5AP) • 72
- Probe tool group • 4
- Probe tool properties • 4
- Probing tools • 4

## R

- Ramp angle • 22
- Ramp distance • 22
- Rapid on curve (TURN) • 87
- Remove area less than (3D HSM) • 63
- Remove unsafe areas (3D HSM) • 63
- Reserved words • 50
- Reset all colors • 20
- Retract point • 22
- Reverse slug transfer (TURN) • 85
- Revolved surface boundary • 91
- Right-hand tool • 19
- Round stock • 30
- Rounding mill improvements • 3
- Roundness tolerance • 30

## S

- Scallop height (3D HSM) • 70
- Semi-finish pass (TURN) • 90
- Send part files • 25
- Shaded grey style • 16
- Show all in Setup • 24
- Show features • 24
- Show Flutes on End Mills • 19
- Show tool holder • 19
- Side liftoff angle (TURN) • 38
- Sim. speed slider • 49
- Simulation improvements • 30
- Simulation performance • 30
- Simulation speed slider • 49
- Simulation start point buttons • 24
- Slug transfer (TURN) • 85
- Smoothed (5AS) • 79
- Smoothing distance (5AS) • 76

- Snap to STL • 49
- Solid source file • 12
- SolidWorks associativity • 12
- SQL Server Express • 28
- Start curve • 58, 74
- Start point • 22
- Start point(s) (3D MX) • 74
- Stepped on surface (5AS) • 79
- Stepped with links (5AS) • 79
- STL as clamp • 48
- STL curve of revolution (TURN) • 91
- STL files • 48, 49, 91
- Stock curve finder • 12
- Stock model operation • 56
- Stock model options • 34
- Stock model rest roughing • 34
- Stock model settings • 34
- Sub angle • 85
- Sub spindle operations color • 20
- Supported (TURN) • 85

## T

- Tip-to-shoulder length • 3
- Tool axis smoothing (5AS) • 79
- Tool Manager background • 18
- Tool Manager improvements • 18
- Tool shank • 19
- Toolpath edit • 54
- Touch-off at the shoulder • 3
- Touch-off point, rounding mill • 3
- Transfers (TURN) • 85
- Transform (Toolpath feature) • 54
- Turn improvements • 37

## U

- Unigraphics NX 8.0 • 42
- Unit conversion (feed, speed) • 47
- Use axial depth of cut • 34
- Use canned cycle (TURN) • 90
- Use continuous spiral • 53
- Use depth of cut • 34
- Use new stock curve finder • 12
- Use STL as clamp • 48
- User interface improvements • 15

## V

Vertical mill/turn • 10

VerticalMillTurnAddin.bas • 10

## X

XBUILD • 50

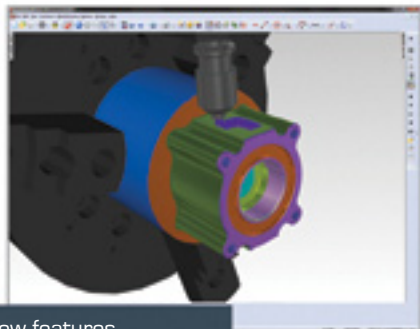
## Z

Z ramp clearance • 22

Zip part files • 25

# FeatureCAM 2013

FeatureCAM 2013



new features

Delcam TV



[www.delcam.tv](http://www.delcam.tv)

FeatureCAM Learning Zone



[www.delcam.tv/lz](http://www.delcam.tv/lz)

FeatureCAM Website



[www.featurecam.com](http://www.featurecam.com)



THE QUEEN'S AWARDS  
FOR ENTERPRISE:  
INNOVATION  
2010



THE QUEEN'S AWARDS  
FOR ENTERPRISE:  
INTERNATIONAL TRADE  
2011

## Powering your productivity

Delcam Headquarters | Small Heath Business Park | Birmingham | B10 0HJ | UK  
+44 (0)121 766 5544 | [marketing@delcam.com](mailto:marketing@delcam.com) | [www.delcam.com](http://www.delcam.com)

To contact your local reseller, visit [www.delcam.com/resellers](http://www.delcam.com/resellers)

© Copyright Delcam plc 2012. All other trademarks are the property of their respective owners.