



Advanced  
Manufacturing  
Solutions

# FeatureCAM 2012

What's New

---

FeatureCAM 2012 R3

# What's New



Copyright © 1995-2012 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: 2012 R3 Date: 13 April 2012 13:51

# Contents

## What's New in FeatureCAM 2012 R3 1

General: Only one user can edit FeatureCAM files at a time .....	1
General: Enter custom G-code into the NC program .....	2
General: Soft errors .....	4
User interface: Set priority for multiple operations on the Op List .....	4
Paste special: Copy attributes from one feature to many.....	4
Stock: Shaded and translucent .....	6
Import: New surface naming scheme groups surfaces .....	7
Import: PTC Creo files supported .....	7
Tool mapping: 8-digit Diameter & Length offset registers .....	7
Toolpath edit: Insert multiple lines of G-code.....	8
2.5D: Rough cutter comp. machining attribute.....	8
2.5D: New technology (NT) toolpaths for Side features .....	9
2.5D: Select Holes by depth or diameter .....	13
Viewing modes: Horizontal milling and vertical lathe .....	14
MTT: Better user interface .....	14
Tool database: FeatureCAM add-in to import TDM tools.....	15
API: Help.....	15

## What's New in FeatureCAM 2012 R2 16

Network database: Shared machining configurations .....	16
Import improvements .....	18
User interface: Copy and paste from PowerSHAPE .....	18
2.5D: New technology (NT) toolpaths .....	18
Differences between NT and traditional toolpaths .....	23
Better Minimize tool retract with NT toolpaths .....	24
2.5D: Separate lead-in and lead-out distances .....	28
2.5D: Thread Mill tool hand.....	29
2.5D: Face Mill tools for Chamfers.....	30
2.5D: Hole cycle G74 and G88 .....	33
3D: Z-level finish: Flat surface support.....	33
3D: Save boundary curve .....	34
5-axis: Turn/Mill B-axis alternative position.....	37
Simulation: Step backward .....	37
Simulation: Faster 3D simulation .....	37
Simulation: New keyboard shortcuts.....	38
Machine simulation: Better collision display.....	38
Turn/Mill: Cylindrical interpolation support .....	39
Turn/Mill: Separate fixture offset for each feature .....	41
XBUILD: Use Delcam post processor files.....	42

Probing: UDF available .....	42
API: New functions .....	43
Other improvements .....	43

## What's New in FeatureCAM 2012 44

User interface: More isometric views .....	44
User interface: Keyboard shortcuts.....	46
User interface: Part view multiple select .....	47
Tooling: Better Face Mill tools.....	48
Tooling: Cutting data by tool .....	49
Tool cutting data example .....	51
Depth of cut precedence .....	56
2.5D: Total stock for finish pass.....	60
2.5D: Toolpath feature improvements.....	61
2.5D: Exact Z step .....	65
2.5D: Back Bore holes .....	67
Simulation: Part compare extended.....	68
3D: Z-level spiral roughing.....	70
Turn: Solid model as tool holder .....	71
Solid tool holder example .....	72
Turn: Better tool holder graphics.....	79
Turning leave allowance .....	80



# What's New in FeatureCAM 2012 R3

FeatureCAM 2012 R3 offers all of the original features of FeatureCAM 2012 R2, but with numerous improvements. The most significant improvements are detailed in the following topics.

---

## General: Only one user can edit FeatureCAM files at a time

In FeatureCAM 2012 R3, a file is *locked* when a user opens it. If a second user tries to open the file, a message is displayed:

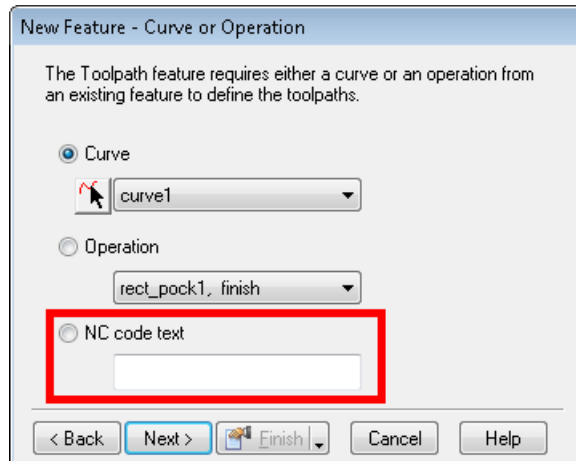
**The requested file is locked by [user]. Would you like to open a read only copy?**

Files that are read-only display **[Read Only]** in the title bar after the name of the file.

If you open a read-only copy, you can't save any changes using **Save**. (You can save a copy of the file using **Save As**.)


# General: Enter custom G-code into the NC program

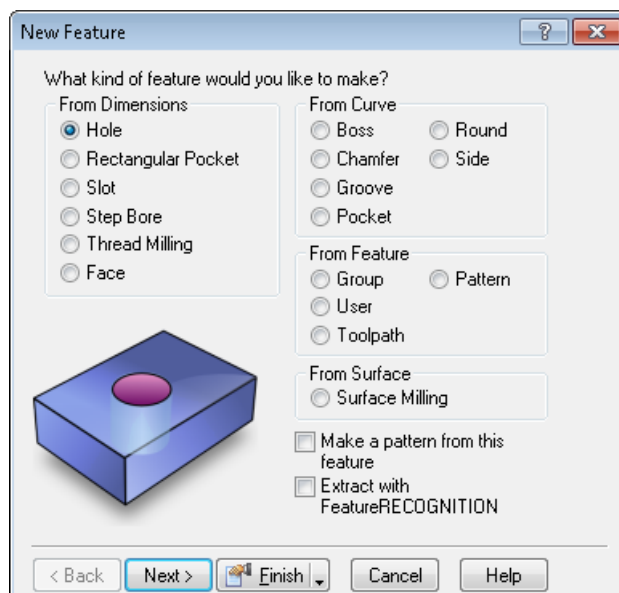
In FeatureCAM 2012 R3, you can enter G-code directly into the NC program, by using the new **NC code text** option for creating a Toolpath feature:



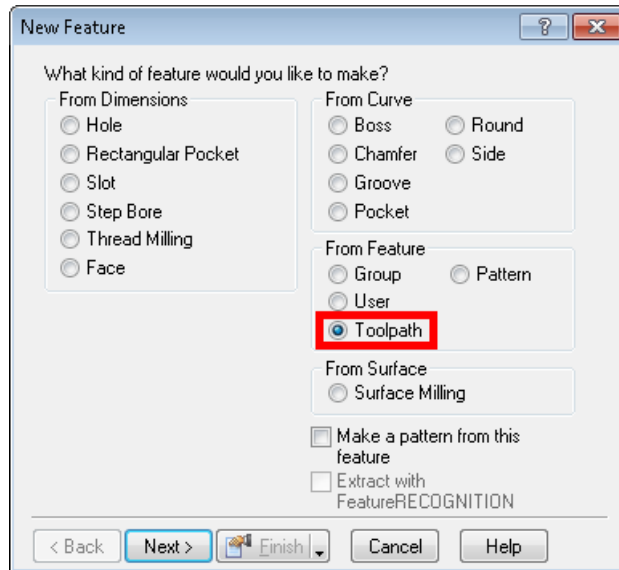
## Program stop code example

To enter program stop code directly into the NC program:

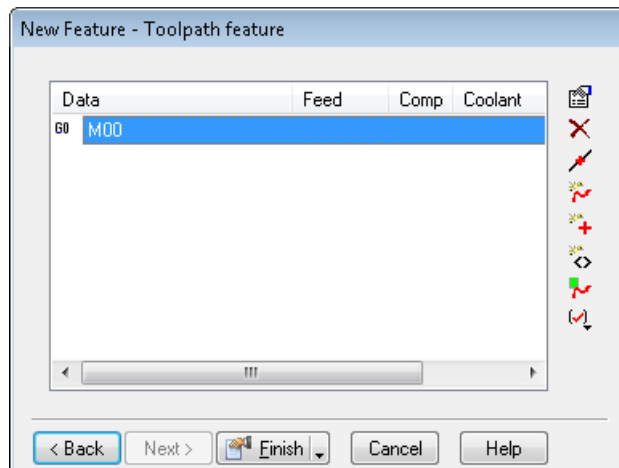
- 1 Click the **Features**  step in the **Steps** panel. The **New Feature** dialog opens.



- 2 In the **From Feature** section, select **Toolpath** and click **Next**. The **New Feature - Curve or Operation** page is displayed.



- 3 Select the **NC code text** option and enter **M00** (or **M01**).
- 4 Click **Next**. The **New Feature - Toolpath feature** page is displayed.



This page has the same toolpath editing buttons as the **Toolpaths** tab in the **Toolpath Properties** dialog.

- 5 Click **Finish**. The Toolpath feature is listed in the **Part View** and on the **Op List** tab.
- 6 Simulate the Toolpath feature. In this example, there are no toolpaths to see, but you must simulate the feature to produce the NC code.
- 7 View the NC code on the **NC Code** tab.



*If a Toolpath feature consists only of G-code, FeatureCAM does not display the **Tools** or **F/S** tabs in the **Toolpath Properties** dialog.*

---

## General: Soft errors

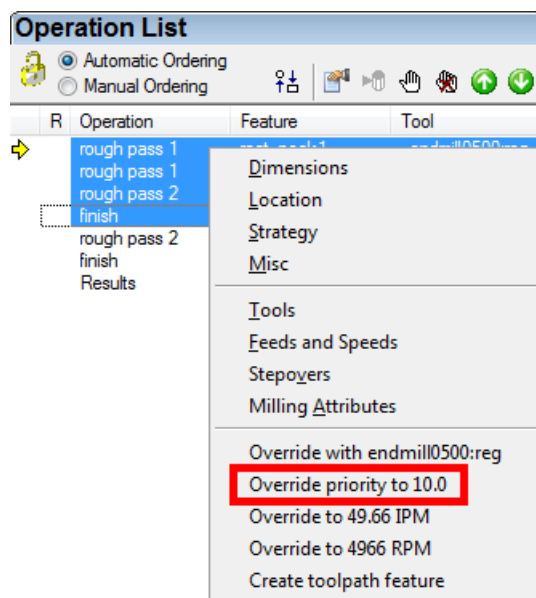
FeatureCAM 2012 R3 introduces *soft errors*. A soft error does not prevent you from running a simulation and generating NC code, but it does prevent you saving the NC code (the **File > Save NC** menu option is unavailable).

Soft errors are suffixed **SE**, for example **FS001SE** is the soft error for *Feed and speed table not found*.

---

## User interface: Set priority for multiple operations on the Op List

In FeatureCAM 2012 R3, you can override the priority for multiple operations in the **Operation List**. To do this, **SHIFT+select** or **CTRL+select** multiple operations on the **Op List** tab, right-click and select **Override priority to...** in the context menu:



---

## Paste special: Copy attributes from one feature to many

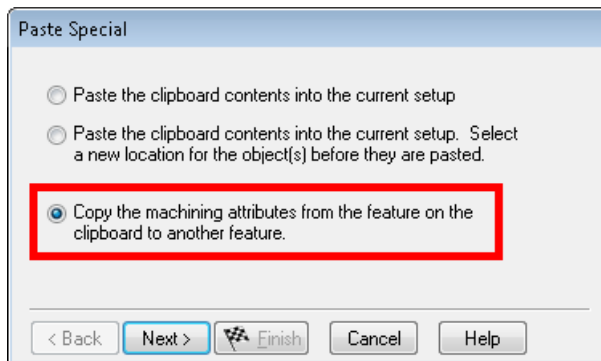
In FeatureCAM 2012 R3, you can copy the attributes from one feature to many features.

To do this:

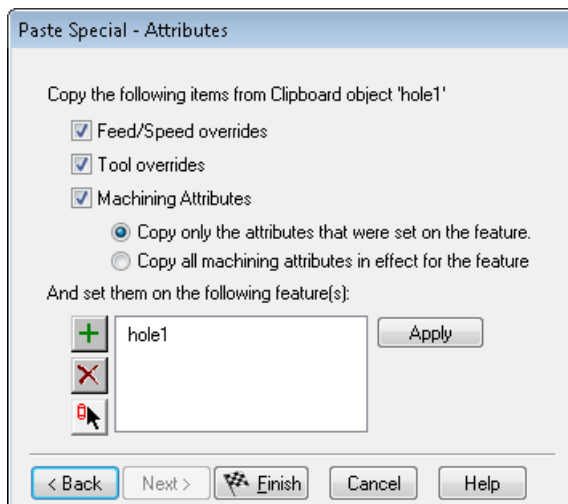
- 1 Select the feature whose attributes you want to copy.
- 2 Select **Edit > Copy** from the menu.
- 3 Select **Edit > Paste Special** from the menu.

The **Paste Special** wizard is displayed.

- 4 In the **Paste Special** wizard, select **Copy the machining attributes from the feature on the clipboard to another feature.**



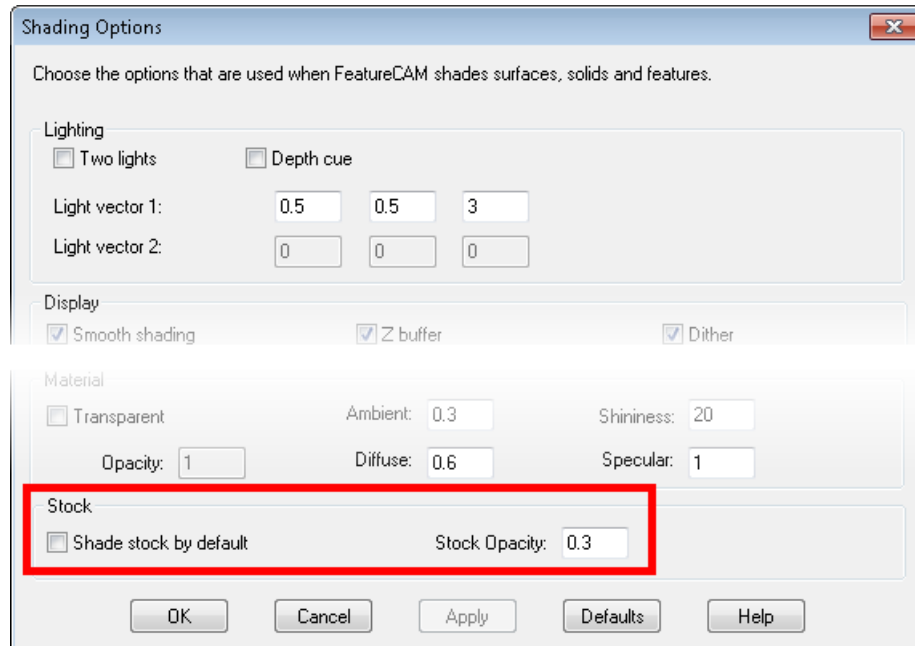
- 5 Click **Next**. The **Paste Special - Attributes** page is displayed:



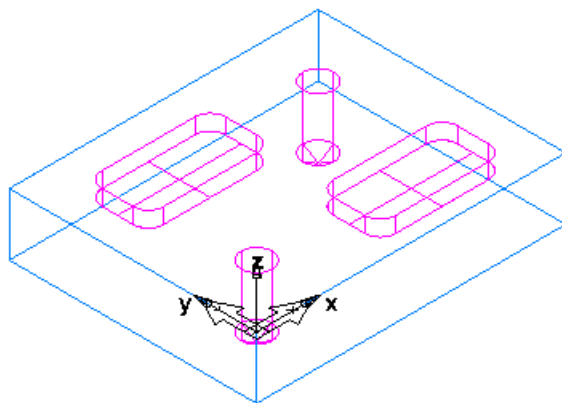
- 6 Select the types of attribute you want to copy from:
  - **Feed/Speed overrides**
  - **Tool overrides**
  - **Machining Attributes**
- 7 Select whether you want to **Copy only the attributes that were set on the feature** or **Copy all machining attributes in effect for the feature.**
- 8 Select the features you want to copy the attributes to.
- 9 Click **Finish**.

# Stock: Shaded and translucent

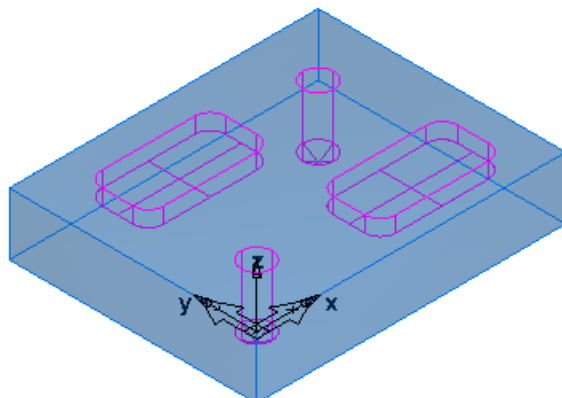
FeatureCAM 2012 R3 can display stock as shaded and translucent. To do this, select **Shade stock by default** in the **Shading Options** dialog and enter a **Stock Opacity** of between **0** and **1**.



This is an example part with **Shade stock by default** deselected:



And with **Shade stock by default** selected with a **Stock Opacity** of **0.3**:



---

## Import: New surface naming scheme groups surfaces

In FeatureCAM 2012 R3, imported surfaces are named according to which solid they belong to. For example, the surfaces imported from a solid named **solid1** are named **solid1\_1**, **solid1\_2**, **solid1\_3**, and so on in the **Part Tree**; and the surfaces from a solid named **solid2** are named **solid2\_1**, **solid2\_2**, **solid2\_3**, and so on.

This makes it easier to see which surfaces belong to which solid and to select all the surfaces from just one solid. This is useful if you want to stitch imported surfaces back into solids.

---

## Import: PTC Creo files supported

In FeatureCAM 2012 R3, you can import PTC Creo files.

---

## Tool mapping: 8-digit Diameter & Length offset registers

FeatureCAM 2012 R3 supports more machine types because you can enter 8 digits for the tool **Length offset register** and **Diameter offset register** in the **Tool Mapping** dialog:

The **Tool Mapping** dialog box is shown, used to map tools from a part to tool slots in a machine. The table below lists the tools and their parameters:

+	Name	Diameter	Length	ID	Crib	Time	Dist	Holes
*1	center_M3150-1250	*12345678	*12345678	1				
2	TD_05000_112:J	2	2	2				
3	endmill0500:reg	3	3	3				
4	endmillM1000:reg	4	4	4				

The detailed view for the tool **center\_M3150-1250** is shown below. The **Diameter offset register** and **Length offset register** fields are highlighted with a red box, showing the 8-digit values **12345678** and **12345678** respectively. The **Tool number** is **1** and the **Tool ID** is **1**.

Slots for center\_M3150-1250

Tool number: 1

Diameter offset register: 12345678

Length offset register: 12345678

Tool ID: 1


Tool life for center\_M3150-1250

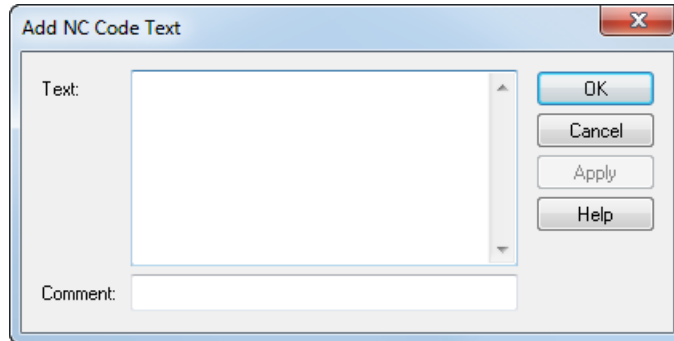
Enable/disable tool change to new tool after specified time, number of operations, etc



Buttons: Set, Save in Crib, Clear in Crib, Set All, Reset All, Show All Tools Saved in Crib, Show empty tools slots.

---

## Toolpath edit: Insert multiple lines of G-code

In FeatureCAM 2012 R3, you can enter multiple lines of G-code. Click the **Add NC code text**  button on the **Toolpaths** tab to open the **Add NC Code Text** dialog:

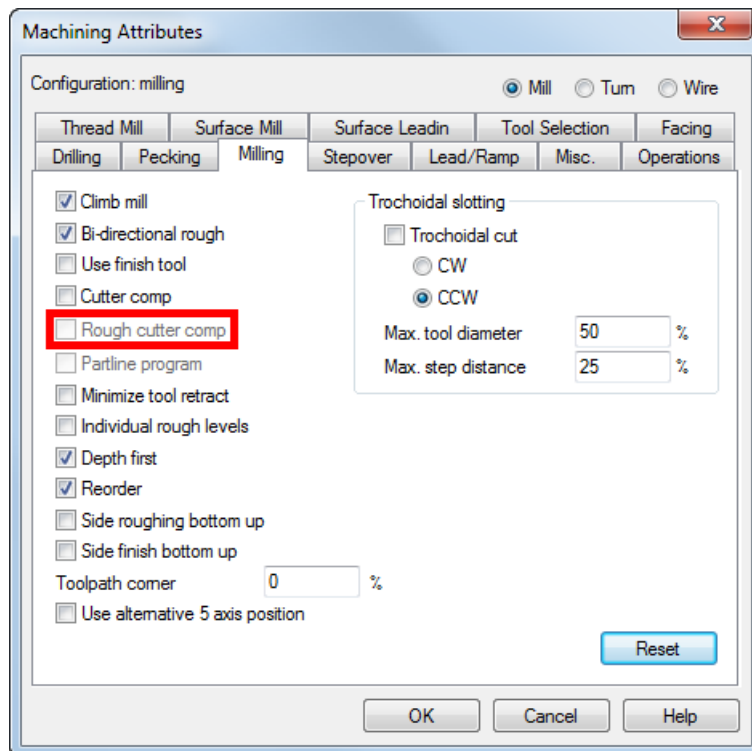


Single lines of NC code have the  icon on the **Toolpaths** tab and multiple lines of code have the  icon.

---

## 2.5D: Rough cutter comp. machining attribute

In FeatureCAM 2012 R3, there is a new **Rough cutter comp** option on the **Milling** tab of **Machining Attributes**:

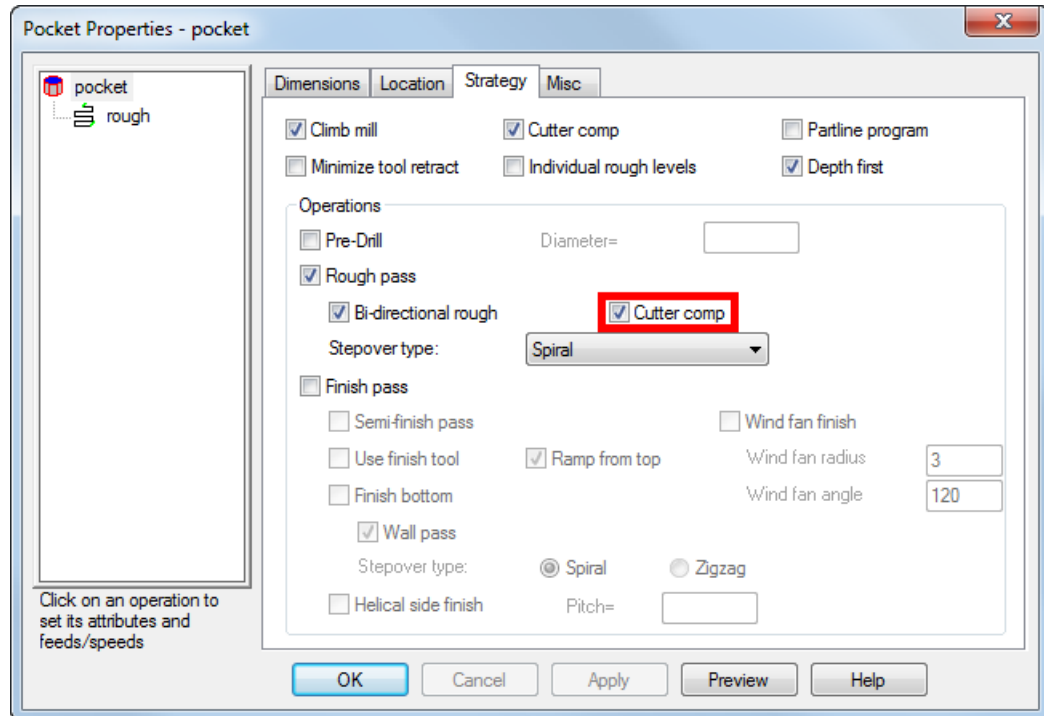


Select the **Rough cutter comp** option to enable cutter compensation for the rough pass of all milled features in the current document by default.



*The **Rough cutter comp** option is not available until you select the **Cutter comp** option.*

You can disable it at feature-level on the **Strategy** tab:



## 2.5D: New technology (NT) toolpaths for Side features

In FeatureCAM 2012 R3, the NT toolpaths that were introduced in FeatureCAM 2012 R2 for Pocket and Boss features, are also available for Side features.

The three new roughing Stepover types are:

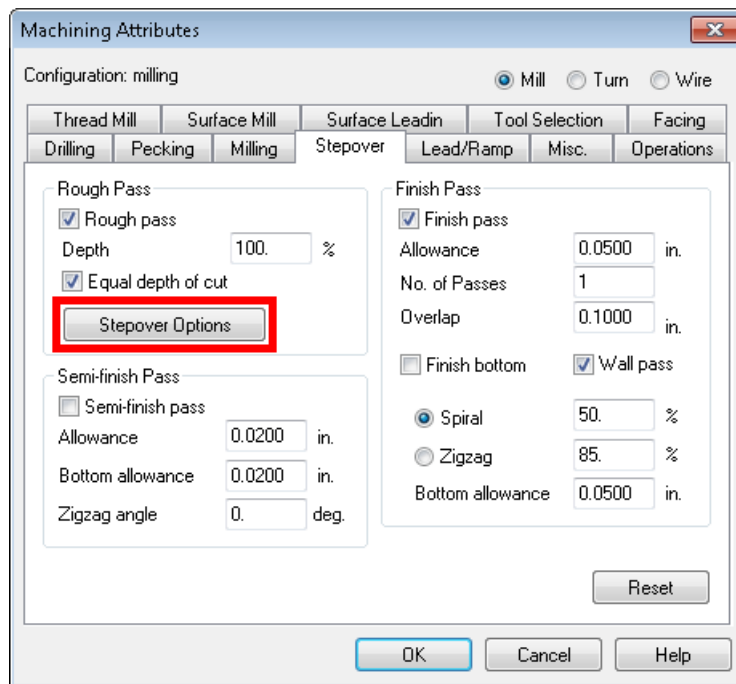
- **NT Spiral** - can use stepovers larger than 50%
- **NT Zigzag** - uses an angle that is calculated automatically to cut the longest toolpaths



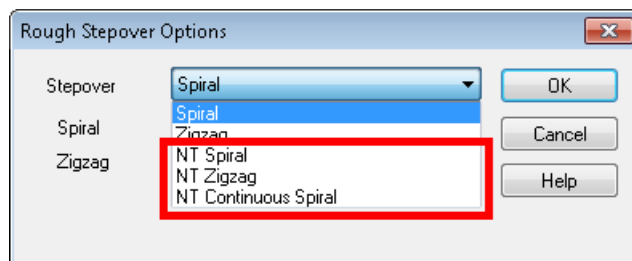
*You can use NT Zigzag toolpaths only with features made from closed curves. If they are not closed and you select this option, FeatureCAM uses NT Spiral.*

- **NT Continuous Spiral** - eliminates nearly all stepovers

Set the default type in **Machining Attributes** on the **Stepover** tab:

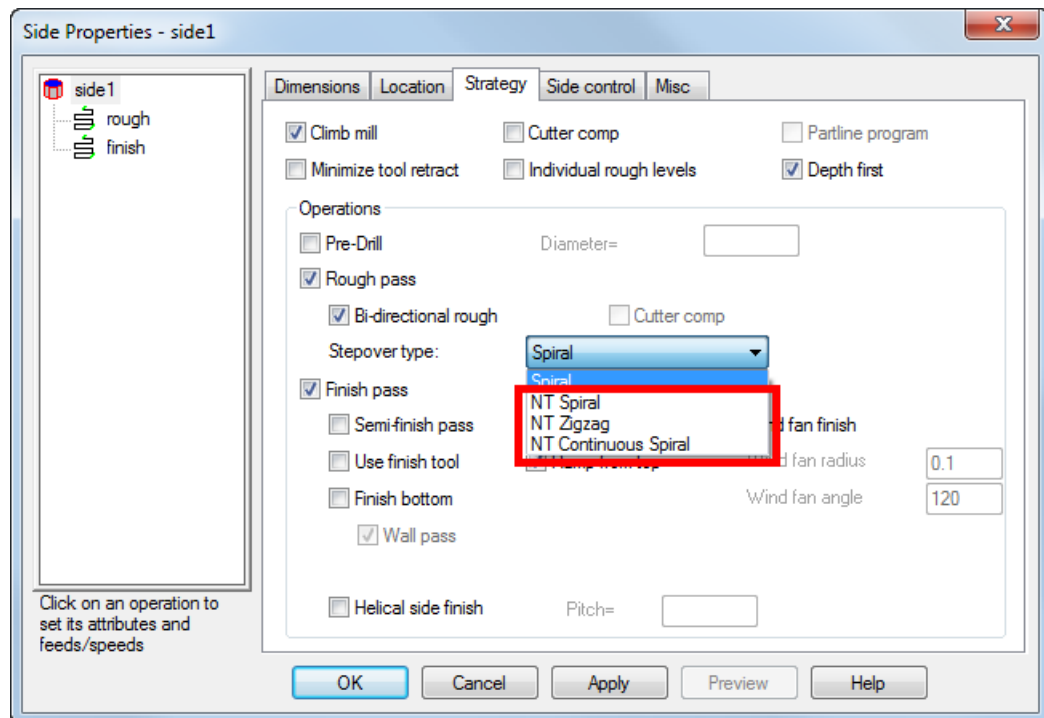


Click the new **Stepover Options** button to open the new **Rough Stepover Options** dialog:

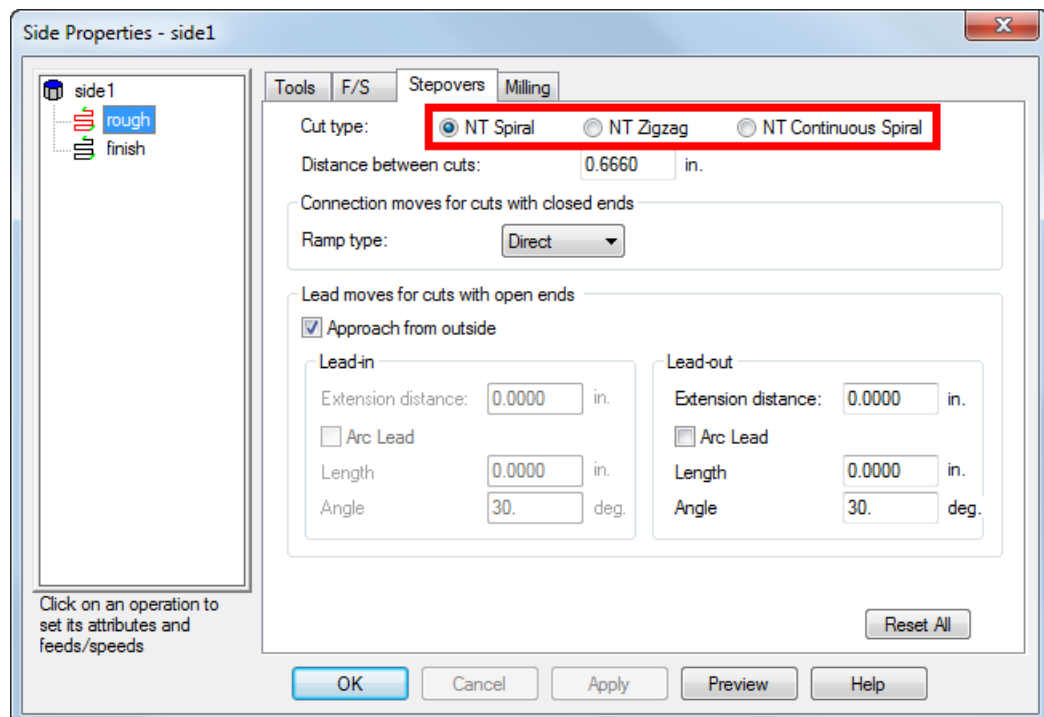


*A **Stepover** type of **Zigzag** is not available for Side features.*

At feature level, set the **Stepover type** in the new menu on the **Strategy** tab of the feature's **Properties** dialog:



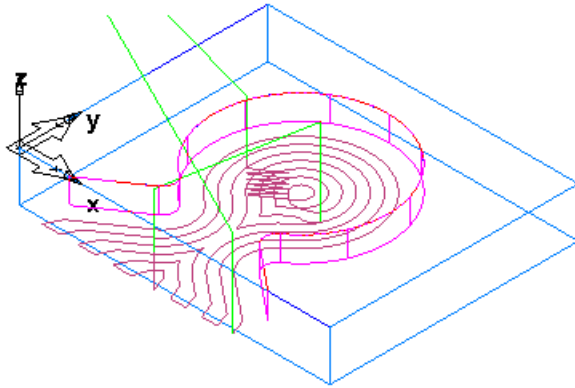
You can override this at operation level on the **Stepovers** tab:



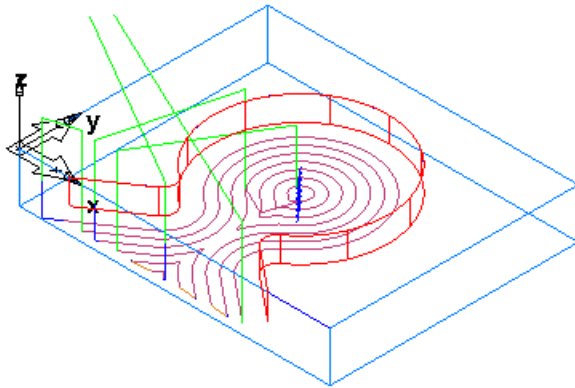
If you are using **Individual rough levels**, you can set the **Cut type** for each individual rough pass.

### NT Side example

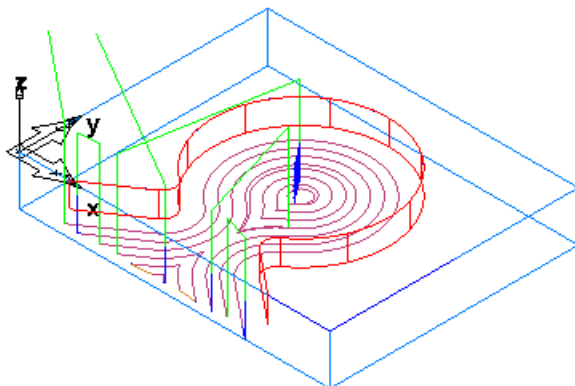
This example shows a centerline simulation of a Side feature using the traditional **Spiral** stepover type:



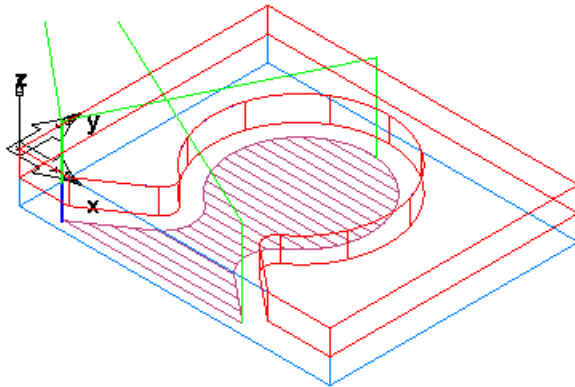
Using **NT Spiral** gives a similar result, but the toolpaths and stepovers are smoother:



Using **NT Continuous Spiral**, the closed part of the toolpath at the center of the feature (the teardrop shape) uses a continuous spiral and the rest of the toolpath uses a non-continuous spiral:



**NT Zigzag** applies only to closed curves, so with this example, the Side feature is made from a closed curve that goes around the edge of the Stock:

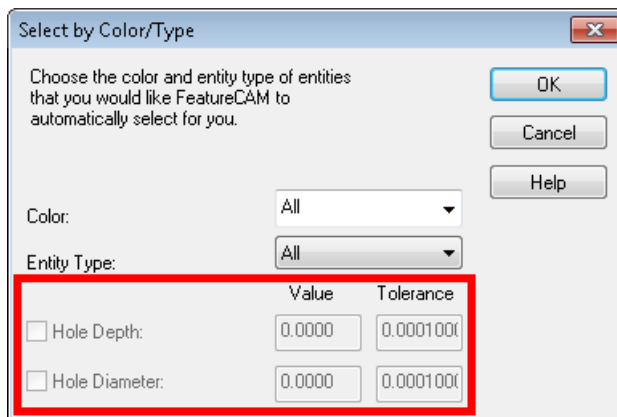


*If you select a **Stepover Type** of **NT Zigzag** for an open curve, FeatureCAM uses **NT Spiral**.*

## 2.5D: Select Holes by depth or diameter

In FeatureCAM 2012 R3, you can select all Holes in the part with the same depth and/or diameter.

The **Select by Color/Type** dialog has two new options, **Hole Depth** and **Hole Diameter**. Select **Edit > Select by Color/Type** to open the dialog.



**Hole Depth** — To select all Holes of a particular depth, first select a type of hole in the **Entity Type** menu, then enter the Hole Depth **Value**, and a **Tolerance** for how close to the **Value** the depth must be.

**Hole Diameter** — To select all Holes of a particular depth, first select a type of hole in the **Entity Type** menu, then enter the Hole Diameter **Value** and a **Tolerance** for how close to the **Value** the diameter must be.

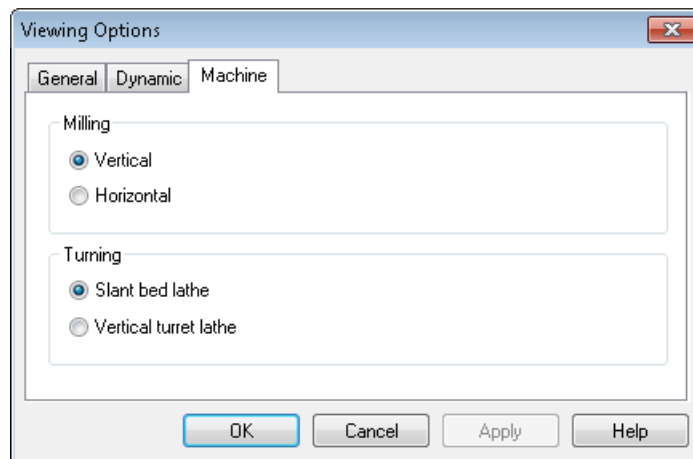


*Select a Hole on your part before opening the dialog to pre-populate the fields.*

---

## Viewing modes: Horizontal milling and vertical lathe

In FeatureCAM 2012 R3, there is a new **Machine** tab in the **Viewing Options** dialog:




**Milling** — Select the type of milling machine from **Vertical** (the default and previous behavior) and **Horizontal**, to improve viewing during machine simulation.

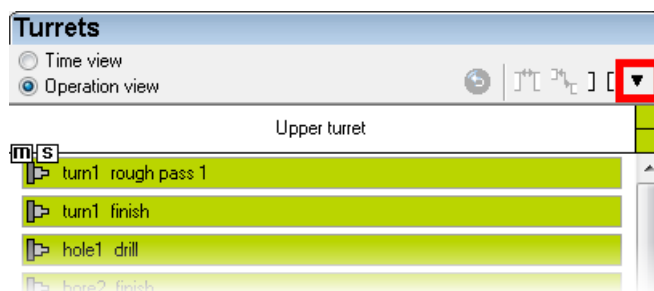
**Turning** — Select the type of turning machine from **Slant bed lathe** (the default and previous behavior) and **Vertical turret lathe**, to improve viewing during machine simulation.

---

## MTT: Better user interface

In FeatureCAM 2012 R3, it is easier to access options that were previously available only on the right-click context menu.

Click the new  button on the **Turrets** tab to access the options **Reset all opers to default turret** and **Remove all oper sync points**:



---

## Tool database: FeatureCAM add-in to import TDM tools

A new FeatureCAM add-in is available that can be run independently of the FeatureCAM MS-Access network database. It uses a tab-delimited file to read the FeatureCAM-TDM (Tool Data Management) tool mapping, and uses that to import tools from TDM into the FeatureCAM network database. These are a few benefits:

- The script works with any version of FeatureCAM, as long as the map is correct for that version.
- The FeatureCAM-TDM tool map and import script have been separated, making the script tool-agnostic. Also you don't have to deal with SQL and other technical details which can be frustrating.
- The script creates a log of all the import failures.
- The script is a FeatureCAM add-in, making it easy to run and automate if necessary.

---

## API: Help

All new documentation for FeatureCAM's API is now available from the FeatureCAM help file, making writing FeatureCAM addins easier than ever before. Select **Help > FeatureCAM API Help** from the menu to access it.

# What's New in FeatureCAM 2012 R2

FeatureCAM 2012 R2 offers all of the original features of FeatureCAM 2012, but with numerous improvements. The most significant improvements are detailed in the following topics.

---

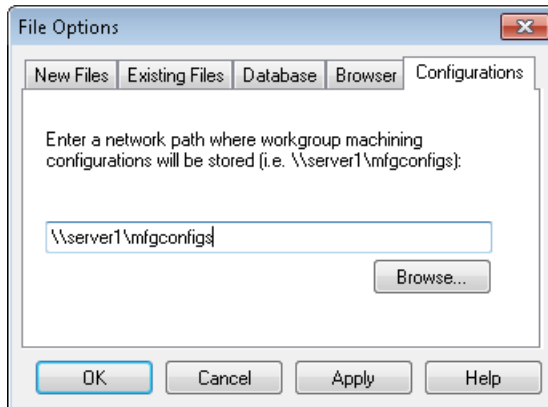
## Network database: Shared machining configurations

In FeatureCAM 2012 R2, you can share machining configurations across a network.

There are now three types of configuration:

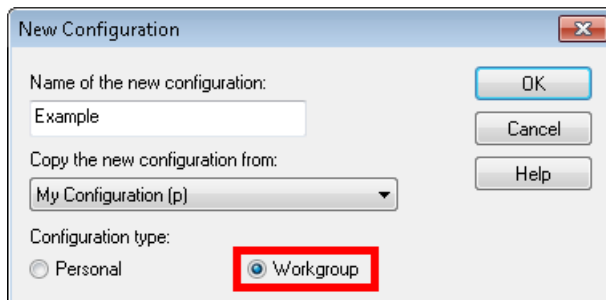
- 📁 **File (f)** - Configurations for files that are currently open.
- 👤 **Personal (p)** - Configurations that are independent of a particular file and are stored only on your PC.
- 👥 **Workgroup (w)** - Configurations that are independent of a particular file and are stored on a network for sharing with colleagues.

To create a **Workgroup** configuration, first enter a valid network path in the **File Options** dialog on the **Configurations** tab:

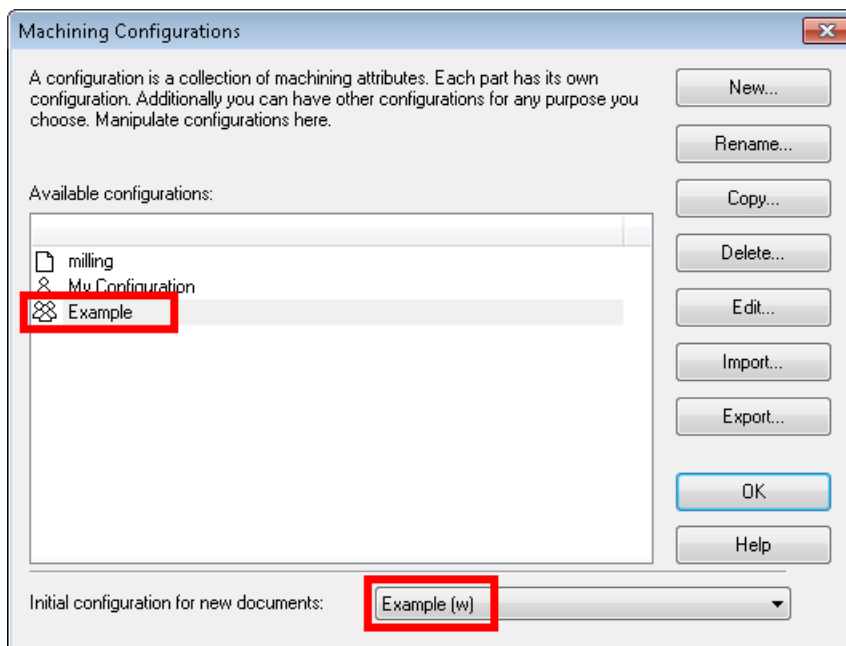


In the **Machining Configurations** dialog, click **New** to open the **New Configuration** dialog

You can now set a **Configuration type** of **Workgroup**:



After you click **OK**, the new configuration is shown in the list of available configurations, with a group icon and is followed by **(w)** in menu items:



You can set the new Workgroup configuration as the **Initial configuration for new documents**.

---

## Import improvements

FeatureCAM 2012 R2 can read Pro/E Family Tables, which are multiple configurations of a given part.

SolidEdge ST4 is now supported.

---

## User interface: Copy and paste from PowerSHAPE

You can now copy and paste solids and surfaces from PowerSHAPE into FeatureCAM. Surfaces, curves, lines, and arcs are supported.

You can also copy and paste solids from FeatureCAM into PowerSHAPE.

---

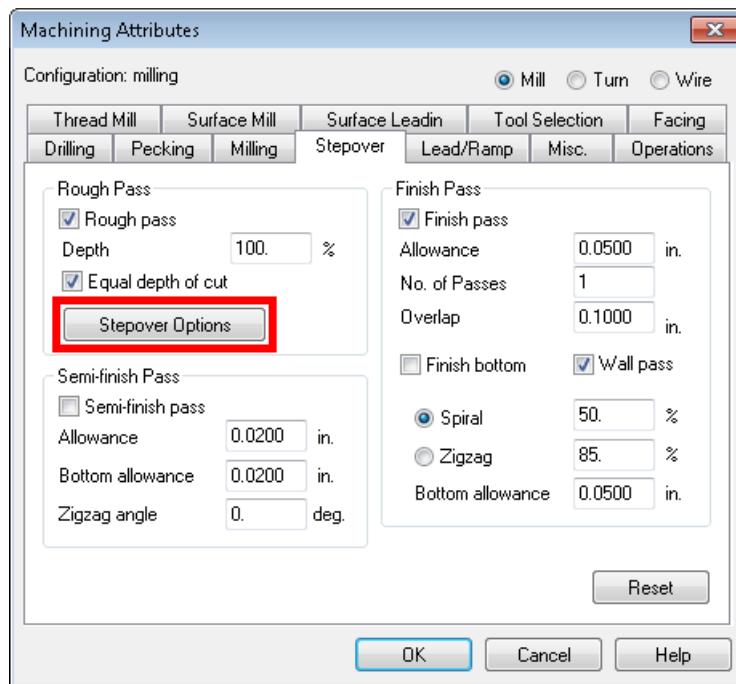
## 2.5D: New technology (NT) toolpaths

In FeatureCAM 2012 R2, there are three new roughing Stepover types:

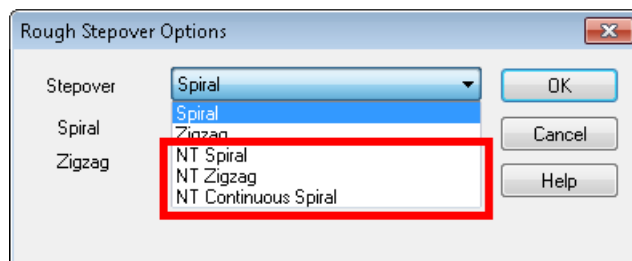
- **NT Spiral** - can use stepovers larger than 50%
- **NT Zigzag** - uses an angle that is calculated automatically to cut the longest toolpaths
- **NT Continuous Spiral** - eliminates nearly all stepovers

The **NT** (new technology) toolpaths support Pocket, Rectangular Pocket, and Boss features.

Set the default type in **Machining Attributes** on the **Stepover** tab:

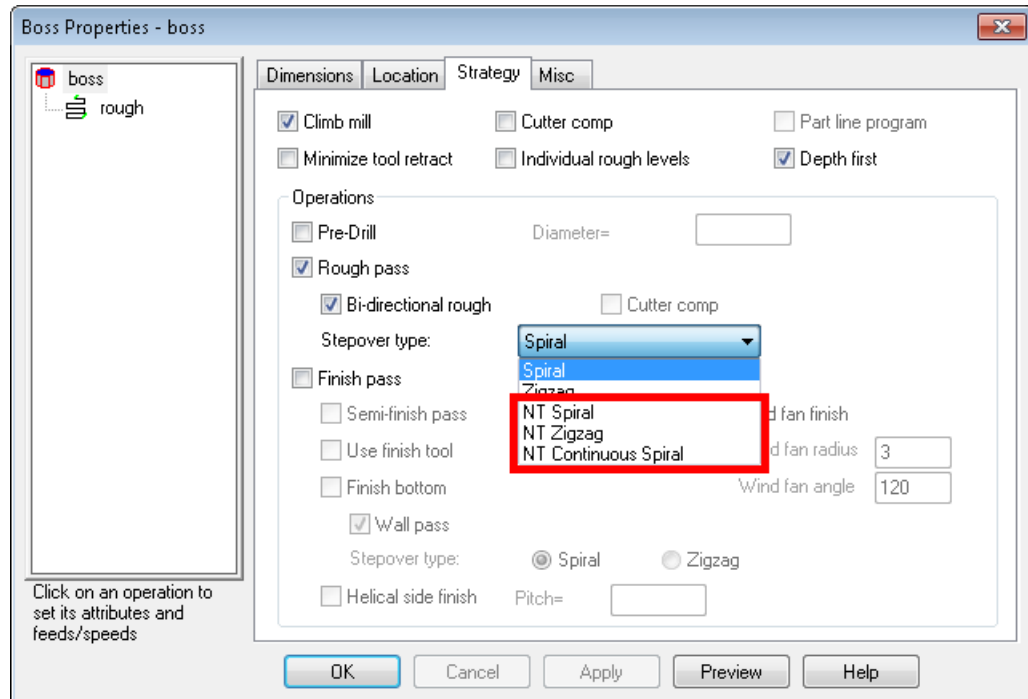


Click the new **Stepover Options** button to open the new **Rough Stepover Options** dialog:

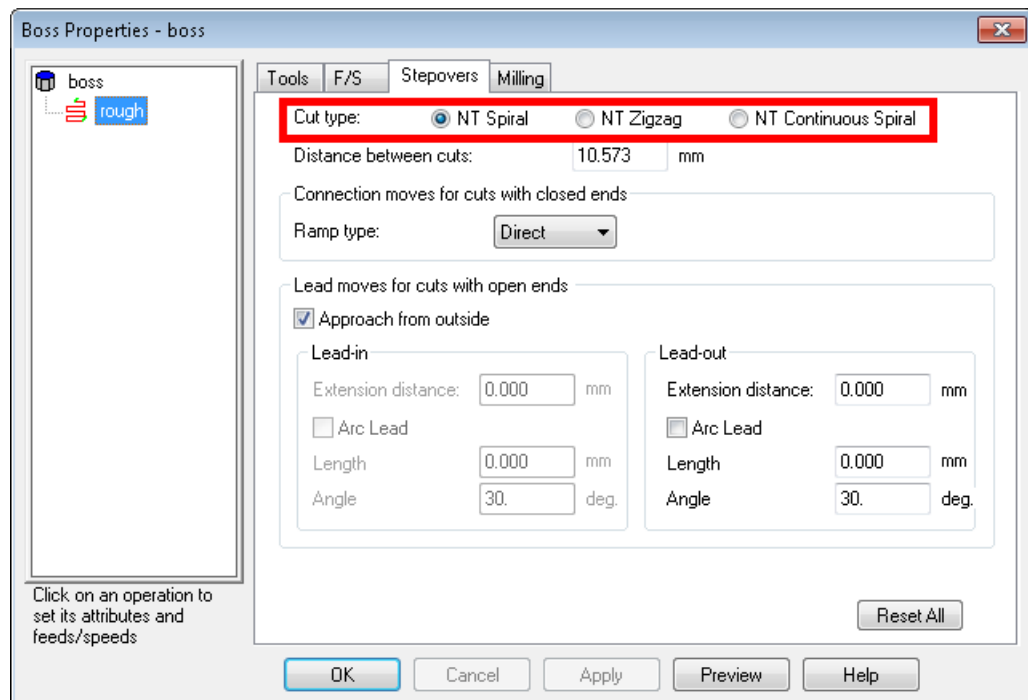


The **NT** toolpaths are available in the **Stepover** menu along with the traditional **Spiral** and **Zigzag** toolpaths.

At feature level, set the **Stepover type** in the new menu on the **Strategy** tab of the feature's **Properties** dialog:



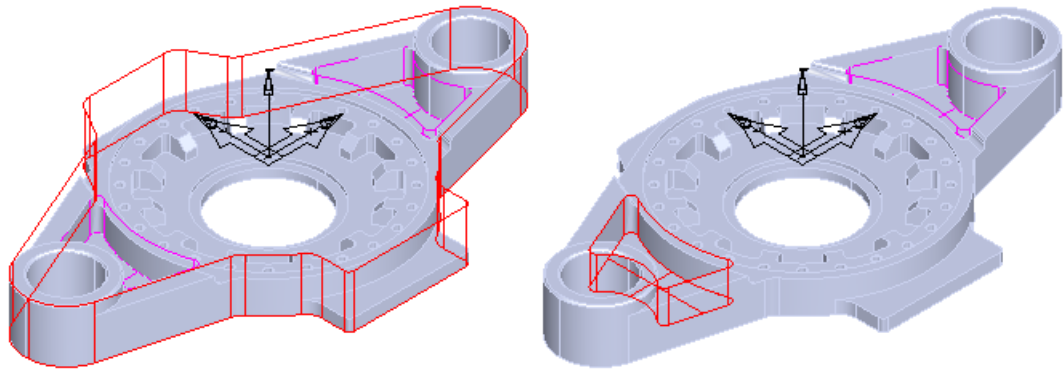
You can override this at operation level on the **Stepovers** tab:



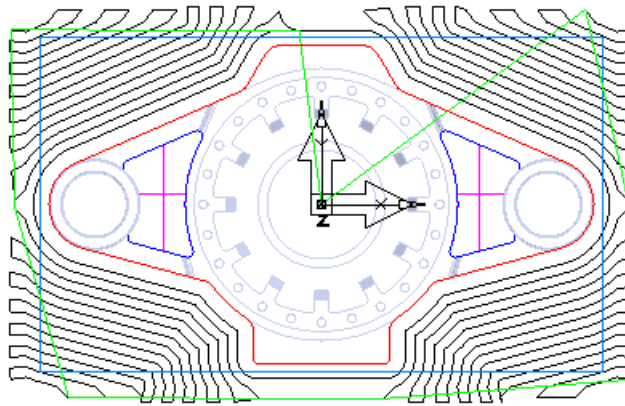
If you are using **Individual rough levels**, you can set the **Cut type** for each individual rough pass.

### NT Spiral example

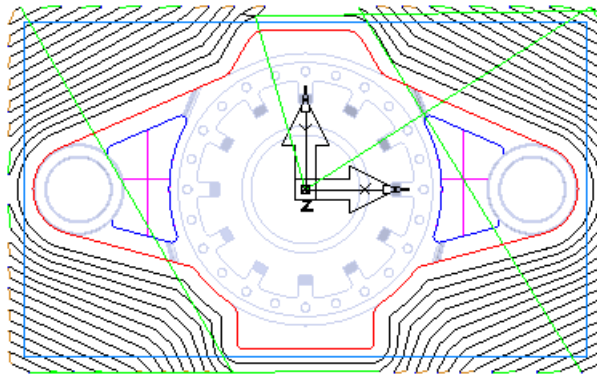
This example part includes a Boss feature and a Pocket feature, outlined in red:



This is a centerline simulation of the Boss feature using the traditional **Stepover type** of **Spiral**:

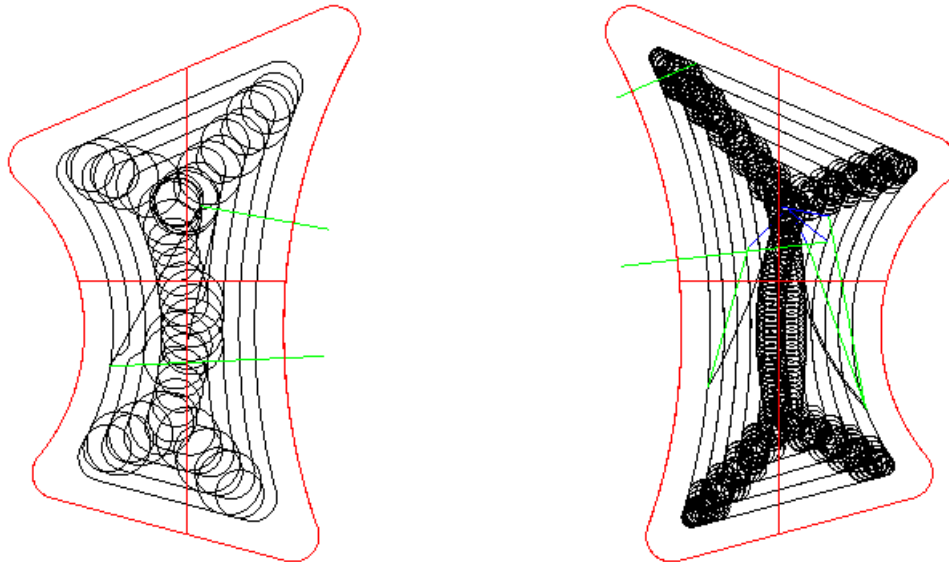


This is the same feature cut using the new technology **Stepover type** of **NT Spiral**:

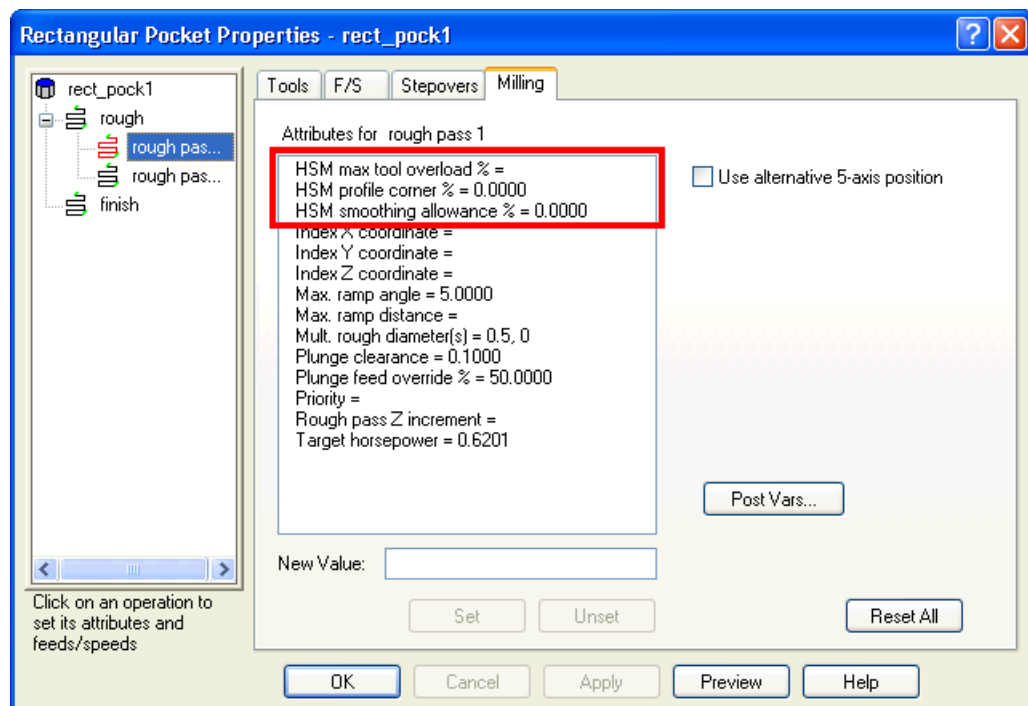


This is a centerline simulation of the left Pocket feature using the traditional **Stepover type** of **Spiral**:

This is the right Pocket cut using the new technology **Stepover type** of **NT Spiral**:



There are three new attributes available on the roughing operation **Milling** tab of the Feature Properties dialog:



**HSM max tool overload %** - instigates trochoidal machining. As the tool gets towards overload, FeatureCAM automatically puts in a trochoidal path to take away the overload..

**HSM profile corner %** - This allows the arc fitting of profile to avoid sharp changes in direction in internal corners.

**HSM smoothing allowance %** - replaces the standard offset with a smoother one that can achieve higher feed rates.

- Sharp rounded corners replace sharp corners.

- The stepover is changed from a fixed to a variable distance. The percentage defines the maximum deviation from the specified stepover. The maximum percentage is **40%** of stepover. So, if you have a 10 mm stepover the maximum deviation from the original to the smoothed offset is 4 mm.

This replaces the standard offset with a smoother one that can achieve higher feed rates.

- Sharp rounded corners replace sharp corners.

The stepover is changed from a fixed to a variable distance. The percentage defines the maximum deviation from the specified stepover. The maximum percentage is **40%** of stepover. So, if you have a 10 mm stepover the maximum deviation from the original to the smoothed offset is 4 mm.

## Differences between NT and traditional toolpaths

### Pre-drill differences

There are some differences with how the **Pre-drill** operation works with the NT toolpaths compared to the traditional toolpaths.

- You can use a **Pre-drill** operation with the **NT Zigzag** toolpath. It is not available for the traditional **Zigzag** toolpath.
- With the NT toolpaths, zigzag ramping is automatically disabled when you use a **Pre-drill** operation. This is not supported for the original **Spiral** toolpath and you must set the **Max. ramp angle** to **0** to disable zigzag ramping when using a **Pre-drill** operation.
- The **Pre-drill** operation for the NT toolpaths includes the tops of multi-height islands. This is not supported for the original toolpaths.
- The traditional **Spiral** toolpath supports manual plunge points for the pre-drill locations using the **Plunge point(s)** attribute. The NT toolpaths support only automatic locations.

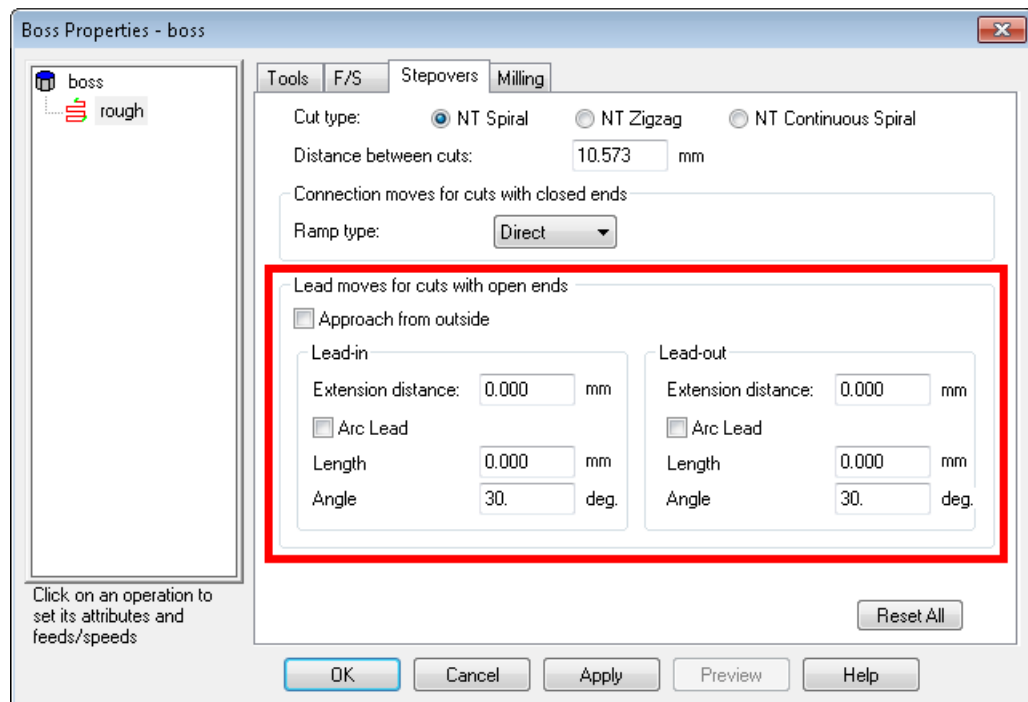
### Stepover differences

There are some differences in the attributes available for NT toolpaths on the feature **Stepover** tab:

There are two **Ramp type** options for the NT toolpaths:

- **Smooth** - similar to the traditional toolpath option **S-type**
- **Direct** - similar to the traditional toolpath option **Line**.

You can specify separate **Lead-in** and **Lead-out** options for NT toolpaths:



The NT toolpaths have a new attribute **Approach from outside**. This option automatically adds a very small lead-in whenever the tool is plunging off the stock. When you select this option, the lead-in options are disabled. Deselect it to manually specify the lead-in as a line, arc, and/or toolpath extension.



*The **Angle** attribute for NT toolpaths supports only positive angles. The direction relative to the toolpath is automatically determined.*

### Other differences

On the **Milling** tab, NT toolpaths have a new attribute, **Stepover rapid distance** that controls when to retract and plunge on Boss stepovers. This is an absolute distance and replaces the **Min. rapid distance %** attribute used by the traditional toolpaths.

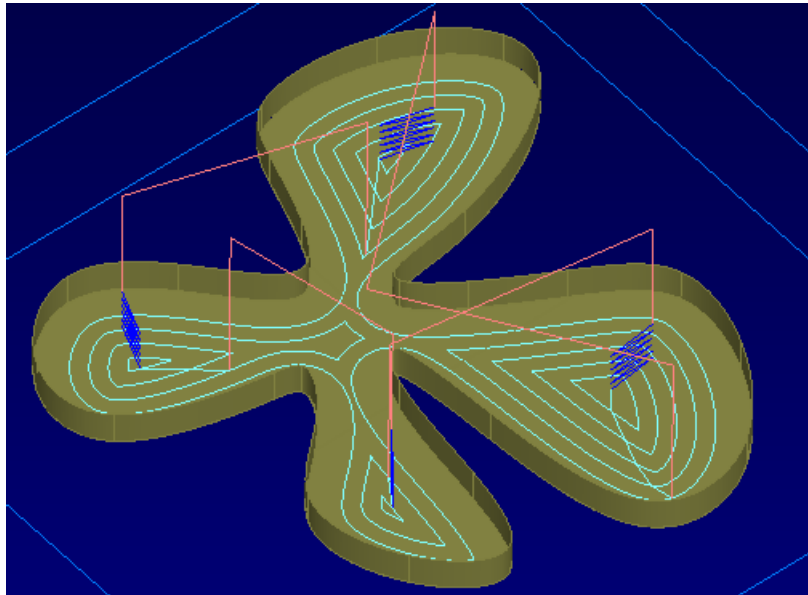
## Better Minimize tool retract with NT toolpaths

Using the **Minimize tool retract** options with the NT toolpaths gives better results than with the traditional toolpaths.

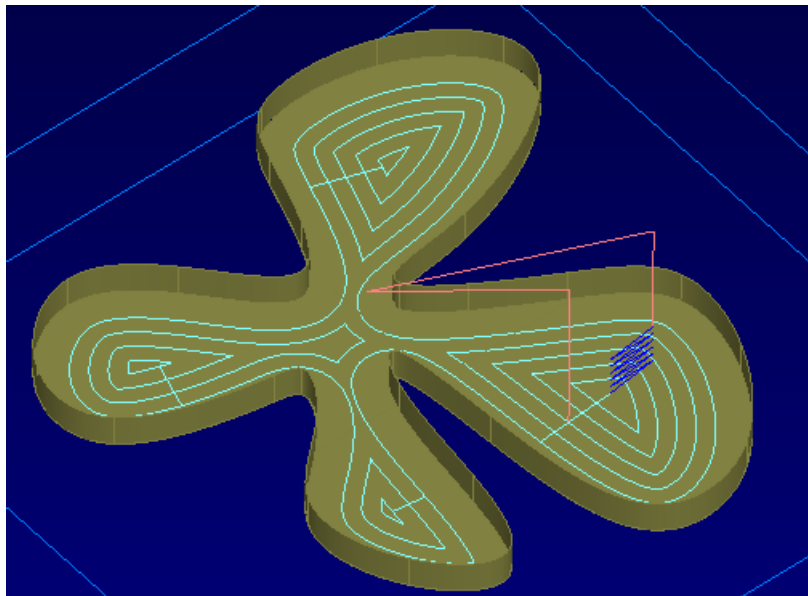
### Pocket feature

For a Pocket feature, the toolpath slot cuts following the offsets instead of just a straight line. There are fewer plunges than with traditional toolpaths.

Traditional toolpath example:



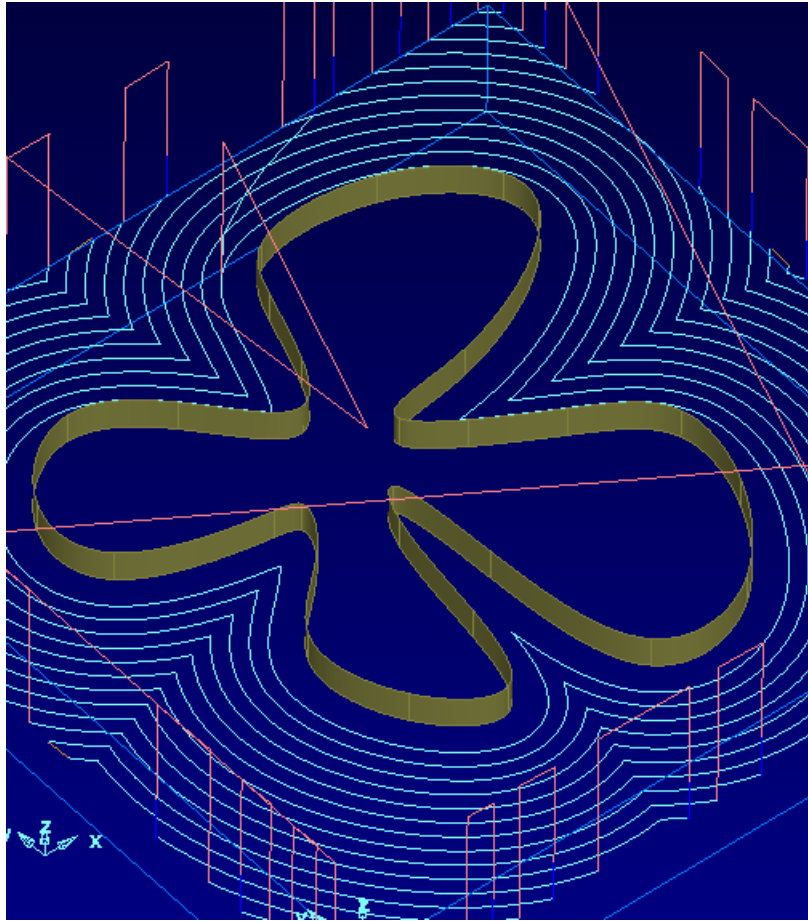
NT toolpath example:



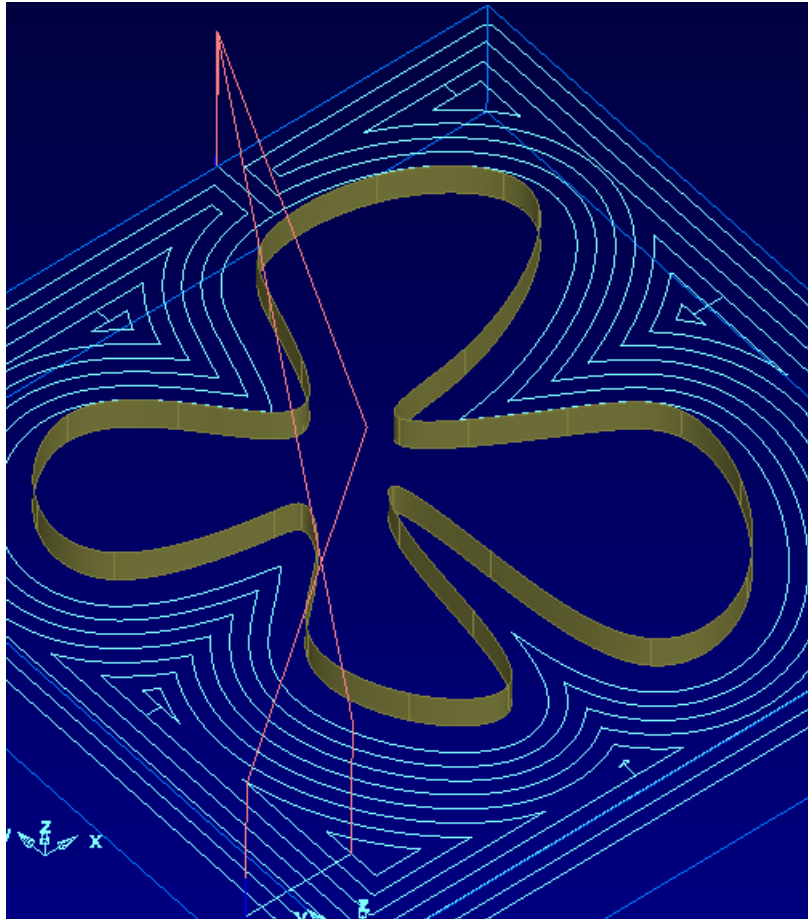
### **Boss feature**

For a Boss feature, the toolpath has a lot fewer retracts and plunges at the edge of the Stock.

Traditional toolpath example:

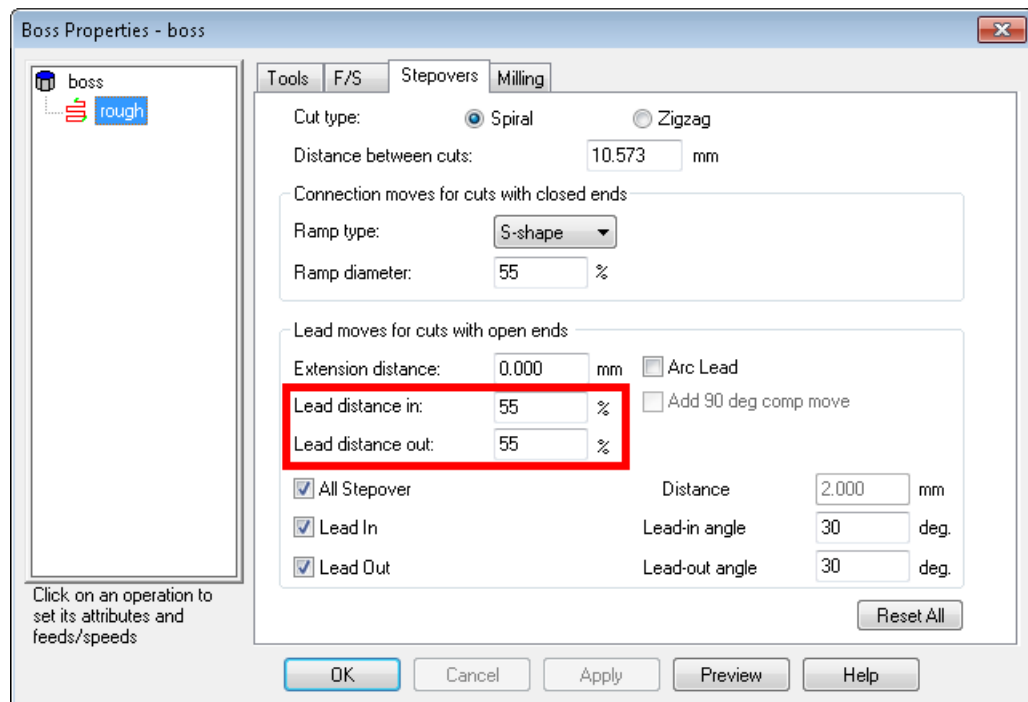


NT toolpath example:



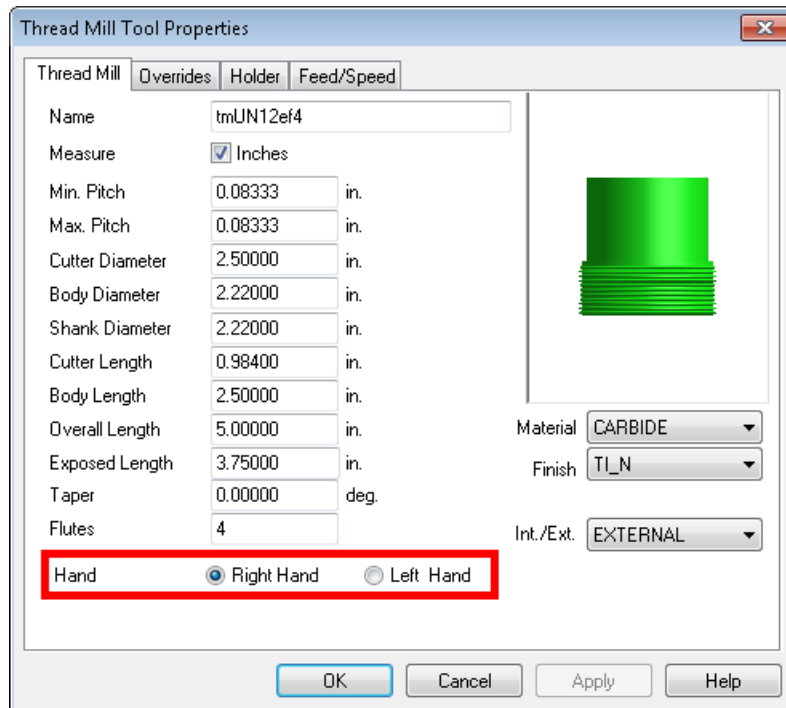
## 2.5D: Separate lead-in and lead-out distances

In FeatureCAM 2012 R2, you can set lead-in and lead-out distances separately:



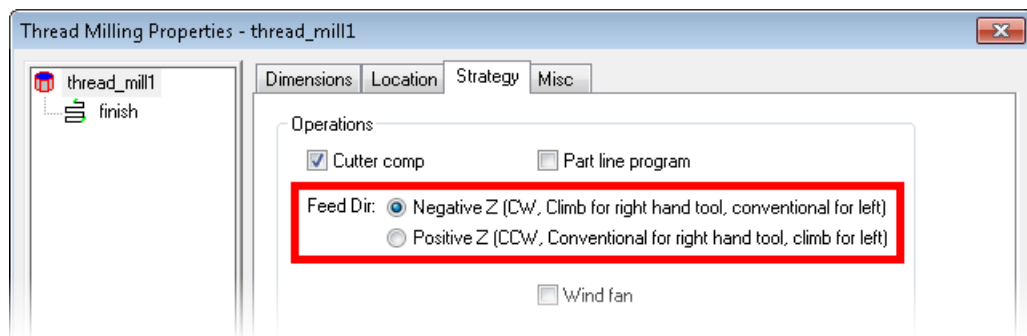
## 2.5D: Thread Mill tool hand

FeatureCAM 2012 R2 supports either right-hand or left-hand thread milling tools:



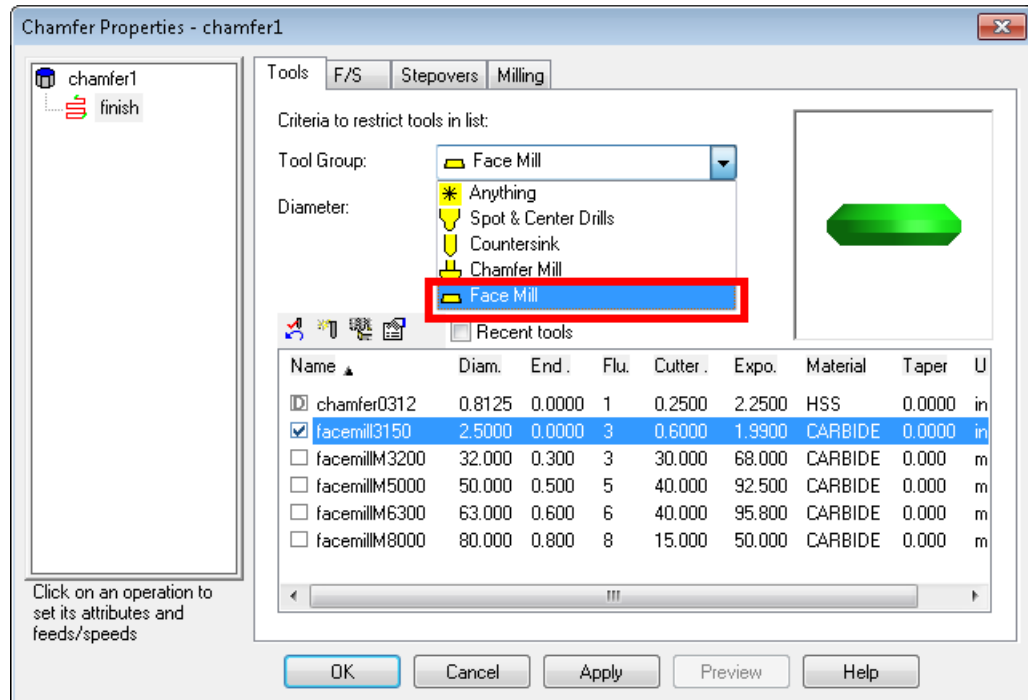
Select **Right Hand** or **Left Hand** for the **Hand** option.

The **Feed Dir** attribute on the **Strategy** tab has also been updated:



## 2.5D: Face Mill tools for Chamfers

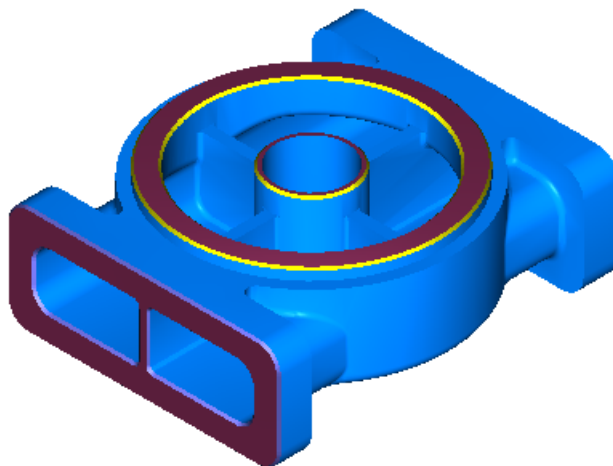
In FeatureCAM 2012 R2, you can use a chamfered Face Mill tool to cut a Chamfer feature. **Face Mill** tools are now available in the **Tool Group** menu on the **Tools** tab:



The benefit of this is that you can use fewer tools to machine a part.

### Example

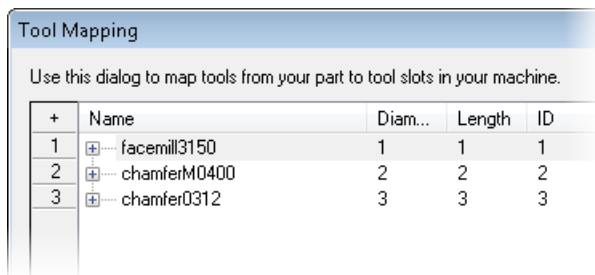
This example part has a Face and Chamfer features on the top and on the side:



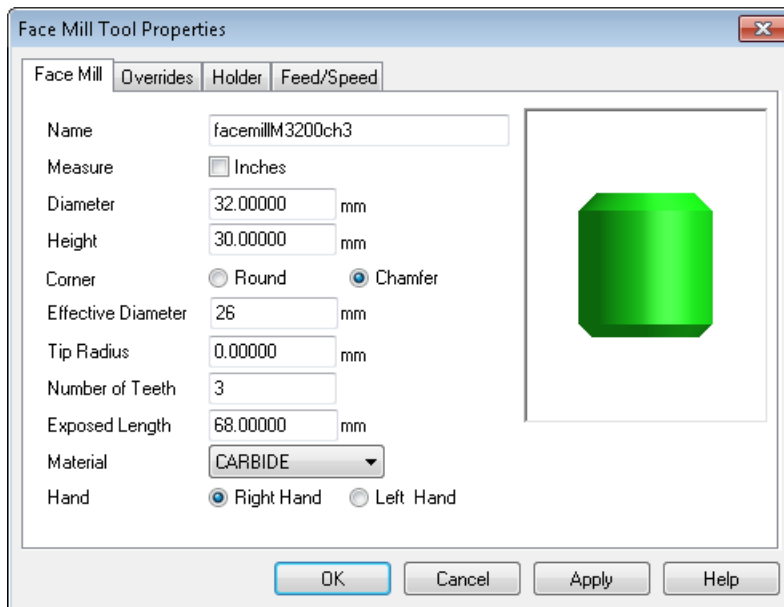
By default, this part uses three tools:

- a Face Mill to cut the top Face, then indexed around to cut the side Face.
- a Chamfer tool to cut the Chamfer around the top Face

- a Chamfer tool to cut the Chamfer around the side Face

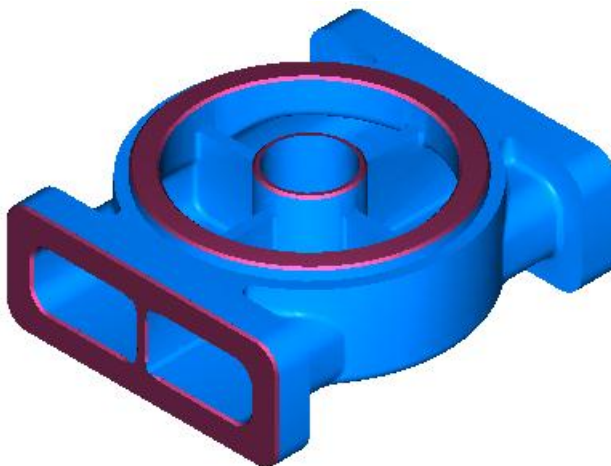


Instead of the default Face Mill, you can create a new Face Mill with a **Chamfer**, for example:

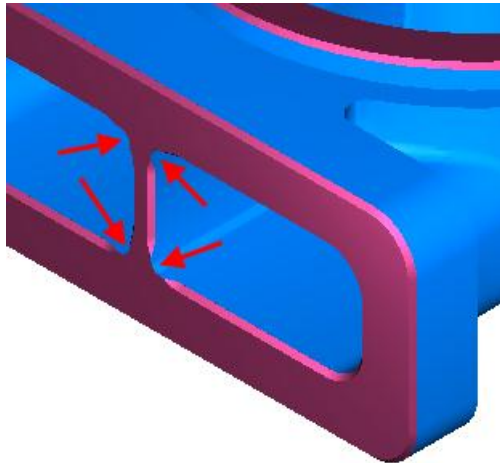


Set this new tool as the override for the Face feature, and then use it to cut the Chamfer features, by overriding the default tools in the Op List.

After simulating the part again, you can see that the new Face Mill tool cuts both the Face features and the Chamfers:

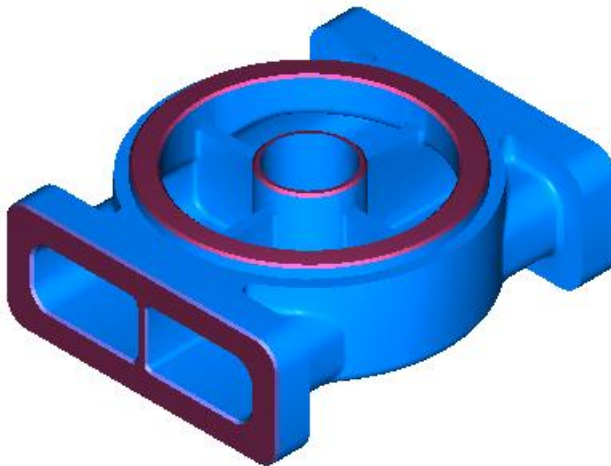


In this example, the chamfered Face Mill tool is too large to cut the Chamfer on the side of the part:



FeatureCAM give a warning and you can change the tool for this Chamfer back to the default tool.

Using a chamfered Face Mill tool, you can cut this particular part with two tools instead of three:



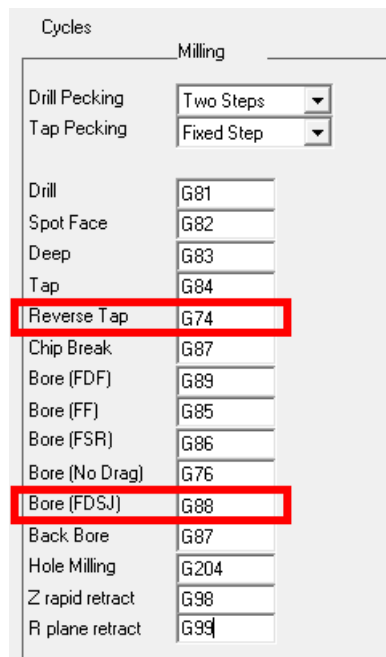
- a chamfered Face Mill to cut the top Face, the top Chamfer, then indexed around to cut the side Face
- a Chamfer tool to cut the Chamfer around the side Face

Tool Mapping				
Use this dialog to map tools from your part to tool slots in your machine.				
+	Name	Diam...	Length	ID
1	facemillM3200ch3	1	1	1
2	chamferM0400	2	2	2

## 2.5D: Hole cycle G74 and G88

FeatureCAM now supports the hole cycles G74 (left-handed tapping) and G88 (boring cycle).

Enter the G-codes in the **NC Codes** dialog in XBUILD:

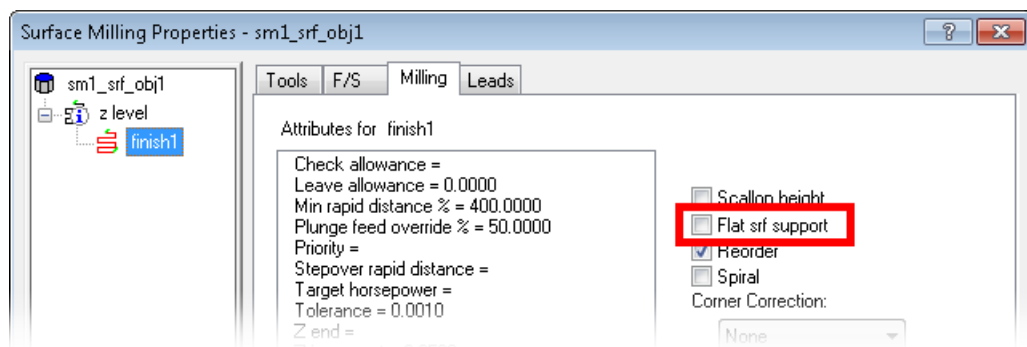


*If you do not enter a G-code for the **Reverse Tap** cycle, computed canned moves are performed. If you do not enter a G-code for the **Bore (FDSJ)** cycle, the **Bore (FF)** cycle is used.*

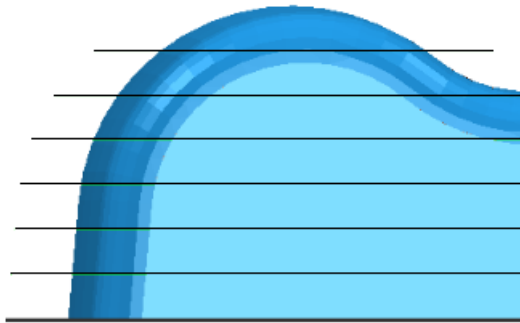
The new **Reverse Tap** cycle is available on the **Cycle** tab of a Hole feature for a Tap operation and the **Bore (FDSJ)** cycle is available for a Bore operation.

## 3D: Z-level finish: Flat surface support

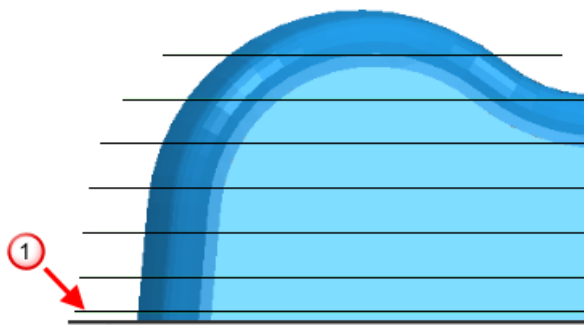
In FeatureCAM 2012 R2, you can now use flat surface support with Z-level finishing. This option inserts a final profile pass exactly at the base of a flat pocket. There is a new option, **Flat srf support**, in the **Surface Milling Properties** dialog on the **Milling** tab:



This example shows previous behavior, with **Flat srf support** disabled:

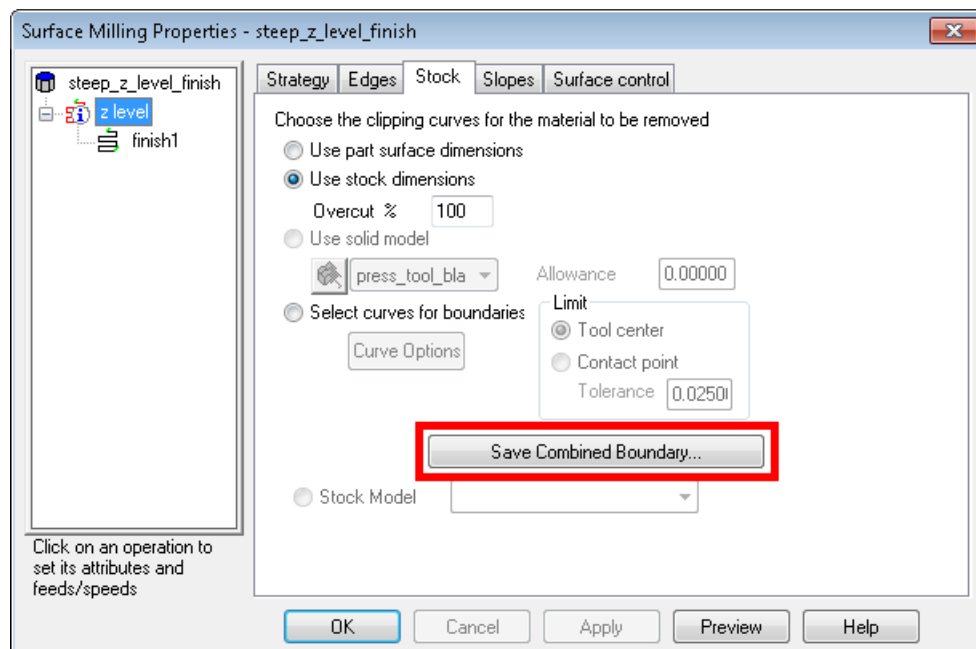


With **Flat Srf support** enabled, FeatureCAM adds a final profile pass, **1**:



## 3D: Save boundary curve

You can now save the boundary curve that has been calculated for an operation to use in subsequent operations. To save the boundary click the new **Save Combined Boundary** button on the **Stock** tab:



The boundary is saved as **Curves**, which are listed in the **Part View**.

To use some or all of these curves as the boundary of another operation:

- 1 Select the **Select curves for boundaries** option on the **Stock** tab of the second operation.
- 2 Click the **Curve Options** button to open the **Boundary Curve** dialog.
- 3 Click the **Boundaries** button to open the **Select Boundary Curves** dialog.
- 4 Select the box(es) next to the curve(s) you want to use.



*Hold down the Shift key to select multiple consecutive curves.*

- 5 Click **OK** to save and close the **Select Boundary Curves** dialog.
- 6 Click **OK** to save and close the **Boundary Curve** dialog.
- 7 Simulate the second operation.

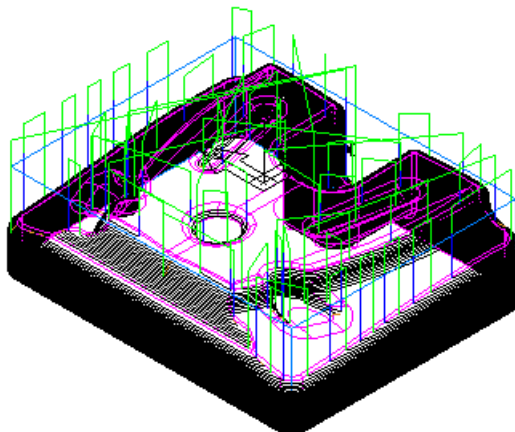
The simulation is quicker because FeatureCAM does not need to calculate the boundary again.

### Example

This example part uses a Z-level finish operation restricted to the steep areas only, using the **Vertical only** option on the **Slopes** tab, followed by a 3D spiral operation on the shallow areas.

- 1 Simulate the Z-level finish operation.

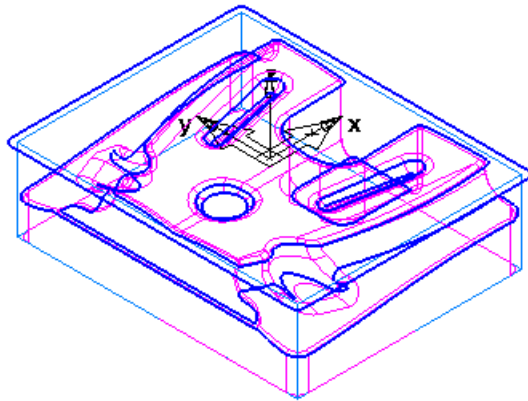
FeatureCAM takes a short time to calculate the boundary and you see a **Calculating boundary** message on the progress bar. When it is complete, the toolpath looks like this:



You can save this boundary to use in subsequent operations.

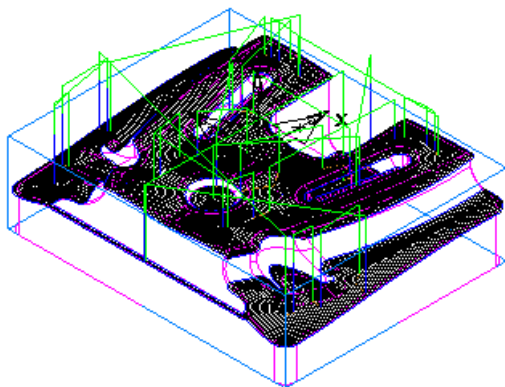
- 2 On the **Stock** tab for the Z-level operation, click the **Save Combined Boundary** button.

- 3 Click **Yes** to save the boundary. The boundary curves are displayed in dark blue:



For the next operation, we want to use a 3D spiral operation on the shallow areas of the part.

- 4 On the **Stock** tab of the 3D spiral operation, select the **Select curves for boundaries** option.
- 5 Click the **Curve Options** button to open the **Boundary Curve** dialog.
- 6 In the **Boundary Curve** dialog, click the **Boundaries** button to open the **Select Boundary Curve** dialog, which displays a list of the curves you saved.
- 7 Select the boxes next to the curves you want to use for the boundary. In this case we want to use all except the first curve, which runs around the top of the stock.
- 8 Click **OK** to save and close the **Select Boundary Curves** dialog.
- 9 Click **OK** to save and close the **Boundary Curve** dialog.
- 10 Simulate the 3D spiral operation. FeatureCAM does not need to calculate the boundary again and goes straight into calculating the toolpath:



---

## 5-axis: Turn/Mill B-axis alternative position

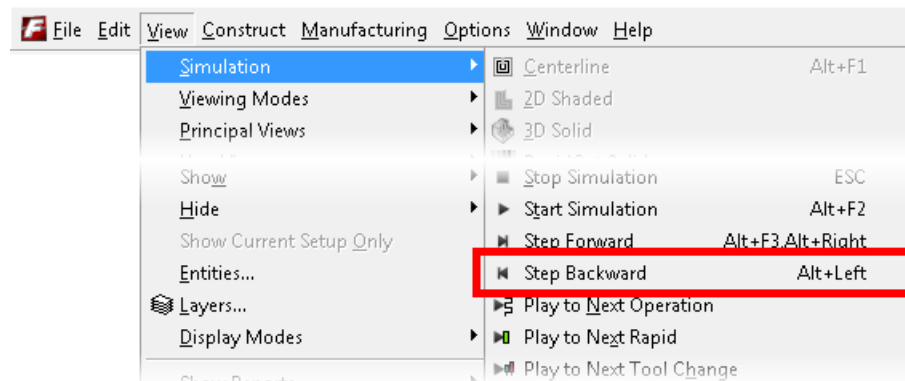
In FeatureCAM 2012 R2, the **Use alternative 5-axis position** attribute is available on the **Drilling** and **Milling** tab for 2.5D milling operations in Turn/Mill parts.

This means that you could cut 2.5D operations at, for example, B=-10 instead of the default +10 if the B-head has more clearance tilting up.

---

## Simulation: Step backward

You can now step backward through a simulation. Select **View > Simulation > Step Backward** from the menu to go back one step, or use the keyboard shortcut **Alt+Left**.



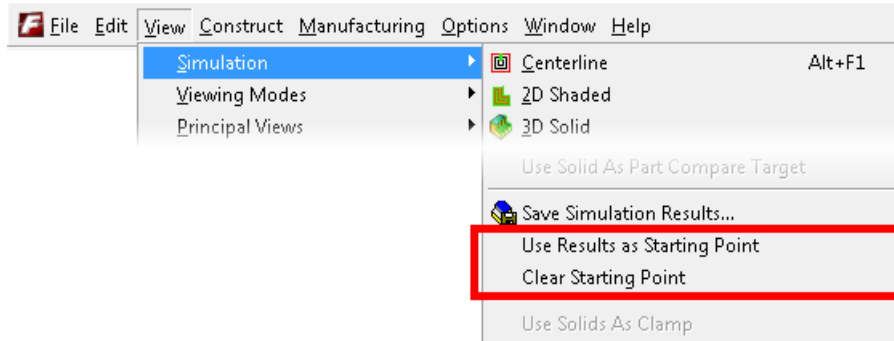
*This option is available only after you have used **Step Forward** from this menu or the **Single Step**  button on the **Simulation** toolbar.*

---

## Simulation: Faster 3D simulation

In FeatureCAM 2012 R2, 3D simulation and machine simulation is faster thanks to the use of multiple cores when doing simulation. Multiple cuts are performed simultaneously using multiple cores, and you see the end result much sooner.

You can also save the results of a 3D simulation to use as a starting point, to save calculation time during subsequent simulations. Select **View > Simulation > Use Results as Starting Point** to use the current simulation results. To clear the simulation results, select **View > Simulation > Clear Starting Point**.



*To use this feature, you must select **Save result files during 3D sim** on the **2D/3D Shaded** tab of the **Simulation Options** dialog.*

---

## Simulation: New keyboard shortcuts

There are two new keyboard shortcuts that you can use during simulation:

**Alt+Right** - Press and hold the **Alt** key then click the right cursor arrow key to step forward once.

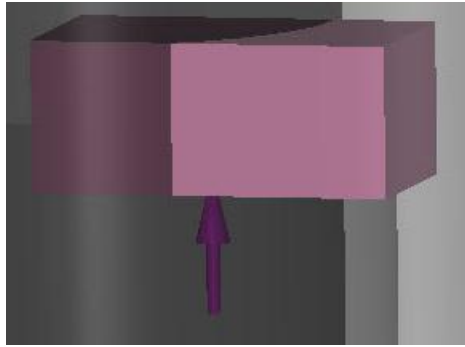
**Alt+Left** - Press and hold the **Alt** key then click the left cursor arrow key to step backward (see page 37) once.

---

## Machine simulation: Better collision display

FeatureCAM 2012 R3 has improved collision detection for machine simulation.

Whenever a named part of the machine collides with another solid, a flashing arrow is displayed and the display turns transparent, for example:



You must enable ***Pause on gouge*** on the ***2D/3D Shaded*** tab of ***Simulation Options*** to see this feature.

*The first collision point only is marked.*

*The display returns to normal when you continue the simulation.*

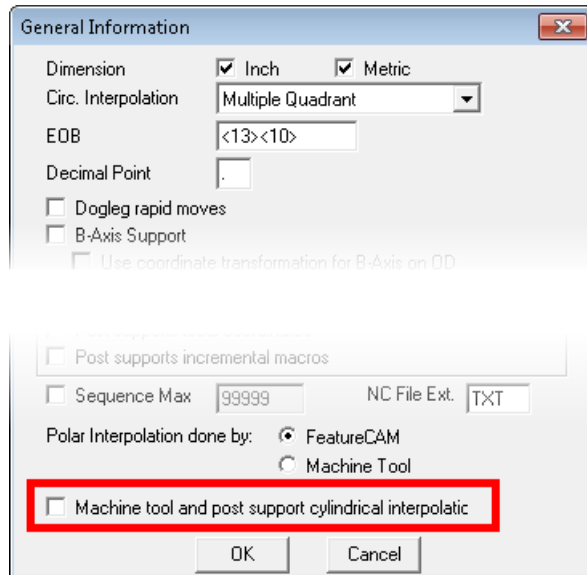
*The arrow is not shown for tools colliding with stock.*

---

## Turn/Mill: Cylindrical interpolation support

FeatureCAM 2012 R2 supports cylindrical interpolation in the NC code for Turn/Mill and 4th-axis parts. This enables features that are wrapped around one of the index axes to use a cylindrically interpolated toolpath. The result is shorter NC code and better interpretation of cutter compensation.

To use it, select the new option **Machine tool and post support cylindrical interpolation** in the **General Information** dialog in XBUILD:

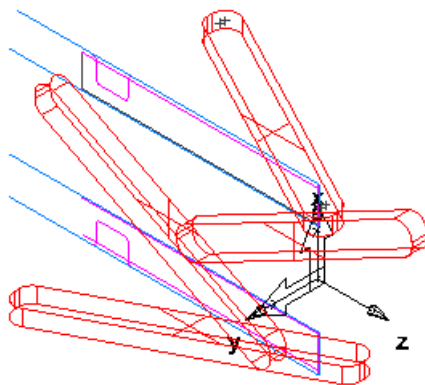


Cylindrical interpolation uses these new reserved words:

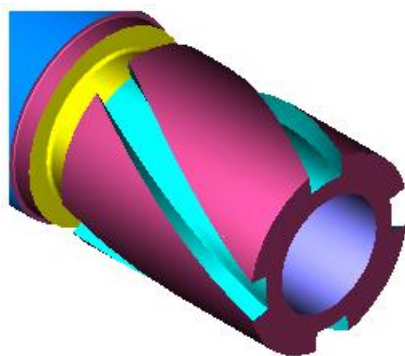
`<CYL-INTERP>` - logical, `TRUE` if cylindrical interpolation is used

`<WRAP-RADIUS>` - numeric, the wrapping radius

For example, this part has four Pockets wrapped around the Z axis:



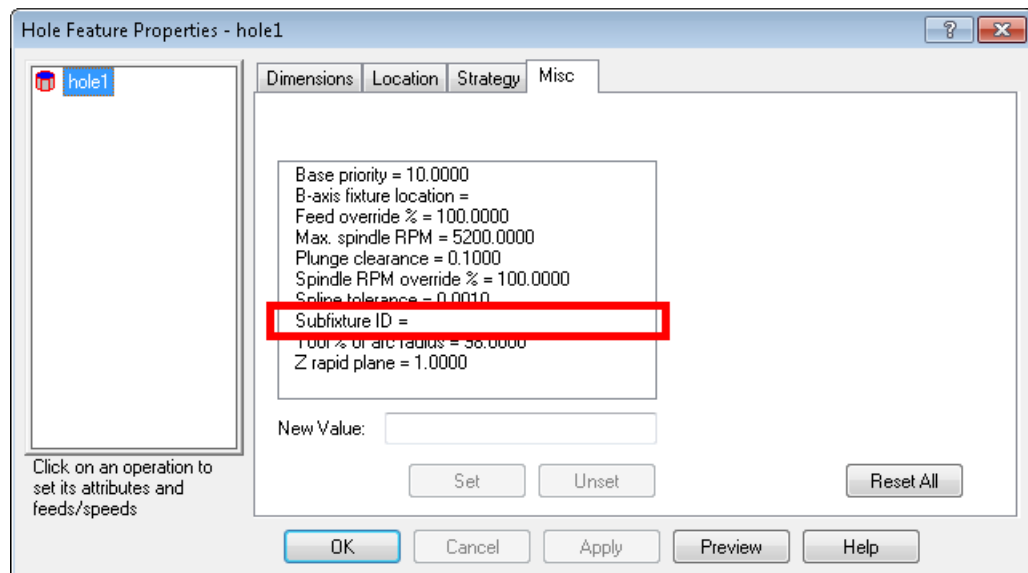
A 3D simulation looks like this:



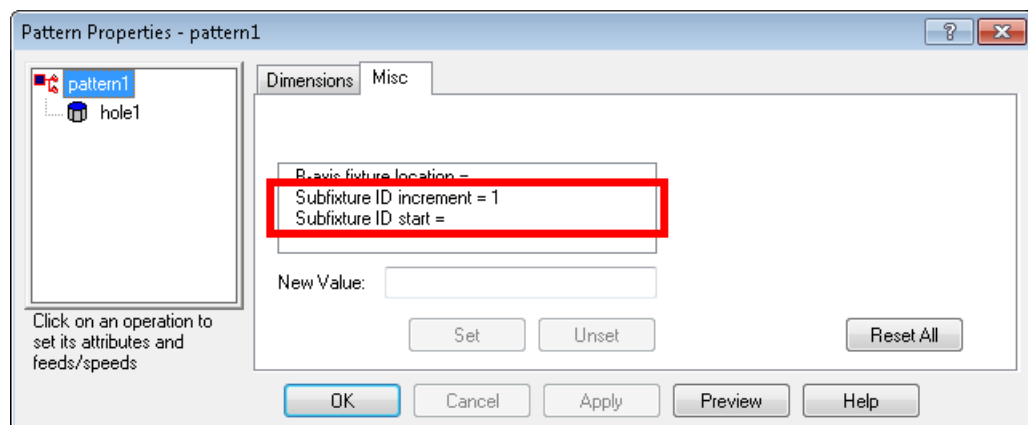
After selecting the **Machine tool** and **post support cylindrical interpolation** option, the NC code uses `G07` to turn on cylindrical interpolation for the Pocket operations. This makes the NC code shorter, and smaller in memory size.

## Turn/Mill: Separate fixture offset for each feature

There are new attributes in FeatureCAM 2012 R3, on the **Misc** tab of the feature's Properties dialog, which let you use a separate fixture offset for each B-axis turn/mill Hole (or other feature).



**Subfixture ID** - applies to B-axis turn/mill features and corresponds to the `<SUBFIXTURE>` reserved word in XBUILD. This lets you use a separate fixture offset for each Hole (or other feature). For example, if the **Fixture ID** is **54** and you enter a **Subfixture ID** of **1**, the output is `54.1`.

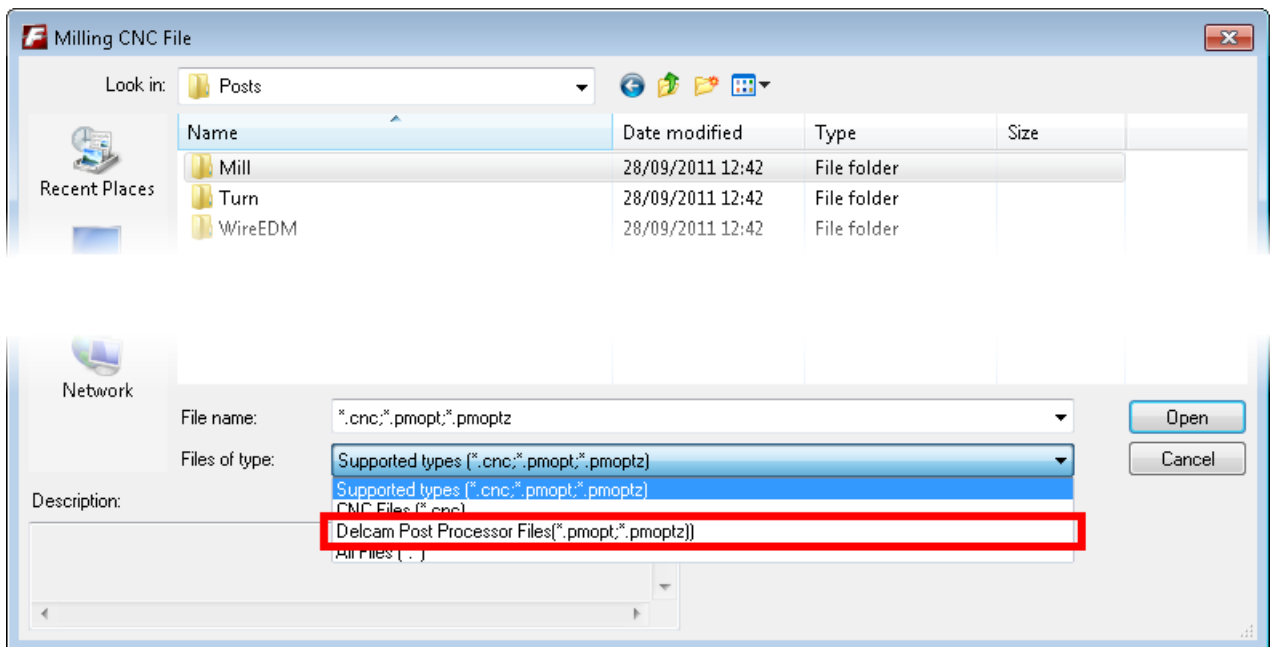


For patterns, enter a **Subfixture ID start** value and a **Subfixture ID increment** value. For example, if the **Fixture ID** is **54** and you enter a **Subfixture ID start** of **1** and a **Subfixture ID increment** of **1**, the output is **54.1, 54.2, 54.3**, and so on.

---

## XBUILD: Use Delcam post processor files

You can now use \*.pmopt and \*.pmoptz files with XBUILD:



*Some PMOPT capabilities are not supported by XBUILD. For example, XBUILD supports much wider variety of NC output for 2-axis milling than is currently unavailable in PMOPT. So if you choose PMOPT you lose such control of your NC code. PMOPT files do not support Turning and Wire EDM.*

---

## Probing: UDF available

In FeatureCAM 2012 R2 you can preview a UDF-based probing capability that is available to purchase as an optional module. This module supports a variety of touch-probing capabilities such as setting of part location fixture offsets, setting of tool offsets, probing-and-recut, and so on. The software is developed as an add-in that works with a modified post processor, and is available by request to the Post department.

---

## API: New functions

This is a summary of the new API functions, for more information please see the API help file.

`FMToolpathPts.DeleteItem` - Delete a toolpath point from list of points.

`FMToolpathPt.Dwell` - Return and set the dwell time associated with the toolpath point.

`FMToolpathArc.Dwell` - Return and set the dwell time associated with the toolpath arc.

`FMToolpathPts.AddtoolpathNCtext` - Create a toolpath NC text segment.

`FMToolpathNCtext` - A new class to represent the NC text segment.

`FMFeatures.AddToolpathFromOperation` - Create a toolpath feature from an operation.

`Application.SetGraphicsWinLowerColor` - Set graphics window lower background color.

`Application.SetGraphicsWinUpperColor` - Set graphics window upper background color.

`FMCurves.AddFunctionCurve` - Create curve from function.

`FMSurfaces.AddSurfaceCoons` - Create a coons surface.

---

## Other improvements

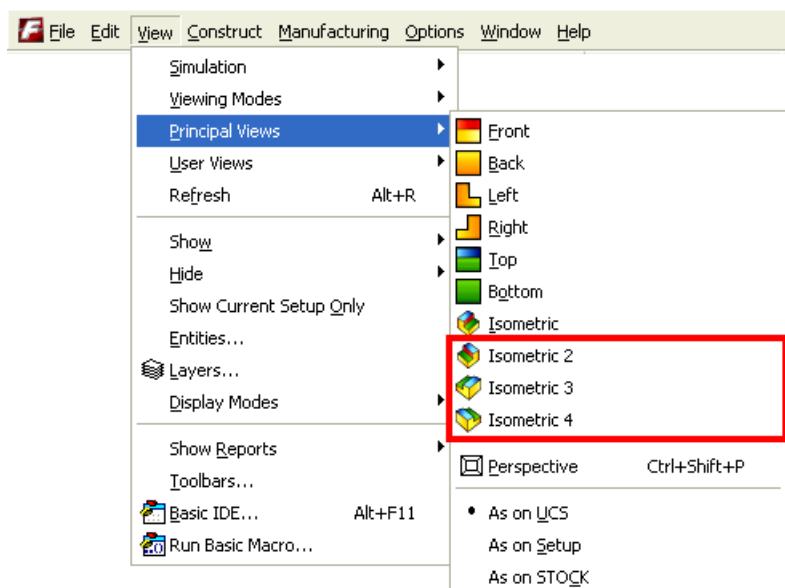
- Network database - The network database is now more reliable.
- 2.5D: Performance improvement for large STL stock.
- Turn/Mill - The **Disable Macros** option is now available on the **Turn/Mill** tab in the **Post Options** dialog.

# What's New in FeatureCAM 2012

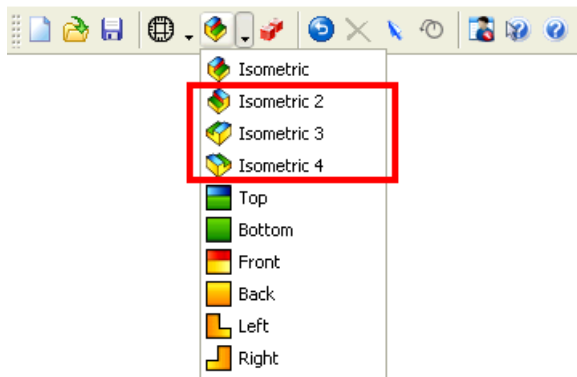
FeatureCAM 2012 offers all of the original features of FeatureCAM 2011 R3 (V17.8), but with numerous improvements. The most significant improvements are detailed in the following topics.

## User interface: More isometric views

There are now four isometric views available. As well as the existing **Isometric** view, there are also **Isometric 2**, **Isometric 3**, and **Isometric 4** views. Access the new views from the **View > Principle Views** menu:

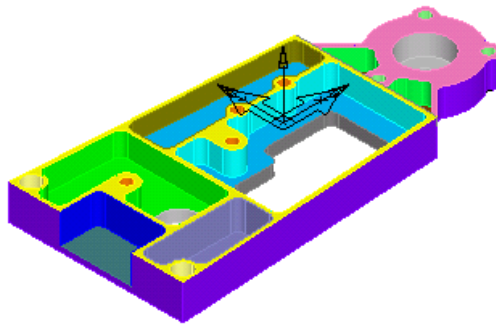


Or from the **Principle View** menu on the **Standard** toolbar:

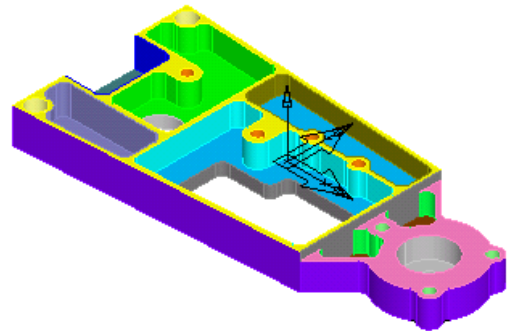


The following example shows all four isometric views:

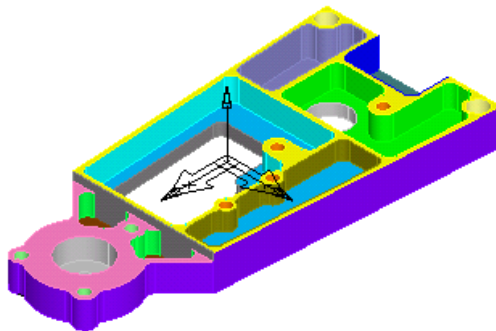
**Isometric:**



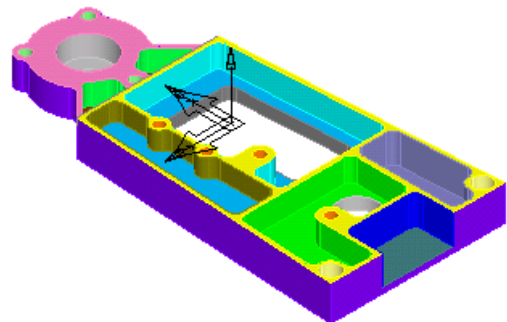
**Isometric 2:**



**Isometric 3:**

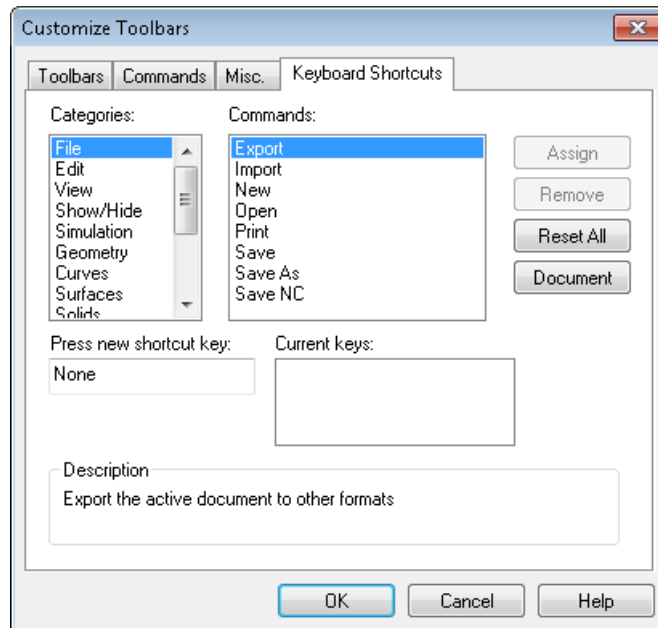


**Isometric 4:**



# User interface: Keyboard shortcuts

There is a new tab in the **Customize Toolbars** dialog, **Keyboard Shortcuts**:



This tab lets you assign your own shortcut keys to the FeatureCAM menu commands.

To set a new shortcut key:

- 1 Select a category from the **Categories** list.
- 2 Select a command from the **Commands** list.

If the command already has a shortcut, it is listed under **Current keys**.

- 3 To assign a new key, select the **Press new shortcut key** field and press the keys that you want to use as the shortcut.



*Press the keys on the keyboard, for example, press and hold down the **Alt** key and press the **N** key; do not type **ALT + N**.*



*A warning displays if the shortcut you entered is already assigned to a command.*

- 4 Click the **Assign** button to save the new shortcut.



*If you click **Assign** after receiving a warning that the shortcut is already assigned to a command, the shortcut is removed from the existing command and assigned to the new command.*

**Remove** — Click this button to remove the shortcut assigned to the selected command.

**Reset All** — Click this button to reset all shortcuts to the FeatureCAM defaults.

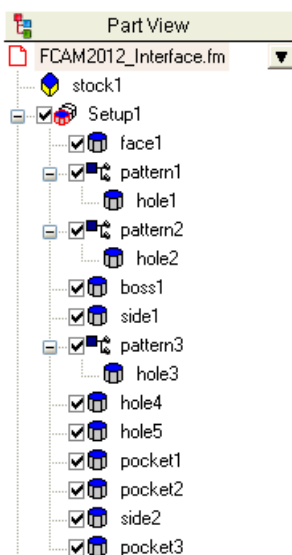
**Document** — Click this button to open a \*.txt file, containing a list of the current shortcuts, in your text editor. You can print or save this file.

---

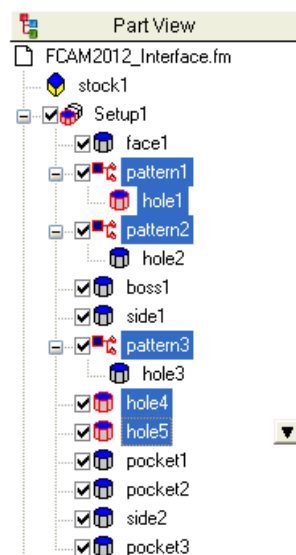
## User interface: Part view multiple select

You can now select multiple items in the **Part View** using **Shift + click** to select consecutive items and **Ctrl + click** to select non-consecutive items.

In the following example, we want to select all of the Hole features to add to a group:

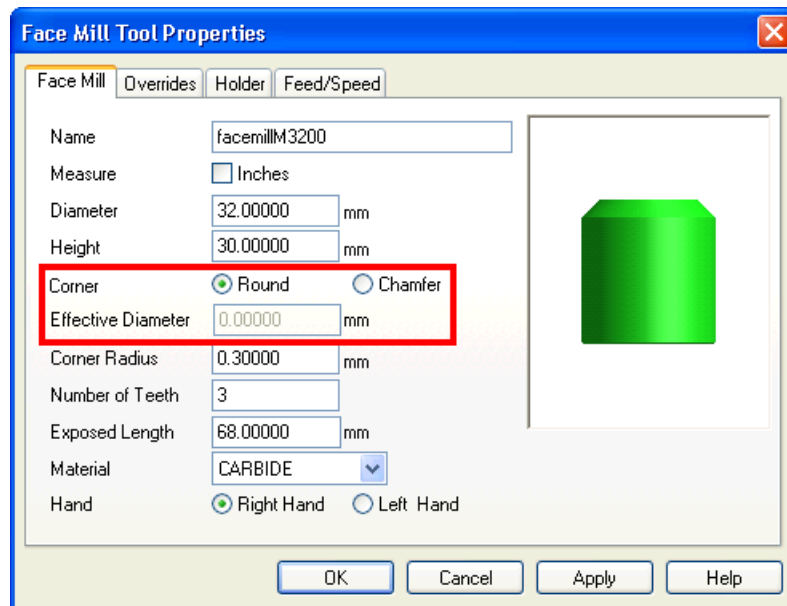


To do this, select **pattern1**, hold down the **Shift** key and click **pattern2**, hold down the **Ctrl** key and click **pattern3**, **hole4**, and **hole5**:



# Tooling: Better Face Mill tools

There is a new option to specify a 45° chamfer for Face Mill tools:



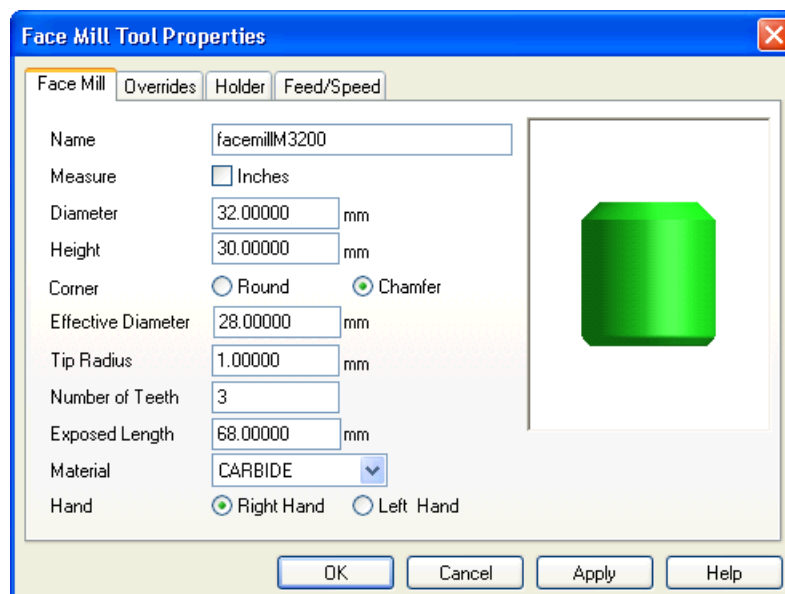
This new option allows more accurate simulation and machining.

Existing tools are classified as **Round** by default.

To set a 45° chamfer on a Face Mill tool:

- 1 Select **Chamfer** as the **Corner** type.
- 2 Enter an **Effective Diameter** greater than zero.
- 3 Enter a **Tip Radius** greater than or equal to zero.
- 4 Click the **Apply** button.

The tool preview image on the right of the dialog updates to show the chamfer, for example:



# Tooling: Cutting data by tool

You can associate cutting data with individual tools in the tool database at two levels:

- On the **Overrides** tab of the **Tool Properties** dialog. You can set cutting data for the tool for all materials and operations.
- On the **Feed/Speeds** tab of the **Tool Properties** dialog. You can set cutting conditions by material, operation, and cut type. These override any settings on the **Overrides** tab.



*You can override these settings at feature operation level.*

## Tool overrides for all materials and operations

The new parameters on the **Overrides** tab of the **Tool Properties** dialog are **Depth** of cut (Z-step increment), **Stepover** distance, and **Ramp angle**.

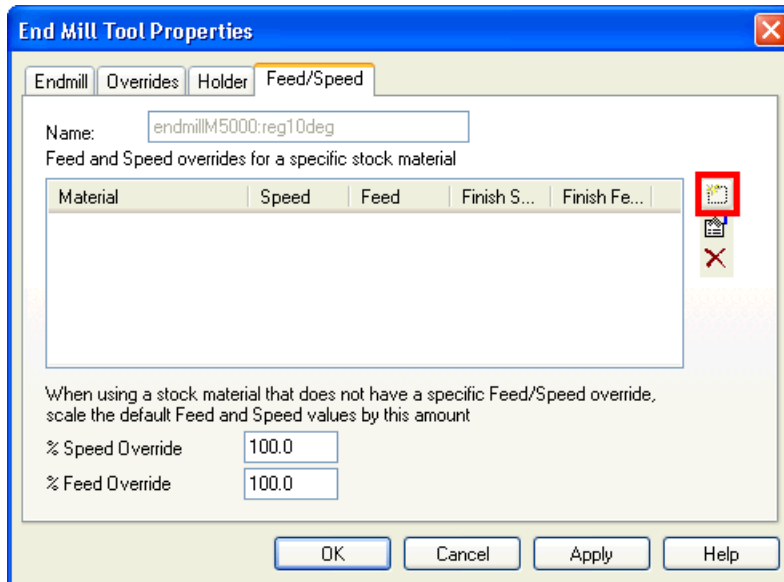
Enter values to use with the current tool for all materials, except those overridden on the **Feed/Speed** tab, for example:

The screenshot shows the 'End Mill Tool Properties' dialog box with the 'Overrides' tab selected. The 'Name' field contains 'endmillM5000:reg45deg'. Under 'Default tool registers', 'Tool number' is 0 with a checked 'Same' box, 'Diameter offset register' is 0, 'Length offset register' is 0, and 'Tool ID' is 0. On the right, 'Operations' is set to 'Rough operation' and 'Coolant' is 'flood'. A red rectangle highlights the 'Depth' and 'Stepover' settings: 'Depth' is checked and set to 3.000 mm, and 'Stepover' is checked and set to 15.000 mm. The 'RampAngle' checkbox is unchecked. At the bottom is a 'Comments' text area and buttons for 'OK', 'Cancel', 'Apply', and 'Help'.

## Tool overrides by material, operation, and cut type

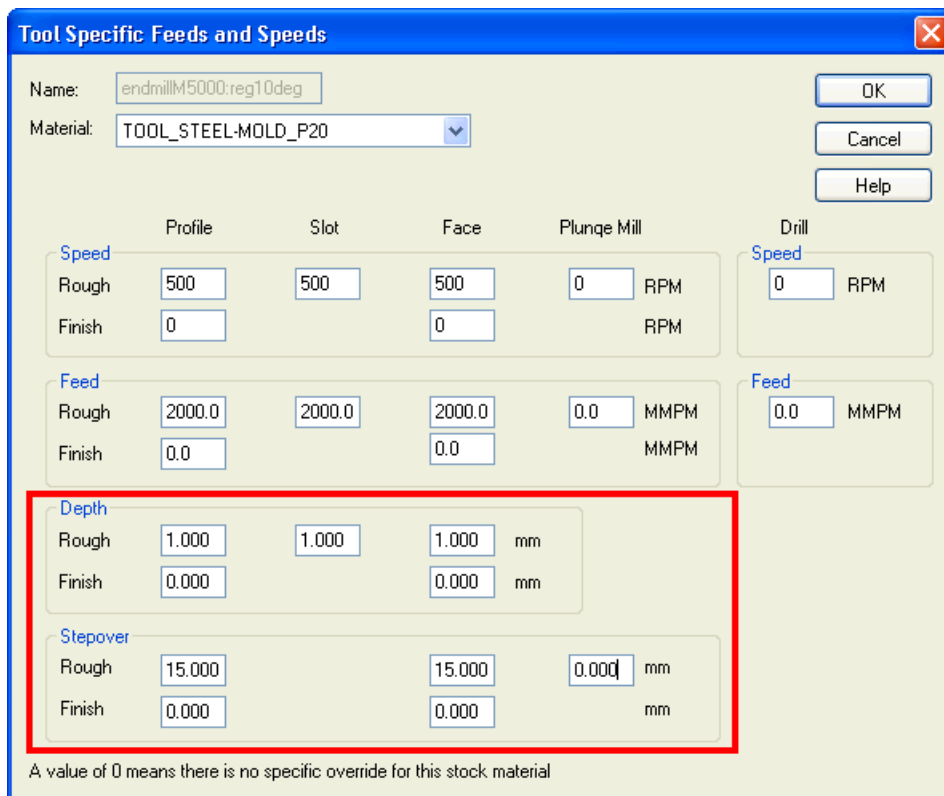
As well as the previously available **Speed** and **Feed** settings, you can set **Depth** (of cut) and **Stepover** by **Material**, operation (**Rough** and **Finish**) and type of cut (**Profile**, **Slot**, **Face**, and **Plunge Mill**).

To do this, select the **Feed/Speed** tab of the **Tool Properties** dialog and click the **Create new item**  button:

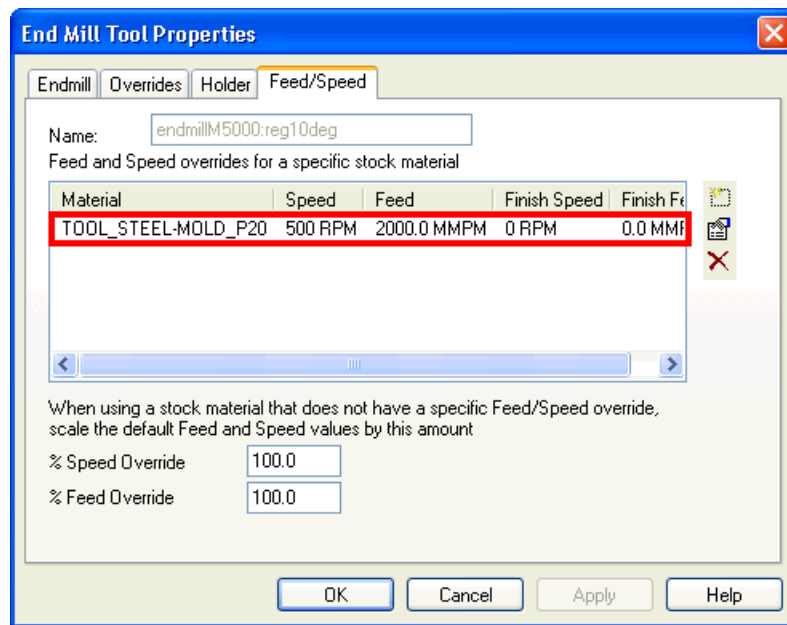


The **Tool Specific Feeds and Speeds** dialog opens.

Select the **Material** name from the list and enter any **Depth** and **Stepover** overrides by operation and type of cut, for example:



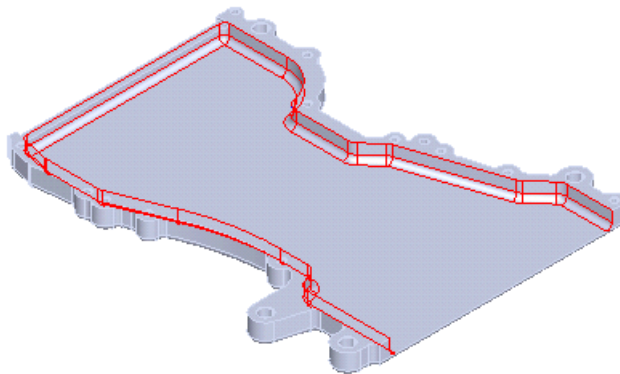
Click **OK**. The material is listed on the **Feed/Speed** tab:



These material-based settings take priority over any on the global **Overrides** tab.

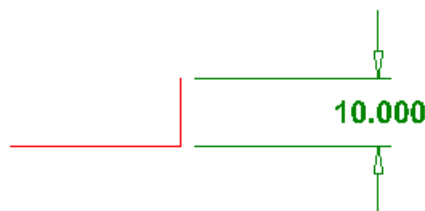
## Tool cutting data example

This example part has a Side feature with a Depth of 15 mm:

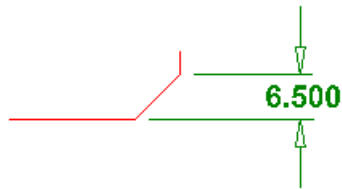


We have the choice of three form tools to cut this Side feature:

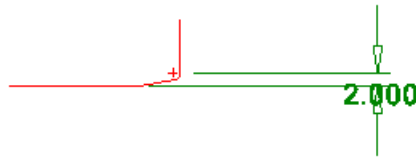
- A 90° insert with a cut depth of 10 mm:



- A 45° insert with a cut depth of 6.5 mm:



- A 10° insert with a cut depth of 2 mm:



Each of these tools has a different feed rate associated with it as well as a different cut depth.

For the 90° insert tool with the depth of cut of 10 mm, we created a material-based maximum **Depth** (of cut) override of **8** mm for the **Rough** operation for **ALUMINUM**:

**Tool Specific Feeds and Speeds**

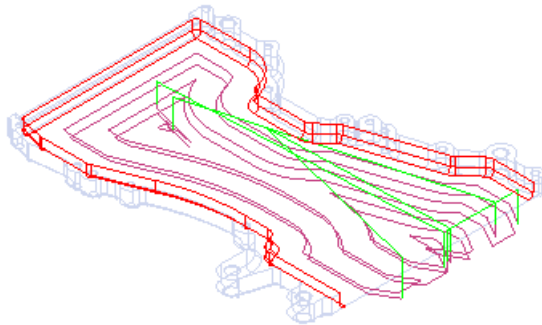
Name:  OK Cancel Help


Material:

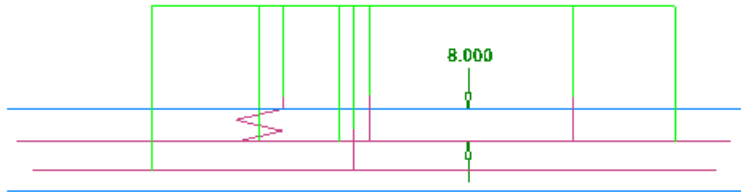
	Profile	Slot	Face	Plunge Mill	Drill
<b>Speed</b>					
Rough	<input type="text" value="500"/>	<input type="text" value="500"/>	<input type="text" value="500"/>	<input type="text" value="0"/> RPM	<input type="text" value="0"/> RPM
Finish	<input type="text" value="0"/>		<input type="text" value="0"/>		
<b>Feed</b>					
Rough	<input type="text" value="1250.0"/>	<input type="text" value="1250.0"/>	<input type="text" value="1250.0"/>	<input type="text" value="0.0"/> MPPM	<input type="text" value="0.0"/> MPPM
Finish	<input type="text" value="0.0"/>		<input type="text" value="0.0"/>		
<b>Depth</b>					
Rough	<input type="text" value="8.000"/>	<input type="text" value="8.000"/>	<input type="text" value="8.000"/> mm		
Finish	<input type="text" value="0.000"/>		<input type="text" value="0.000"/> mm		
<b>Stepover</b>					
Rough	<input type="text" value="25.000"/>		<input type="text" value="25.000"/>	<input type="text" value="0.000"/> mm	
Finish	<input type="text" value="0.000"/>		<input type="text" value="0.000"/>		

A value of 0 means there is no specific override for this stock material

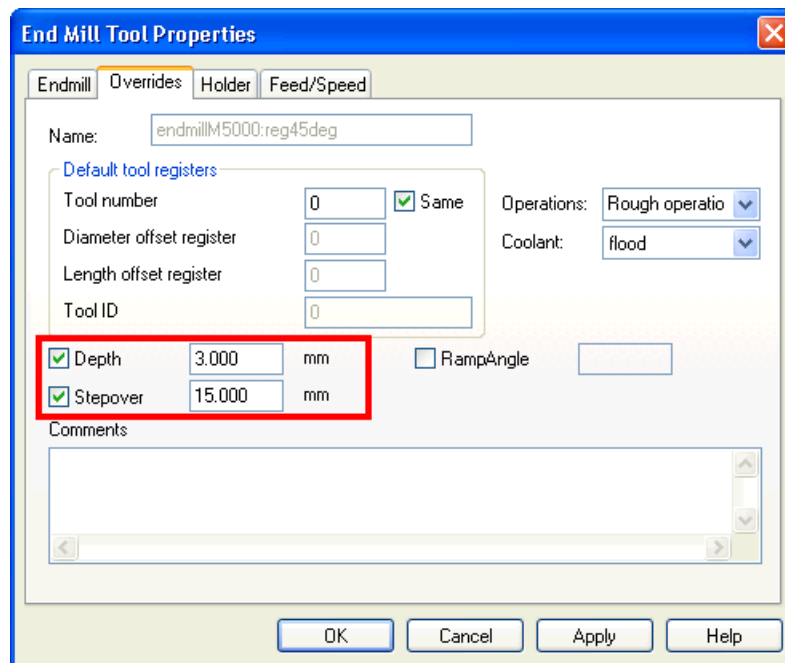
This is the centerline simulation of the 90° tool cutting the rough operation of the Side feature, with two toolpath passes:



Viewing the simulation of the toolpath from the front, we can use the **Vertical Distance**  button on the **Geometry** toolbar to check that the 8 mm **Depth** override has been applied to the rough pass:

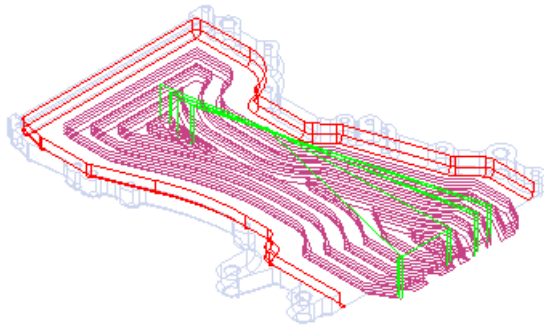


For the 45° insert with a cut depth of 6.5 mm, we created a tool-specific maximum **Depth** (of cut) override of **3** mm and a **Stepover** override of **15** mm on the **Overrides** tab:

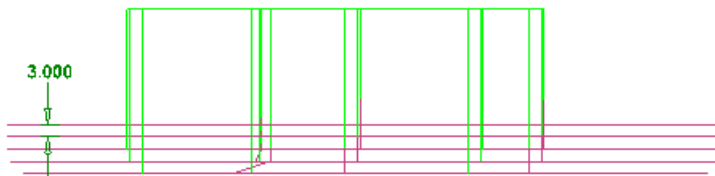


These values are used unless you set any material-based overrides on the **Feed/Speed** tab.

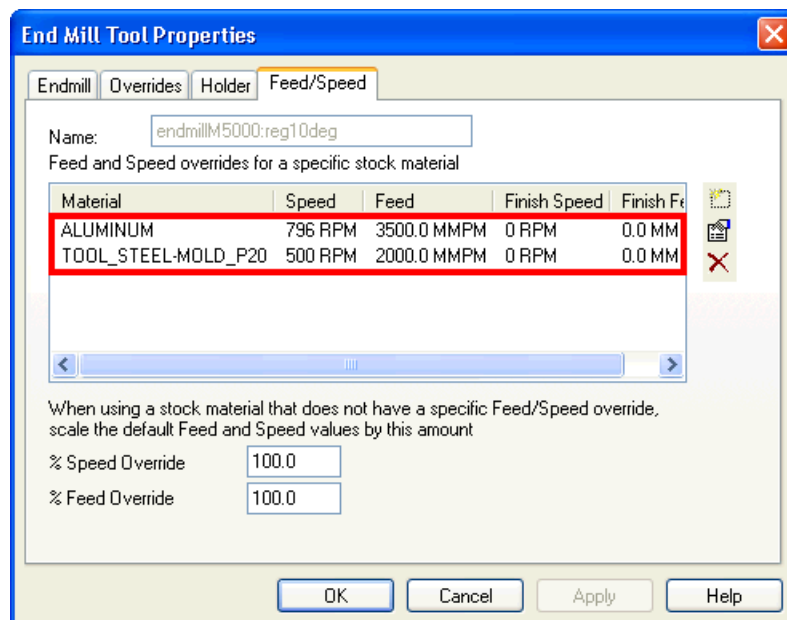
This is the centerline simulation of the 45° tool cutting the rough operation of the Side feature:



Using the same method as before, we can check that the 3 mm maximum **Depth** (of cut) override has been applied:



For the 10° insert with a cut depth of 2 mm, we created two material-based overrides on the **Feed/Speed** tab:



For **ALUMINUM**, we set quite a high **Feed** rate, and a **1.5 mm** maximum cut **Depth**:

**Tool Specific Feeds and Speeds**

Name:

Material:

OK Cancel Help

	Profile	Slot	Face	Plunge Mill	Drill
<b>Speed</b>					
Rough	796	796	796	0 RPM	0 RPM
Finish	0		0	RPM	
<b>Feed</b>					
Rough	3500.0	3500.0	3500.0	0.0 MMPM	0.0 MMPM
Finish	0.0		0.0	MMPM	
<b>Depth</b>					
Rough	1.500	1.500	1.500 mm		
Finish	0.000		0.000 mm		
<b>Stepover</b>					
Rough	15.000		15.000	0.000 mm	
Finish	0.000		0.000	mm	

A value of 0 means there is no specific override for this stock material

For **TOOL\_STEEL-MOLD\_P20**, we set a lower cut **Depth** of **1 mm**, a lower **Feed** rate and a lower spindle **Speed**:

**Tool Specific Feeds and Speeds**

Name:

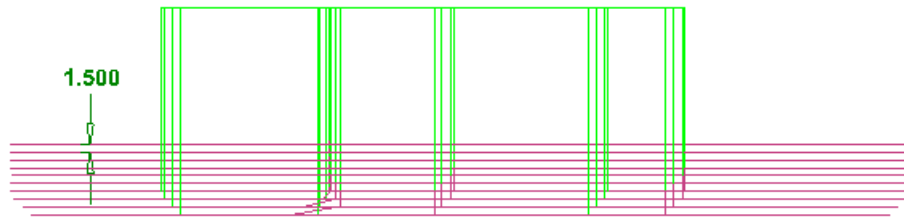
Material:

OK Cancel Help

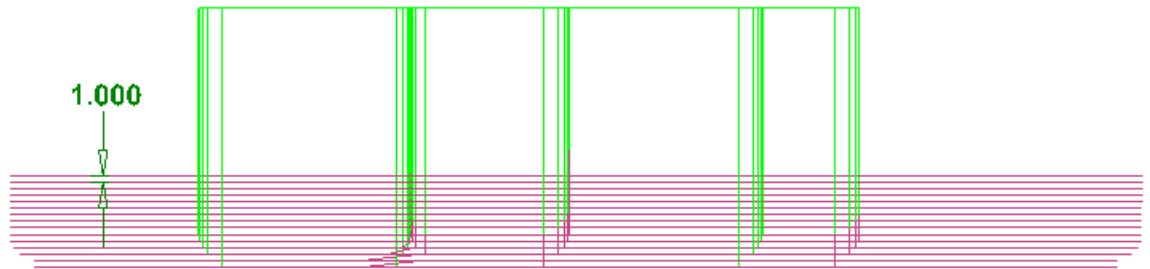
	Profile	Slot	Face	Plunge Mill	Drill
<b>Speed</b>					
Rough	500	500	500	0 RPM	0 RPM
Finish	0		0	RPM	
<b>Feed</b>					
Rough	2000.0	2000.0	2000.0	0.0 MMPM	0.0 MMPM
Finish	0.0		0.0	MMPM	
<b>Depth</b>					
Rough	1.000	1.000	1.000 mm		
Finish	0.000		0.000 mm		
<b>Stepover</b>					
Rough	15.000		15.000	0.000 mm	
Finish	0.000		0.000	mm	

A value of 0 means there is no specific override for this stock material

Again, we can check that the 1.5 mm maximum **Depth** (of cut) override has been applied to the **ALUMINUM** stock:

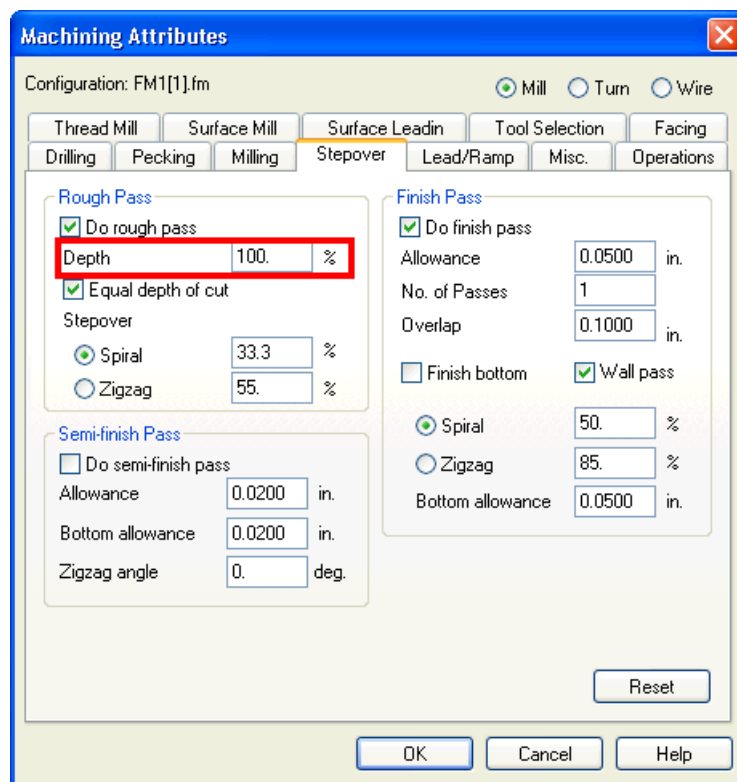


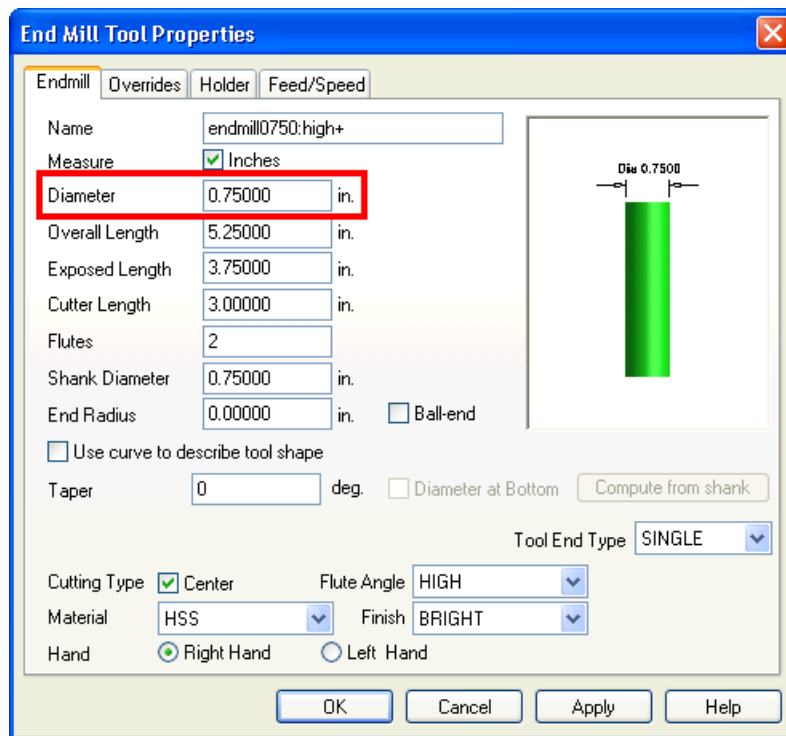
If we change the stock to **TOOL\_STEEL-MOLD\_P20** and run the simulation again, we see that the 1 mm maximum cut **Depth** override has been applied:



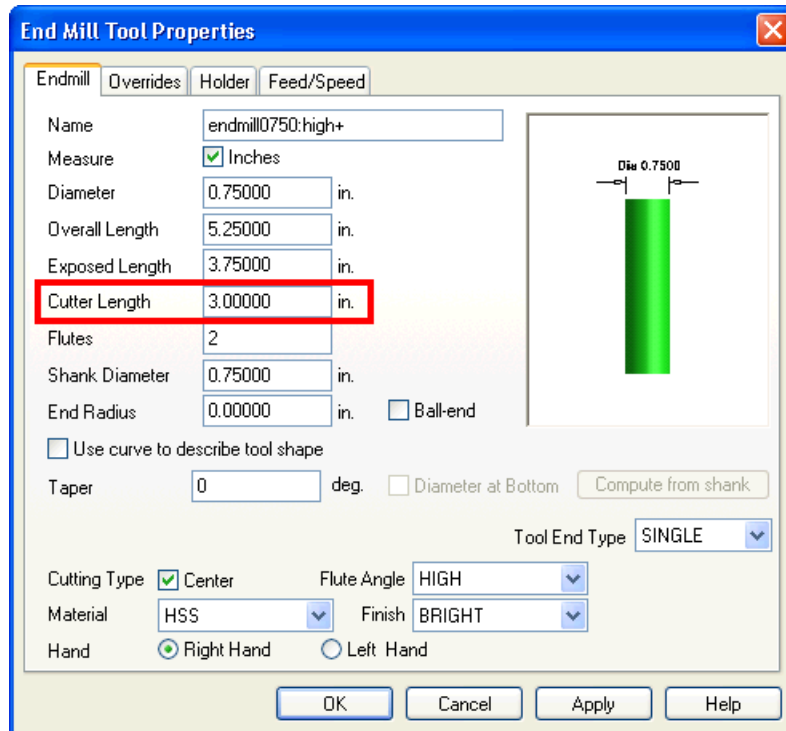
## Depth of cut precedence

The Rough depth of cut is normally calculated as the tool **Diameter** multiplied by the **Depth %** attribute set on the **Stepover** tab of **Machining Attributes**.





The default Finish behavior is a single pass around the profile of the feature, up to the **Cutter Length** of the tool. This is to avoid leaving a mark on the side wall of the feature.



You can override both the Rough and Finish depth of cut in several places, from lowest to highest precedence:

- Tool level: **Depth** attribute on the **Overrides** tab of the **Tool Properties** dialog.



- Tool level: Material-specific **Depth** attribute on the **Feed/Speed** tab of the **Tool Properties** dialog.

**End Mill Tool Properties**

Endmill Overrides Holder **Feed/Speed**

Name:

Feed and Speed overrides for a specific stock material

Material	Speed	Feed	Finish S...	Finish Fe...
ALUMINUM	0 RPM	0.00 IPM	0 RPM	0.00 IPM

When using a stock material that does not have a specific Feed/Speed override, scale the default Feed and Speed values by this amount

% Speed Override

% Feed Override

OK Cancel Apply Help

**Tool Specific Feeds and Speeds**

Name:

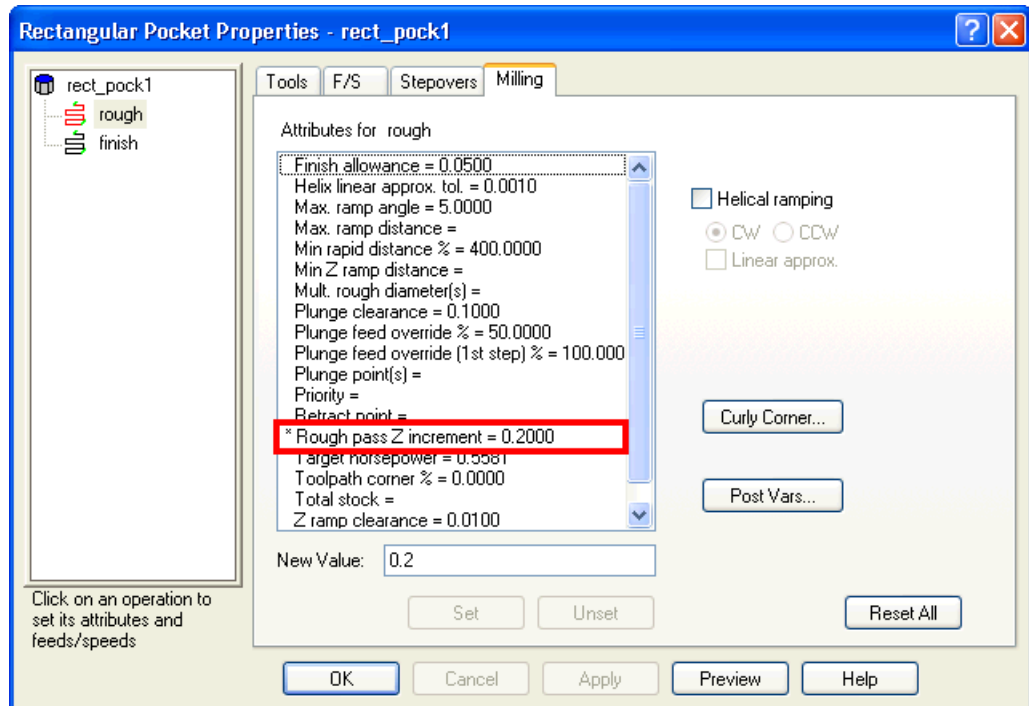
Material:

OK Cancel Help

	Profile	Slot	Face	Plunge Mill	Drill
<b>Speed</b>					
Rough	<input type="text" value="0"/>	<input type="text" value="0"/>	<input type="text" value="0"/>	<input type="text" value="0"/> RPM	<input type="text" value="0"/> RPM
Finish	<input type="text" value="0"/>		<input type="text" value="0"/>	RPM	
<b>Feed</b>					
Rough	<input type="text" value="0.00"/>	<input type="text" value="0.00"/>	<input type="text" value="0.00"/>	<input type="text" value="0.00"/> IPM	<input type="text" value="0.00"/> IPM
Finish	<input type="text" value="0.00"/>		<input type="text" value="0.00"/>	IPM	
<b>Depth</b>					
Rough	<input type="text" value="0.25"/>	<input type="text" value="0.25"/>	<input type="text" value="0.25"/> in.		
Finish	<input type="text" value="0.0000"/>		<input type="text" value="0.0000"/> in.		
<b>Stepover</b>					
Rough	<input type="text" value="0.0000"/>		<input type="text" value="0.0000"/>	<input type="text" value="0.0000"/> in.	
Finish	<input type="text" value="0.0000"/>		<input type="text" value="0.0000"/>	in.	

A value of 0 means there is no specific override for this stock material

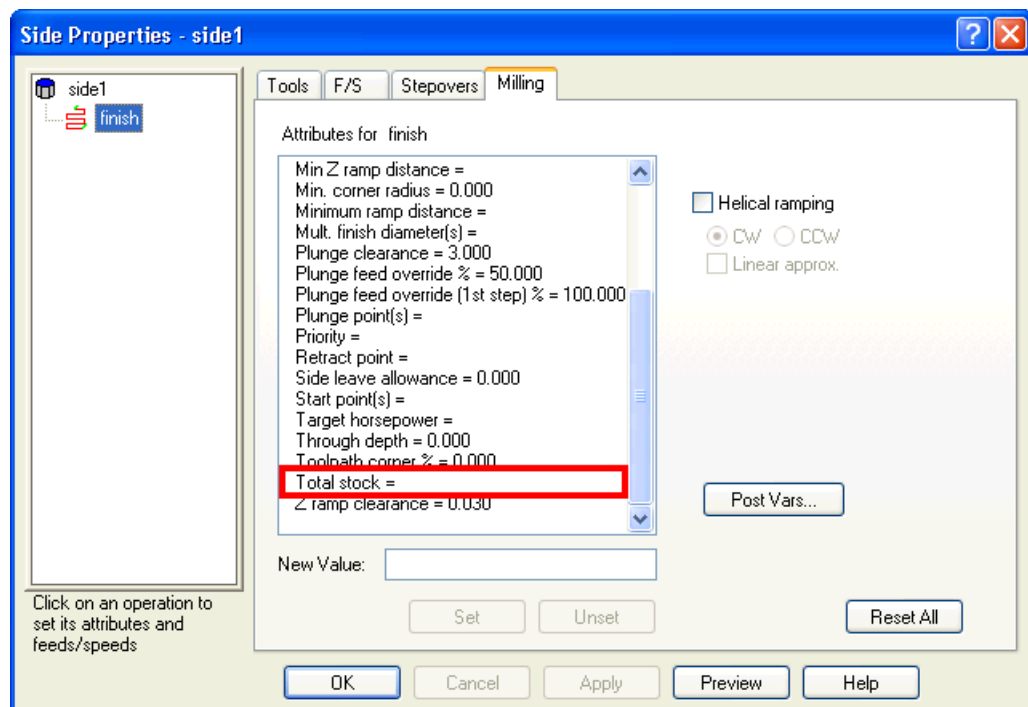
- Feature operation level: **Rough/Finish pass Z increment** on the **Milling** tab of the **Feature Properties** dialog.



The final depth of cut depends on whether you have selected **Equal depth of cut** on the **Misc.** tab of the **Feature Properties** dialog.

## 2.5D: Total stock for finish pass

In FeatureCAM 2012, you can set a **Total stock** attribute for finish operations. Set it on the **Milling** tab for the finish operation:

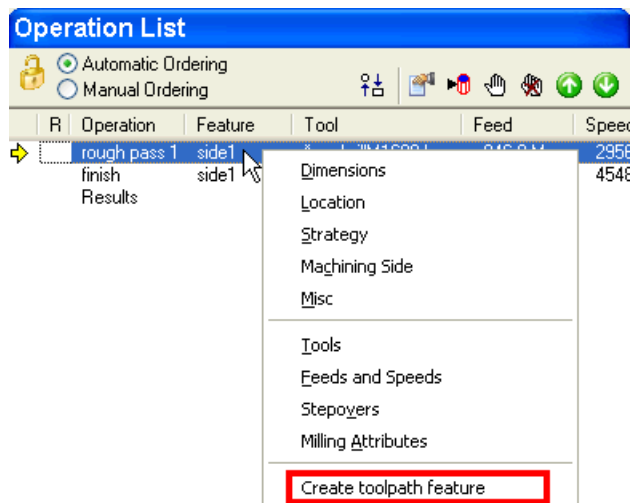


Previously this attribute was available only for rough operations.


## 2.5D: Toolpath feature improvements

### Quickly create a Toolpath feature from an operation in the Op List

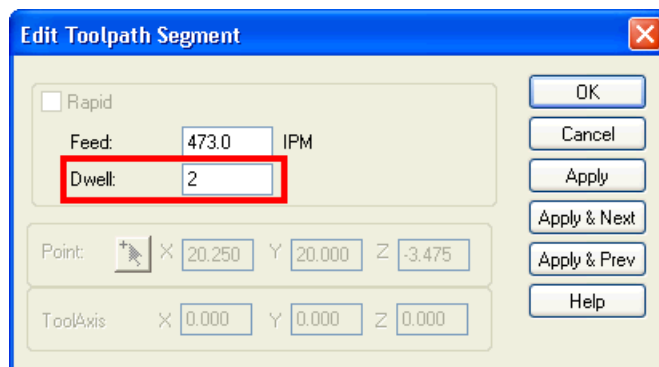
In FeatureCAM 2012, you can create a Toolpath feature by right-clicking on the operation in the **Op List** tab and selecting the new option, **Create toolpath feature** from the context menu, for example:



### Add a Dwell value to a toolpath point

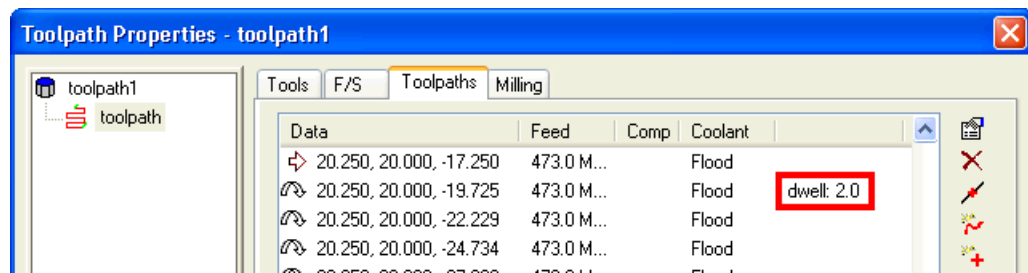
You can now add a dwell value to a toolpath point. To do this, open the **Toolpath Properties** dialog, and select the **Toolpaths** tab for the toolpath operation. Select the toolpath segment where you want to add the dwell and click the **Edit segment**  button.

The new **Dwell** option is in the **Edit Toolpath Segment** dialog:



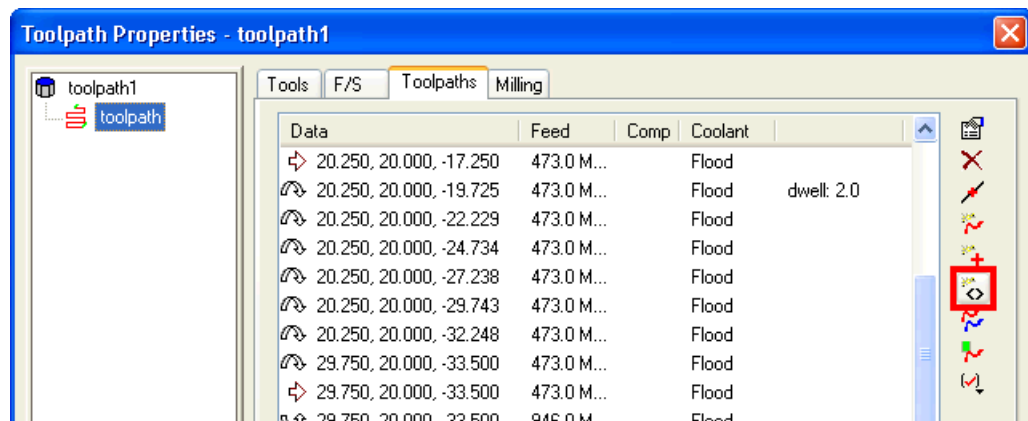
Enter the **Dwell** value in seconds.

The dwell amount is displayed on the **Toolpaths** tab, for example:

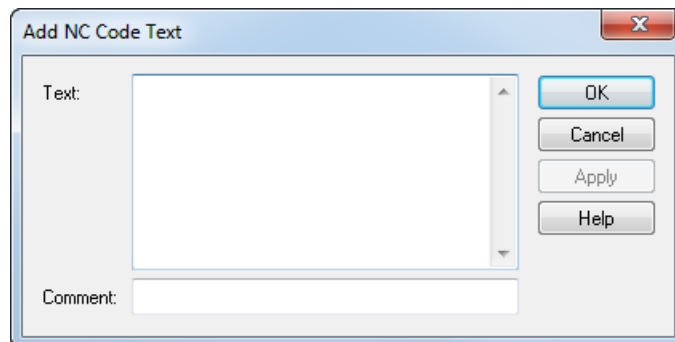


### Add NC code text to a Toolpath feature

There is a new button on the **Toolpaths** tab,  **Add NC code text:**

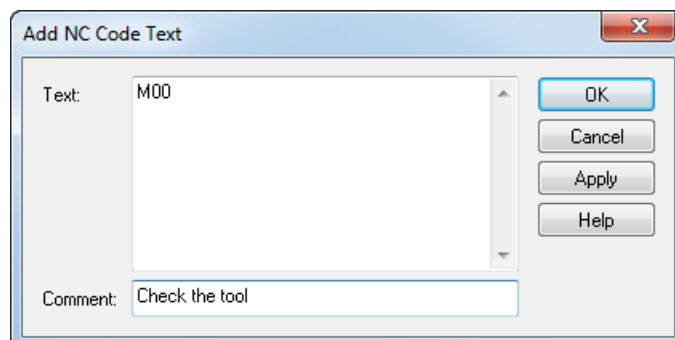


This button opens the **Add NC Code Text** dialog:

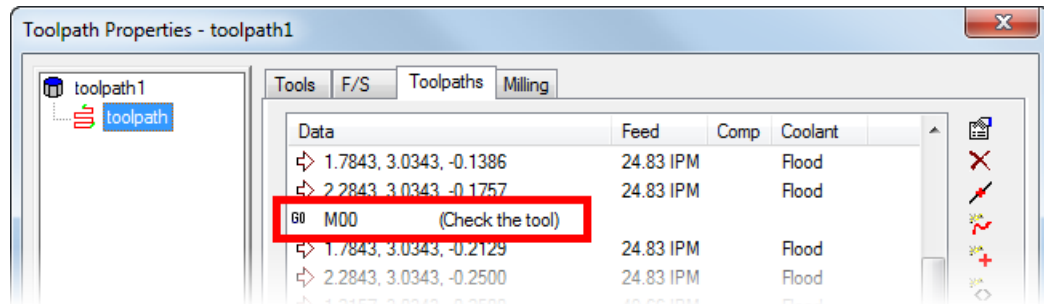


The **Add NC Code Text** dialog lets you insert NC code before the selected point and optionally add a comment.

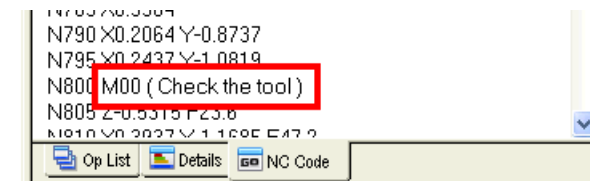
For example, you could enter a **M00** conditional stop at a particular point in order to check the tool:



Click the **Apply** button, then the **OK** button, and the NC code and comment are added to the **Toolpaths** tab to the row above the selected row, for example:



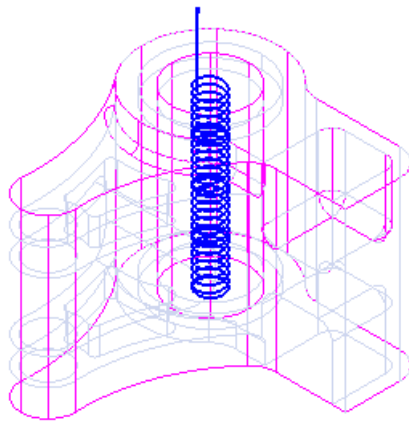
And after simulation, you can see the code on the **NC Code** tab, for example:



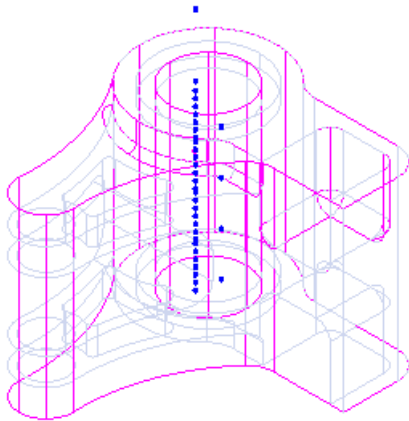
*Your post must support dwell and EDF text to use these new features.*


### Toolpath feature displayed as lines

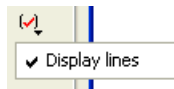
When you create a Toolpath feature and select the **Toolpaths** tab, you can now see the full lines of the toolpath, for example:




Previously, toolpath points only were displayed, for example:

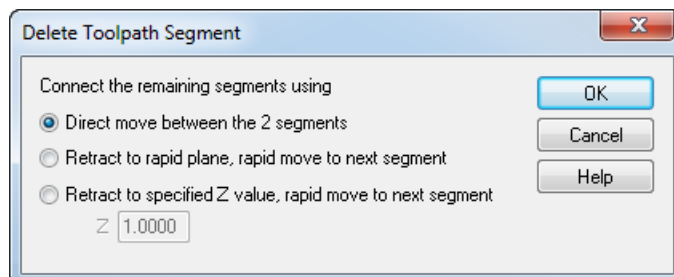


Toolpath lines are displayed by default, but you can turn them on and off in the context menu of the new **Options**  button. Select or deselect **Display lines**:



### Connection options after deleting toolpath segments


In FeatureCAM 2012, when you delete toolpath segments, there are more options for connecting the remaining segments. The **Delete Toolpath Segment** dialog opens after you click the **Delete segment**  button:



Select from:

- **Direct move between the 2 segments** (previous behavior)
- **Retract to rapid plane, rapid move to next segment**
- **Retract to specified Z value, rapid move to next segment**, and enter a Z value

### Undo toolpath edits

You can now use the **Undo**  button on the **Standard** toolbar to undo edits that you make to a Toolpath feature. For example, if you delete some toolpath points, you can click the **Undo** button to restore the points.

## 2.5D: Exact Z step

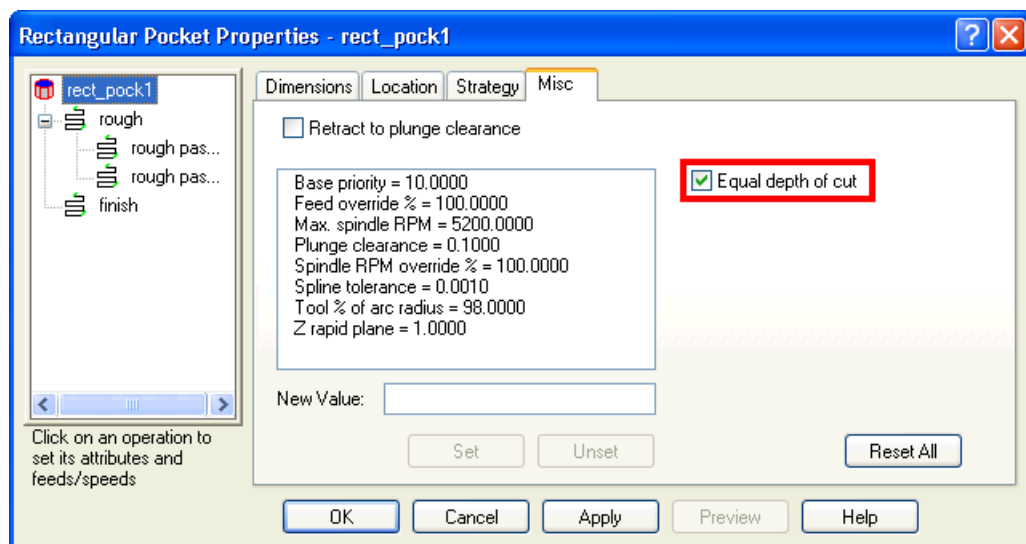
Currently FeatureCAM calculates the depth of cut like this:

- 1 Divides the feature depth, for example 10 mm, by the depth of cut you set to get the number of steps.
- 2 Divides the depth by the number of steps to get the actual depth of cut (the depth of cut for each step is equal).

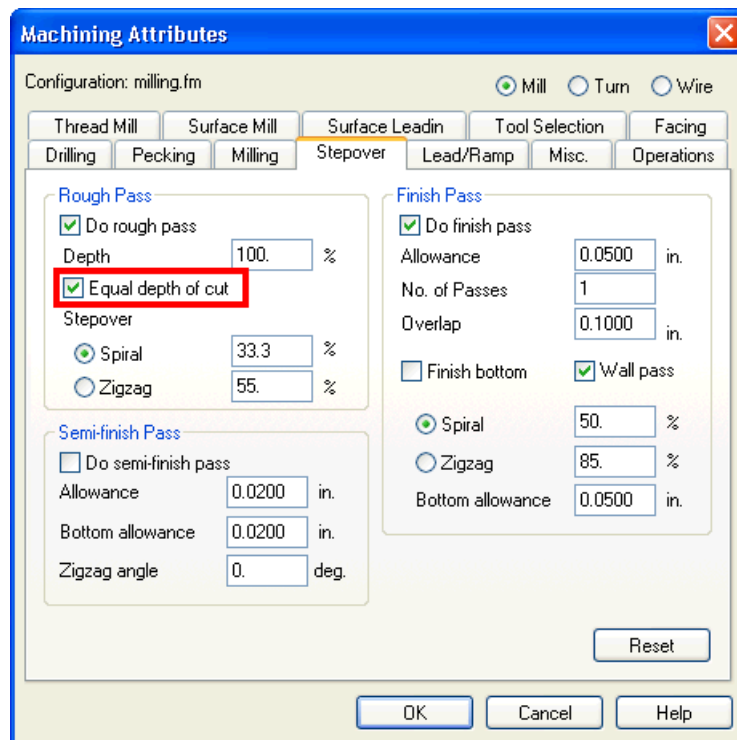
For example:

If the feature depth is 10 mm and you set a depth of cut of 3 mm, the number of steps is calculated as  $(10/3) + 1 = 4$  steps. The actual depth of cut is  $10 / 4 = 2.5$  mm. The steps are cut at 2.5 mm, 5 mm, 7.5 mm, and 10 mm.

There is a new option, **Equal depth of cut**, at feature level:



And at machine level:



The new option is selected by default, so that FeatureCAM keeps previous behavior.

To set an exact depth of cut, deselect **Equal depth of cut**.

Using the same example, with **Equal depth of cut** deselected:

If the feature depth is 10mm and you set a depth of cut of 3mm, the number of steps is calculated as  $(10/3) + 1 = 4$  steps.

FeatureCAM uses the actual depth of cut you set (3 mm) for each pass, then cuts any remainder depth. The steps are cut at 3 mm, 6 mm, 9 mm, and 10 mm.

These features support exact depth of cut:

- Profile boss
- Profile pocket
- Profile side
- Profile groove (Face groove only)
- Slot
- Rectangular pocket
- Step bore
- Counter bore hole
- Plain hole with drill/mill roughing
- Tapped hole with drill/mill roughing

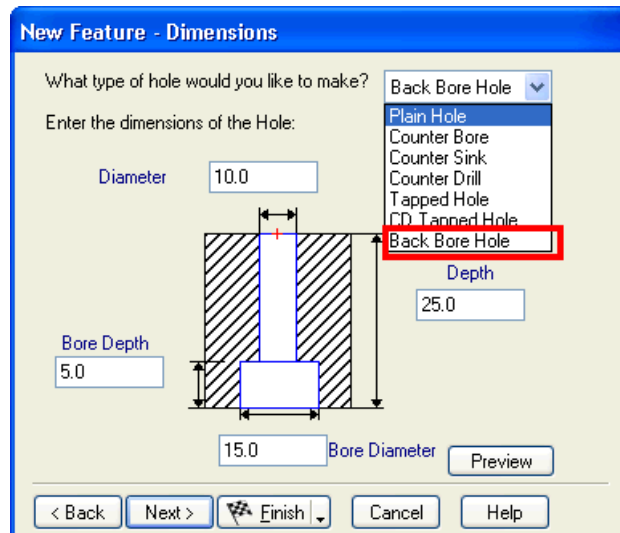
## 2.5D: Back Bore holes

In FeatureCAM 2012, you can create a back bore cycle (such as a Fanuc G87). The Back Bore hole type lets you machine back bore holes using a single Setup.

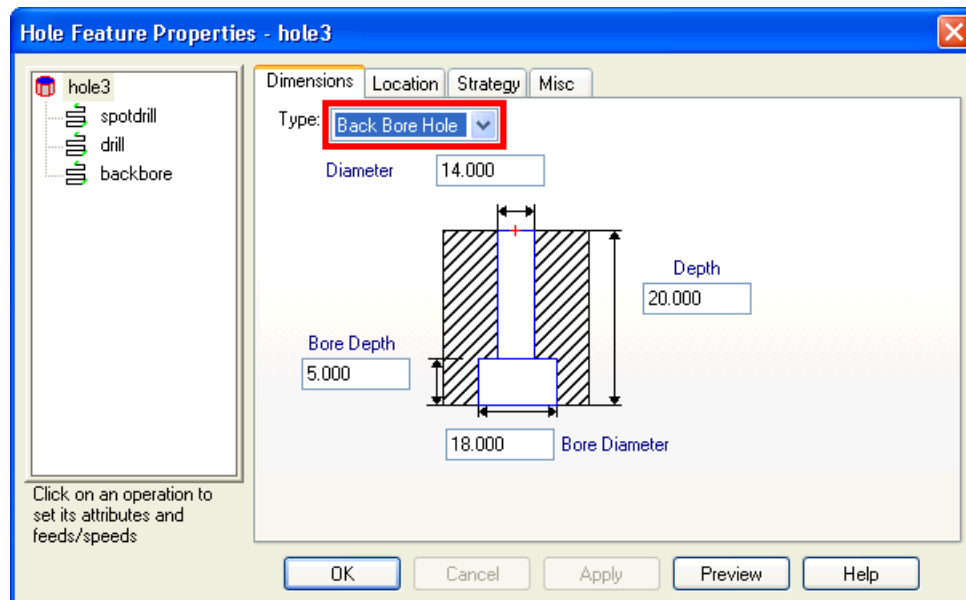


*You must use a post processor that supports back bore holes.*

**Back Bore Hole** is a new hole type available in the Feature wizard:

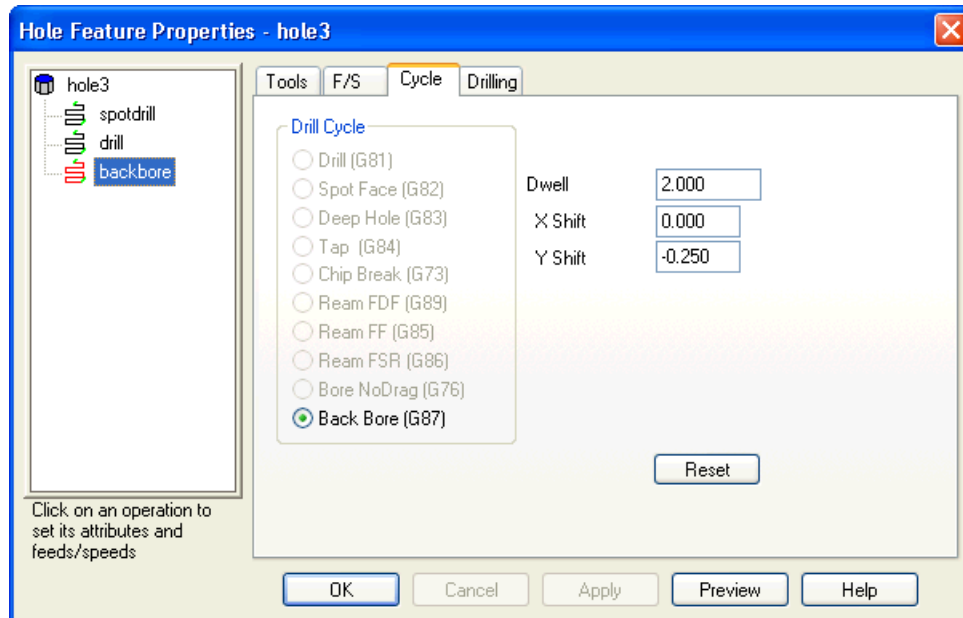


And on the **Dimensions** tab of the **Feature Properties** dialog:



You must enter the **Diameter**, **Depth**, **Bore Depth**, and **Bore Diameter** dimensions for the Back Bore Hole.

When you create a Back Bore Hole, a **backbore** operation is created, with a **Back Bore** cycle on the **Cycle** tab:



If your post processor supports it, you can select a **Drill Cycle** type and enter **Dwell**, **X Shift**, and **Y Shift** values.

---

## Simulation: Part compare extended

In FeatureCAM 2012, Part compare is available for many more part types. You can now use Part Compare with these modules:

- 4-axis
- 5-axis positioning
- 5-axis simultaneous
- Turning
- Turn/mill
- Wire

To do a part compare:

1 Right-click the solid part you want to use for the part compare and select **Use Solid As Part Compare Target** from the context menu.

2 Run a 3D or RapidCut simulation on the part.

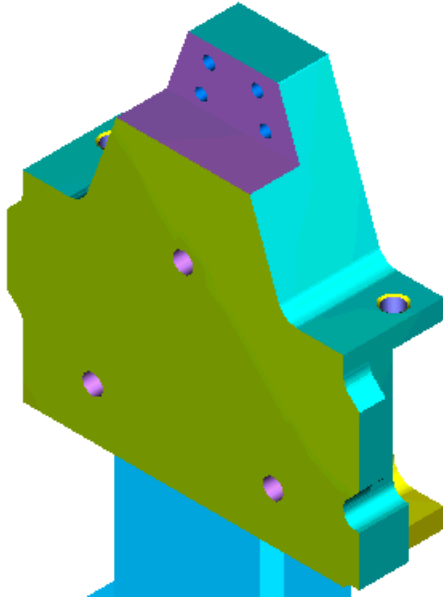


*You can not run a RapidCut simulation on turned, indexed, or multiple-fixture parts.*

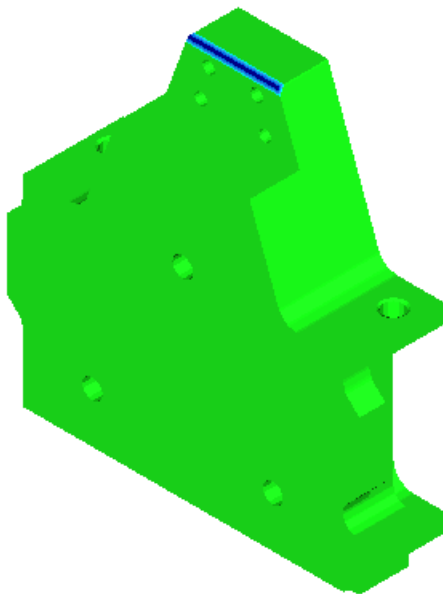
3 Select **View > Simulation > Show Part Compare** from the menu.

- 4 Check if there are any areas of your part that are shown in blue, these parts have not been cut or are undercut. Areas that have been cut correctly are shown in green.

This example shows a 5-axis part after a 3D simulation:

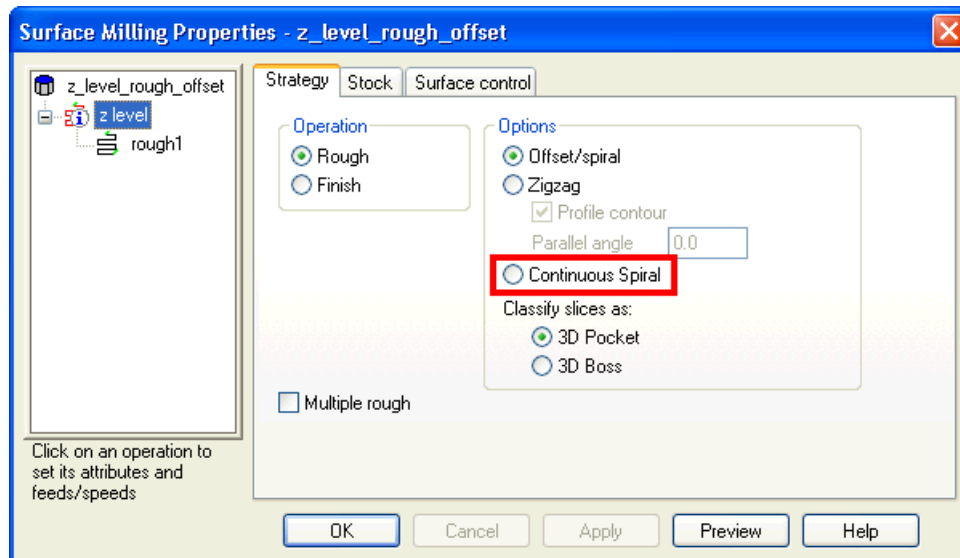


By using part compare, you can see that a chamfer has been missed when programming the part. This is the area shown in blue:



## 3D: Z-level spiral roughing

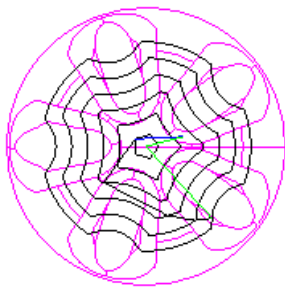
There is a new option on the **Strategy** tab for Z-level roughing, **Continuous Spiral**:



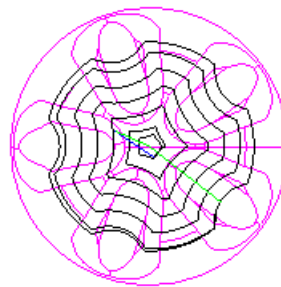
The **Continuous Spiral** option lets the tool move in a continuous, smooth motion, which reduces deceleration and acceleration, keeps a more constant chip load, and increases the tool life.

This blow mold base example shows a single Z-level rough slice:

Using the **Offset** option:

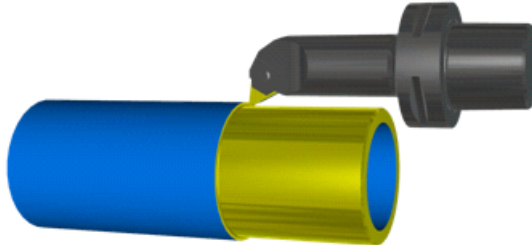


Using the **Continuous Spiral** option:



## Turn: Solid model as tool holder

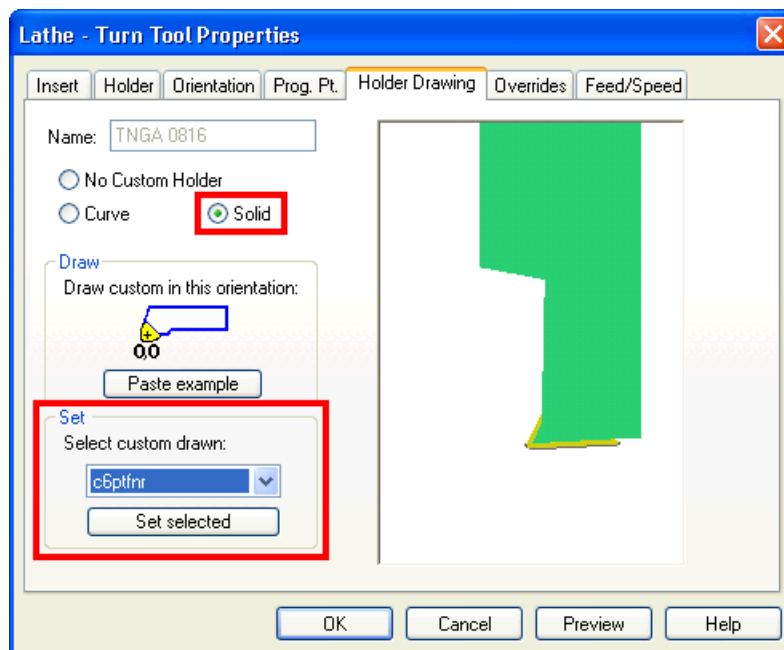
In FeatureCAM 2012 R3, you can set solid models as turning tool holder geometry. This gives much more accurate simulation of turning feature operations and collision checking, for example:



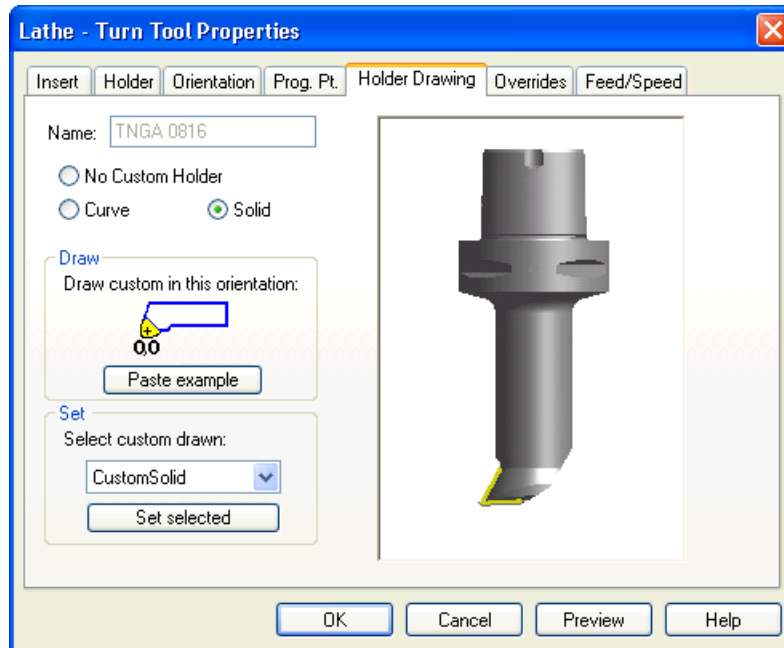
There is a new option, **Solid**, on the **Holder Drawing** tab for turn tools.

To set a solid model as a tool holder:

- 1 Import the solid model that represents the tool holder.
- 2 Position the solid model (see page 72) so that the point zero is in the correct place.
- 3 On the **Holder Drawing** tab in the **Tool Properties** dialog, select **Solid**, and in the **Set** section, select the name of the solid holder you imported and click the **Set selected** button.



- 4 The solid holder is displayed in the preview window on the tab and you can see the insert position relative to the holder:



The solid model holder is used in simulation.

## Solid tool holder example

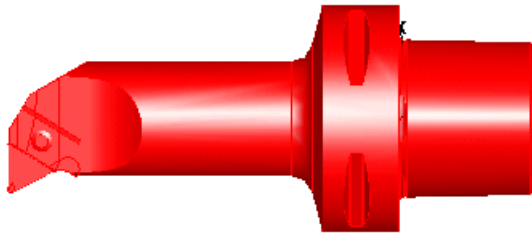
Here is an example of a FeatureCAM turning tool holder:



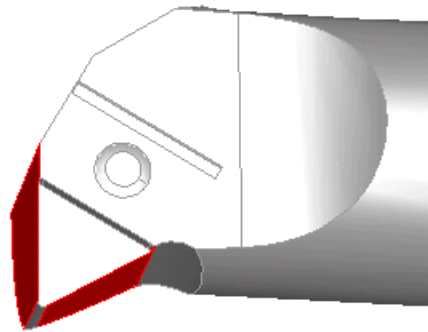
To use a solid model of the tool holder that represents it more accurately:

- 1 Import the solid model file of the tool holder. Click the **Cancel** button when the **Solid Wizard** opens.

In this example, a \*.x\_t solid file of the tool holder was imported.



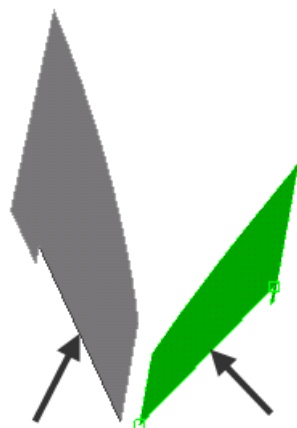
- 2 Optionally rename the solid in the **Part View**.
- 3 You must now make sure that the imported solid model tool holder is positioned correctly. First, find the zero point (mounting point) of the solid model holder. You can do this using the following method:
  - a Select the two faces that point towards the zero point, for example:



- b Hide everything that is unselected.

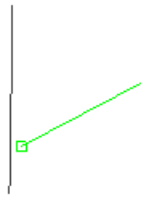


- c Using **Line from two points**, create two sketch lines along the mounting edge of the faces, for example:

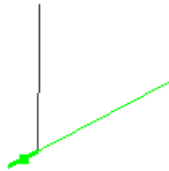


- d Hide the two faces to show only the two sketch lines.

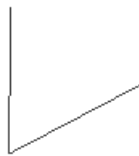
- e Use the **Trim/Extend** tool to extend the lines until they intersect:



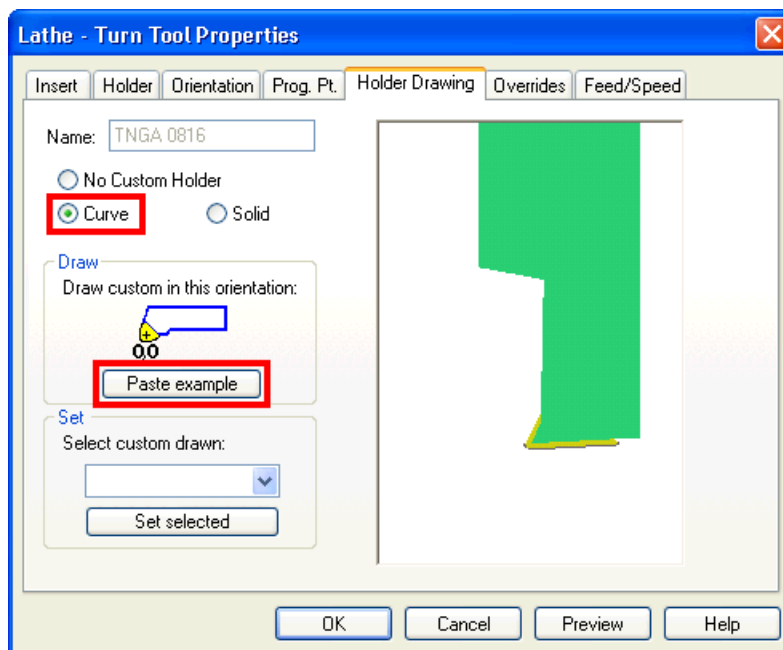
- f Use the **Clip** tool to clip the ends of the lines beyond the intersection:



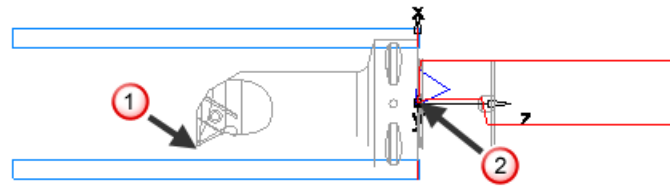
You are left with a sketch of the exact zero point to use to position the holder correctly.



- 4 You now need a reference point to move the holder to. You can do this using the following method:
  - a Select the **Holder Drawing** tab in the current **Tool Properties** dialog.
  - b Select **Curve** and click the **Paste Example** button:



You can now see the pasted curve and the solid holder in the Graphics window:



Turn off shading of the solid to compare them more easily.

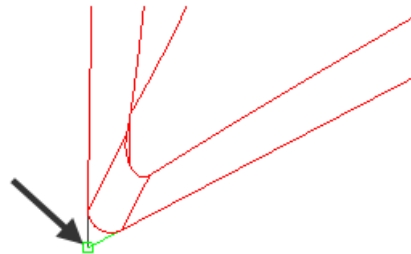
We need to move the zero point of the solid holder (1) that we sketched earlier, to the zero point of the pasted holder curve (2).

5 Use the Transform tool to do this:

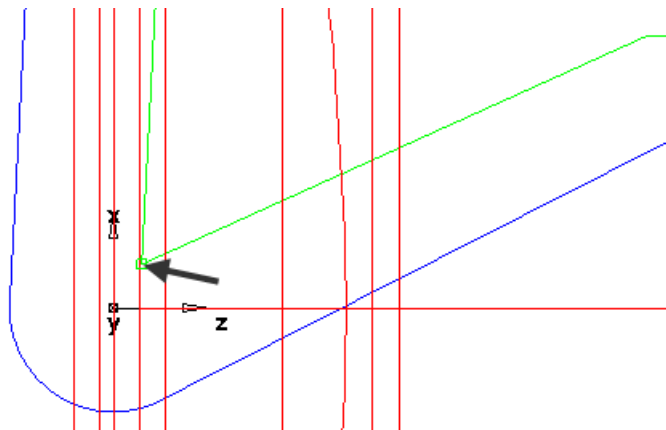
- a Select the solid holder.
- b Select **Edit > Transform** from the menu.

The **Transform** dialog opens.

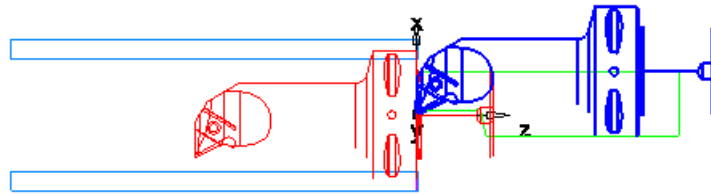
- c Select **Translate** and **Move** and under **Distance from 2 points**, click the **From** pick location button and select the zero point you sketched earlier:



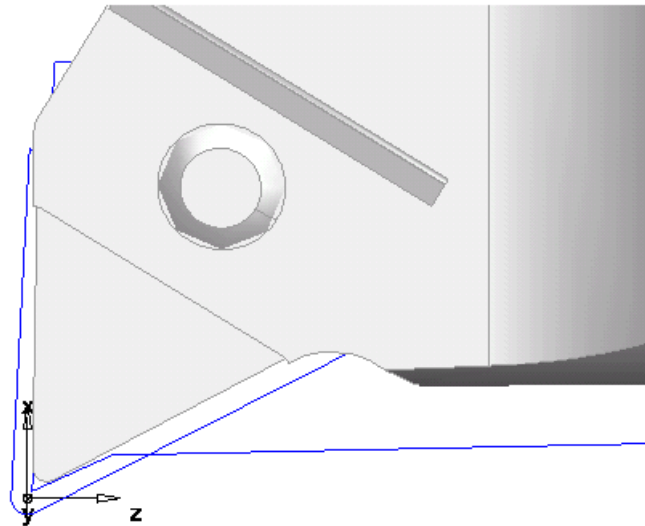
- d Click the **To** pick location button and select the zero point of the pasted holder curve:



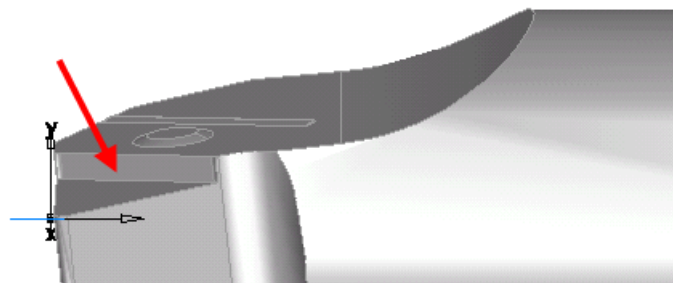
- e Click the **Preview** button to check the new location of the solid holder. The preview is shown in blue:



- f Click **OK** to accept the new location.
- 6 Turn the solid holder shading back on and you can see the position of the tool insert curve relative to the holder:

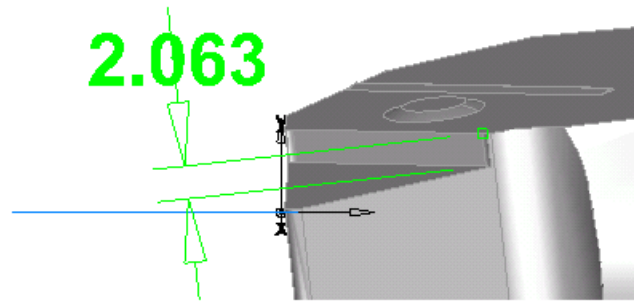


- 7 You can now delete the holder curves you pasted earlier.
- 8 If you look at the solid holder from the front, there is a recess where the insert sits:



- 9 You must take this recess into account to achieve accurate simulation. To do this:

- a Using the **Linear Distance** tool, take a measurement of the depth of the recess:



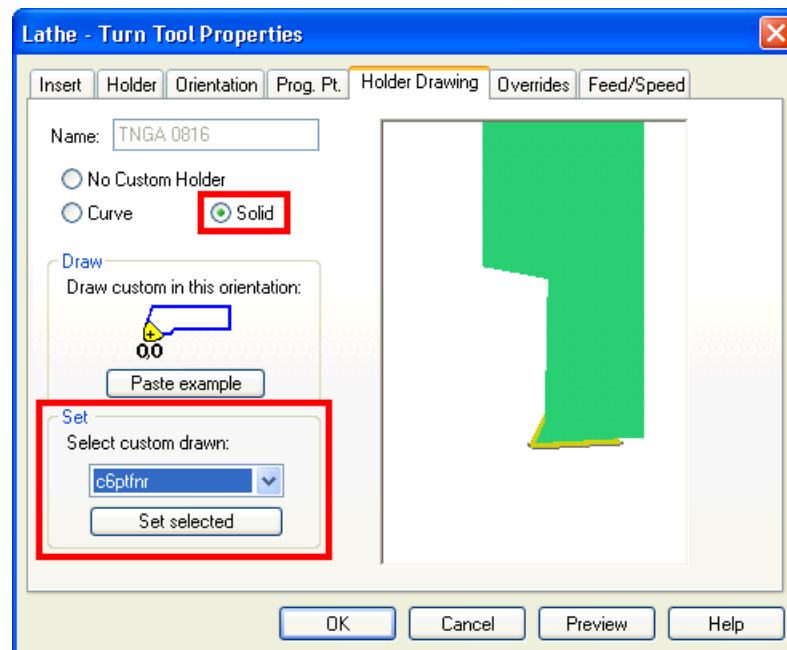
In this example, it is approximately 2mm.

- b Select the solid holder.
- c Select **Edit > Transform** from the menu to open the **Transform** dialog.
- d In the **XYZ Distance** section, enter 0 for X, -2 for Y, and 0 for Z to move the holder down by 2mm in the Y direction.
- e Click **OK**.

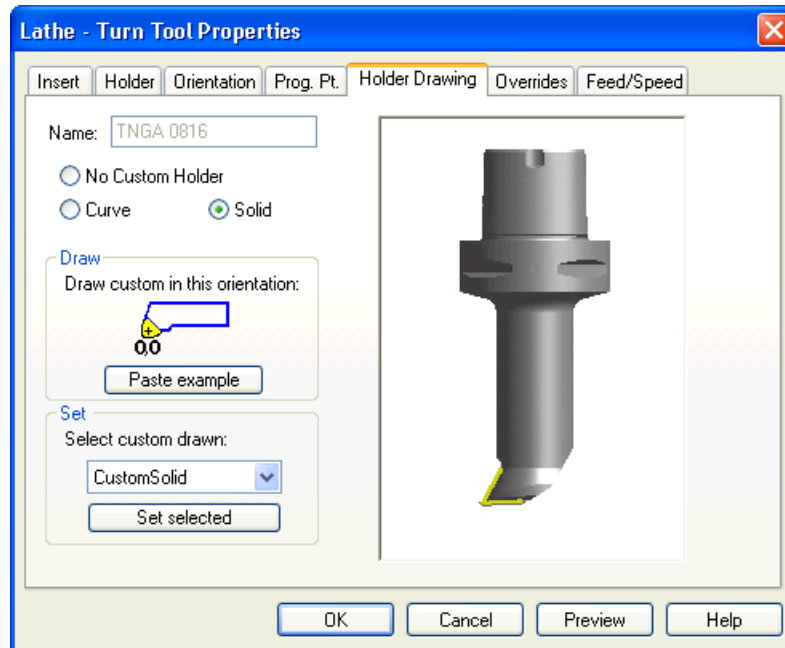
The solid holder is now in the correct position and ready to set as the holder on the tool's **Holder Drawing** tab.

- 10 To set the solid as the holder for the tool:

- a Select the **Holder Drawing** tab in the **Tool Properties** dialog.
- b Select **Solid**, and in the **Set** section, select the name of the solid holder you imported and click the **Set selected** button.

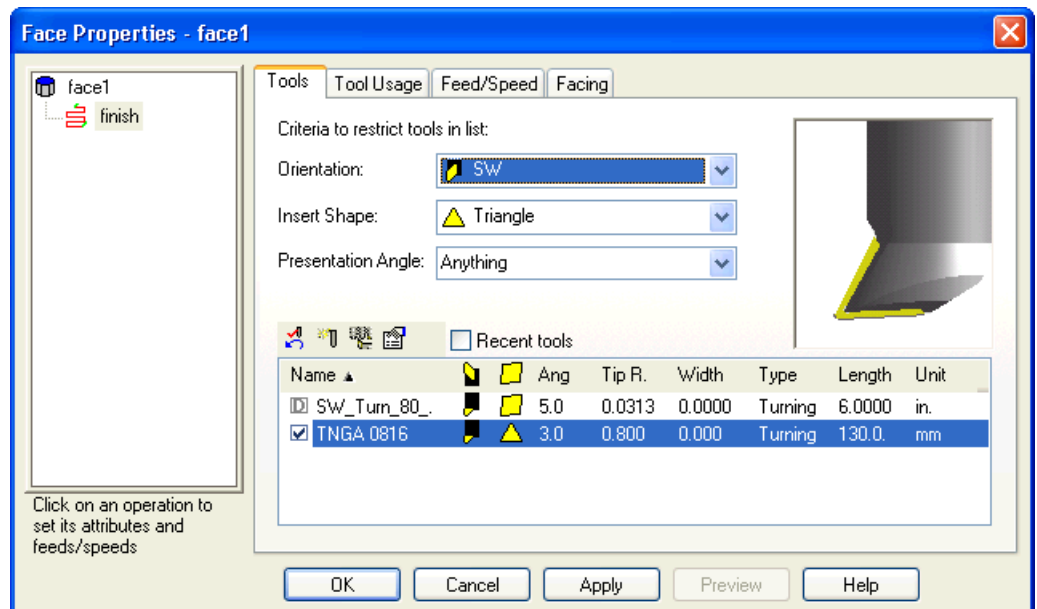


- c The solid holder is displayed in the preview window on the tab and you can see the insert position relative to the holder:



- d Click **OK** to accept the solid holder.

The preview on the **Tools** tab updates too:



- e Click **OK** to continue.

- 11 Hide the imported solid and show the part solid.

- 12 Run a 3D simulation. The solid model that you imported is now used as the tool holder and the simulation is more accurate:



---

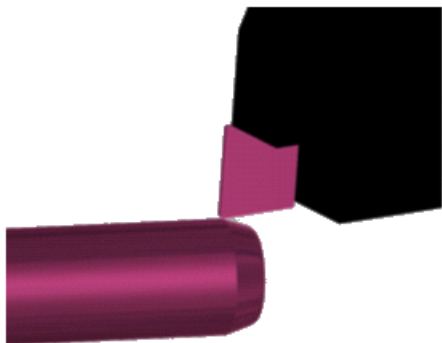
## Turn: Better tool holder graphics

In FeatureCAM 2012 R3, the graphical representation of tool holders is improved.

The material behind the insert on the tool holder is now shown, for example:



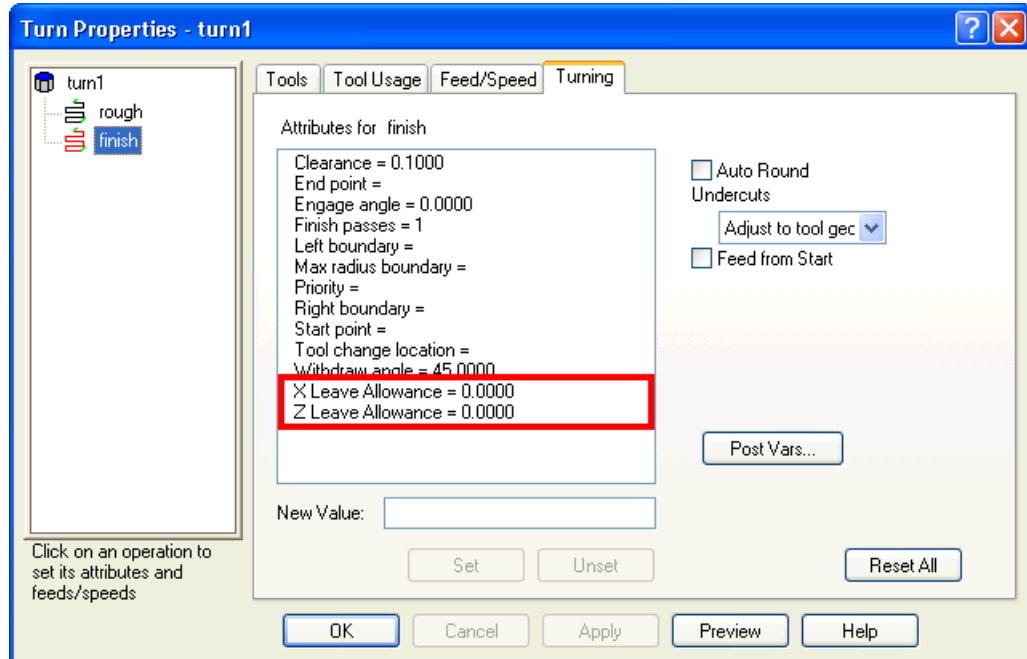
In previous versions, this material was not shown, for example:



The new tool holder graphics increase the realism of simulations with the default tooling.

# Turning leave allowance

In FeatureCAM 2012, you can set **X Leave Allowance** and **Z Leave Allowance** attributes for a Turn feature finish pass, on the **Turning** tab of the **Turn Properties** dialog:



This means you can undercut or overcut a part without having to remodel it. This is essential for situations such as fitting allowance.

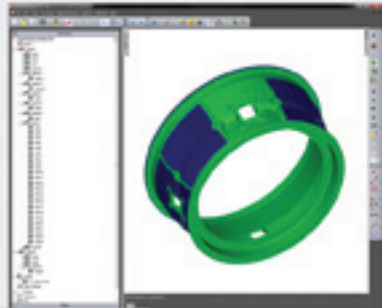
**X Leave Allowance** is the amount of material to leave after the finish pass, in the X direction, and **Z Leave Allowance** is the amount of material to leave after the finish pass, in the Z direction.

The **X Leave Allowance** and **Z Leave Allowance** values can be positive or negative, but both must have the same sign (both positive or both negative).



# FeatureCAM 2012

FeatureCAM 2012



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)