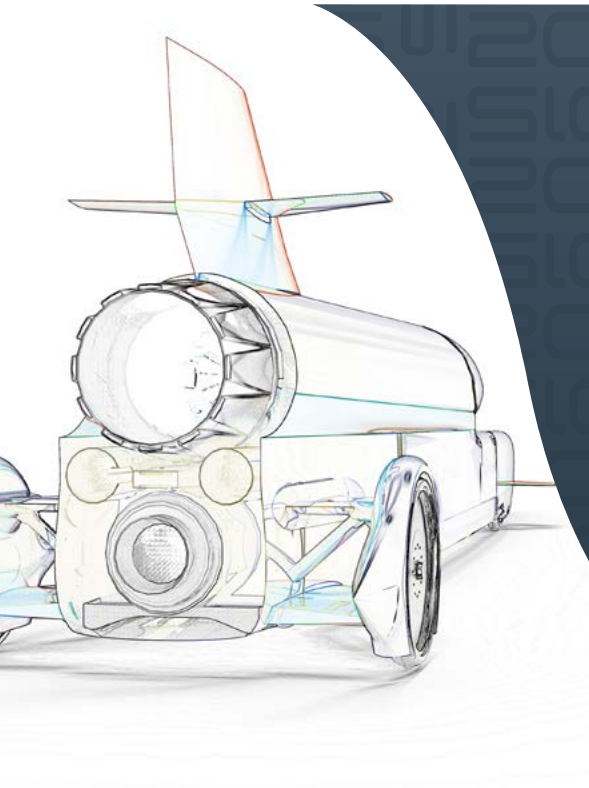


Delcam Advanced Manufacturing Solutions

FeatureCAM 2015 R3

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

www.featurecam.com

What's New

FeatureCAM 2015 R3

What's New

Copyright © 1995-2015 Delcam Ltd. All rights reserved.

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

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

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

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

Acknowledgements

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

FeatureCAM Version: 2015 R3 Date: 26 March 2015 12:22

Contents

What's New 1

What's new in FeatureCAM 2015 R3 2

User Interface 4
- Improved icons 5
- Selecting objects 6
- Inserting Part Library features 7
- Pasting features using polar coordinates 9
- Deleting unused curves 10

Importing 11
- Importing SolidWorks 2015 files 12
- Importing SolidEdge ST7 files 13

Milling 14
- Creating Face features using IFR 15
- Automatic tool selection 17
- Simulating tool holders 19
- Machine movement limits 20

Turning and Turn/Mill 22
- Indexing using a stock solid 23
- Turnmilling program point 25

5-axis machining 26
- C-axis indexing 27
- 5-axis engraving 29

Machine Design 31
- Specifying machine movement limits 32
- Testing machine movements 33
- Simulating mini-turrets 35

What's new in FeatureCAM 2015 R2 36

User Interface 39
- Changing the point size 40
- Customizing the snapping cursor 41
- Measuring curve length 42
- Creating internal gears 43
- Combining solids (SOLID) 44
- Hiding rapid moves in centerline simulations 45
- Chip recognition size 46
- Recognizing features on large parts 47

Milling 48
- Automatic tool selection 49

Vortex approaches flats from outside stock 51
Output options for 2D NT toolpaths 53
Previewing the toolpath point distribution 55
Tool pecking depths 56
Changing the Posting tolerance 57
Helical side finish 58
Turning 59
Removing undercuts in no-drag turning features 60
Controlling steady rests 61
Simulating bar stock 63
Custom turret names (MTT) 64
XBUILD 65
Generating Post documentation 65
Using macros in the post processor 66
Add-ins and extensions 67
Using Setup Sheets 68
FeatureCAM to CAMplete TruePath add-in 69
Probing update options 71
Support for Microsoft SQL server 2014 71
Machine Simulation Design 72
Multi-tool blocks 73
Selecting the tool block 76
Protecting solids in Machine Design documents 78

What's new in FeatureCAM 2015 R1 79

User interface improvements 83
Installing FeatureCAM 84
SQL server authentication 85
Accessing machining attributes 86
Customizing the Toolbox 86
Adding new stock materials 88
Improved feature recognition 89
Recognizing Face features 90
Snapping modes 91
Creating solids 92
Polygonal curves 94
Creating gears 96
Command line options 98
Selecting surfaces 99
Milling Improvements 100
Tool holder collision avoidance 101
Machine maximum stock 103
Isoline and flowline features 104
Configuring counterbore operations 106
Default auto-chamfer tool 108
Setting toolpath boundaries 109
Creating 5-axis patterns 111
Turning improvements 113
Improved machining times for undercuts 114

Synchronizing operations 115
Turning curves 117
Tool properties for multi-turret machining 119
Multiple part catchers 120
Wire EDM improvements 121
Improved reliability for no-core machining 122
Selecting cutting data 123
Importing 125
Using FeatureCAM with Inventor 2015 126
Importing ProE files 127
Importing JT files 127
Importing PDF files 128
Add-ins and extensions 129
Using add-ins 130
The Add-in Library 132
Turn-curve tolerance 134
Exporting NC programs to NCSIMUL 137
Extended machine support 138
XBUILD improvements 139
Custom Formats 140
Customizing cutting data for Wire EDM 141
Saving CNC files 144
Configuring reserved words 146
Using the Reserved Word dialog 147

Index **149**

What's New

FeatureCAM issues a major enhancement release in September, January, and May every year. For details of our developments in the last twelve months, see:

What's New in FeatureCAM 2015 R3 (see page 2)

What's New in FeatureCAM 2015 R2 (see page 36)

What's New in FeatureCAM 2015 R1 (see page 79)

Also, visit our website at http://www.delcam.tv/lz/ for up-to-the-minute news and videos; and visit http://updates.delcam.com to download our latest service packs and enhancements.

What's new in FeatureCAM 2015 R3

FeatureCAM 2015 R3 contains the following new features and enhancements:

User interface

- **Improved icons** (see page 5) — Several icon graphics across FeatureCAM have been updated.
- **Selecting objects** (see page 6) — There is a new **Select Partial** mode.
- **Inserting Part Library features** (see page 7) — You can now use the **New Feature** wizard to insert Part Library features into a document.
- **Pasting features using polar coordinates** (see page 9) — You can now use polar coordinates to locate features in the **Paste Special** wizard.
- **Deleting unused curves** (see page 10) — There is a new add-in that you can use to identify all unused curves in a model and delete them.

Importing

- **Importing SolidWorks 2015 files** (see page 12) — You can now import SolidWorks 2015 files into FeatureCAM.
- **Importing SolidEdge ST7 files** (see page 13) — You can now import SolidEdge ST7 files.

Milling

- **Creating Faces features using IFR** (see page 15) — You can now automatically round the corners of Face features that are created using Interactive Feature Recognition.
- **Automatic tool selection** (see page 17) — You can now specify an additional clearance on the automatic tool selection to ensure the tool holder does not collide with the stock.
- **Simulating tool holders** (see page 19) — You can now simulate complicated tool holders more accurately.
- **Machine movement limits** (see page 20) — You can now display a warning message when the machine exceeds its limits of movement during machine simulation.

Turning and Turn/Mill

- **Indexing using a stock solid** (see page 23) — You can now calculate the index height directly from the stock solid, instead of calculating it above a square bounding box.
- **Setting the program point for turnmilling tools** (see page 25) — You can now specify the program point for turnmilling tools.

5-axis machining

- **C-axis indexing** (see page 27) — You can now specify the C-axis position of the part in the machine at the start of an operation.
- **5-axis engraving** (see page 29) — You can now create 5-axis engraving features with the tool axis normal to the surface.

Machine Design

- **Specifying machine movement limits** (see page 32) — You can now set the limits of movement for solids in a Machine Design file.
- **Testing machine movements** (see page 33) — You can now use machine jogging to simulate the movement of solids in a Machine Design document without having to use an FM file.
- **Simulating mini-turrets** (see page 35) — You can now simulate mini-turrets, which are tools with multiple inserts where the tool rotates around the b-axis to access each tool.

User Interface

FeatureCAM 2015 R3 contains the following changes and improvements to the user interface:

- **Improved icons** (see page 5) — Several icon graphics across FeatureCAM have been updated.
- **Selecting objects** (see page 6) — There is a new **Select Partial** mode.
- **Inserting Part Library features** (see page 7) — You can now use the **New Feature** wizard to insert Part Library features into a document.
- **Pasting features using polar coordinates** (see page 9) — You can now use polar coordinates to locate features in the **Paste Special** wizard.
- **Deleting unused curves** (see page 10) — There is a new add-in that you can use to identify all unused curves in a model and delete them.

Improved icons

Several icon graphics across FeatureCAM have been updated to improve the user interface, including:

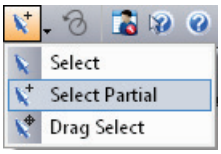
- **Feature Properties** dialog
- **Part View**
- **Part Library**
- **Tombstone Process Plan** dialog

Selecting objects

There is a new **Select Partial** mode, which enables you to box-select items by partially selecting them. This improves work-flow when selecting multiple items, and enables you to select items more easily in complicated documents.

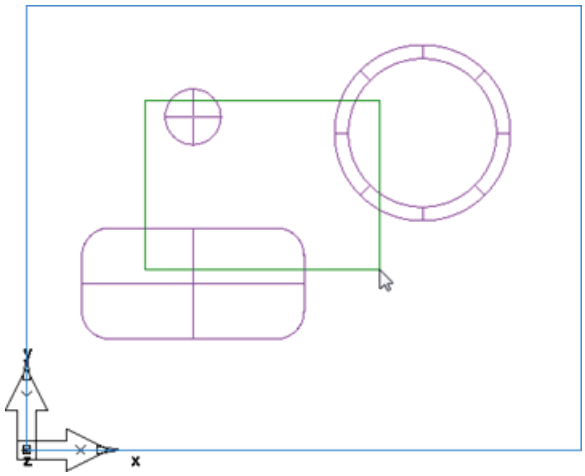
To enable the new **Select Partial** mode:

- In the **Standard** toolbar, in the **Select Menu**, select **Select Partial**.



- Select the **Edit > Select > Box Select Partial** menu option.

Click and drag to select objects. You do not need to enclose an object to select it. In the example below, all three features are selected:



Inserting Part Library features

You can now use the **New Feature** wizard to insert Part Library features into a document.

Adding features to the Part Library

To add a feature to the Part Library:

1. Select the **Construct > Part Library** menu option.

   The **Part Library** dialog is displayed.
2. Select a feature in the **Tree View** or graphics window.
3. In the **Part Library** dialog, click **Add Selected**.

   The selected feature is added to the list in the **Part Library** dialog.
4. Click **OK** to close the dialog.

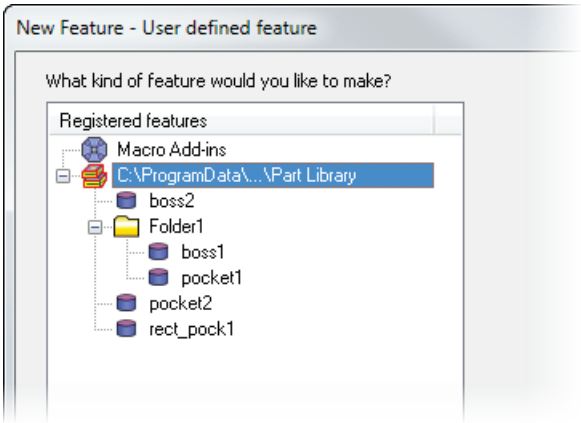
Inserting features from the Part Library into the document

To use the Part Library in the **New Feature** wizard:

1. Click the **Features** step in the **Steps** panel to display the **New Feature** wizard.
2. In the **New Feature** wizard, under **From Feature**, select **User** and click **Next**.

   The **User defined feature** page is displayed.

   The Part Library features are displayed in the **Registered features** list.



3. Select a feature in the list and click **Next**.

   The **Paste Special** dialog is displayed.
4. Use the **Paste Special** dialog to insert the selected feature into the document, then click **Finish** to close the dialog.
5. The **User defined feature** page of the **New Feature** wizard is displayed.
6. Use the wizard to insert more features from the Part Library, or click **Cancel** to close the wizard.

Pasting features using polar coordinates

You can now use polar coordinates to locate features in the **Paste Special** wizard. This gives you more control when duplicating features and when using the Part Library.

You can only use polar coordinates to locate individual features.

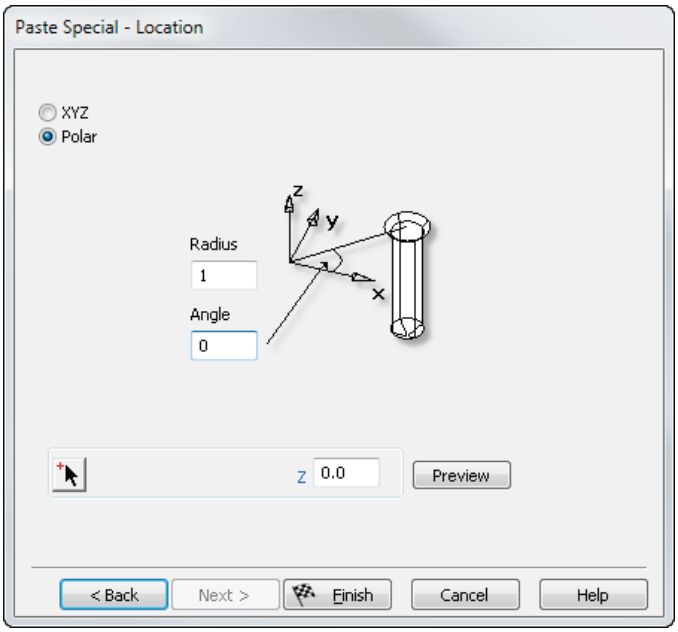
To duplicate a feature into a new location using polar coordinates:

1. Right-click a feature in the **Part Tree**, and select **Copy**.
2. Select the **Edit > Paste Special** menu option.

   The **Paste Special** wizard is displayed.
3. Select **Paste the clipboard contents. Select a new location** and click **Next**.

   The **Reference** page of the **Paste Special** wizard is displayed.
4. Click **Next**. You do not need to specify a reference location.

   The **Location** page of the **Paste Special** wizard is displayed.
5. Select the new **Polar** option.



6. Enter the **Radius** and **Angle** to offset the new feature from the setup location.
7. Enter the **Z** value to specify the Z height of the feature above the setup.
8. Click **Preview** to display a preview of the new feature in the graphics window.
9. Click **Finish** to create the new feature and close the wizard.

Deleting unused curves

There is a new delete_curves.bas add-in that you can use to identify all unused curves in a model and delete them.

To delete all unused curves in a document:

1. Load the delete_curves.bas add-in using the **Macro Add-ins** dialog.
2. In the **Utilities** toolbar, click **deletecurves**.
3. The **Curve Information** dialog is displayed, which contains a list of unused curves in the document.
4. Click **OK** to delete all unused curves and close the dialog.

Importing

FeatureCAM 2015 R3 contains these changes and improvements to importing files from external applications:

- **Importing SolidWorks 2015 files** (see page 12) — You can now import SolidWorks 2015 files into FeatureCAM.
- **Importing SolidEdge ST7 files** (see page 13) — You can now import SolidEdge ST7 files.

Importing SolidWorks 2015 files

You can now import SolidWorks 2015 files into FeatureCAM.

To import a SolidWorks file:

1. Select the **File > Import** menu option.

   The **Import** dialog is displayed.
2. Browse to the folder containing the file you want to open.
3. In the **Files of type** list, select **SolidWorks (*.sldprt;*.sldasm)**.

   Only SolidWorks documents are displayed in the **Import** dialog.
4. Select a file to display a preview image on the right of the dialog.
5. Click **Open** to import the file and close the dialog.

   The **Import Results** wizard is displayed, which you can use to specify the setup location and stock size, and to recognize features.

Importing SolidEdge ST7 files

You can now import SolidEdge ST7 files.

To import a SolidEdge file:

1. Select the **File > Import** menu option.

   The **Import** dialog is displayed.

2. Browse to the folder containing the file you want to open.

3. In the **Files of type** list, select **SolidEdge (*.par;*.psm:*.asm)**.

   Only SolidEdge files are displayed in the **Import** dialog.

4. Select a file to display a preview image on the right of the dialog.

   Click **Open** to import the selected file and close the dialog.

   The **Import Results** wizard is displayed, which you can use to specify the setup location and stock size, and to recognize features.

Milling

FeatureCAM 2015 R3 contains these changes and improvements to Milling:

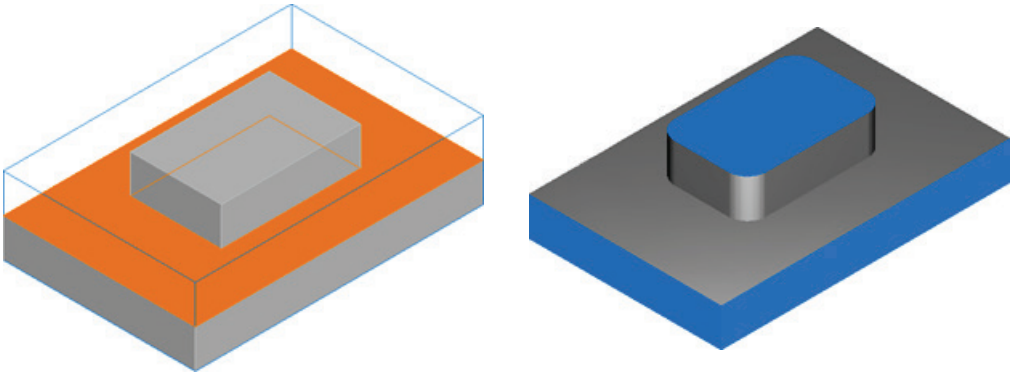
- **Creating Face features using IFR** (see page 15) — You can now automatically round the corners of Face features that are created using Interactive Feature Recognition.
- **Automatic tool selection** (see page 17) — You can now specify an additional clearance on the automatic tool selection to ensure the tool holder does not collide with the stock.
- **Simulating tool holders** (see page 19) — You can now simulate complicated tool holders more accurately.
- **Machine movement limits** (see page 20) — You can now display a warning message when the machine exceeds its limits of movement during machine simulation.

Creating Face features using IFR

You can now automatically round the corners of Face features that are created using Interactive Feature Recognition.

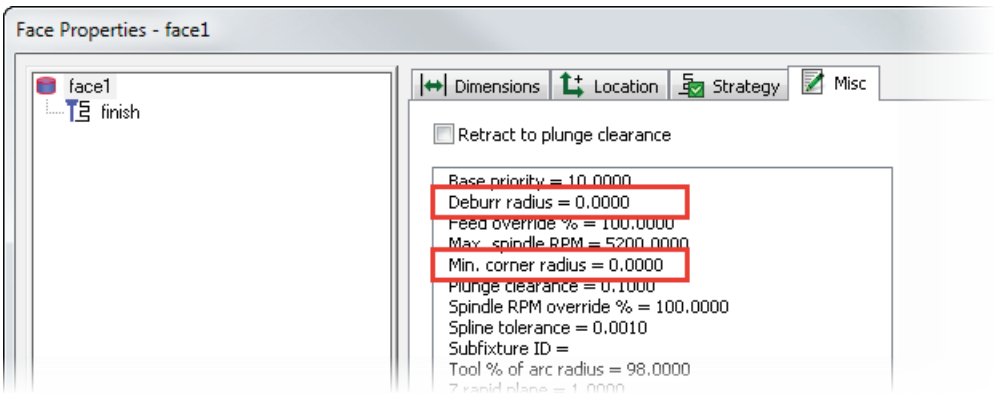
This enables you to use a Face feature instead of a Side feature in some situations, which can reduce the machining time by reducing air cutting. This is especially useful when using cutter compensation and partline programming.

In the example below, a Face feature is created on the orange surface, and a deburr radius is applied to round the outside corners of the feature during machining:



To specify a corner radius for a Face feature:

1. Create a Face feature using Interactive Feature Recognition.
2. In the **Face Properties** dialog, in the **Misc** tab, select one of the new options.



Use the **Deburr radius** option to round the outside corners of the feature.

Use the **Min corner radius** option to round the inside corners of the feature.

3 In the **New Value** field, enter a new value and click **Set** to override the selected option.

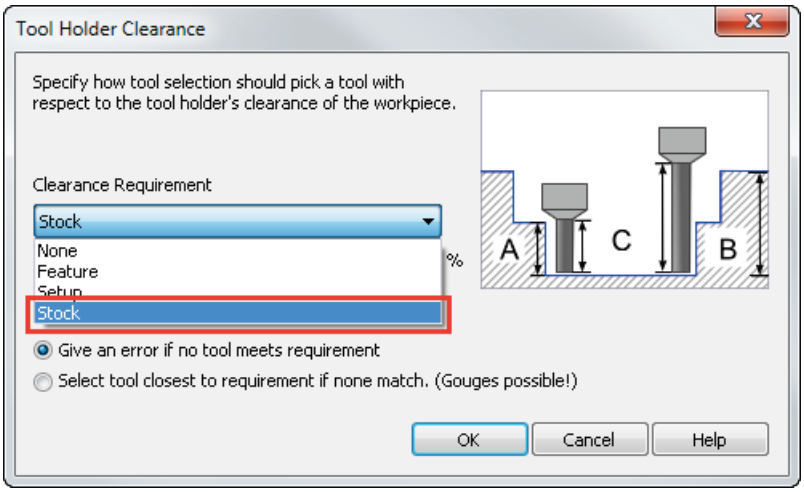
4 Click **OK** to accept your changes to the feature and close the dialog.

*For Face features with multiple operations, you can set the **Min. corner radius** option separately for the rough operation and for each finish pass.*

Automatic tool selection

In FeatureCAM 2015 R2, the **Tool Holder Clearance** dialog (see page 49) was added, which enables you to specify an additional clearance on the automatic tool selection to prevent tool holder gouges.

There is a new **Stock** option in the **Clearance Requirement** list. Select this option to ensure the tool is long enough for the tool holder to clear the total depth into the stock.



No additional clearance:

Clearance above the stock:

To specify a tool holder clearance above the stock for automatic tool selection:

1. Select the **Manufacturing > Machining Attributes** menu option to display the **Machining Attributes** dialog.
2. On the **Tool Selection** tab of the **Machining Attributes** dialog, click **Tool Holder Clearance**.

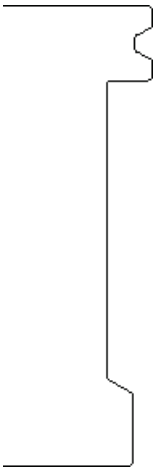
   The new **Tool Holder Clearance** dialog is displayed.
3. In the **Clearance Requirement** list, select **Stock**.
4. Click **OK** to close the dialog.

Simulating tool holders

You can now simulate complicated tool holders more accurately. This enables you to create better toolpaths that can access more material and check for gouges more accurately when using automatic tool holder clearance.

Previously, the shape of the tool holder was approximated using either a cylinder or a cone.

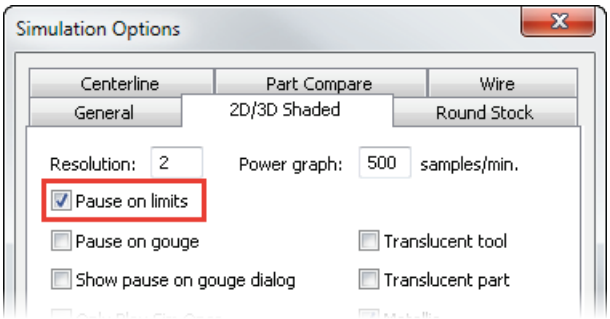
The tool holder shape does not take into account undercuts, for example:

If the tool holder has this curve:	The toolpaths are calculated with this tool holder shape:
	

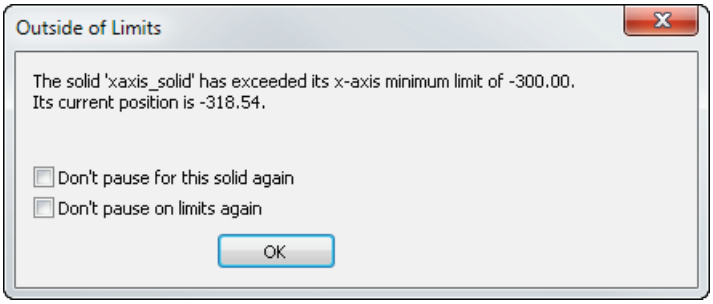
Machine movement limits

You can now display a warning message when the machine exceeds its limits of movement during machine simulation.

There is a new **Pause on limits** option in the **2D/3D Shaded** tab of the **Simulation Options** dialog. Select this option to pause the machine simulation if the machine moves outside the limits specified in the MD file.



A new message dialog is displayed when the machine moves outside the specified limits.



The **Outside of Limits** message dialog displays:

- The solid name that has exceeded its limits.
- The axis of movement.
- The specified limit.
- The solid's current position.

You can select these options in the **Outside of Limits** message dialog:

- **Don't pause for this solid again** — Select this option to continue the machine simulation without pausing when this solid exceeds its limits.
- **Don't pause on limits again** — Select this option to run machine simulations without pausing when any solids exceed their limits. This is the same as deselecting **Pause on limits** in the **Simulation Options** dialog.

To specify a machine's limits of movement (see page 32), you need to edit the Machine Design file.

Turning and Turn/Mill

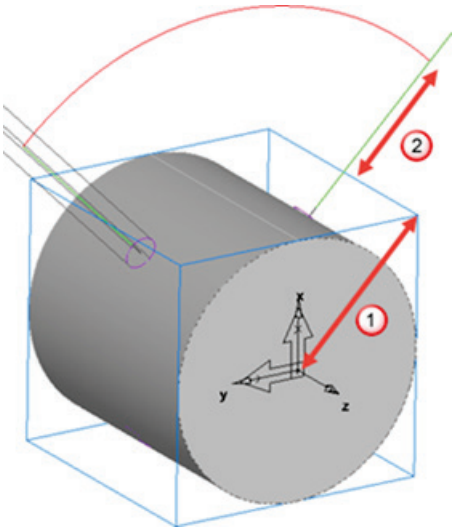
FeatureCAM 2015 R3 contains these changes and improvements to Turning and Turn/Mill:

- **Indexing using a stock solid** (see page 23) — You can now calculate the index height directly from the stock solid, instead of calculating it above a square bounding box.
- **Setting the program point for turnmilling tools** (see page 25) — You can now specify the program point for turnmilling tools.

Indexing using a stock solid

You can improve machining times for turn/mill parts with stock solids by reducing the distance above the part at which the tool indexes. There is a new **Calculate index radius from solid stock outline** option that enables you to calculate the index height directly from the stock solid, instead of calculating it above a square bounding box:

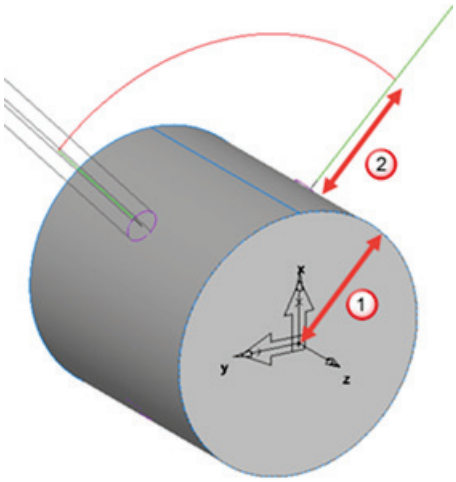
Calculate index radius from solid stock outline off:



1 — Distance to bounding box

2 — **Z rapid level** attribute value

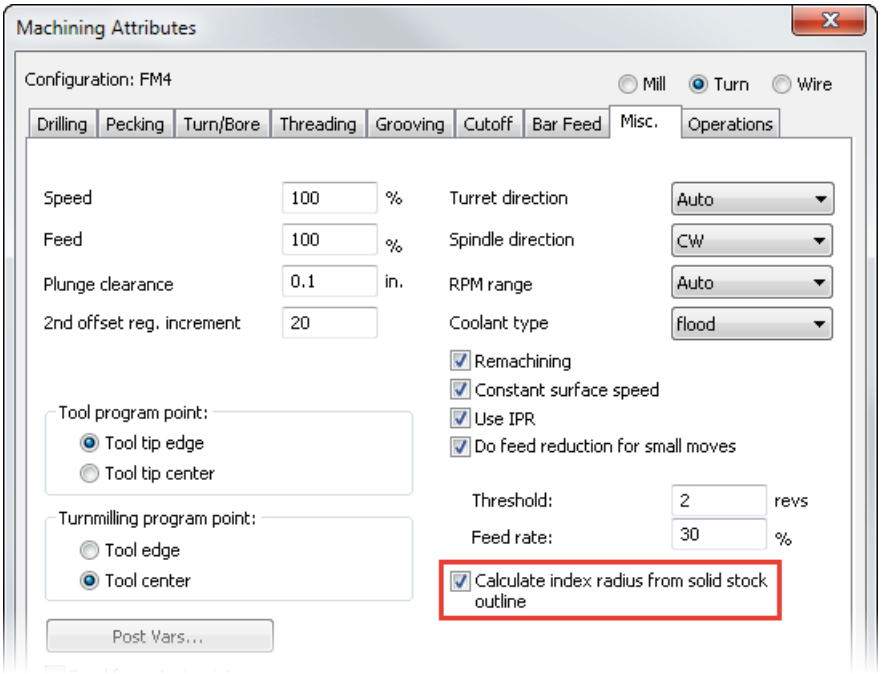
Calculate index radius from solid stock outline on:



1 — Distance to stock boundary

2 — **Z rapid level** attribute value

The new option is displayed on the **Misc** tab of the **Machining Attributes** dialog.

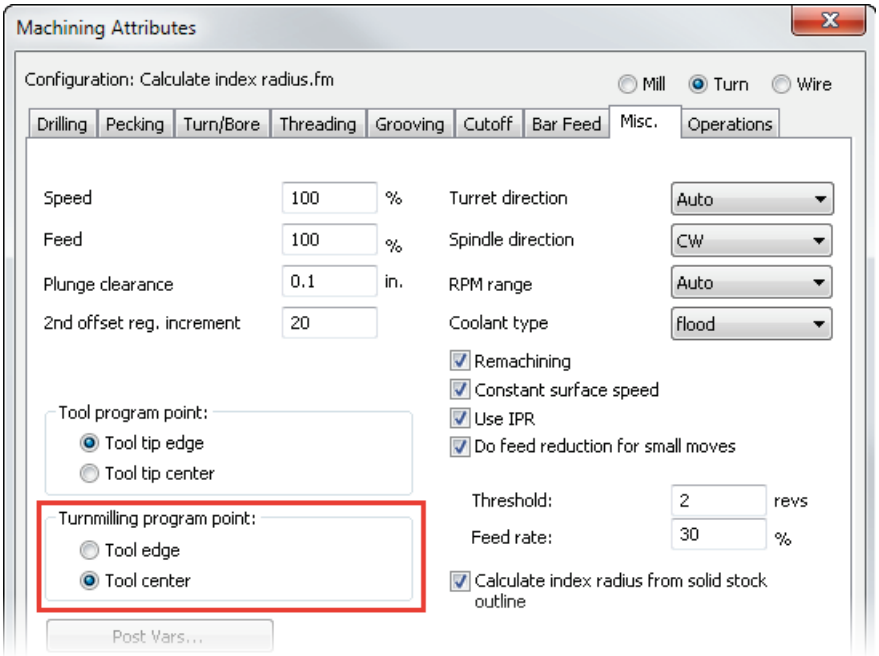


Turnmilling program point

You can now specify the program point for turnmilling tools. This enables you to touch-off the tool at the edge, and specify the insert radius compensation at the machine instead of in FeatureCAM. Previously, this option was available for turning tools only.

In the **Machining Attributes** dialog, on the **Misc** tab, select an option under **Turnmilling program point**:

- **Tool tip edge** — Select this option to adjust the tool program point by the tool radius in the NC code.
- **Tool tip center** — Select this option to adjust the tool program point by the tool radius at the machine.



This option does not affect turnmilling tools that are used for milling.

To use a turnmilling tool, select the **Turnmilling** option in the **Strategy** tab of the **Feature Properties** dialog for a Turn feature.

5-axis machining

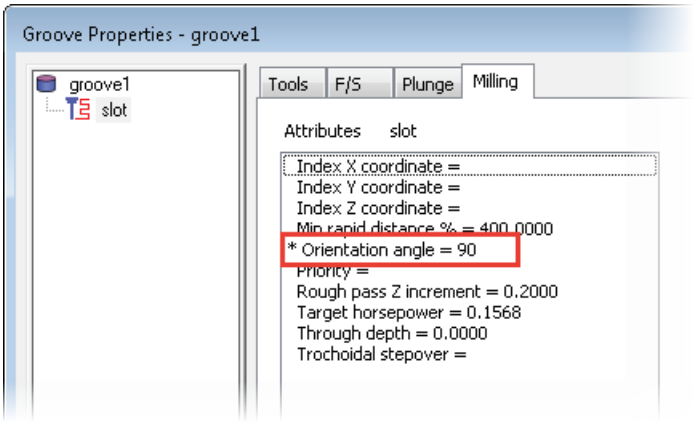
FeatureCAM 2015 R3 contains these changes and improvements to 5-axis machining:

- **C-axis indexing** (see page 27) — You can now specify the C-axis position of the part in the machine at the start of an operation.
- **5-axis engraving** (see page 29) — You can now create 5-axis engraving features with the tool axis normal to the surface.

C-axis indexing

You can now specify the C-axis position of the part at the start of an operation. This is useful for large parts where the machine has limited travel, or to prevent machine collisions.

There is a new **Orientation angle** option in the **Milling** or **Drilling** tab of the **Feature Properties** dialog.



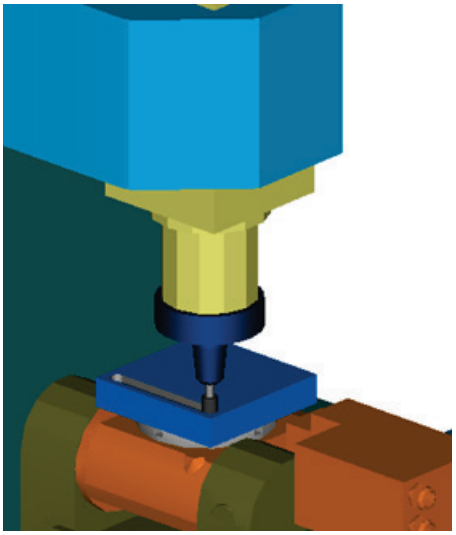
Use this option to specify the rotation of the X and Y axes about the Z-axis. This option only applies if the machine tool starts at the singularity (where the machine tool's Z-axis is aligned with the setup's Z-axis).

If the machine tool is not at the singularity, you can specify the C-axis orientation using these methods:

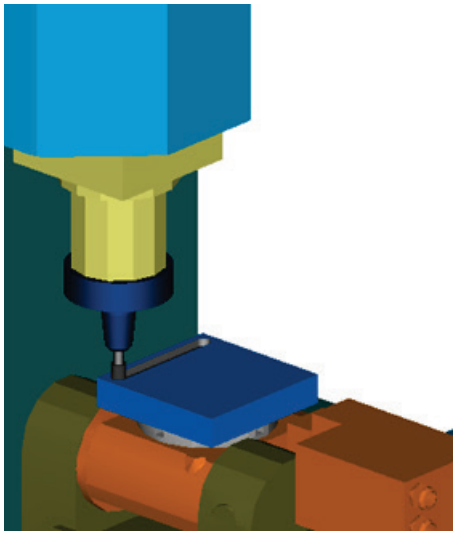
- Use the **Alternative 5-axis position** option to specify a C-axis orientation of either 0 or 180 degrees.
- Use the **Use Origin of this Setup as the Touch-off Point** option in the **5 Axis Fixture Location** dialog. This method applies the C-axis orientation to all setups in the part, instead of to individual operations.

For example:

With an orientation angle of 0, the groove is cut in the machine's Y direction.

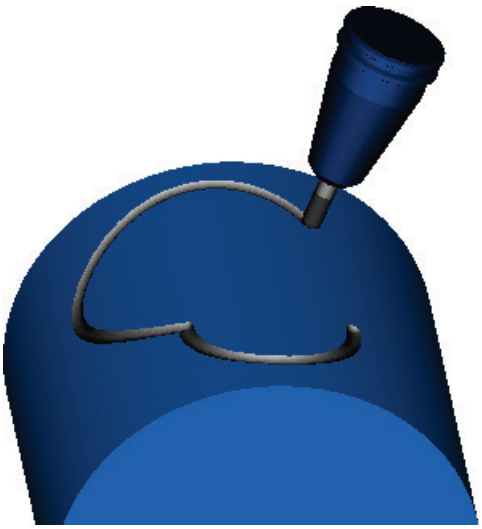


With an orientation angle of 90, the groove is cut in the machine's X direction.



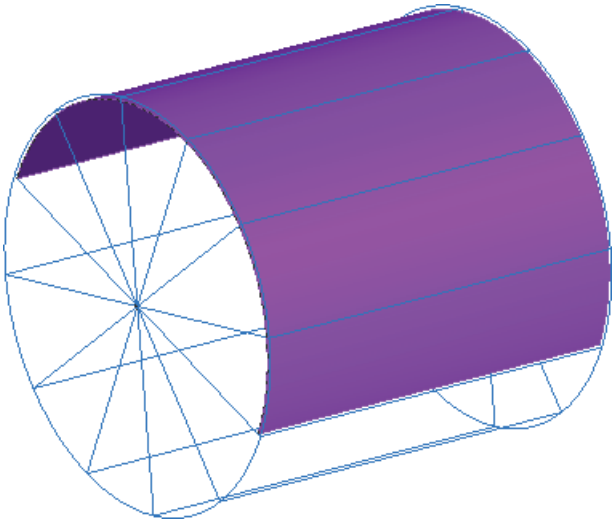
5-axis engraving

The **5-Axis** tab of the **Feature Properties** dialog is now available for 2d spiral operations, which you can use to create 5-axis engraving features. This enables you to use a tool axis angle normal to the surface to ensure the engraving has a uniform cross-section and depth of cut.

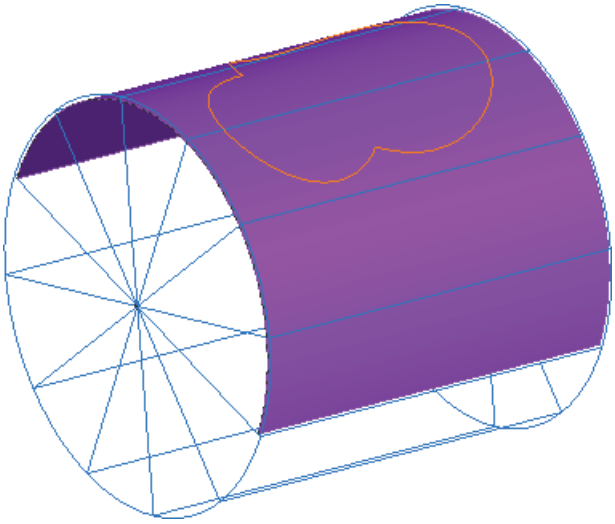


To create a 5-axis surface engraving feature:

1 Create a surface below the stock boundary to determine the depth of the engraving, for example:



2 Create the curve to define the shape of the engraving feature, for example:



*You can use the **Construct > Curve > From Surface > Project onto Surface** menu option to project a curve onto a surface.*

3 Create a surface milling feature with a 2D spiral operation.

4 Double-click the feature in the Part View to display the **Feature Properties** dialog, and select **spiral2d** in the Tree View.

5 On the **Stock** tab, under **Choose the drive curve**, select **Select curves for boundaries** and click **Curve Options**.

The **Boundary Curve** dialog is displayed.

6 Under **Boundary curve type**, select **Wall only**.

This creates a toolpath along the curve.

7 Under **Boundary curves**, click **Boundaries**.

The **Select Boundary Curves** dialog is displayed.

8 Select the curve that defines the shape of engraving, and click **OK** to close the **Select Boundary Curves** dialog.

9 Click **OK** to close the **Boundary curves** dialog.

10 On the **5-Axis** tab, select **Use Lead and Lean**, and in the **from** list, select **Contact normal**.

This keeps the tool axis normal to the surface.

11 Click **OK** to close the **Feature Properties** dialog.

Machine Design

FeatureCAM 2015 R3 contains these changes and improvements to Machine Design:

- **Specifying machine movement limits** (see page 32) — You can now set the limits of movement for solids in a Machine Design file.
- **Testing machine movements** (see page 33) — You can now use machine jogging to simulate the movement of solids in a Machine Design document without having to use an FM file.
- **Simulating mini-turrets** (see page 35) — You can now simulate mini-turrets, which are tools with multiple inserts where the tool rotates around the b-axis to access each tool.

Specifying machine movement limits

You can now set the limits of movement for solids in a Machine Design file. This gives you more control of machine simulations, and enables you to ensure the machine does not move beyond its limits during a simulation.

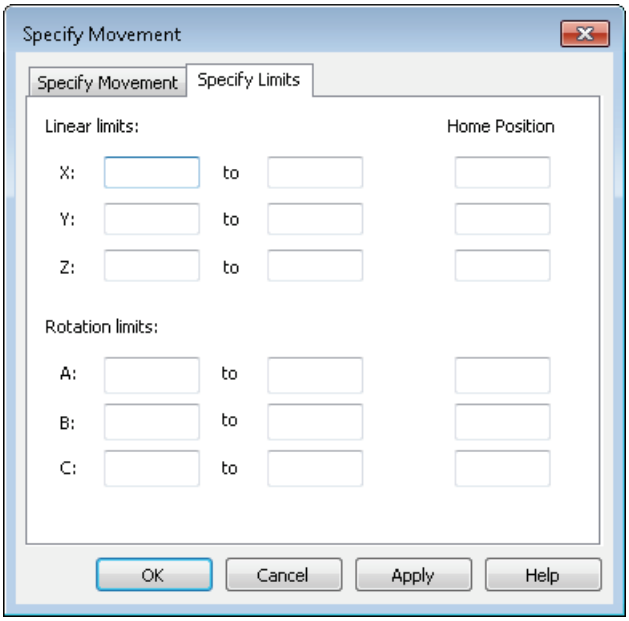
To specify the limits of movement for a solid:

1. In a Machine Design document, select the **Machine Design > Specify Movement** menu option.

   The **Specify Movement** dialog is displayed.

2. Use the **Specify Movement** tab to select a solid and specify how it can move.

3. Use the new **Specify Limits** tab to specify the **Home Position** and the linear and rotational limits of movement for the selected solid.



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

FeatureCAM can display a warning if a solid exceeds these limits during machine simulation (see page 20).

There are several new example MD files that have limits of movement set. These are located in \FeatureCAM\Examples\Machine Design\Axis Limits.

Testing machine movements

You can now use machine jogging to simulate the movement of solids in a Machine Design document without having to use an FM file. This enables you to test the movement and ensure the limits are set correctly.

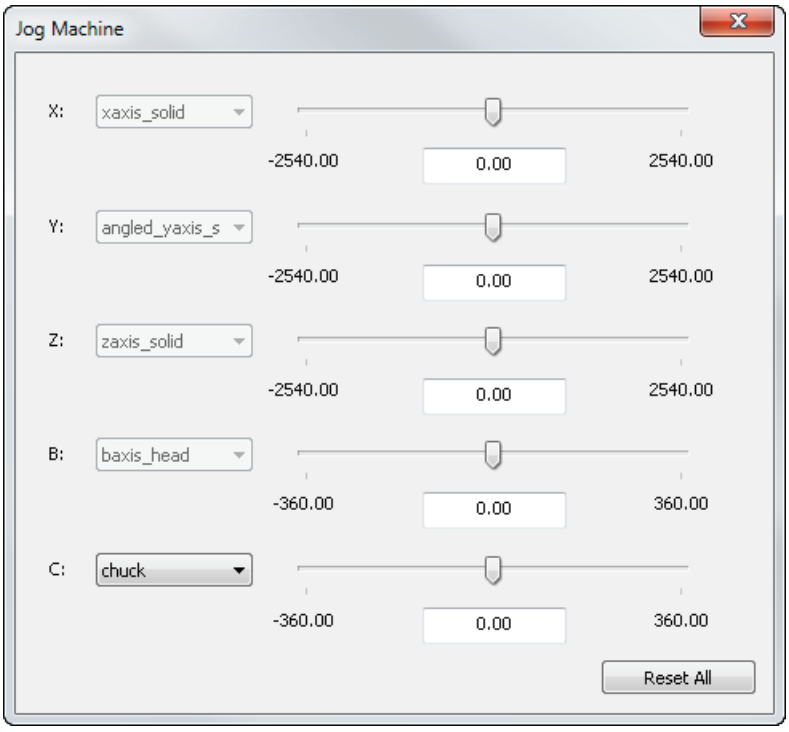
To use machine jogging to test machine movements:

1 Save any changes to your MD document.

Any unsaved changes to the document are not displayed in the jogging simulation.

2 Select the **Machine Design > Jog Machine** menu option.

The new **Jog Machine** dialog is displayed.



Each row displays an axis in which solids in the document can move.

3 Select a solid in the list next to an axis name.

The list is unavailable if there is only one solid that can move in that axis.

4 For the selected solid, move the slider between the minimum limit and the maximum limit.

The solid's current position is shown in the middle field, and the solid's movement is simulated in the graphics window.

*You can set the limits and home positions of the solids using the **Specify Movement** dialog (see page 32).*

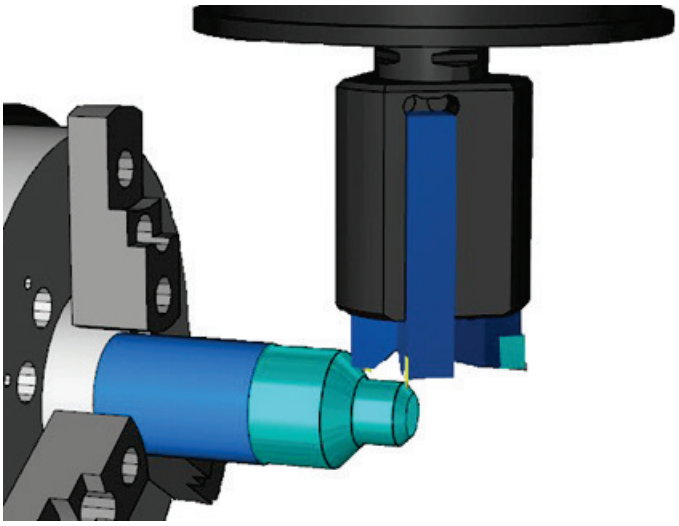
5 To hide solids in the jogging simulation, click **Select** in the **Standard** toolbar, then click solids in the graphics window.

6 To return all solids to their default positions, click **Reset All**.

7 When finished, close the dialog. The jogging simulation is cleared, and all solids are returned to their default positions.

Simulating mini-turrets

You can now simulate mini-turrets, which are tools with multiple inserts where the tool rotates around the b-axis to access each insert. This enables you to create a more flexible tooling setup and perform faster tool changes. The tools are simulated simultaneously, which enables you to check for gouges with the tools that are not currently in use.



To simulate a mini-turret:

1. In the Machine Design file, create a solid and UCS to represent the mini-turret. The X-axis of the UCS must point towards the main spindle.
2. Create a UCS for each tool location. The difference between the X-axis of the tool location and the turret UCS determines the angle that the mini-turret is rotated to use the tool.
3. In the **Tool Block** dialog:
   a. Select the turret solid under **This solid is a tool block for solids**.
   b. Select the turret UCS under **This UCS will match up with the tool location on the turret**.
   c. Use the **Tool Locations** tab to add each tool location UCS as a separate sub slot in the tool block.
   d. Click **OK** to close the dialog.
4. Save the Machine Design file.
5. In the FM document, use the **Tool Mapping** dialog to specify which tool block to use for each feature.

What's new in FeatureCAM 2015 R2

FeatureCAM 2015 R2 contains the following new features and enhancements:

User interface

- **Changing the point size** (see page 40) — You can now change the size of geometry point objects to make them easier to see.
- **Customizing the snapping cursor** (see page 41) — You can now change the size and color of the snapping cursor.
- **Measuring curve length** (see page 42) — You can now find the length of curves and geometry segments.
- **Creating internal gears** (see page 43) — You can now perform analysis on internal gears curves.
- **Combining solids** (see page 44) — You can now combine multiple solids.
- **Hiding rapid moves in centerline simulations** (see page 45) — You can now hide rapid moves in centerline simulations.
- **Chip recognition size** (see page 46) — You can now specify the size at which detached pieces of stock are considered to be chips and hidden from simulation.
- **Recognizing features on large parts** (see page 47) — IFR now works more quickly for large parts.

Milling

- **Automatic tool selection** (see page 49) — You can now specify an additional clearance on the automatic tool selection to prevent tool holder gouges.

- **Vortex approaches flats from outside stock** (see page 51) — Vortex toolpaths can approach flat areas from outside of stock instead of always ramping into it.
- **Output options for 2D NT toolpaths** (see page 53) — The **Output Options** dialog is now available for 2D toolpaths that use NT or Vortex stepover types.
- **Previewing the toolpath point distribution** (see page 55) — You can now preview the points of a surface milling toolpath to help with editing the point distribution.
- **Tool pecking depths** (see page 56) — You can now specify the pecking depths for individual tools.
- **Changing the posting tolerance** (see page 57) — You can now create more precise toolpaths, which is useful for machining small parts.
- **Helical side finish** (see page 58) — Helical side finish operations are now machine at the feed rate, instead of the plunge feed rate.

Turning

- **Removing undercuts in no-drag turning features** (see page 60) — You can now remove undercuts in no-drag turning features to prevent gouges and simplify toolpaths.
- **Controlling steady rests** (see page 61) — You can now open and close the jaws of a steady rest without moving it to the home position.
- **Simulating bar stock** (see page 63) — You can now simulate bar stock in FeatureCAM by specifying the length of the stock displayed in simulation.
- **Custom turret names (MTT)** (see page 64) — You can now use customized turret names to make FeatureCAM more consistent with your machine.

XBUILD

- **Generating Post documentation** (see page 65) — You can now output your CNC data file as HTML or XML to make it understandable.
- **Using macros in the post processor** (see page 66) — The **Disable Macros** option in the **Post Options** dialog is now selected by default.

Add-ins and extensions

- **Using Setup Sheets** (see page 68) — There are new tags which you can use to create more detailed setup sheets.

- **FeatureCAM to CAMplete TruePath add-in** (see page 69) — You can export documents to CAMplete TruePath, which you can use to analyze, modify, optimize, simulate and post 5-Axis toolpaths.
- **Probing update options** (see page 71) — When creating probing features, you can now select multiple update options, which enables you to perform multiple actions from a single probing cycle.
- **Support for Microsoft SQL 2014** (see page 71) — FeatureCAM now supports Microsoft SQL server 2014.

Machine Simulation Design

- **Multi-tool blocks** (see page 73) — You can now create multi-tool blocks and double-sided tool blocks in Machine Design files and use them in machine simulations.
- **Selecting the tool block** (see page 76) — You can now select which tool block holds each tool within an FM file, which enables you to create accurate machining simulations more easily.
- **Protecting Machine Design documents** (see page 78) — You can protect your Machine Design files from the extraction of solids, so that you can share them for simulation without anyone being able to extract the solids.

User Interface

FeatureCAM 2015 R2 contains the following changes and improvements to the user interface:

- **Changing the point size** (see page 40) — You can now change the size of geometry point objects to make them easier to see.
- **Customizing the snapping cursor** (see page 41) — You can now change the size and color of the snapping cursor.
- **Measuring curve length** (see page 42) — You can now find the length of curves and geometry segments.
- **Creating internal gears** (see page 43) — You can now perform analysis on internal gears curves.
- **Combining solids** (see page 44) — You can now combine multiple solids.
- **Hiding rapid moves in centerline simulations** (see page 45) — You can now hide rapid moves in centerline simulations.
- **Chip recognition size** (see page 46) — You can now specify the size at which detached pieces of stock are considered to be chips and hidden from simulation.
- **Recognizing features on large parts** (see page 47) — IFR now works more quickly for large parts.

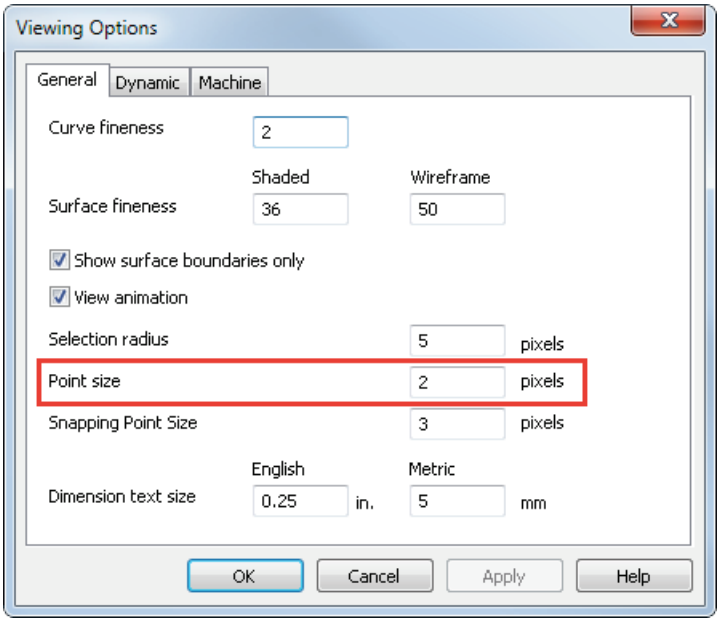
Changing the point size

You can now change the size of geometry point objects to make them easier to see.

To change the point size:

1 Select the **Options > Viewing** menu option.

The new **Point size** option is displayed in the **Viewing Options** dialog.



2 Enter a **Point size** to specify the size of geometry point objects in the graphics window.

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

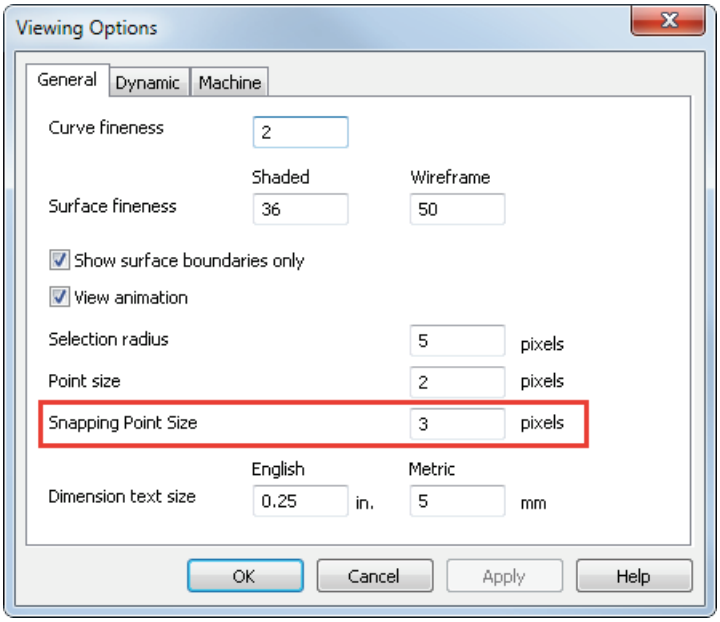
Customizing the snapping cursor

The snapping cursor shows you which object you are snapping to. You can now change the size and color of the snapping cursor to improve work flow when working with large, small or complex parts.

To change the size of the snapping cursor:

1 Select the **Options > Viewing** menu option.

The new **Snapping Point Size** option is displayed in the **Viewing Options** dialog.



2 Enter a **Snapping Point Size** to specify the size of the snapping cursor in the graphics window.

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

To change the color of the snapping cursor:

1 Select the **Options > Coloring > Default Colors** menu option.

The **Default Colors** dialog is displayed.

2 Select **Highlight** in the list.

3 Click **More Colors** to display the **Color** dialog.

4 Select a color and click **OK** to close the dialog.

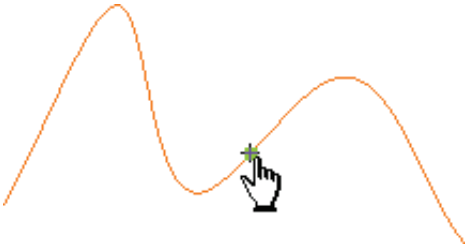
The selected color is displayed in the **Default Colors** dialog.

5 Click **Apply** to apply the color to the selected item.

6 Click **Done** to close the dialog.

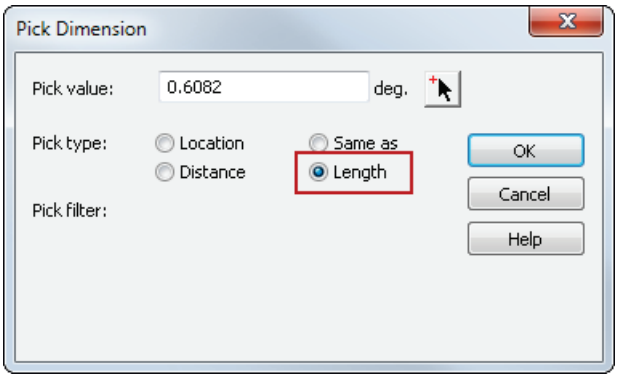
Measuring curve length

You can now find the length of curves and geometry segments, which enables you to find important dimensions of complicated shapes easily.



To find the length of a curve:

1. Select the **Construct > Dimension > Interrogation** menu option. The **Pick Dimension** dialog is displayed.
2. Under **Pick type**, select the new **Length** option.

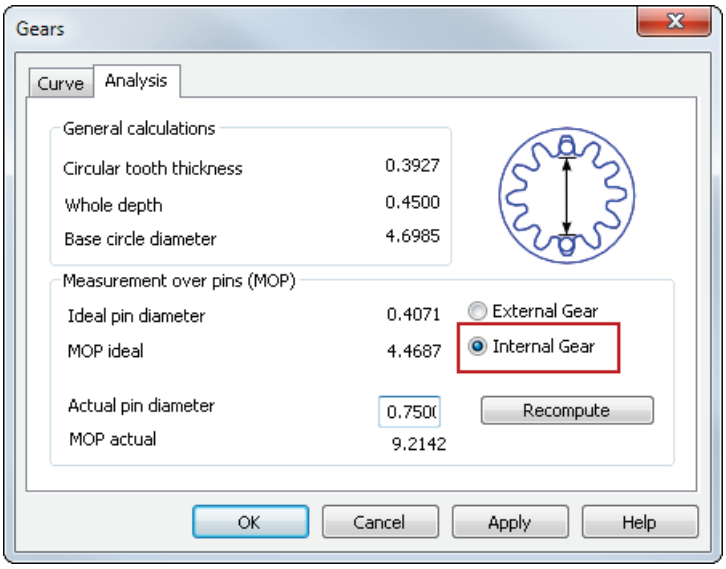


3. Click **Pick Location** and select the curve you want to measure in the graphics window.

   The length of the selected curve is displayed in the **Pick value** field.
4. To measure another curve, repeat the previous step.
5. When you have finished, click **OK** to close the dialog.

Creating internal gears

In FeatureCAM 2015 R1, the **Analysis** tab was added to the **Gears** dialog (see page 96), which enables you to view the calculated dimensions of the gear, the calculated pin gauge diameter and the outside measurement over pins diameter. In FeatureCAM 2015 R2, the **Analysis** tab has been extended to enable you to find the measurement over pins value of an internal gear.



To create and analyze an internal gear curve:

1. Select the **Construct > Curve > Other Methods > Gears** menu option.

   The **Gears** dialog is displayed.

2. In the **Curve** tab, specify the properties of the gear.

3. Select the **Analysis** tab.

   The calculated dimensions of the gear are displayed under **General calculations**.

4. Under **Measurement over pins (MOP)**, select **Internal Gear**.

   The **Ideal pin diameter** and **MOP ideal** values are displayed.

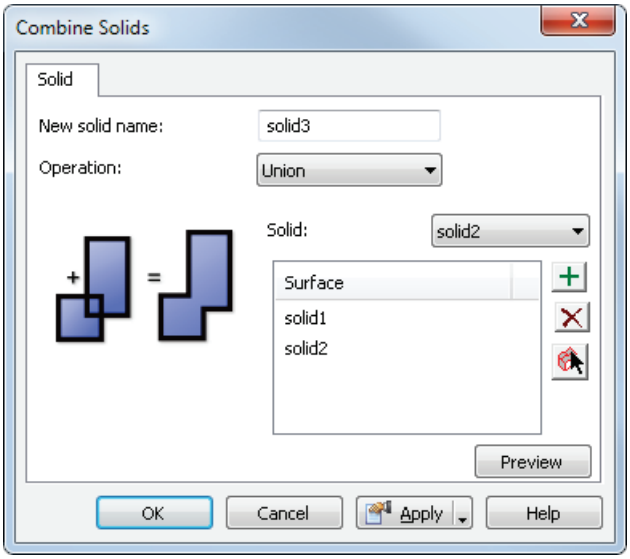
5. Enter the **Actual pin diameter** and click **Recompute**.

   The calculated measurement over pins value is displayed in the **MOP actual** field.

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

Combining solids (SOLID)

You can now combine multiple solids in different ways to create complicated shapes more easily. Previously, you could combine only two solids at once.



To combine solids:

1. Select the **Construct > Solid > Modifiers > Combine solids** menu option.

   The **Combine Solids** dialog is displayed.

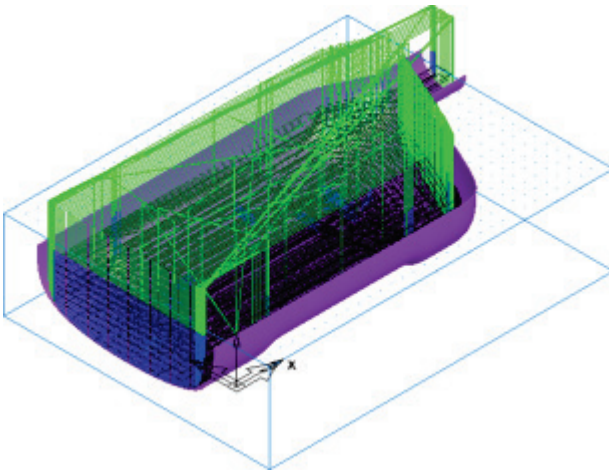
   *This dialog was previously called the **Boolean** dialog.*

2. Enter a **New solid name**.
3. Select an **Operation**:
   - **Difference** — Subtract a solid from another solid. You can combine only two solids with this operation.
   - **Union** — Merge multiple solids together.
   - **Intersection** — Create a solid at the intersection of multiple solids.
4. Add the solids you want to combine:
   - To add a solid by name, select it in the **Solid** list and click **Add item from list** .
   - To add solids graphically, click **Pick solid** and select them in the graphics window.
5. Click **OK** to close the dialog.

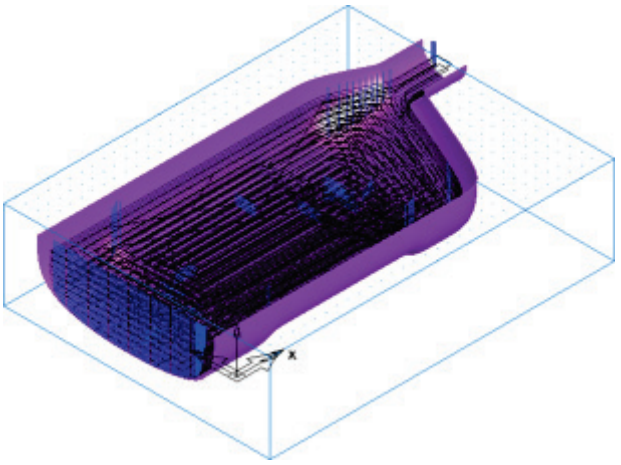
Hiding rapid moves in centerline simulations

You can now hide rapid moves in centerline simulations. This enables you to see the cutting moves more clearly in complicated toolpath simulations.

Rapid moves shown:



Rapid moves hidden:

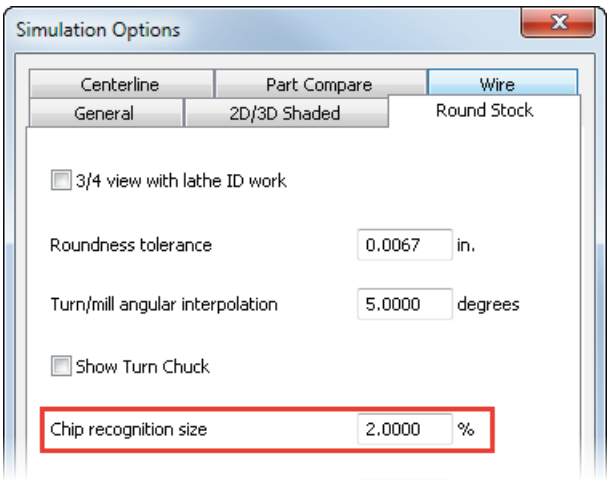


To hide rapid moves in centerline simulations, deselect the **View > Simulation > Show Centerline Rapids** menu option. The next time you run a centerline simulation, rapid moves are not displayed.

Chip recognition size

You can now specify the size at which detached pieces of stock are considered chips and hidden from simulation.

There is a new **Chip recognition size** option on the **Round Stock** tab of the **Simulation Options** dialog.



For parts with large stock, you may want to reduce this value to ensure cutoffs are not hidden.

Recognizing features on large parts

Interactive Feature Recognition (IFR) now works more quickly on large parts.

Previously, IFR would validate the solid when recognizing features, which could take a long time for large parts.

To verify that a solid is valid, select the solid in the **Automatic Feature Recognition** dialog and click **Verify**.

Milling

FeatureCAM 2015 R2 contains the following changes and improvements to Milling:

- **Automatic tool selection** (see page 49) — You can now specify an additional clearance on the automatic tool selection to prevent tool holder gouges.
- **Vortex approaches flats from outside stock** (see page 51) — Vortex toolpaths can approach flat areas from outside of stock instead of always ramping into it.
- **Output options for 2D NT toolpaths** (see page 53) — The **Output Options** dialog is now available for 2D toolpaths that use NT or Vortex stepover types.
- **Previewing the toolpath point distribution** (see page 55) — You can now preview the points of a surface milling toolpath to help with editing the point distribution.
- **Tool pecking depths** (see page 56) — You can now specify the pecking depths for individual tools.
- **Changing the posting tolerance** (see page 57) — You can now create more precise toolpaths, which is useful for machining small parts.
- **Helical side finish** (see page 58) — Helical side finish operations are now machine at the feed rate, instead of the plunge feed rate.

Automatic tool selection

You can now specify an additional clearance on the automatic tool selection to prevent tool holder gouges.

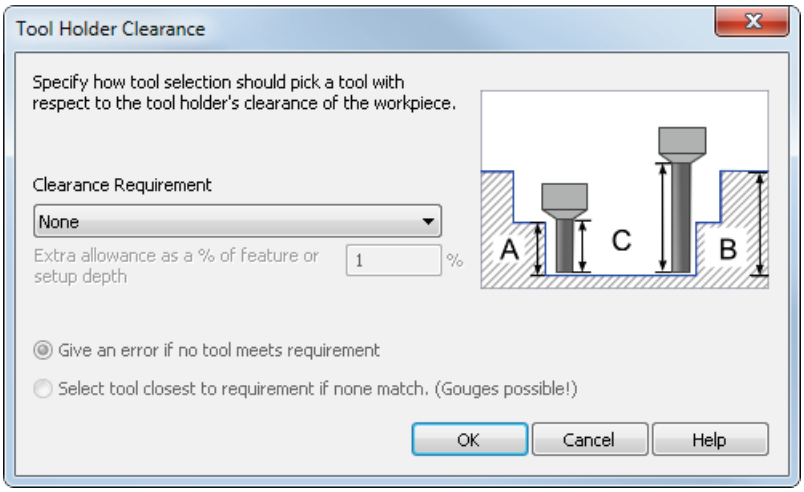
No additional clearance:

Clearance above the Setup:

To specify a tool holder clearance for automatic tool selection:

1. Select the **Manufacturing > Machining Attributes** menu option to display the **Machining Attributes** dialog.
2. On the **Tool Selection** tab of the **Machining Attributes** dialog, click **Tool Holder Clearance**.

   The new **Tool Holder Clearance** dialog is displayed.

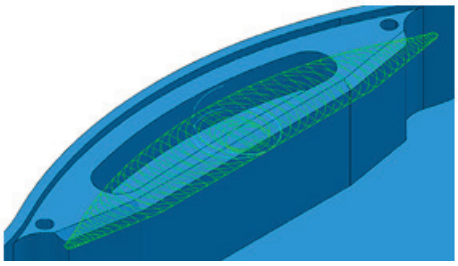


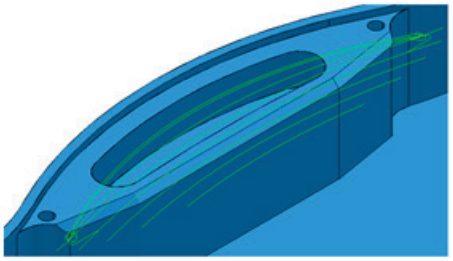
3. In the **Clearance Requirement** list, select the clearance you want between the tool holder and the part. Select from:
   - **None** — Select this option to leave no additional clearance. Old part files still select the same tools as before.
   - **Feature** — Select this option to ensure the tool is long enough for the tool holder to clear the feature.
   - **Setup** — Select this option to ensure the tool is long enough for the tool holder to clear the total depth into the setup.
4. Enter an **Extra allowance as a % of feature or setup depth** to leave extra clearance of the tool holder above the feature or Setup.
5. Select how tool selection is affected if no matching tool is found:
   - **Give an error if no tool meets requirements** — FeatureCAM does not select a tool for the operation, so an error is shown during NC code generation. In the **Operation List**, a red exclamation point ❗ is displayed beside operations with no tool selected.
   - **Select tool closest to requirements if none match** — this enables you to generate NC code, but it may result in tool holder gouges because a smaller tool may be used.
6. Click **OK** to close the dialog.

Vortex approaches flats from outside stock

Vortex toolpaths can approach flat areas from outside of stock instead of always ramping into it. This enables you to create Vortex toolpaths that are faster to machine and are compatible with a wider selection of tools.

FeatureCAM 2015 R2	FeatureCAM 2015 R3
The tool ramps into the part	The tool approaches from the outside

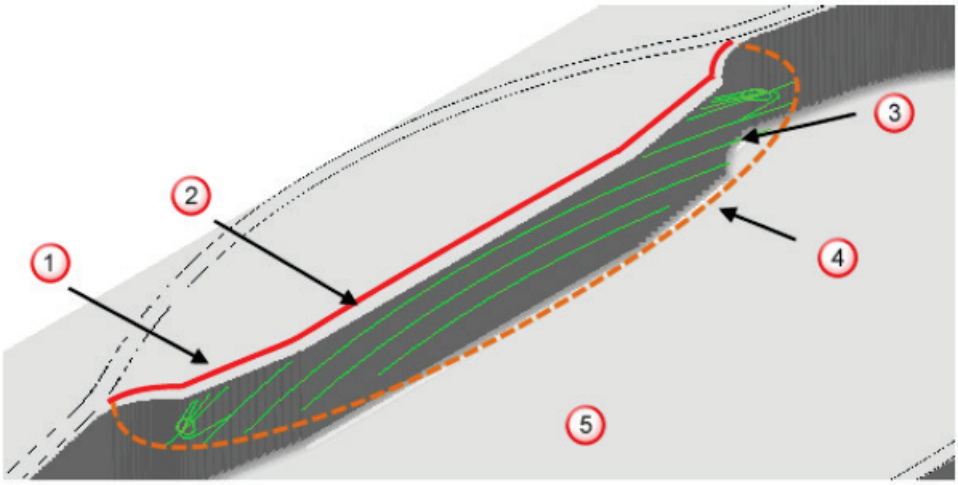




To approach from outside the stock, FeatureCAM:

- extends a section of the flat area beyond the stock and into an area already machined; and;
- fills the extended section with cutting moves.

By extending the section to an area already machined, the machine tool can approach the flat area in open space.



1. Edge of flat area
2. Stock
3. Extended section of flat area filled with cutting moves
4. Outside edge of extended section
5. Area already machined

Criteria for toolpath to approach from outside stock

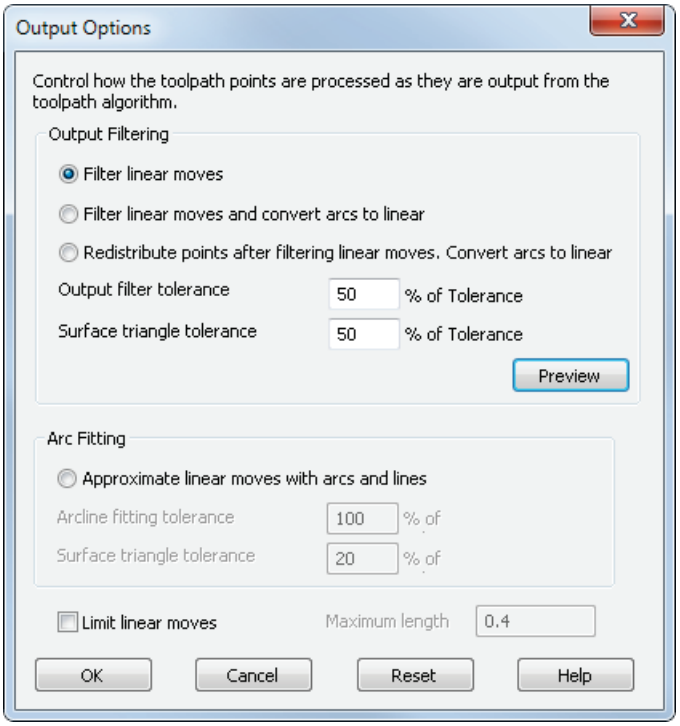
FeatureCAM only extends a section of the flat area if the extended section:

- has an outside edge that the machine tool can approach.
- does not gouge the model.
- is wide enough to be profile smoothed successfully.
- can reach an area already machined within the distance of one tool diameter.

If the extended section fails to meet the criteria, FeatureCAM does not extend the flat area and instead uses a ramp move to approach the toolpath.

Output options for 2D NT toolpaths

The **Output Options** dialog is now available for 2D toolpaths that use NT or Vortex stepover types. This enables you to control how the points of 2D vortex and New Technology toolpaths are processed in the NC program. Previously, this was only available for 3D toolpaths.



To display the **Output Options** dialog, click **Output Options** on the **Milling** tab of the **Feature Properties** dialog.

The dialog contains these output options:

- **Filter linear moves** — This automatically removes unnecessary points in the toolpath while maintaining tolerance. The points are not equispaced because unnecessary points are deleted.
- **Filter linear moves and convert arcs to linear** — This is similar to the first option except that all arcs are replaced by straight line segments. This option is suitable for machine tools which do not handle arcs well.
- **Redistribute points after filtering linear moves. Convert arcs to linear** — This option allows the insertion of new points. This ensures a constant distance between points, only inserting extra points if they are necessary to keep tolerance. This option may increase toolpath creation time, but reduce time on the machine tool. This option is suitable for machine tools that can handle large numbers of equispaced points.

- **Approximate linear moves with arcs and lines** — Select this option to create an arc line approximation for toolpaths that are contained in the XY, YZ, and XZ plane. This allows 3D programs to be smaller and to result in smoother surface finishes for certain types of parts.
- **Limit linear moves** — Select this option to limit the distance between linear move points to the **Maximum length** value.

Previewing the toolpath point distribution

You can now preview the points of a surface milling toolpath to help with editing the point distribution.

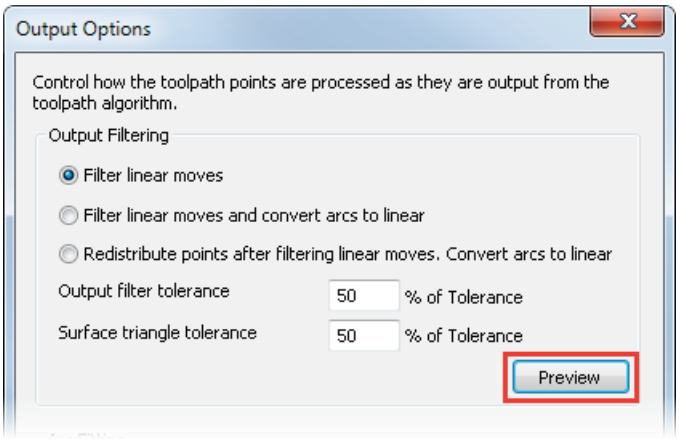
To redistribute toolpath points:

1. In the **Surface Milling Properties** dialog, on the **Milling** tab, click **Output Options**.

   The **Output Options** dialog is displayed.

2. Under **Output Filtering**, specify the options for filtering linear moves in the toolpath.

3. Click the new **Preview** button. The points of the toolpath are shown in blue in the graphics window.



4. Modify the view in the graphics window to see the toolpath preview.

5. When you have finished, click **OK** to close the dialog.

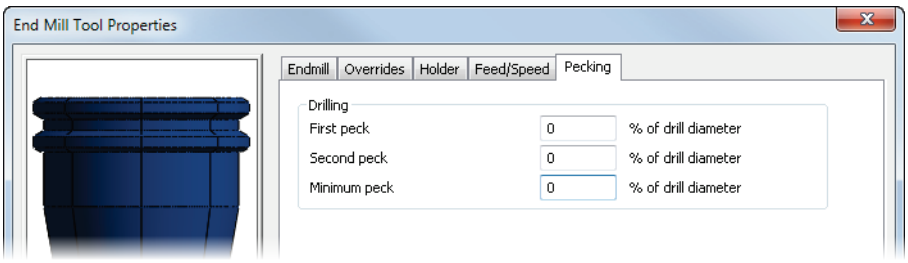
Tool pecking depths

Pecking is used when drilling deep holes, where the tool retracts multiple times while drilling to clear debris from the hole.

You can now specify the pecking depths for individual tools. Previously, you had to change the pecking depths for individual operations to override the global settings.

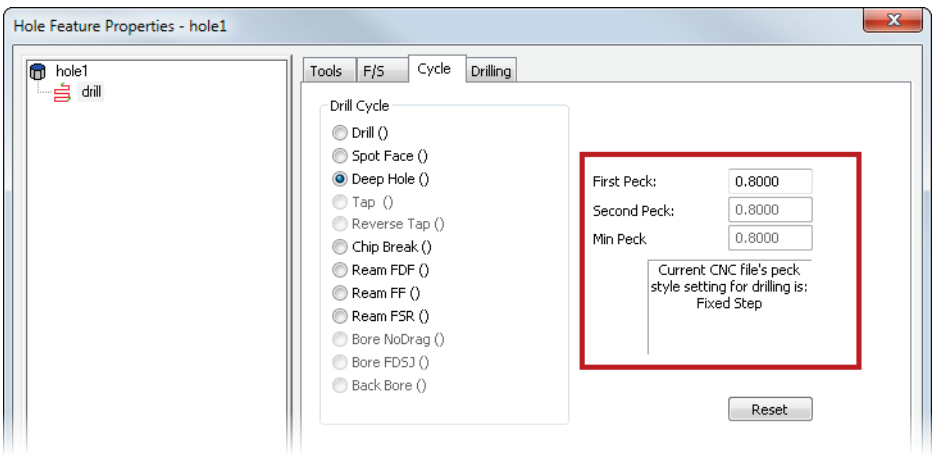
To specify the pecking depths at which the drill retracts:

- You can change the global pecking depths on the **Pecking** tab of the **Machining Attributes** dialog.
- You can change the pecking depths for each tool on the new **Pecking** tab of the **Tool Properties** dialog:



*Leave the values at 0% to use the global values in the **Machining Attributes** dialog.*

- You can see and edit each operation's absolute pecking depths on the **Cycle** tab of the **Hole Feature Properties** dialog.

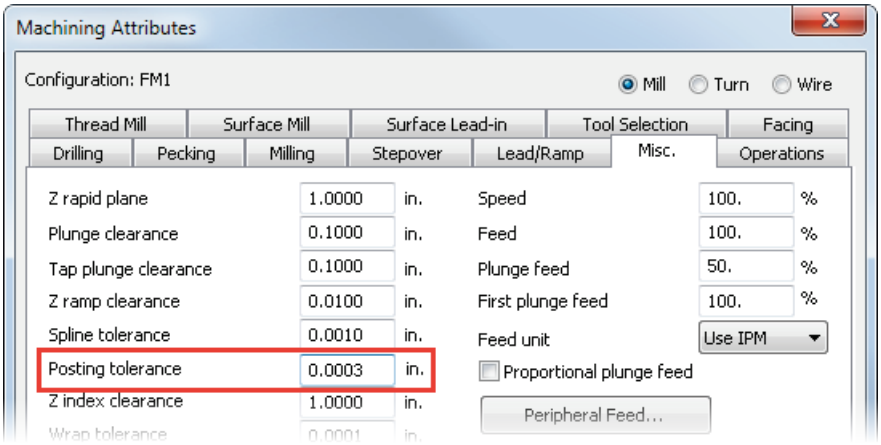


The peck style is specified in the .cnc file. This determines which of the tool's pecking values are used to calculate the absolute pecking depths for the operation.

Changing the Posting tolerance

You can now control the tolerance with which toolpaths are created. This enables you to create more precise toolpaths, which is useful for machining small parts.

There is a new **Posting tolerance** option in the **Misc** tab of the **Machining Attributes** dialog.

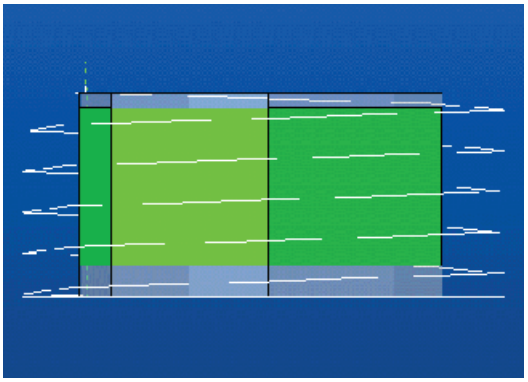


Reduce the **Posting tolerance** for small parts to create more precise toolpaths. You must also adjust your post processor to output more digits. For example, if you adjust the posting tolerance from 0.001 to 0.0001, then you must adjust the digit format in the post processor so that the extra decimal place is used in the NC code.

Reducing the posting tolerance creates additional lines of NC code, so you should only do this for high-precision NC machines that can use the high-precision coordinates, when required for an application.

Helical side finish

You can use the **Helical side finish** option to create a continuous spiral finishing toolpath for a 2.5D milling feature, which prevents tool marks on the surface.



Helical side finish operations are now machined at the feed rate specified on the **F/S** tab of the **Feature Properties** dialog, previously they were machined using the plunge feed rate, specified by the **Plunge feed override** on the **Plunge** tab.

To enable this option, select **Helical side finish** on the **Strategy** tab of the **Feature Properties** dialog, and enter a **Pitch** for the spiral.

Turning

FeatureCAM 2015 R2 contains the following changes and improvements to Turning:

- **Removing undercuts in no-drag turning features** (see page 60) — You can now remove undercuts in no-drag turning features to prevent gouges and simplify toolpaths.
- **Controlling steady rests** (see page 61) — You can now open and close the jaws of a steady rest without moving it to the home position.
- **Simulating bar stock** (see page 63) — You can now simulate bar stock in FeatureCAM by specifying the length of the stock displayed in simulation.
- **Custom turret names (MTT)** (see page 64) — You can now use customized turret names to make FeatureCAM more consistent with your machine.

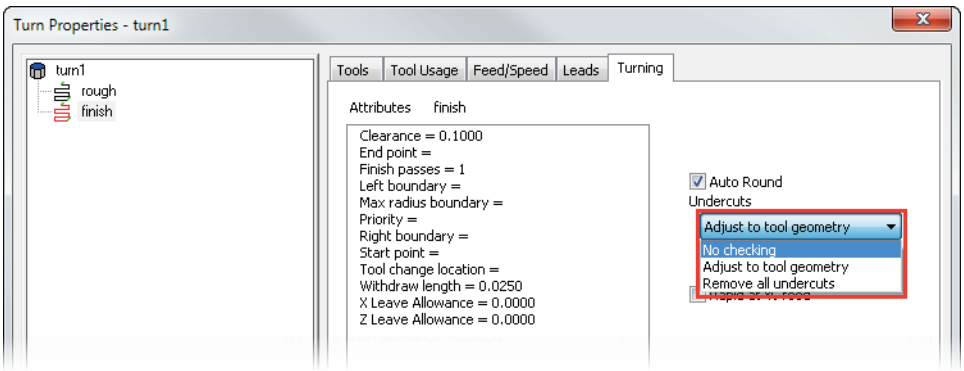
Removing undercuts in no-drag turning features

You can create turning features with a no-drag finishing strategy, where the tool does not cut the feature in a single movement, which reduces tool wear and prevents chips being dragged along the surface.

You can now remove undercuts in no-drag turning features to prevent gouges and simplify toolpaths, which improves machining time and prevents unnecessary tool wear. Previously, this option was available only for features which use a conventional finishing strategy.

To create a no-drag turning feature and remove undercuts:

1. Create a Turn feature.
2. In the **Turn Properties** dialog, on the **Strategy** tab, select **No-Drag**.
3. Select an operation in the Tree View.
4. On the **Turning** tab, select an option in the **Undercuts** list:
   - **No checking** — Select this option to not check the feature for undercuts.
   - **Adjust to tool geometry** — Select this option to clip undercuts that cause the tool to gouge the part.
   - **Remove all undercuts** — Select this option to remove all undercuts.

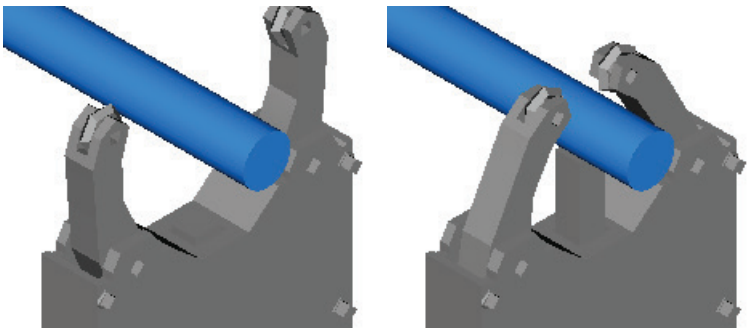


Controlling steady rests

You can now open and close the jaws of a steady rest without moving it to the home position.

This is useful for turning operations on long parts. For example, you can machine up to the steady rest, open the steady rest and machine past it, then close the steady rest and machine to the end of the part.

Steady rest open: Steady rest closed:



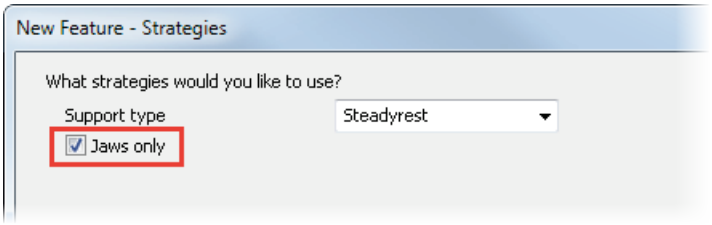
To control the jaws of a steady rest without moving it:

1. Click the **Features** step in the **Steps** panel to display the **New Feature** wizard.
2. Select **Part Handling** and click **Next**.

   The **Dimensions** page is displayed.
3. To close a steady rest, select **Part Support On**.

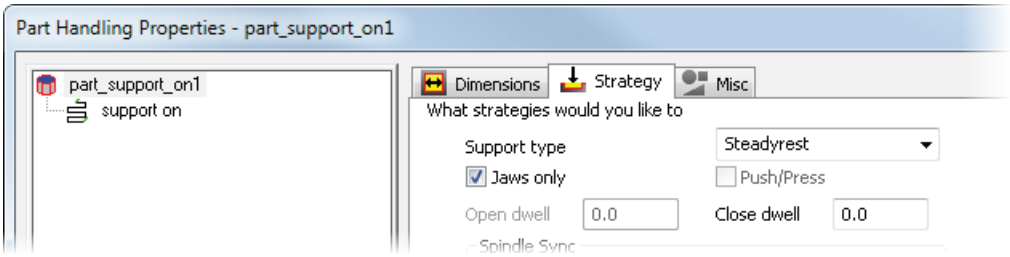
   To open a steady rest, select **Part Support Off**.
4. Click **Next**.

   The **Strategies** page is displayed.
5. In the **Support type** list, select **Steadyrest**.
6. Select the new **Jaws only** option.



7. Click **Finish** to close the wizard.
8. To specify when the Part Handling feature is performed, click and drag the feature in **Part View**, or change the **Base Priority** attribute on the **Misc** tab of the **Part Handling Properties** dialog.

The **Jaws only** option is also displayed on the **Strategy** tab of the **Part Handling Properties** dialog.

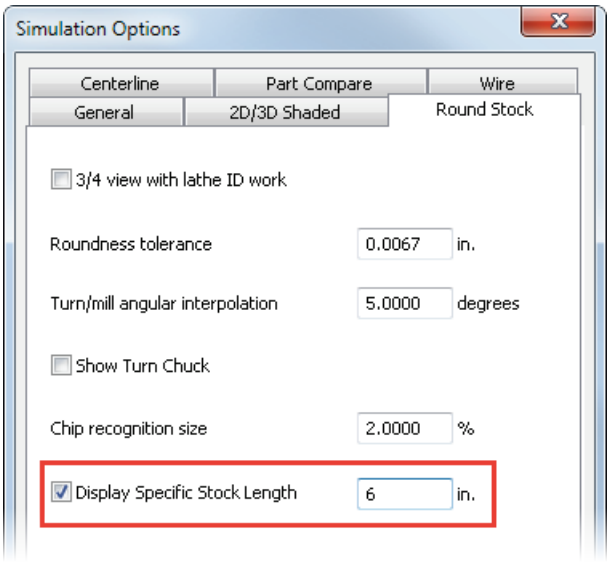


There is a new <JAWS-ONLY> reserved word in XBUILD. You must update your post Formats to use the reserved word to obtain the correct NC code that matches the new simulation.

Simulating bar stock

You can now simulate bar stock in FeatureCAM by specifying the length of the stock displayed in simulation without changing the stock dimensions.

Select the **Options > Simulation** menu option, and use the new **Display Specific Stock Length** option on the **Round Stock** tab of the **Simulation Options** dialog.

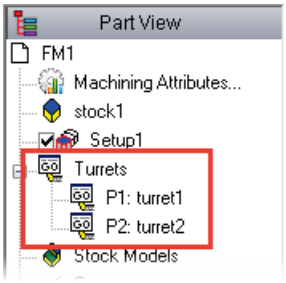


Custom turret names (MTT)

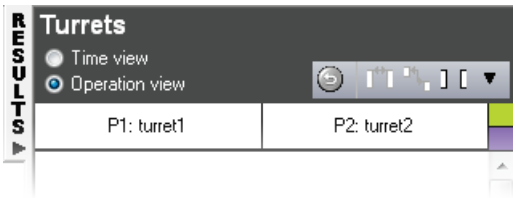
You can now use customized turret names to make FeatureCAM more consistent with your machine. Previously, all turrets were called upper and lower turrets.

The turret names are taken from the names of the turret or gang solids in the Machine Design file, and are used throughout FeatureCAM:

- In the **Part View** panel.



- In the **Turrets** tab of the **Results** panel.



- In the **Details** tab of the **Results** panel.
- In the NC program.

XBUILD

FeatureCAM 2015 R2 contains the following XBUILD improvements:

- **Generating Post documentation** (see page 65) — You can now output your CNC data file as HTML or XML to make it understandable.
- **Using macros in the post processor** (see page 66) — The **Disable Macros** option in the **Post Options** dialog is now selected by default.

Generating Post documentation

In XBUILD, you can output your CNC data file as a text file. This file has a .cnx extension, and you can edit and print it with any text editor.

You can now output your CNC data file as HTML or XML to make it more understandable.

Select one of these **File > Document CNC** menu options:

- **CNX (Text)** — Creates a .cnx file which can be read by any text editor.
- **HTML** — Creates a .html file. This is the most readable document.
- **XML** — Creates an .xml file.

Using macros in the post processor

The **Disable Macros** option in the **Post Options** dialog is now selected by default. You must deselect this option to enable macro generation for the NC code.

Add-ins and extensions

FeatureCAM 2015 R2 contains the following changes to add-ins and extensions:

- **Using Setup Sheets** (see page 68) — There are new tags which you can use to create more detailed setup sheets.
- **FeatureCAM to CAMplete TruePath add-in** (see page 69) — You can export documents to CAMplete TruePath, which you can use to analyze, modify, optimize, simulate and post 5-Axis toolpaths.
- **Probing update options** (see page 71) — When creating probing features, you can now select multiple update options, which enables you to perform multiple actions from a single probing cycle.
- **Support for Microsoft SQL 2014** (see page 71) — FeatureCAM now supports Microsoft SQL server 2014.

Using Setup Sheets

You can use the SetupSheet.dll add-in to generate html setup sheets from your document to give information to the machine operator about the manufacturing, tooling, and toolpaths of a part.

There are new tags which you can use to create more useful setup sheets:

- Machining allowances:
  - `{operation.finish_allowance}` — Finish Allowance.
  - `{operation.bottom_finish_allowance}` — Bottom Finish Allowance.
  - `{operation.surface_finish_leave_allowance}` — Finish Leave Allowance.
- Stepover:
  - `{operation.stepover}` — 3D milling stepover.
- Z increment:
  - `{operation.finish_z_increment}` — Finish Z Increment
  - `{operation.surface_finish_z_increment}` — Surface Finish Z Increment.
  - `{operation.surface_rough_z_increment}` — Surface Rough Z Increment.
- Coolant:
  - `{tool.coolant_override}` – The tool's coolant override.
- Depth of cut:
  - `{operation.depth_of_cut}` — Rough Z Pass Increment.
  - `{operation.turn_depth_of_cut}` — Rough Depth of Cut for turning operations.

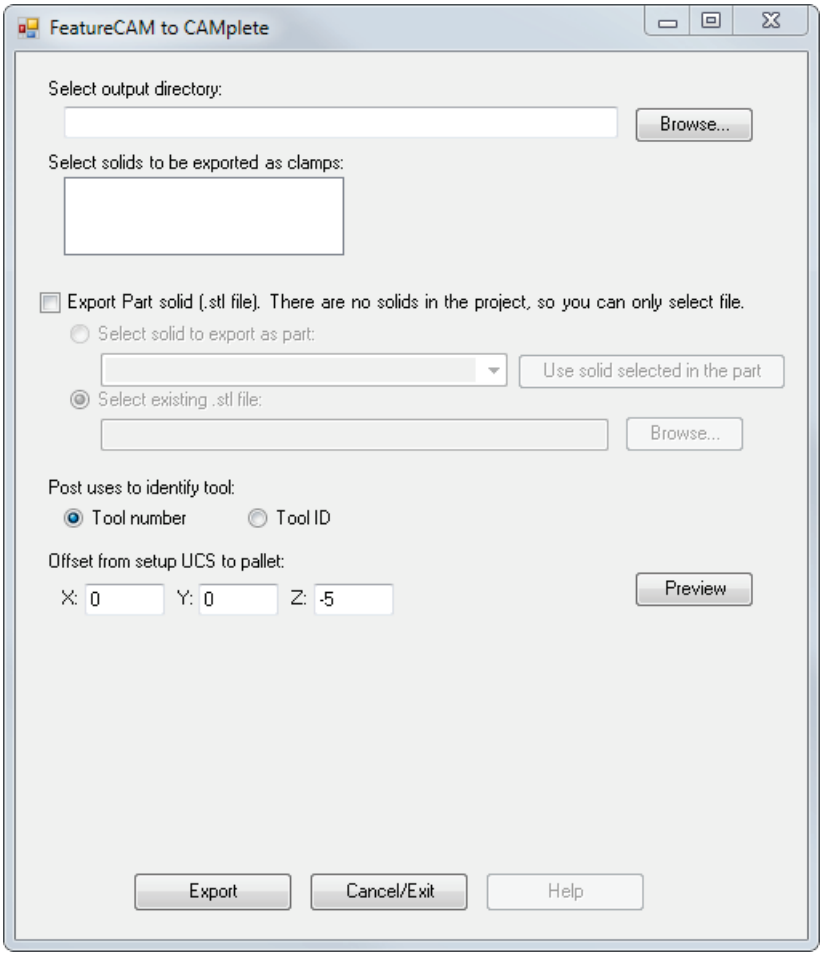
FeatureCAM to CAMplete TruePath add-in

CAMplete TruePath is an application that you can use to analyze, modify, optimize, simulate and post 5-Axis toolpaths.

To export a document to use with CAMplete TruePath:

1. License the **CAMplete verification** product component, and ensure it is selected in the **Evaluation Options** dialog.
2. In the **Post Options** dialog, select the CAMplete_TruePath.cnc post in the /Posts/Mill/5-Axis folder.
3. Load the FeatureCAMToCAMplete.dll add-in.
4. In the **Utilities** toolbar, click **FeatureCAMToCAMplete** .

   The **FeatureCAM to CAMplete** dialog is displayed.



5. Click **Browse** and select where you want to save the exported files.
6. In the solids list, select solids you want to export as clamps.
7. Click **Export Part solid** to export a solid to use as the part in CAMplete.

- To export a solid in the document as an .stl file, select **Select solid to export as part**, and select a solid from the list.
- To use an .stl file, select **Select existing .stl file** and click **Browse** and select it in the **Select part .stl file** document.

8 Select whether you want the post to use the **Tool number** or **Tool ID** to identify tools.

9 Under **Offset from the setup UCS to pallet**, enter the offset distances in the X, Y and Z directions.

10 Click **Preview** to display a point in the graphics window which shows the offset from the setup UCS.

11 Click **Export** to export the document and close the dialog. The files are created in the selected output directory.

Probing update options

When creating probing features, you can now select multiple update options. This enables you to perform multiple actions from a single probing cycle.

To create a probing feature with multiple update options:

1. Select the **Options > Add-Ins** menu option and use the **Macro Add-ins** dialog to load the StandardProbing2.dll add-in.

   This replaces the StandardProbing.dll add-in, but it is still available to enable backwards compatibility.

2. Create a User Defined Feature, and select the feature type you want to measure, then click **Next**.
3. Select an item in the **Dimension** list, enter a **New Value**, and click **Set**.

   You can enter the following values:

   - **MCS** — Enter the Machine Coordinate System you want to update.
   - **Tool** — Enter the tool number for the tool you want to update.
   - **Store** — Specify whether to store the results.
   - **Print** — Specify whether to print the results.
4. Complete the wizard and click **Finish** to create the probing feature.

Support for Microsoft SQL server 2014

FeatureCAM now supports Microsoft SQL server 2014.

You can use SQL server to improve the performance and reliability of your network database in FeatureCAM.

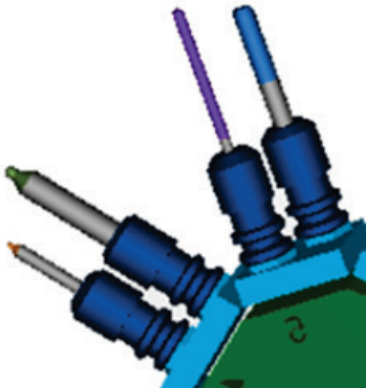
Machine Simulation Design

FeatureCAM 2015 R2 contains the following changes and improvements to Machine Simulation design:

- **Multi-tool blocks** (see page 73) — You can now create multi-tool blocks and double-sided tool blocks in Machine Design files and use them in machine simulations.
- **Selecting the tool block** (see page 76) — You can now select which tool block holds each tool within an FM file, which enables you to create accurate machining simulations more easily.
- **Protecting Machine Design documents** (see page 78) — You can protect your Machine Design files from the extraction of solids, so that you can share them for simulation without anyone being able to extract the solids.

Multi-tool blocks

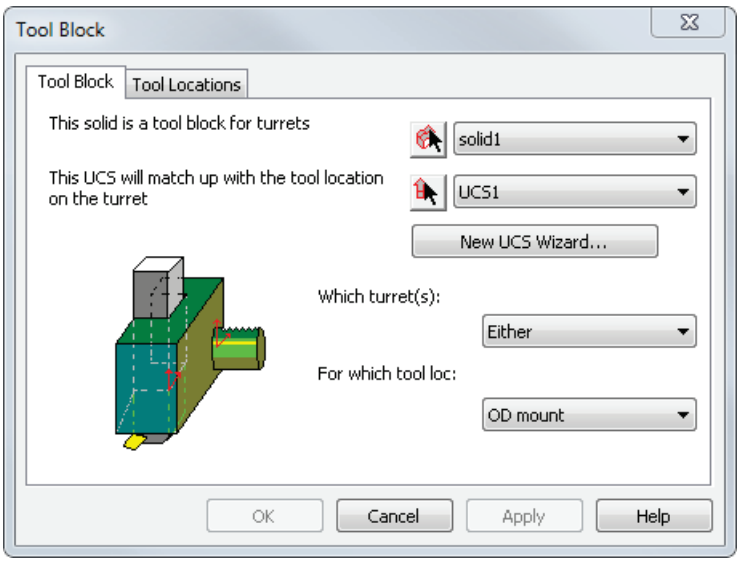
You can now create multi-tool blocks and double-sided tool blocks in Machine Design files and use them in machine simulation. Previously, each tool block could hold only one tool.



To create a multi-tool block:

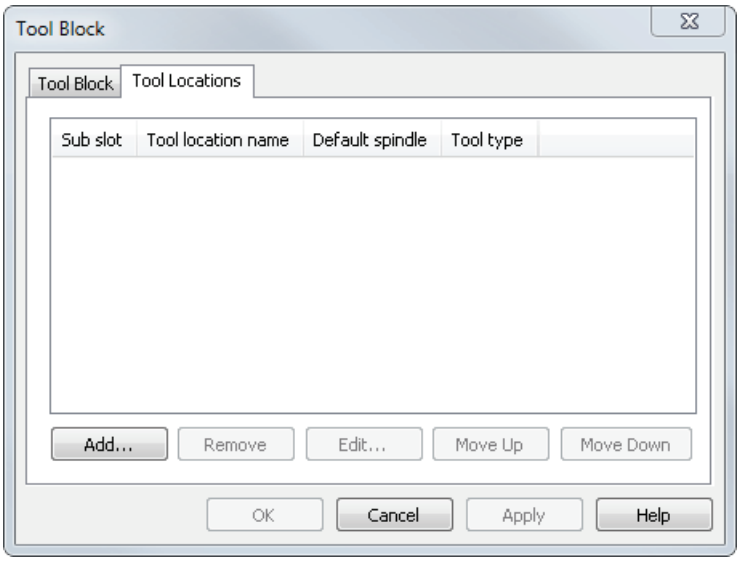
1. Ensure that the **Machine Design > Enable Millturn UI** menu option is selected to access the lathe design options.
2. Select the **Machine Design > Tool Block for Turret** menu option.

   The **Tool Block** tab of the **Tool Block** dialog is displayed. This tab has been updated.



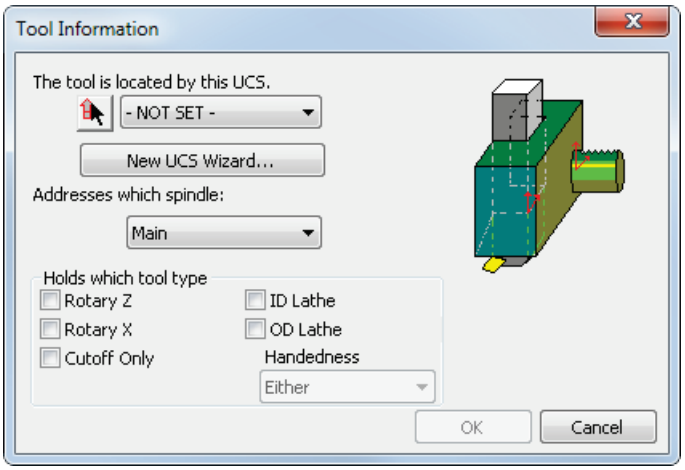
3. Select a solid to use as the tool block.
4. Select a UCS for the tool location on the turret, or click **New UCS Wizard** and create one.
5. Select which turrets the tool block can address.
6. Select which tool locations on the turret the tool block can attach to.

7 Select the new **Tool Locations** tab.



8 Click **Add** to add a new tool location.

The new **Tool Information** dialog is displayed.



9 Select a UCS to define the tool location, or click **New UCS Wizard** and create one.

10 Select which of the spindles the tool addresses.

11 Under **Holds which tool type**, select the tool types that can be used at this tool location. For OD Lathe tools, select the **Handedness** of the tool.

12 Click **OK** to add the tool location and close the **Tool Information** dialog.

13 To remove a tool location, select it and click **Remove**.

14 To edit a tool location, select it and click **Edit**, or double-click it.

15 Use the **Move Up** and **Move Down** buttons to change the order of the tool locations in the dialog.

16 Click **OK** to close the **Tool Block** dialog.

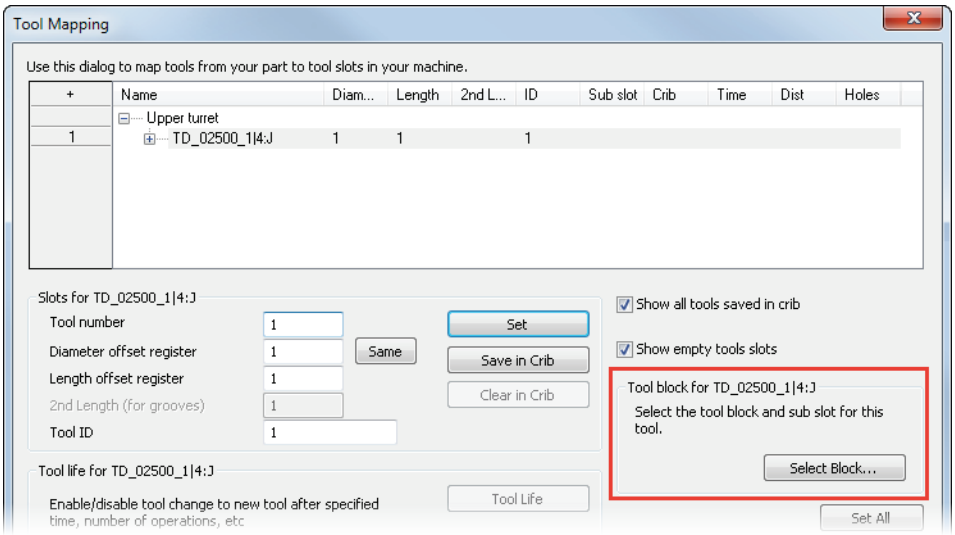
Selecting the tool block

You can now select which tool block holds each tool within an FM file, which enables you to create accurate machining simulations more easily.

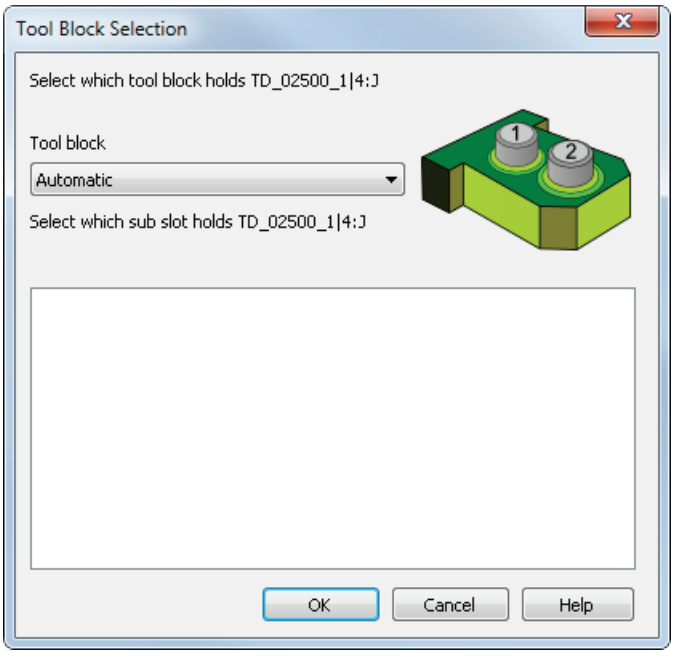
To select the tool block for a tool:

1. Open a Turning or Turn/Mill document that contains toolpaths.
2. Select the **Manufacturing > Tool Mapping** menu option.

   The **Tool Mapping** dialog is displayed.
3. Select a tool in the list, and click the new **Select Block** button.



The new **Select Block** dialog is displayed.

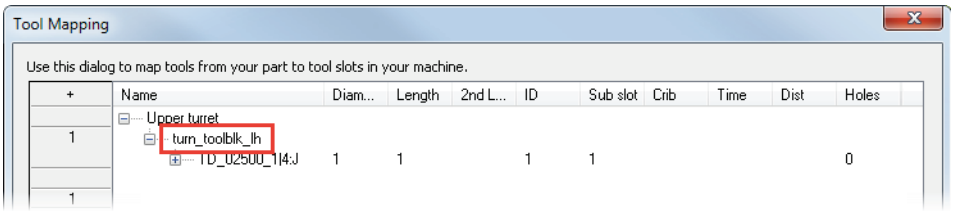


4 In the **Tool block** list, select the tool block solid from the Machine Design file that you want to hold the selected tool.

5 Select the sub slot in the list that you want to hold the tool.

6 Click **OK** to close the **Tool Block Selection** dialog.

The tool block you selected is displayed in the tool mapping tree.

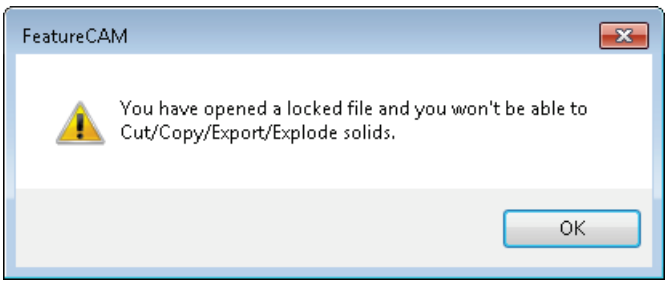


7 Click **OK** to close the **Tool Mapping** dialog and save your changes.

Protecting solids in Machine Design documents

You can protect your Machine Design files from the extraction of solids, so that you can share them for simulation without anyone being able to extract the solids.

FeatureCAM comes with some locked Machine Design files. You can use these for simulation, but you cannot extract solids from them. When you open a locked file, a dialog is displayed explaining the restriction.

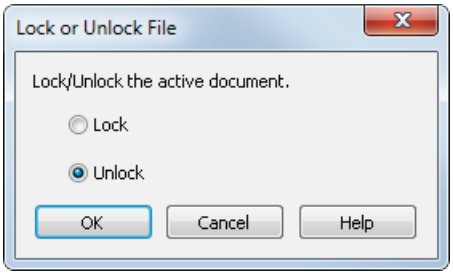


You can lock your files, but you cannot unlock them. You should not lock an original file, instead create a copy of it and lock the copy.

To lock a file:

1. Use the **File > Save As** menu option to save a copy of your file.
2. Select the **File > Lock/Unlock** menu option.

   The **Lock or Unlock File** dialog is displayed.



3. Select **Lock**.

   If you lock a file, you cannot unlock it.

4. Click **OK** to close the dialog and save your changes.

What's new in FeatureCAM 2015 R1

FeatureCAM 2015 R1 contains the following new features and enhancements:

User interface

- **Installing FeatureCAM** (see page 84) — During installation, a dialog reminds you to prepare to update to Windows 8.
- **SQL server authentication** (see page 85) — FeatureCAM can now record the credentials for accessing the tools databases held in SQL Server.
- **Accessing machining attributes** (see page 86) — The **Machining Attributes** dialog can be opened directly from the **Part View** panel.
- **Customizing the Toolbox** (see page 86) — The width each view in the Toolbox can be customized separately.
- **Adding new stock materials** (see page 88) — The **New Stock Material** dialog prevents the creation of invalid stock names.
- **Improved feature recognition** (see page 89) — Feature recognition automatically excludes clamps from solids.
- **Recognizing Face features** (see page 90) — You can now recognize multiple Face features simultaneously.
- **Snapping modes** (see page 91) — The **Snap Mode** toolbar includes a new option that allows the use of the centre points of cylinders and cones.
- **Creating solids** (see page 92) — The **Finish** button of the **Solid Wizard** is configurable in the same way as the **Features Wizard**.
- **Polygonal curves** (see page 94) — A new option is available for creating polygonal curves.

- **Creating gears** (see page 96) — The **Gears** dialog now includes alternative methods of specifying curve information. A separate tab reports the properties of the calculated gear.
- **Command line options** (see page 98) — FeatureCAM's command line interface has been extended.
- **Selecting surfaces** (see page 99) — Selecting the surfaces of long narrow models has been improved.

Milling

- **Tool Holder collision avoidance** (see page 101) — You can now automatically clip toolpaths which would cause a tool holder collision or a gouge.
- **Machine maximum stock** (see page 103) — A new machine maximum stock setting enables you to machine as much stock as possible without retracting the tool.
- **Isoline and Flowline features** (see page 104) — A new sequence option for isoline and flowline milling gives greater control over the creation of toolpaths.
- **Configuring counterbore operations** (see page 106) — A new **Touch off at the shoulder** option enables you to choose the z-zero position for a counterbore operation.
- **Default auto-chamfer tool** (see page 108) — The **Machining Attributes** dialog includes the ability to set a default tool for chamfer operations.
- **Setting toolpath boundaries** (see page 109) — You can use a clipping curve to restrict z-level roughing to a specific area of a part.
- **Creating 5-axis patterns** (see page 111) — 5-axis milling patterns can be specified for any axis.

Turning

- **Improved machining time for dips** (see page 114) — FeatureCAM uses a rapid move to machine dips smaller than the depth of cut.
- **Synchronizing operations** (see page 115) — Improved synchronization points for multi-turret machines make it easier to manage and update synchronization groups.
- **Turning curves** (see page 117) — Follow and pinch turning are now available for the face and back-face cycles of curves.
- **Tool properties for multi-turret machining** (see page 119) — You can assign operations to specific turrets in the **Tool Properties** dialog.

- **Multiple part catchers** (see page 120) — Multiple part catcher support is supported through the **Cutoff Properties** dialog and the FeatureCAM API.

Wire EDM

- **Improved reliability for no-core machining** (see page 122) — FeatureCAM can create no core toolpaths that ensure no material remains after machining.
- **Selecting cutting data** (see page 123) — Two new conditions are available for selecting data.

Importing

- **Using Inventor 2015** (see page 126) — FeatureCAM has been certified for use with Autodesk Inventor 2015.
- **Importing Pro/E files** (see page 127) — The feature tree of imported Pro/E files can be displayed.
- **Importing JT files** (see page 127) — You can load solids from JT files in addition to surfaces.
- **Importing PDF files** (see page 128) — You can import curves from Acrobat files using Delcam Exchange.

Add-ins and extensions

- **Using add-ins** (see page 130) — The **Macro Add-ins** dialog has been updated to improve the management of add-ins and macros.
- **The Add-in Library** (see page 132) — FeatureCAM now includes a catalog of add-ins that were previously only available on the website. The **Add-in Library** is a new feature that enables you to quickly view and select add-ins from the catalog.
- **Turn-curve tolerance** (see page 134) — A new TurnCurveTolerance.bas add-in enables you to tolerance multiple segments in turned or internal bore features.
- **Exporting NC programs to NCSIMUL** (see page 137) — The FeatureCAMToNCSIMUL.dll add-in can now export turn/mill programs as well as milling programs.
- **Extended machine support** (see page 138) — Add-in support for traveling steady rests, bar fed mills, and the Mori Seiki NTX1000 is available for FeatureCAM.

XBUILD

- **Custom formats** (see page 140) — The **Custom Format Name** dialog has an enhanced user interface to support the creation of naturalistic process names.

- **Customizing cutting data from Wire EDM** (see page 141) — You can now configure the headings of the cutting data table for use with Wire EDM.
- **Saving CNC files** (see page 144) — Post numbers can be used to identify individual output files for Fanuc-style multi-turret lathes.
- **Configuring reserved words** (see page 146) —The **Word Info** dialog includes a new option that enables you remove *all* non-significant zeros from numeric outputs.
- **Using the Reserved Words dialog (see page 147)** — The **Reserved Words** dialog is easier and quicker to use.

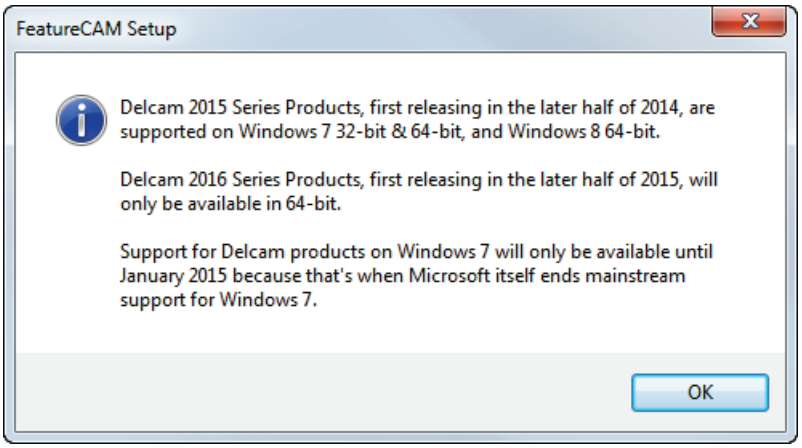
User interface improvements

FeatureCAM 2015 R1 contains the following changes and improvements to the user interface:

- **Installing FeatureCAM** (see page 84) — During installation, a dialog reminds you to prepare to update to Windows 8.
- **SQL server authentication** (see page 85) — FeatureCAM can now record the credentials for accessing the tools databases held in SQL Server.
- **Accessing machining attributes** (see page 86) — The **Machining Attributes** dialog can be opened directly from the **Part View** panel.
- **Customizing the Toolbox** (see page 86) — The width each view in the Toolbox can be customized separately.
- **Adding new stock materials** (see page 88) — The **New Stock Material** dialog prevents the creation of invalid stock names.
- **Improved feature recognition** (see page 89) — Feature recognition automatically excludes clamps from solids.
- **Recognizing Face features** (see page 90) — You can now recognize multiple Face features simultaneously.
- **Snapping modes** (see page 91) — The **Snap Mode** toolbar includes a new option that allows the use of the centre points of cylinders and cones.
- **Creating solids** (see page 92) — The **Finish** button of the **Solid Wizard** is configurable in the same way as the **Features Wizard**.
- **Polygonal curves** (see page 94) — A new option is available for creating polygonal curves.
- **Creating gears** (see page 96) — The **Gears** dialog now includes alternative methods of specifying curve information. A separate tab reports the properties of the calculated gear.
- **Command line options** (see page 98) — FeatureCAM's command line interface has been extended.
- **Selecting surfaces** (see page 99) — Selecting the surfaces of long narrow models has been improved.

Installing FeatureCAM

Following Microsoft's decision to withdraw support for Windows 7 in 2015, Delcam will also be withdrawing support for this operating system. To enable you to prepare for this change, FeatureCAM 2015 R1 displays the following reminder when it detects you are installing it on a computer running Windows 7. or a 32-bit version of Windows 8:



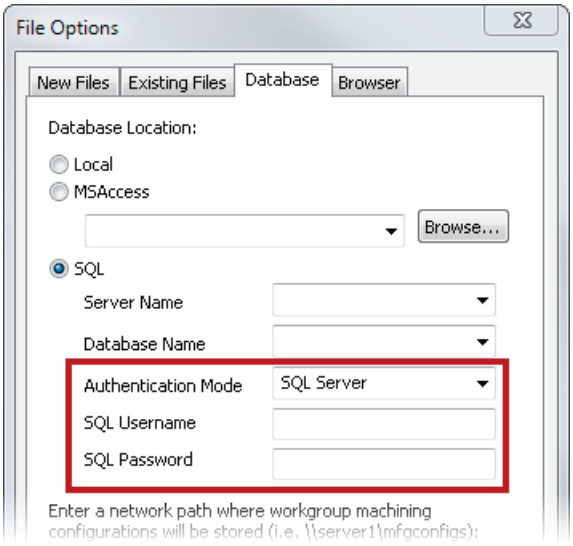
If you want to continue using FeatureCAM updates after June 2015, you must update your computer to a 64-bit version of Windows 8.

SQL server authentication

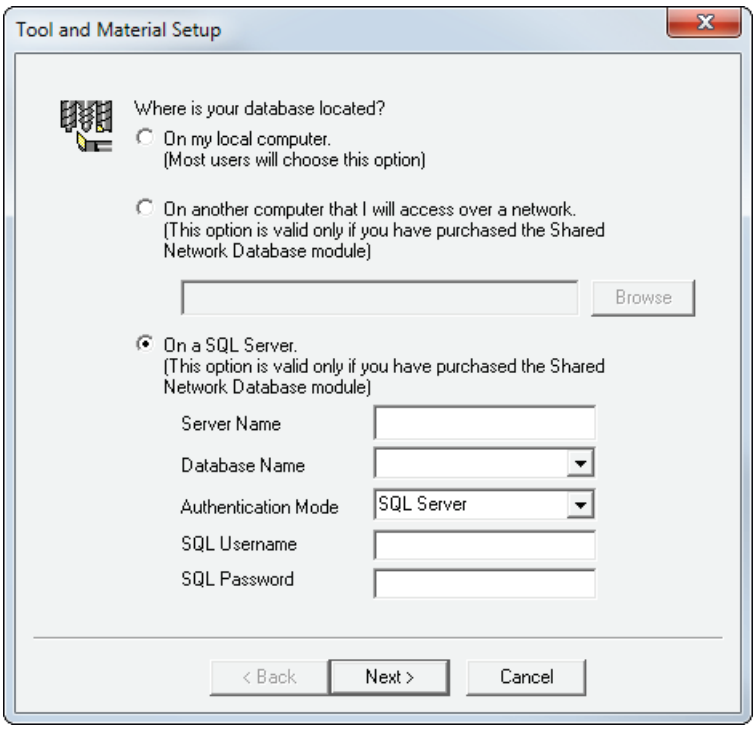
You can now specify a username and password for SQL authentication. When you use SQL authentication, the **Database Name** list only displays databases to which you have access.

There are new options in these places:

- The **Database** tab of the **File Options** dialog:



- In the INITDB window:



To use SQL authentication:

1. Display the **Database** tab of the **File Options** dialog or the INITDB window.

2 Select **On a SQL server**.

3 Enter a **Server Name**.

4 In the **Authentication Mode** list, select **SQL Server**.

5 Enter a **SQL Username** and a **SQL Password**.

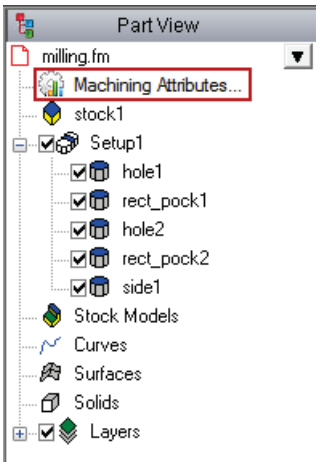
6 In the **Database Name** list, select a database. The list displays only the databases for which you have access rights.

Accessing machining attributes

The **Machining Attributes** dialog can now be opened from the **Parts View** panel as well as from the **Manufacturing > Machining Attributes** menu option. This streamlines the document creation process by enabling you to quickly view and change the attributes of the part you are working with.

To open the **Machining Attributes** dialog from the **Part View**:

1 Double-click the **Machining Attributes** option below the document name. For example:



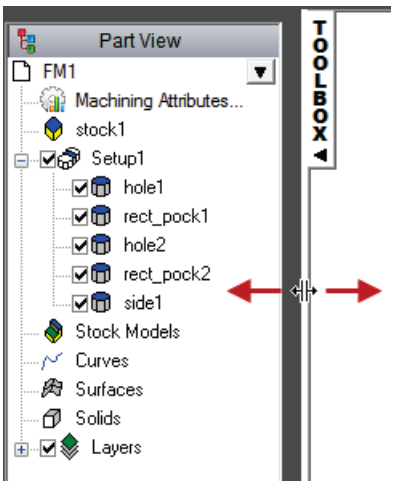
2 Update the machining attributes, and then click **OK** to save your changes and close the dialog.

Customizing the Toolbox

FeatureCAM now remembers the last width change for each view in the **Toolbox** panel. This enables you to customize the display of the **Steps**, **Part View**, and **Browser** panels individually, and then have FeatureCAM automatically change the **Toolbox** width each time you switch between them. The separate widths are saved when you close the program, which means that your changes are remembered between sessions and can be shared across installations using the ezfm_ui.ini file.

To resize the **Toolbox** views:

1. Select **Part View**.
2. Position the cursor over the **Toolbox** border. When the cursor changes to ⇹, left-click and drag the border to its new position. For example:



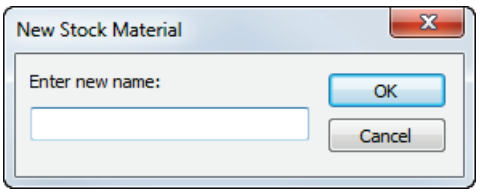
3. Repeat steps **1** and **2** for the **Steps** view and the **Browser** view.

Adding new stock materials

To avoid creating names that cause errors, the names of new materials are now restricted to capital letters (A-Z), numbers (0-9), underscores (_), and dashes (-). To enable you to type normally, lower-case letters are automatically converted to upper-case, and spaces are automatically converted to underscores.

To create a new stock material:

1. Select the **Manufacturing > Feeds/Speeds and Cutting Data Tables** menu option. The **Feeds/Speeds and Cutting Data Tables** dialog is displayed.
2. Click the **New Stock Material** button. The **New Stock Material** dialog is displayed.

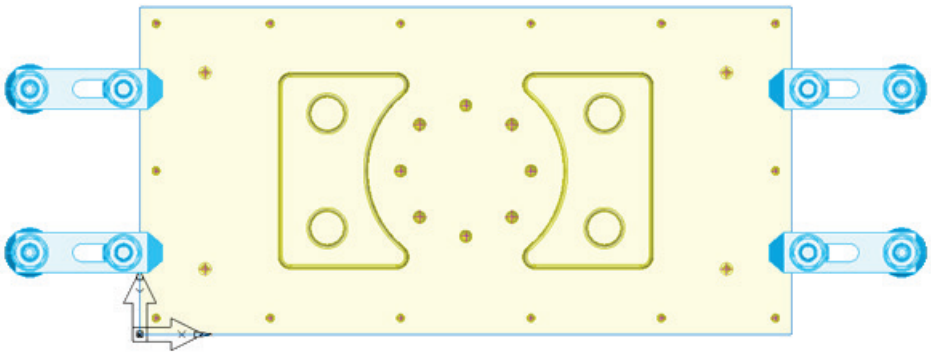


3. Enter a name for the material and click **OK** to save your changes and close the dialog. The name is displayed in the **Material** list.
4. Complete the details of the material in the **Feeds/Speeds and Cutting Data Tables** dialog.
5. Click **OK** to save your changes and close the dialog.

Improved feature recognition

Previously, feature recognition identified and selected clamps as well as features on the part itself. This meant that when using feature recognition on a model that included clamps, you would often need to deselect clamp features.

In FeatureCAM 2015 R1, the feature recognition algorithm has been refined to automatically exclude the features of clamps from the recognition process, so cutting out unnecessary work and enabling you to create documents more quickly.

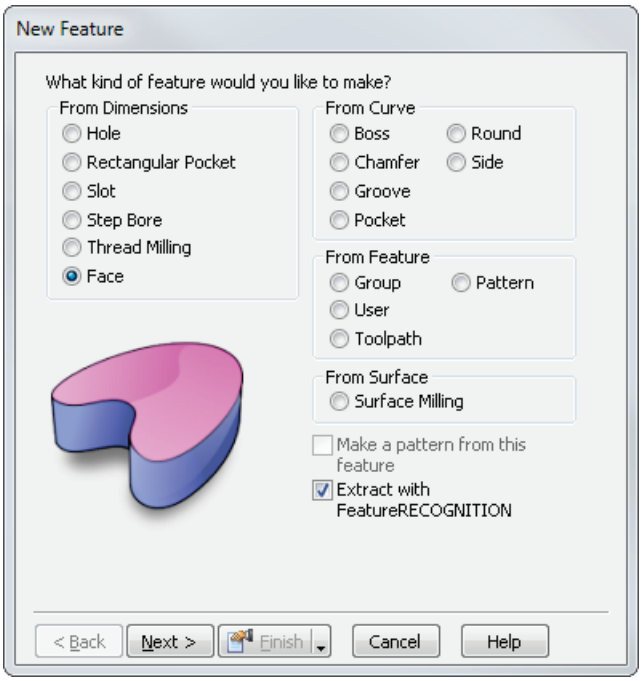


For example, to identify sides using feature recognition:

1. Open the solid.
2. Right-click the clamps and select **Use Solids as Clamp** from the context menu.
3. Open the **New Feature Wizard**.
4. Under **From Curve**, select the **Side** option.
5. Select the **Extract with FeatureRECOGNITION** check box, and click **Next**.
6. In the **Feature Extraction** dialog, select **Automatic recognition** and click **Next**. The **Feature Recognition Options** dialog is displayed, and only the side features on the part are selected.
7. Complete the wizard and click **Finish**.

Recognizing Face features

You can now use FeatureRECOGNITION to recognize multiple Face features simultaneously. This makes it easier to program complex parts, because you no longer need to create Face features individually.



To recognize Face features from surfaces:

1. Open a part that contains multiple flat surfaces.
2. In the **Steps** panel, click the **Features** step. The **New Feature** wizard is displayed.
3. In the **From Dimensions** section, select **Face**.
4. Select the **Extract with FeatureRECOGNITION** check box, then click **Next**. The **Surfaces** page is displayed.
5. Select all flat surfaces you want to recognize in the graphics window, and click + to add the selected surfaces.
6. Click **Next**.

   A message asks if you want to create multiple features.
7. Click **Yes** to create the features and close the wizard.

Snapping modes

A new **Snap to cylinder end points** option has been added to the **Snap Mode** toolbar and the **Snap Modes** dialog. Use it to select and work with the center end points of cylinder and cone axes.

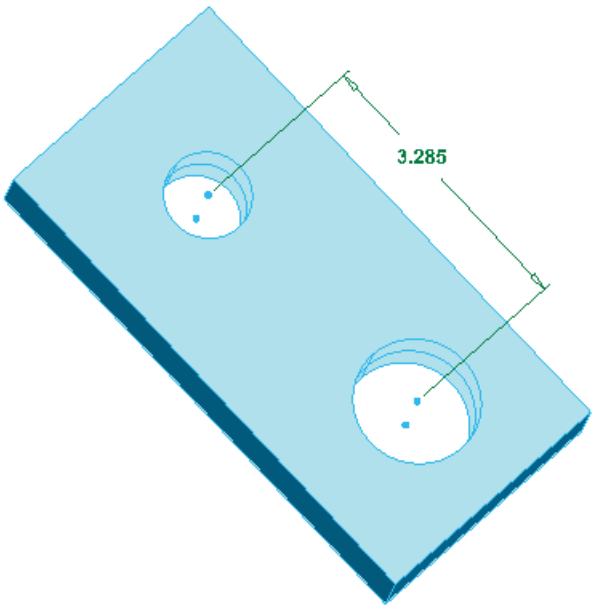
To select the cylinder end points:

1. Use one of these methods to select **Snap to cylinder end points** mode:

   - In the **Snap Modes** dialog, select .
   - In the **Snap Mode** toolbar, click the **Snap to cylinder** button.

   In the Graphics view, selection points are displayed at the center end points on the axis of each cylinder and cone.

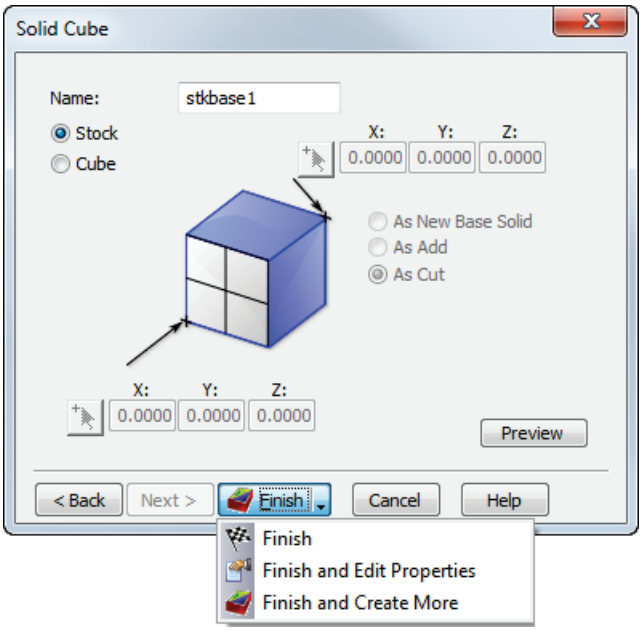
2. Click the selection points to construct geometry or to display the distance between points. For example:



*If the **Snap to cylinder** button is not shown in the **Snap Mode** toolbar, select the toolbar in the **Toolbars** tab of the **Customize Toolbars** dialog, and click **Reset Selected**.*

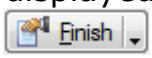
Creating solids

To improve usability, a new **Finish** button has been added to the **Solid Wizard**. It works in the same way as the **Finish** button in the **Features Wizard**, and enables you to speed up the creation of multiple solids. For example:



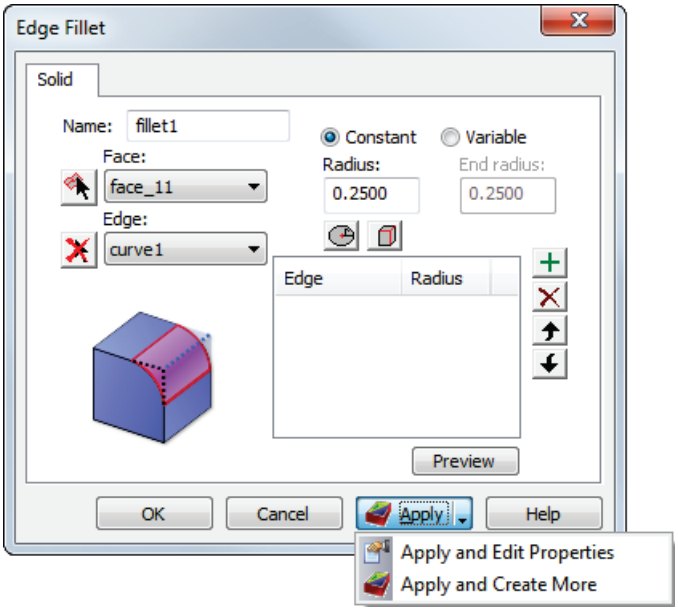
To create solids using the **Solid Wizard**:

1. Open the **Solid Wizard**.
2. Select a method of solid construction.
3. Select a constructor.
4. When you have completed your setting changes, click the down arrow on the **Finish** button and choose an option. Select:
   - **Finish** to create the solid and close the wizard.
   - **Finish and Edit Properties** to close the wizard and open the properties dialog for the solid you created. You can then view and edit the settings.
   - **Finish and Create More** to create the solid and return to the start of the wizard. You can then create a new solid.

   FeatureCAM remembers your selection and, when you next use the wizard, the icon displayed on the button indicates its current state. For example: 

indicates the button is currently in **Finish and Edit Properties** mode.

Similarly, you can create multiple items when using the **Solids** toolbar to create items. For example:



To create solids from the properties dialogs:

1. Select an option in the **Solid** toolbar. The properties dialog for the solid is displayed.
2. Edit the settings.
3. Click the down arrow on the **Apply** button and choose an option. Select:
   - **Apply and Edit Properties** to save your changes and continue editing the current solid.
   - **Apply and Create More** to save your changes. Each time you click **Apply**, FeatureCAM creates a new solid.

   As with the Solid Wizard, FeatureCAM remembers your selection, and an icon indicates the button's current state.

Polygonal curves

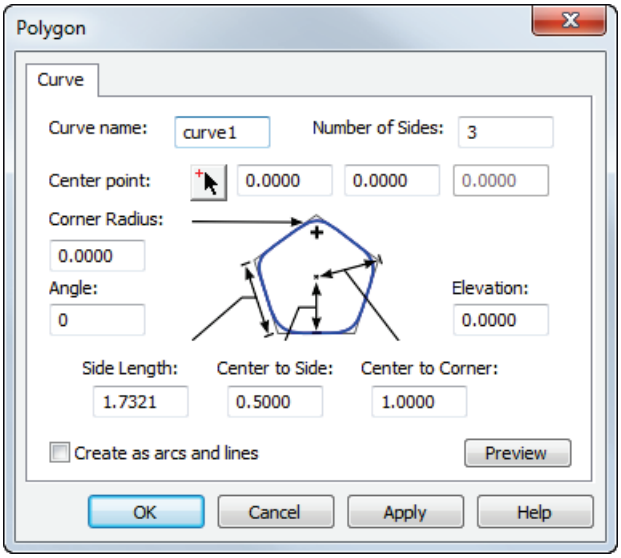
FeatureCAM 2015 R1 contains a new Polygon tool. Use it to create a curve in the shape of a regular polygon.

To create a polygonal curve:

1 Open the **Polygon** dialog in one of these ways:

- In the **Curve Wizard**, select **Other methods**, select **Polygon**, and click **Next**.
- On the **Curves and Surfaces** toolbar, click the **Polygon** button in the **Curve** menu.
- Select **Construct > Curve > Other Methods > Polygon**.

The **Polygon** dialog is displayed.



2 Enter a **Curve name**.

3 Enter the **Number of sides** in the polygon. For example, to create a regular hexagon, type **6**.

4 Enter the x and y coordinates of the **Center point**, or click and select the location in the graphics window.

5 To fillet the corners, enter a **Corner Radius**.

6 To rotate the rectangle counter-clockwise from the X axis, enter an **Angle** in degrees.

7 By default the curve is created on the UCS plane. If you want to position the curve on a parallel plane, enter an **Elevation**.

8 Size the polygon using one of these methods:

- In the **Side length** box, enter the length of the sides.

- In the **Center to side** box, enter the perpendicular distance between the curve center and the sides.
- In the **Center to corner** box, enter the distance between the curve center and the corners.

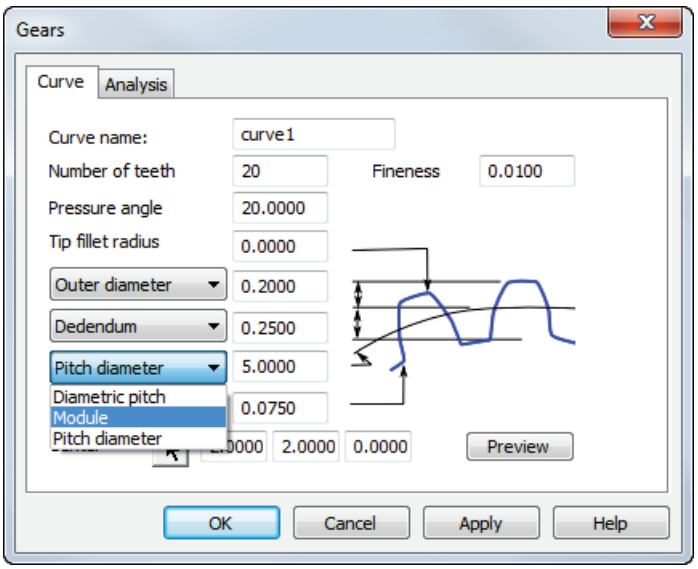
9 To convert the curve into arcs and lines, select the **Create as curves and lines** check box.

10 Click **Preview** to display the polygon in the graphics window.

11 Click **OK** or **Finish** to close the dialog.

Creating gears

The **Gears** dialog has been extended to allow the use of alternative parameters to specify gear curves. You can now specify Outer diameter, Root diameter, and Diametric pitch or Module measurements, making it easier for you to create curves directly from the specification. In addition, a new **Analysis** tab enables you to view the properties of the calculated gear and the measurement over pins values.



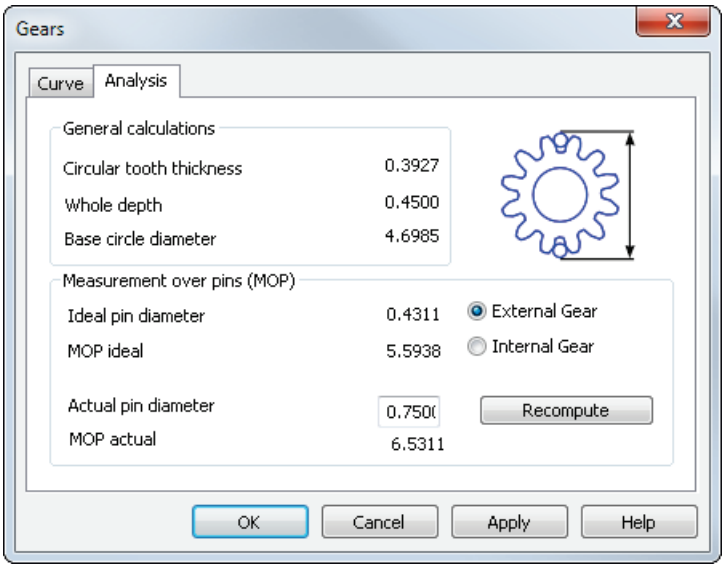
To create a gear curve:

1. Open the **Gears** dialog in one of these ways:
   - In the **Curve Wizard**, select **Other methods**, then **Gears**, and click **Next**.
   - On the **Curves and Surfaces** toolbar, click the **Gears** button in the **Curve** menu.
   - Select the **Construct > Curve > Other Methods > Gears** menu option.

   The **Gears** dialog displays the **Curve** tab.
2. Enter a **Name** for the curve.
3. Enter a **Firmness** value to determine the number of control points in the tooth profile. The smaller the value; the greater the number of control points used to sample the curve.
4. Complete the gear specification:
   - **Number of teeth** — Enter the number of teeth on the gear.
   - **Pressure angle** — Enter the acute angle, in degrees, between the tangent to the two base circles and a normal to the line connecting the gear centers.

- **Tip radius** — Enter the radius of the tooth corner.
- **Addendum** — Enter the radial distance from the pitch circle to the outermost point of the tooth. Alternatively, select **Outer diameter** and enter the outside diameter of the gear.
- **Dedendum** — Enter the radial distance from the depth of the tooth trough to the pitch circle. Alternatively, select **Root diameter** and enter the root diameter of the gear.
- **Pitch diameter** — Enter the diameter of the pitch circle. Alternatively, select **Module** and enter the pitch diameter divided by the number of teeth, or select **Diametric pitch** and enter the number of teeth divided by the pitch diameter.
- **Root fillet radius** — Enter the radius of the fillet at the bottom of each tooth.
- **Center point** —Enter the coordinates for the centre of the gear curve, or click and pick the curve in the Graphics window.

5 Select the **Analysis** tab to view the calculated dimensions of the gear, the calculated pin gauge diameter and the outside measurement over pins diameter.



*The **Analysis** tab is not displayed when you create a gear using the **Curve Wizard**.*

6 If you want to change the calculated diameter of the pin gauge, enter a new value in the **Actual pin diameter** box, and click **Recompute**.

7 Click **OK** (or **Finish** if you are using the wizard).

Command line options

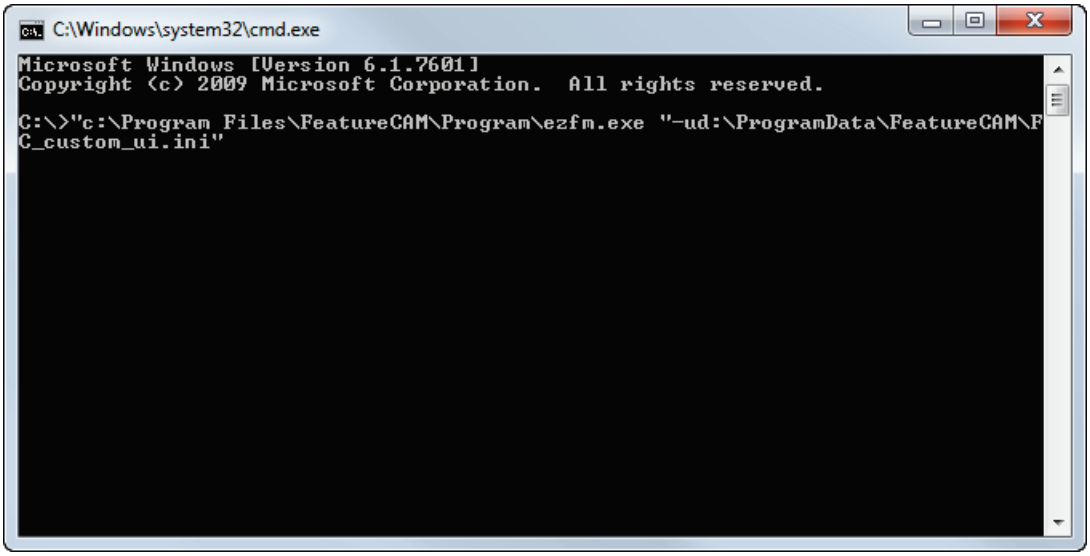
The command line interface for ezfm.exe, the FeatureCAM executable, has been extended. You can use it to control administrative functions, as well as to load customized user-interface and manufacturing setups. The following commands are available:

-b	Load the specified VB add-in from the specified path.
-c	Run the script at the specified path.
-dbug	Send the debug output to the console.
-debugtofile	Send the debug output to file ezfm_debug.txt
-help or -?	List the command line options available for FeatureCAM.
-i	Load the settings from the specified FeatureCAM .ini file.
-l	Specify the network license for FeatureCAM.
-m	Load the manufacturing settings from the specified ezfm_mfg.ini file. For example: ezfm -uc:\ProgramData\FeatureCAM\ezfm_mfg.ini
-noflex or -norms	Skip the license checks.
-regserver	Register the FeatureCAM library
-u	Load the user-interface settings from the specified ezfm_ui.ini file. For example: ezfm -uc:\ProgramData\FeatureCAM\ezfm_custom_ui.ini
-viewal	Send debug graphics to this ezfm instance

To open FeatureCAM using a command line option:

1 Open a **Command Line** window.

2 Type **ezfm.exe**, followed by the parameter. For example:



You must enter the full path of the executable and any specified files when they are not located in the currently selected directory.

3 Press the **Enter** key to execute the command.

Commands are not case-sensitive.

Selecting surfaces

Previously, it could be difficult to select surfaces at each end of a long narrow model. This was because rotating the part would place the view point inside the model causing the outer surfaces to disappear. The problem has been fixed in FeatureCAM 2015 R1.

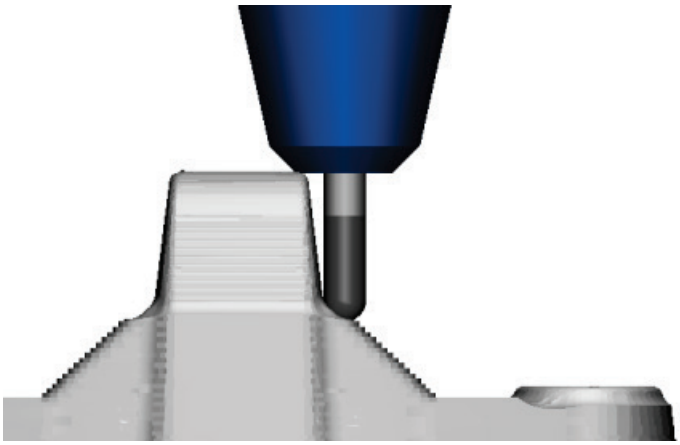
Milling Improvements

FeatureCAM 2015 R1 contains the following changes and improvements to the Milling module:

- **Tool Holder collision avoidance** (see page 101) — You can now automatically clip toolpaths which would cause a tool holder collision or a gouge.
- **Machine maximum stock** (see page 103) — A new machine maximum stock setting enables you to machine as much stock as possible without retracting the tool.
- **Isoline and Flowline features** (see page 104) — A new sequence option for isoline and flowline milling gives greater control over the creation of toolpaths.
- **Configuring counterbore operations** (see page 106) — A new **Touch off at the shoulder** option enables you to choose the z-zero position for a counterbore operation.
- **Default auto-chamfer tool** (see page 108) — The **Machining Attributes** dialog includes the ability to set a default tool for chamfer operations.
- **Setting toolpath boundaries** (see page 109) — You can use a clipping curve to restrict z-level roughing to a specific area of a part.
- **Creating 5-axis patterns** (see page 111) — 5-axis milling patterns can be specified for any axis.

Tool holder collision avoidance

You can now automatically clip toolpaths which would cause a tool holder collision or a gouge. This makes it easier to program complex parts safely, and you can fix problems without having to simulate the toolpaths repeatedly to find the cause of collisions.

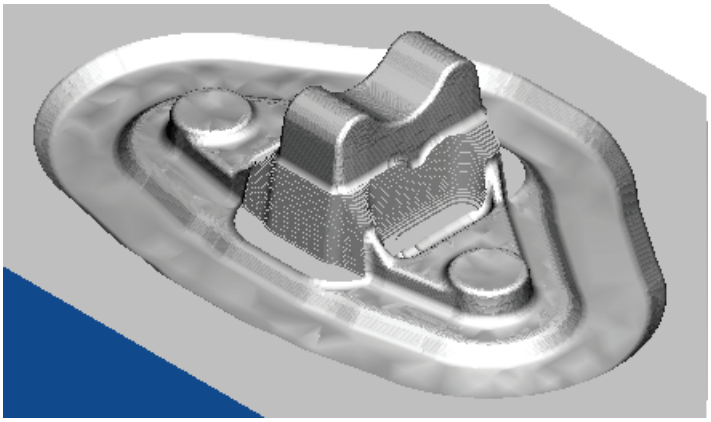


Holder collision clipping only checks for collisions between the tool holder and part surfaces or check surfaces, not remaining stock.

To use tool holder collision and gouge avoidance:

1. Create or open a part containing a Surface Milling feature.
2. Double-click the feature in the **Part View** to display the **Surface Milling Properties** dialog.
3. Select an operation in the Tree View.
4. In the **Strategy** tab, select **Holder collision clipping**.
5. Select a pass of the operation in the Tree View.
6. In the **Milling** tab, enter a **Holder clearance** and **Shank clearance**. The toolpath is clipped when the holder or shank move within these distances of a surface.
7. Click **OK** to accept.

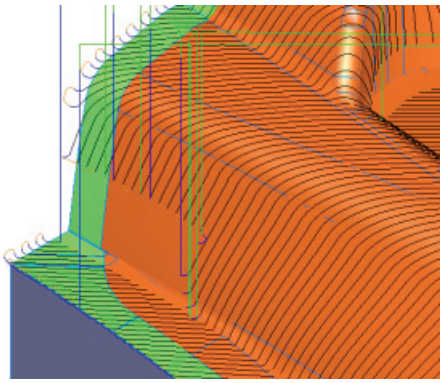
8 Simulate the toolpath to show which areas have not been machined:



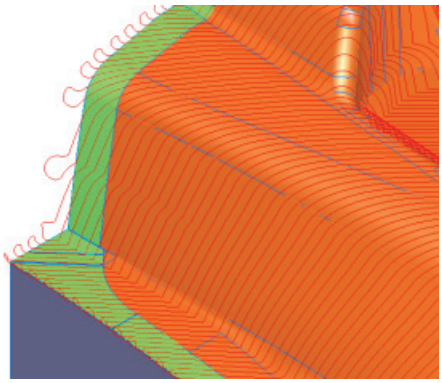
Machine maximum stock

There is a new **Machine maximum stock** setting, which you can use with **Holder collision clipping** (see page 101) to machine as much stock as possible without the tool retracting. This creates smoother toolpaths with fewer retracts, which can improve the surface finish and reduce the machining time.

Machine maximum stock deselected:

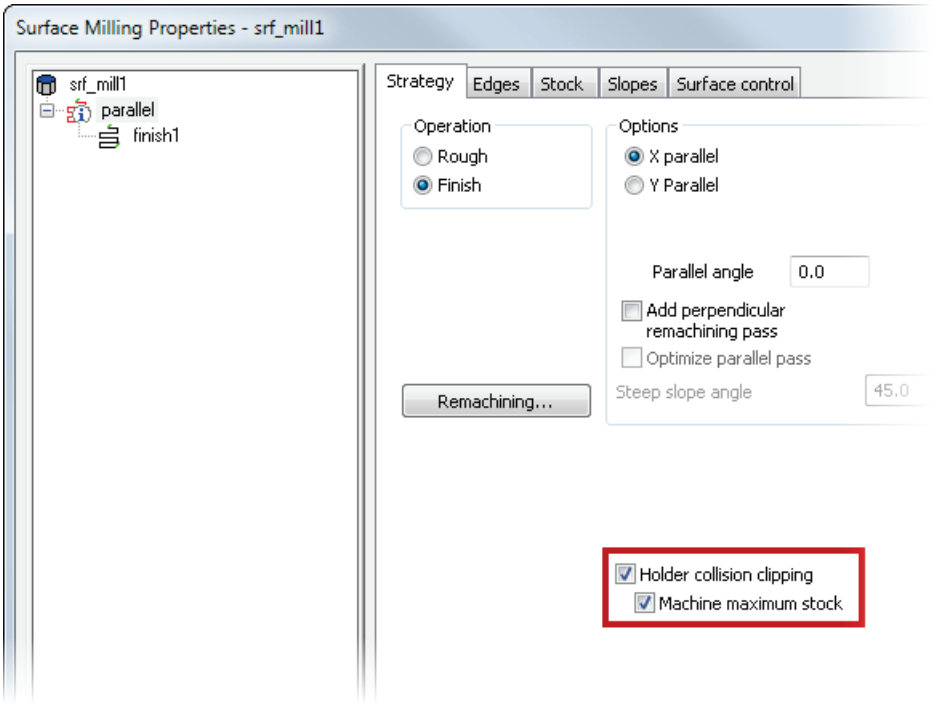


Machine maximum stock selected:



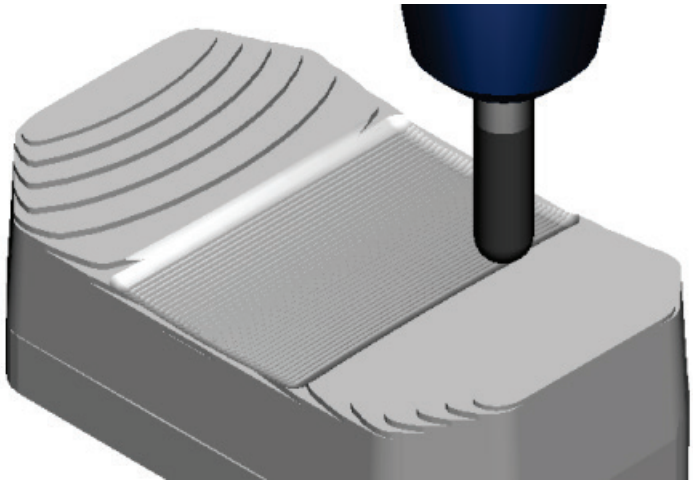
On some parts, this may increase air cutting, which could increase the machining time.

On the **Strategy** tab of the **Surface Milling Properties** dialog, select **Holder collision clipping** and **Machine maximum stock**:



Isoline and flowline features

There is a new sequence option for Isoline and Flowline surface milling features, which gives you more control over the toolpaths. You can now create toolpaths which start at the center of the surface and cut outwards, such as for machining towards and up the walls of cavities.



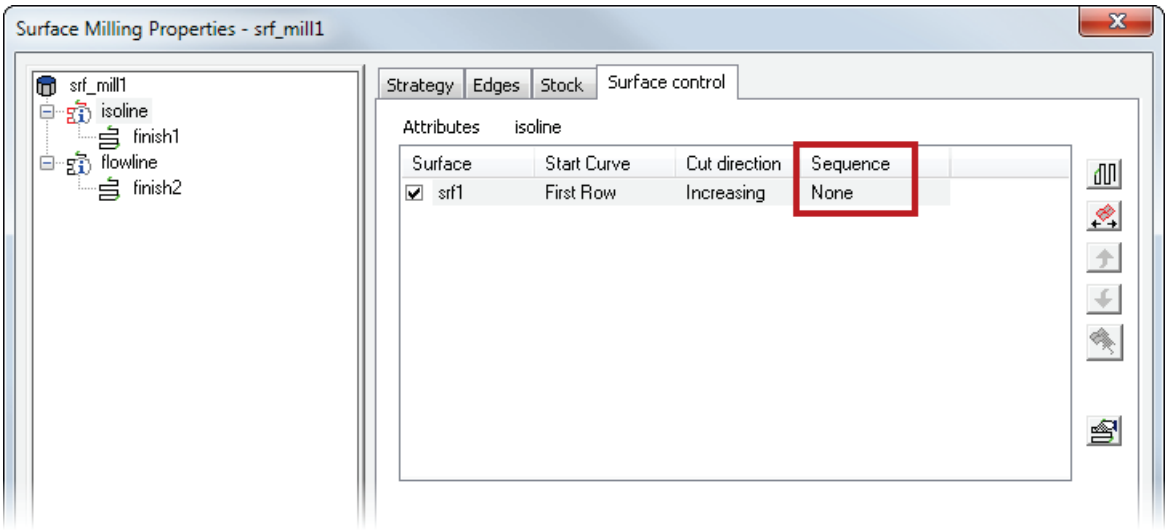
To create an Isoline or Flowline surface milling feature:

1. Import or open a model with surfaces you want to machine.
2. Click **New Feature** on the **Advanced** toolbar. The **New Feature** dialog is displayed.
3. In the **From Surface** area, select **Surface Milling** and click **Next**.
4. In the Graphics window, select the surfaces you want to machine and click . Click **Next**.
5. Select **Choose a single operation** and click **Next**.
6. In the **Finishing Strategies** section, select **Isoline** or **Flowline** and click **Finish**.

To change the **Sequence** for an Isoline or Flowline Surface Milling feature:

1. In the **Part View**, double-click the feature to display the **Surface Milling Properties** dialog.
2. Select the Isoline or Flowline operation in the Tree View.

The **Sequence** is displayed on the **Surface control** tab:



3 Click **Sequence** to cycle through the toolpath sequence options for the selected surfaces. Select:

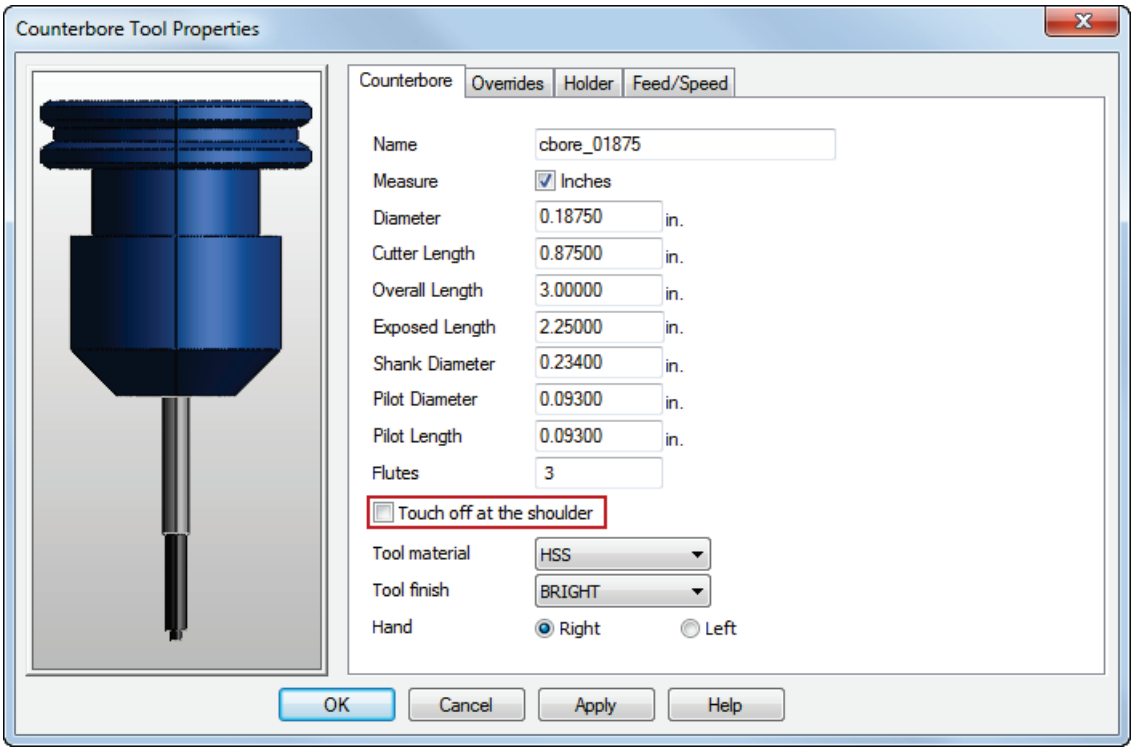
- **None** to create a toolpath that cuts across the surface in the same way as in FeatureCAM 2014 R3. This is selected by default.
- **In to out** to create a toolpath that starts at the center of the surface and cuts outwards.
- **Out to in** to create a toolpath that starts at the outside of the surface and cuts inwards.

*The **Continuous Spiral** option on the **Strategy** tab overrides this setting.*

4 Click **OK** to save your changes.

Configuring counterbore operations

By default, FeatureCAM calculates the z depth of an operation as the sum of the hole's **Bore depth** and the tool's **Pilot length**. In FeatureCAM 2015 R1, the **Counterbore** tab of the **Counterbore Tool Properties** dialog contains a new **Touch off at the shoulder** option, which enables you to locate the tool's z-zero position at the end of the tool flutes (the tool shoulder) instead of at the end of the pilot (the tool tip). This improves the accuracy of the toolpath in 2-axis drilling because it minimizes the effect of the pilot length, which is susceptible to greater variation in tolerance.



To choose the z-zero position for a counterbore operation:

1. Double-click the hole feature in the **Part View**. The **Hole Feature Properties** dialog is displayed.
2. Select the **Counterbore** operation in the Tree view.
3. On the **Tools** tab, double-click the counterbore tool you want to configure. The **Counterbore Tool Properties** dialog is displayed.
4. On the **Counterbore** tab, select the **Touch off at the shoulder** check box to position the tool z-zero at the end of the flutes; deselect the check box to position the tool z-zero at the end of the pilot.
5. Complete any other changes in the dialog.
6. Click **OK** to save your changes and close the **Counterbore Tool Properties** dialog.

7 Click **OK** to save your changes to the **Hole Feature Properties** dialog.

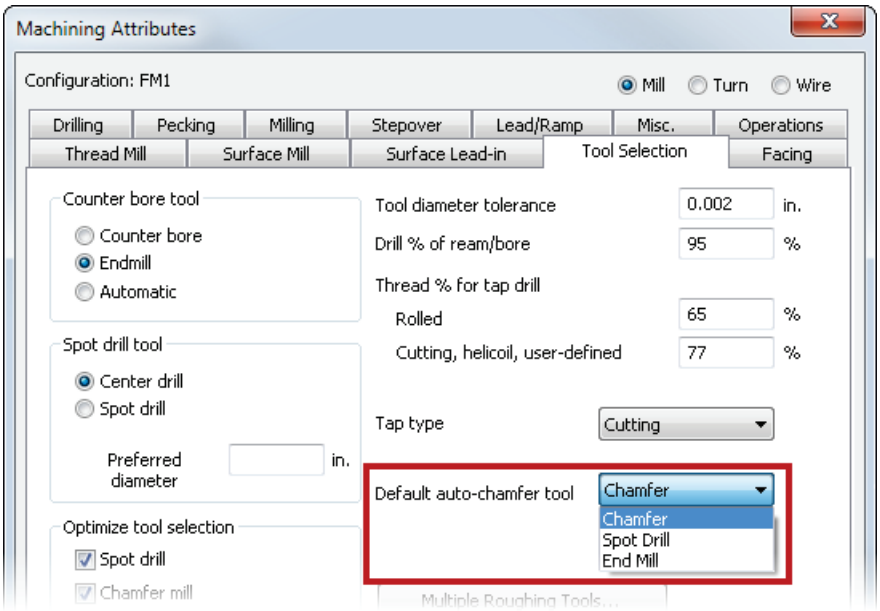
*Selecting the **Touch off at the shoulder** check box has no effect on the simulation of the toolpath.*

Default auto-chamfer tool

You can now set a default tool for chamfer operations. This enables you to create multiple features without having to modify them individually to change the tool.

To set a default chamfer tool:

1. Select **Manufacturing > Machining Attributes** from the menu. The **Machining Attributes** dialog is displayed.
2. In the **Tool Selection** tab, select a **Default auto-chamfer tool** from the list:



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

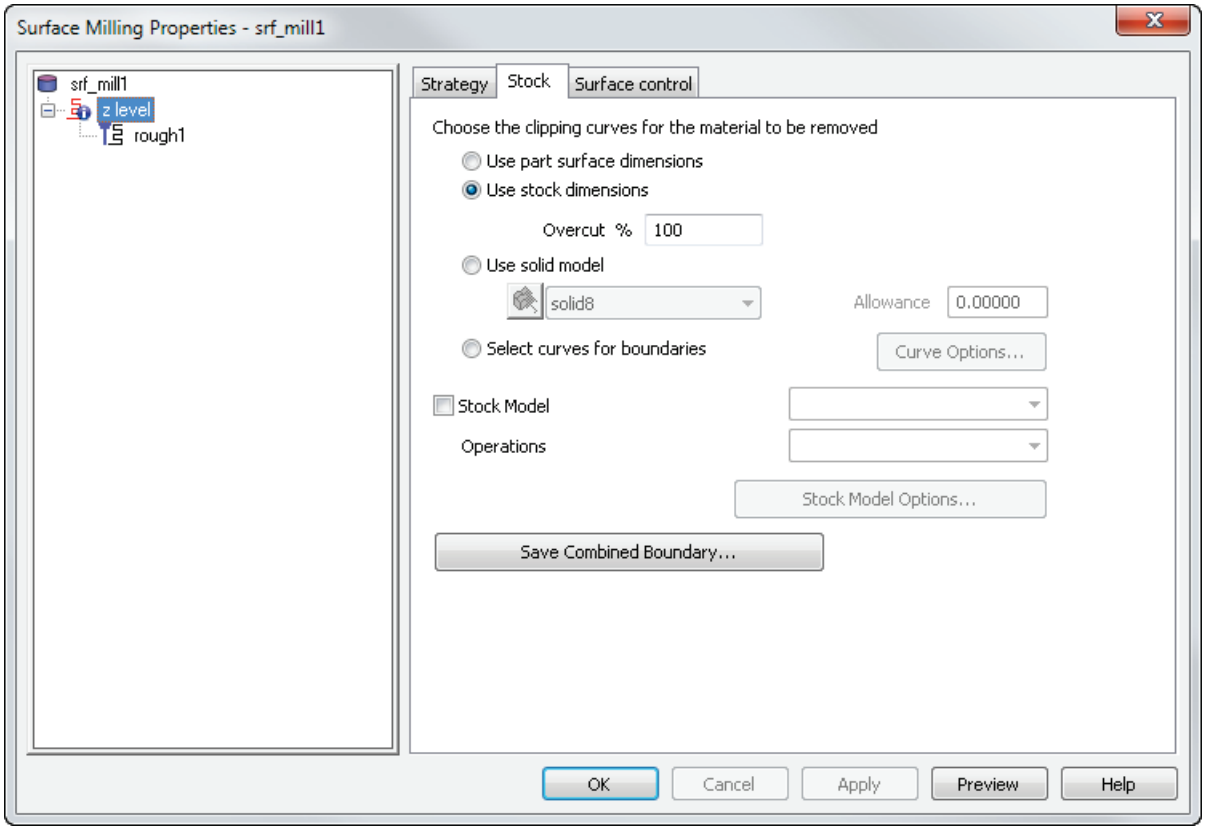
   The selected tool is used for any new chamfer operations you create; any existing chamfer operations that use the default tool are updated.

Setting toolpath boundaries

Previously, in 3D milling, the **Stock Model** option was a clipping-curve alternative in the **Stock** tab of the **Surface Milling Properties** dialog. This meant that when you chose to use a stock model to determine the toolpath boundaries, the machining operation had to be applied to the whole part. In FeatureCAM 2015 R1, the **Stock Model** setting is selectable separately to the clipping curve, enabling you to limit operations to a specific area of a part for z-level rough, parallel rough, and 3-axis finishing strategies.

To specify the boundaries of the toolpath:

1. Double-click the feature in the **Part View**, or right-click the surface and select **Properties**. The **Surface Milling Properties** dialog is displayed.
2. Select the strategy in the tree view. The strategy tabs are displayed.
3. Select the **Stock** tab.



4. Choose a clipping curve. Select:
   - **Use part surface dimensions** to create toolpaths on the surfaces regardless of the location of the stock.
   - **Use stock dimensions** to restrict the toolpaths to the portions of the surface feature that are within the stock.

- **Use solid model** to restrict toolpaths to be within the solid model or STL model that you choose.
- **Select curves for boundaries** to use curves to restrict the toolpath boundaries, or to affect the shape of a spiral toolpath.

5 To use the curve in combination with a stock model:

a Select the **Stock model** check box.

b Select the model from the list.

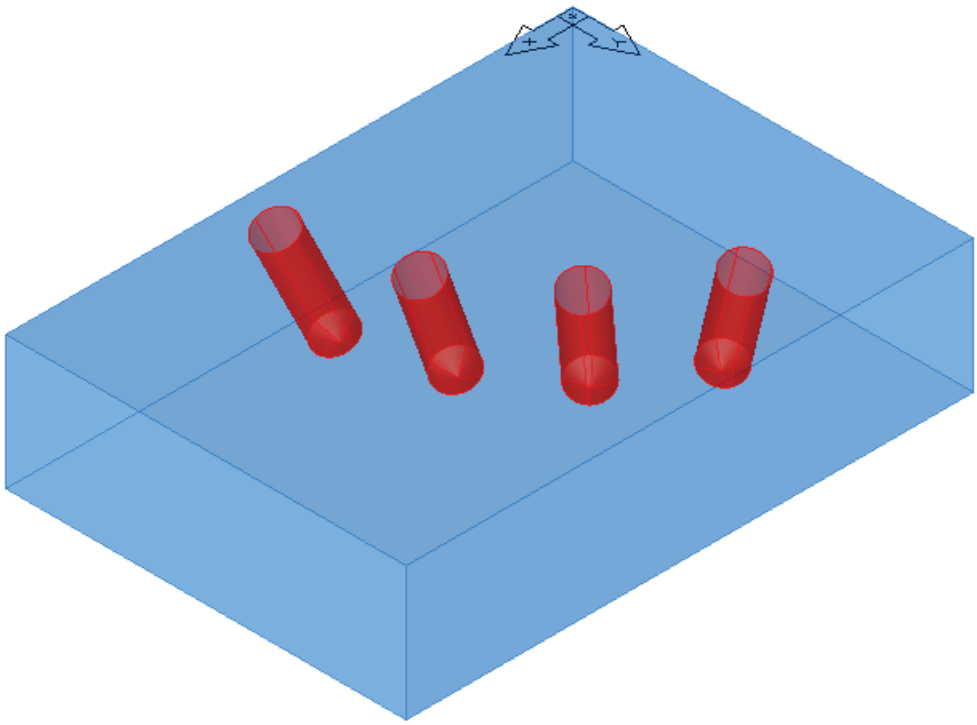
c Select an entry in the **Operations** list.

6 If you want to save the boundary for future use, click **Save Combined Boundary**.

7 Click **OK** to save your changes and close the dialog.

Creating 5-axis patterns

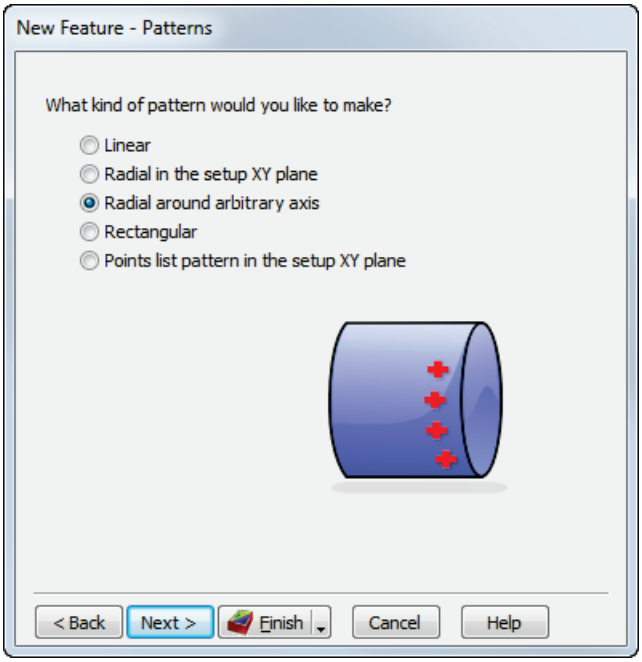
The **New Feature Wizard** now contains an additional option for creating 5-axis milling patterns. With the new **Radial arbitrary index axis** option on the **Patterns** page you can make a hole pattern around any axis, enabling you to program patterns in 5-axis parts quickly and easily. For example:



To create a 5-axis pattern from a new hole:

1. Create a stock using 5th axis positioning in the Stock Wizard, or select **5th axis positioning** in the **Indexing** tab of the **Stock Properties** dialog.
2. In the **New Feature Wizard**, select the **Hole** option, select the **Make a pattern from this feature** check box. Click **Next**.
3. In the **Dimensions** page, enter the details of the hole you want to create. Click **Next**.

4 In the **Patterns** page, select the **Radial arbitrary index axis** option. Click **Next**.



5 In the **Location** page, enter the vector of the central axis and the position of the hole. Click **Next**.

6 In the **Dimensions** page, enter the **Number** of holes in the pattern and the **Spacing angle** between each hole. Click **Next**.

7 In the **Location** page, enter the vector of the indexing axis and the position of its origin. Click **Finish**.

To create a 5-axis pattern from an existing hole:

1 Select **5th axis positioning** in the **Indexing** tab of the **Stock Properties** dialog.

2 In the **New Feature Wizard**, select the **Pattern** option. Click **Next**.

3 In the **Pattern Base** page, select the hole from which you want to create the pattern. Click **Next**.

4 In the **Patterns** page, select the **Radial arbitrary index axis** option. Click **Next**.

5 In the **Dimensions** page, enter the **Number** of holes in the pattern and the **Spacing angle** between each hole. Click **Next**.

6 In the **Location** page, enter the vector of the central axis and the position of the hole. Click **Finish**.

Turning improvements

FeatureCAM 2015 R1 contains the following changes and improvements to the Turning module:

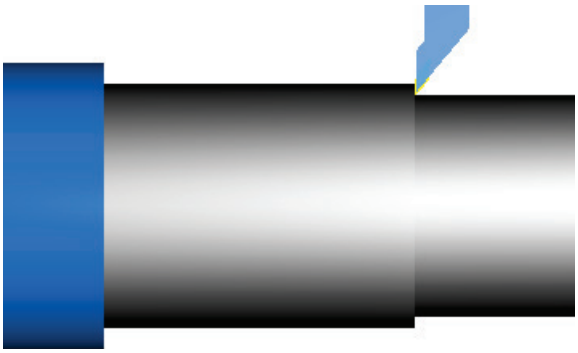
- **Improved machining time for undercuts** (see page 114) — FeatureCAM uses a rapid move to machine undercuts smaller than the depth of cut.
- **Synchronizing operations** (see page 115) — Improved synchronization points for multi-turret machines make it easier to manage and update synchronization groups.
- **Turning curves** (see page 117) — Follow and pinch turning are now available for the face and back-face cycles of curves.
- **Tool properties for multi-turret machining** (see page 119) — You can assign operations to specific turrets in the **Tool Properties** dialog.
- **Multiple part catchers** (see page 120) — Multiple part catcher support is supported through the **Cutoff Properties** dialog and the FeatureCAM API.

Improved machining times for undercuts

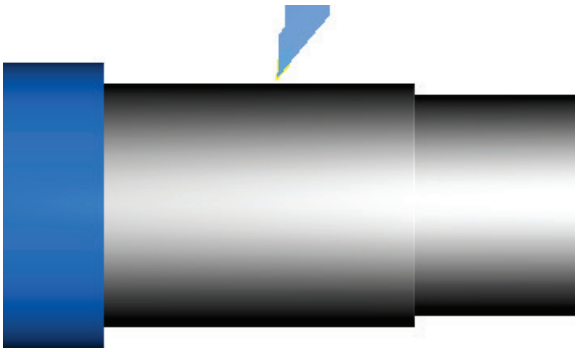
FeatureCAM now creates more efficient toolpaths for features that contain undercuts smaller than the depth of cut. Instead of feeding along a previous cut to machine the undercut, FeatureCAM now uses a rapid move. This prevents the tool from cutting the surface twice, and so reduces machining time, increases tool-life, and prevents problems such as work-hardening on the part.

To machine a turning feature that includes a small undercut in FeatureCAM 2015 R1, the tool:

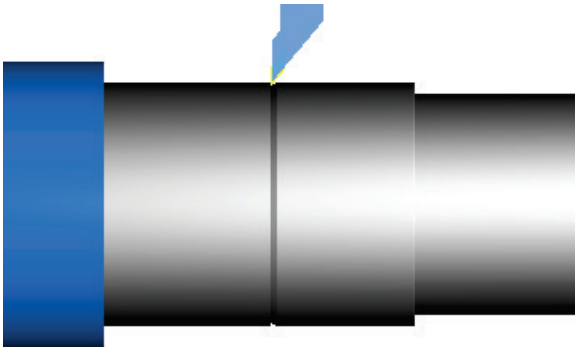
1 Cuts the part to the correct diameters.



2 Retracts and makes a rapid move to the undercut location.



3 Cuts the undercut.

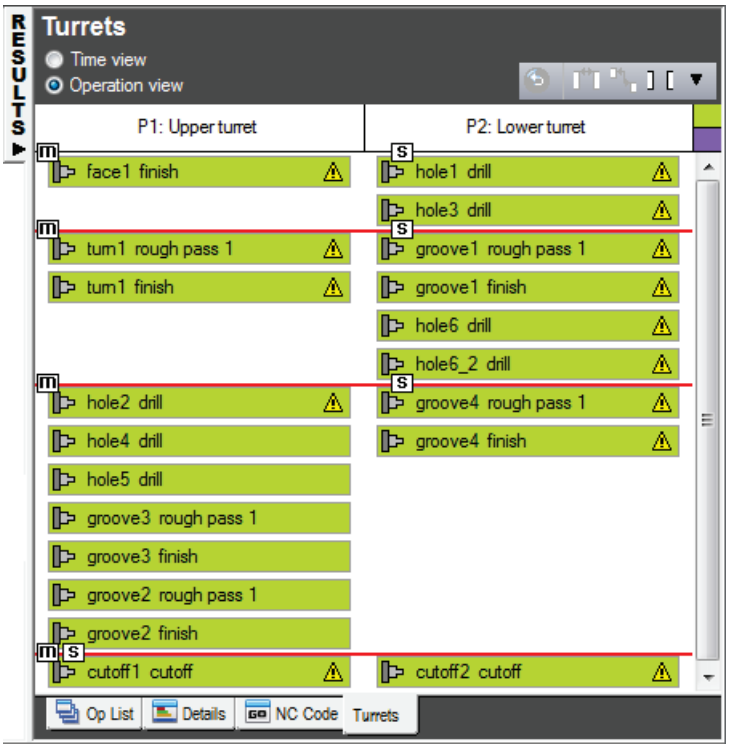


Synchronizing operations

Synchronization points in the **Turrets** tab enable you to specify the order in which operations are performed on multi-turret machines. In FeatureCAM 2015 R1, synchronization points have been redesigned to be independent of operations in the tab, making it easier to change and update synchronization groups, and providing a simplified, more intuitive user interface.

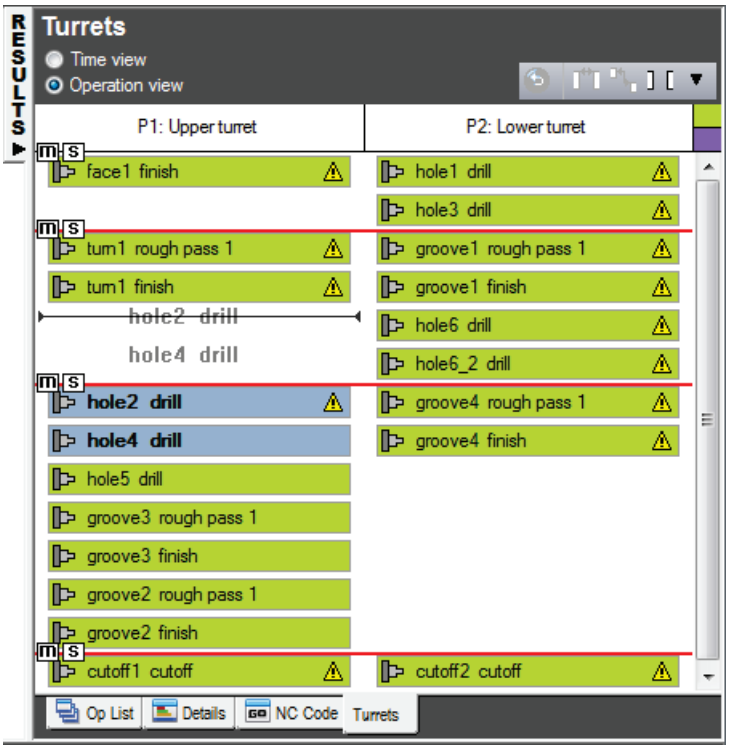
Adding operations to the sequences

When you insert a feature into the **Part View** and **Automatic Ordering** is selected in the **Ops List** tab, FeatureCAM adds the feature's operation to the appropriate synchronization group. For example, a copy of the hole6 feature is automatically assigned to the same synchronization group and turret as the original hole.



Moving operations within the sequence

When you drag operations within the **Turrets** tab, the cursor changes to ⊢——————⊣; when you drop the operations, they are added immediately after it. If the insertion point is at the boundary of two synchronization groups, the position of the mouse cursor above or below the boundary marker determines which group the operations are added to.



In addition, when you drag one operation of an operation pair, both operations are moved together.

Operation pairs are created by FeatureCAM when a feature generates two linked operations. For example, linked operations are created by FeatureCAM's pinch turning feature.

Removing operations from the sequence

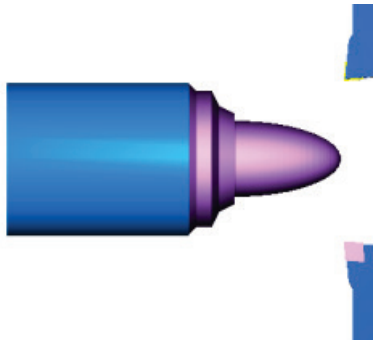
When you delete a feature, or exclude it by deselecting its check box in the **Part View**, FeatureCAM adjusts the remaining operations in the synchronization group. When you reselect a previously excluded feature, it is reinstated at its original position in the synchronization group.

Turning curves

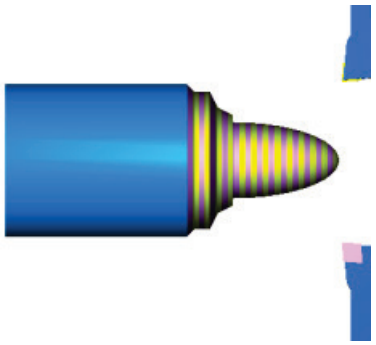
Follow and pinch turning are now available for the face and back-face cycles of curves. This enables you reduce machining time and take full advantage of multi-turret lathes.

To select the cutting method for a curve:

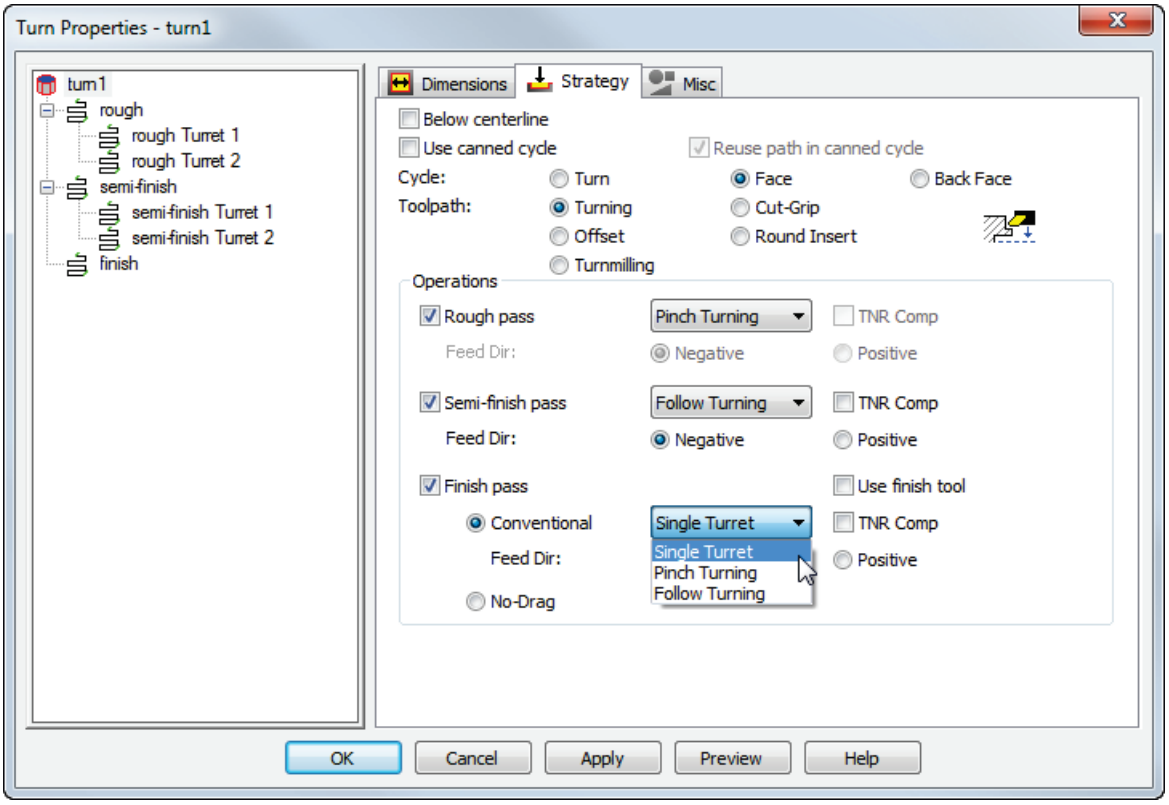
1. Double-click the Turn feature in the **Part View** panel. The **Turn Properties** dialog is displayed.
2. Select the **Strategy** tab.
3. Select the **Turning** option.
4. Under **Operations**, choose a cutting method for each pass you have enabled. Select:
   - **Single turret** to cut the feature using one turret.
   - **Pinch turning** to cut the feature using turrets positioned above and below the axis of rotation. The tools cut simultaneously: the first tool leaves a spiral of material and the second tool cuts the spiral of material left by the first tool. For example:



   - **Follow turning** to cut the feature using turrets positioned above and below the axis of rotation. Each turret uses a standard depth of cut, and the top tool leads the bottom tool. For example:



5 Click **Apply**. The details of the turrets are displayed in the tree view. For example:



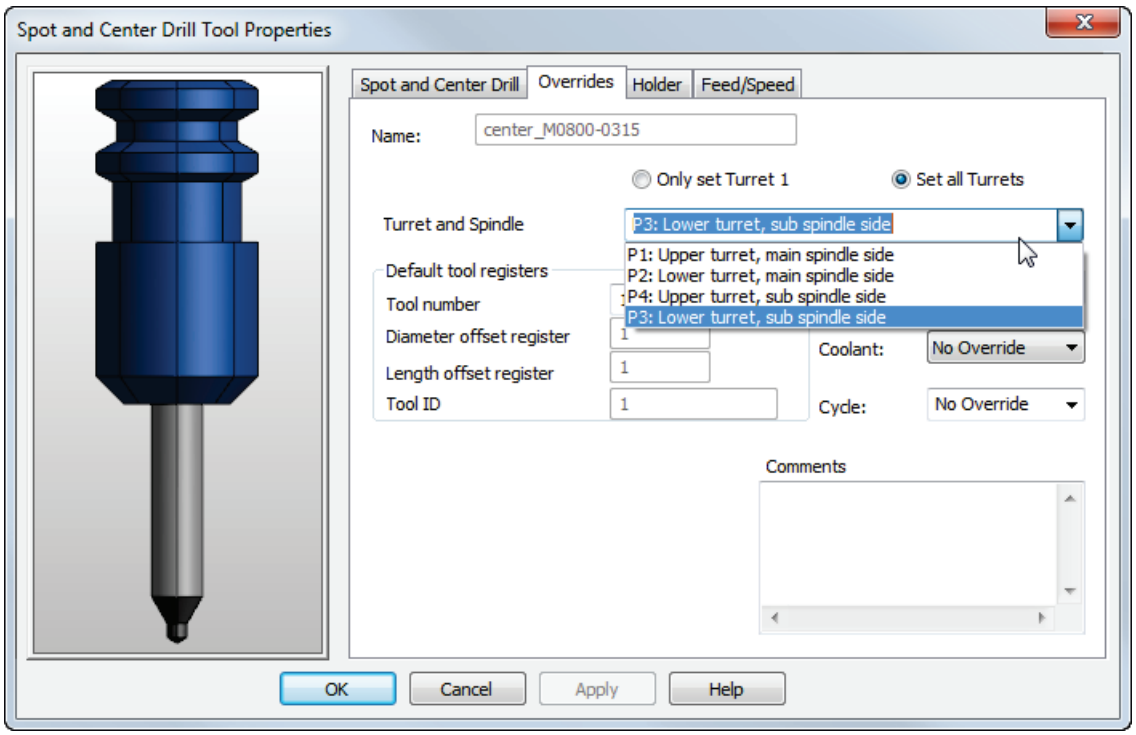
6 Click **OK** to save your changes and close the dialog.

Tool properties for multi-turret machining

The **Override** tab of the **Tool Properties** dialog includes new turret options that enable you to set the tool registers for multiple turrets. Use them to quickly view and edit the overrides of tools that are assigned to different slot numbers on different turrets without using the **Tool Mapping** dialog.

To specify the register settings of turrets in the **Tool Properties** dialog:

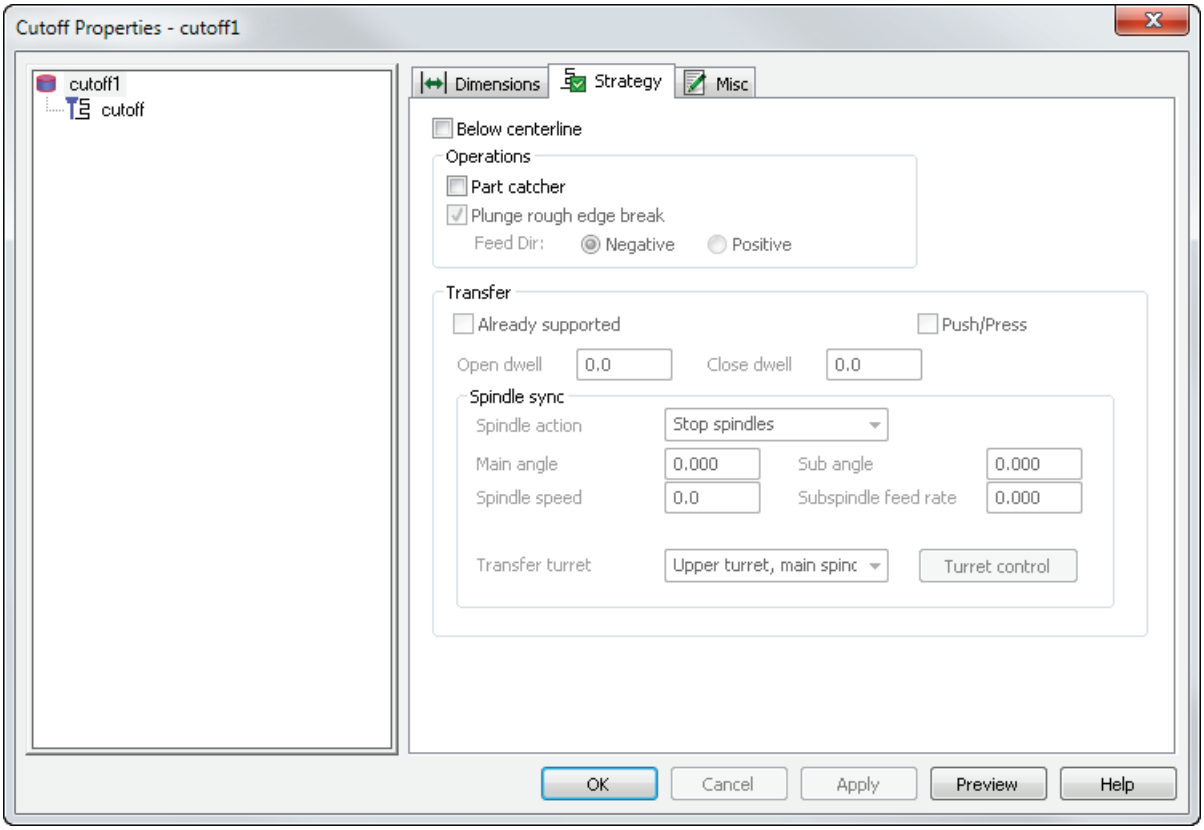
1. Open the **Tool Properties** dialog for the tool.
2. Select the **Overrides** tab.
3. Select the **Set all turrets** option.
4. In the **Turret and spindle** list, select the turret you want to work with. For example:



5. Edit the **Default tool registers** settings for the turret.
6. Repeat steps **4** through **5** for each turret on the machine.
7. Click **OK** to save your changes and close the dialog.
8. If you are asked whether to set the edited tool as override, click **Yes** to use the new tool for the operation; click **No** to continue using the original tool.

Multiple part catchers

FeatureCAM now supports multiple part catchers on Turnmill machines, such as the Nakamura WT-150. Cutoff catchers are supported by selecting the **Part catcher** check box in the **Strategy** tab of the **Cutoff Properties** dialog; other catchers can be implemented by creating user-defined features with FeatureCAM's application programming interface (API).



For more information on implementing multiple part catchers, contact your sales representative.

Wire EDM improvements

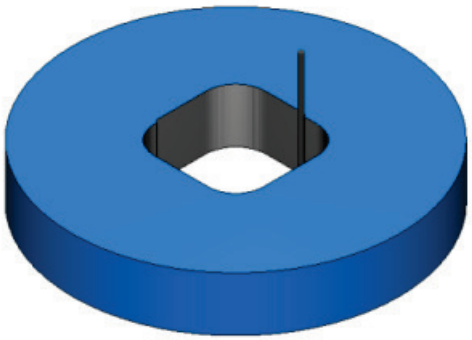
FeatureCAM 2015 R1 contains the following changes and improvements to the Wire EDM module:

- **Improved reliability for no-core machining** (see page 122) — FeatureCAM can create no core toolpaths that ensure no material remains after machining.
- **Selecting cutting data** (see page 123) — Two new conditions are available for selecting data.

Improved reliability for no-core machining

You can create a no-core toolpath that machines an area without creating a slug.

In FeatureCAM 2015 R1, no-core toolpaths are more reliable, and no material remains after machining. This eliminates the risk of loose pieces being left during cutting, which can cause the machine to stop and can damage the nozzle.

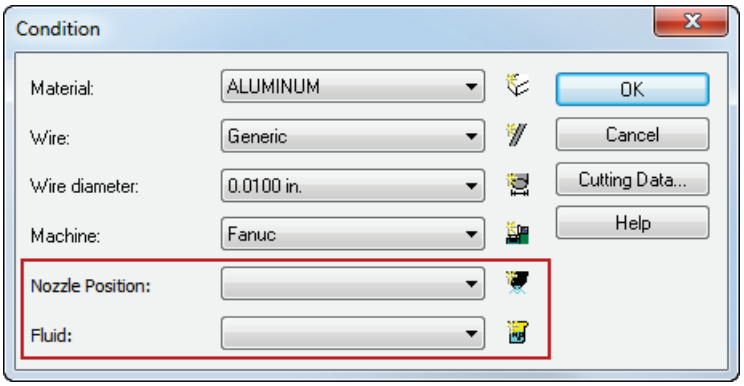


To create a no-core toolpath:

1. Create a 2-Axis Die feature.
2. Display the **Properties** dialog for the Die feature.
3. In the **Strategy** tab, select **Pocketing** or **Zigzag** in the **Operations** list.
4. To add an unmachined area, click **Islands** and select a curve using the **Select Islands** dialog.
5. Click **OK** to close the dialog.

Selecting cutting data

The **Condition** dialog contains two new parameters: **Nozzle Position** and **Fluid**. If your machine supports different nozzle positions or different types of dielectric fluid, you can use these parameters to select machine register settings from the cut-data database.



To select a machine condition for an existing document:

1. Create a Wire EDM document, and open the **Stock Properties** dialog.
2. On the **Dimensions** tab, click the **Condition** button. The new **Nozzle Position** and **Fluid** lists are shown at the bottom of the **Condition** dialog.
3. In each list, select an option to identify the combination of conditions you want to use. If you want to create a new option for a list, click the adjacent button and enter its name. For example, to specify a new **Nozzle Position** option:
   a. Click . The **New Nozzle Position Type** dialog is displayed.
   b. Enter a name for the new position.
   c. Click **OK** to save your change and close the dialog.

   The option is displayed in the **Nozzle Position** list.
4. To check or specify the machine settings for the condition, click the **Cutting Data** button. The **Feeds/Speeds and Cutting Data Tables** dialog is displayed showing the condition you selected:
   - If you have selected a previously defined condition, the machine settings for each pass are displayed in the **Wire EDM** tab. To change a setting, double-click its cell in the table and overtype the value.
   - If you have selected an undefined condition, a message is displayed in the **Wire EDM** tab. Click **New**, select the number of passes required for this condition, and enter the setting values for the condition.

*You can customize the **Wire EDM** table using the **Cutting Condition Names** dialog (see page 141) in XBUILD.*

5 When you have finished, click **OK** to save your changes and close the dialog.

6 In the **Condition** dialog, click **OK**.

*You can also specify the condition in the **Stock** wizard.*

Importing

FeatureCAM 2015 R1 contains the following changes and improvements to importing files from external applications:

- **Using Inventor 2015** (see page 126) — FeatureCAM has been certified for use with Autodesk Inventor 2015.
- **Importing Pro/E files** (see page 127) — The feature tree of imported Pro/E files can be displayed.
- **Importing JT files** (see page 127) — You can load solids from JT files in addition to surfaces.
- **Importing PDF files** (see page 128) — You can import curves from Acrobat files using Delcam Exchange.

Using FeatureCAM with Inventor 2015

FeatureCAM has been certified for use with Autodesk Inventor 2015, an easy-to-use application for the design and documentation of 3D mechanical parts and assemblies. This enables you to use Inventor's latest design tools to create models, and then to quickly import them into FeatureCAM to create features that are ready for machining.

To use Inventor designs in FeatureCAM:

1. Open the **New Part Document** wizard.
2. Select **Open an existing file** and click **Next**. The **Open** dialog is displayed.
3. In the **Files of type** list, select **Inventor (*.ipt, *.iam)**.
4. Select the file you want to work with and click **Open**.
5. In the **New Part Document** wizard, select the type of document you want to create, select the **Wizard** option, and click **OK**. The file is loaded and the **Import Results** dialog is displayed.
6. Select **Use the wizard to establish the initial setup location, stock size, and import features from Inventor**.

   When FeatureCAM is installed on the same computer as Inventor, it accesses the Inventor data and creates hole features as originally designed.
7. Complete the wizard to create the part.

Importing ProE files

FeatureCAM can now display the feature tree of imported Pro/E files. Only solid models are imported from Pro/E; the solid models are imported as surfaces or solids depending on the options you set in the **Import/Export Options** dialog. Only solids are read from a Pro/E file. Any curves or surfaces read must be part of a solid.

*You must have the FeatureCAM **ProE Import** module to import Pro/E files.*

To import CAD files:

1. Open a new or existing part file. You must have a part open to import geometry.
2. Select the **File > Import** menu option. The **Import** dialog is displayed.
3. Select the file you want to import and click **Open**. The **Import Results** dialog is displayed.

*Files must have an extension of ***.prt, *.prt.%number%** (for example foo.prt.8), ***.asm,** or ***.asm.%number%** (for example bar.asm.23).*

Importing JT files

Previously, FeatureCAM was only able to import surfaces from JT files even though the format supports other information, such as boundary representation surfaces (NURBS), Product and Manufacturing Information (PMI), and metadata. In FeatureCAM 2015 R1, the interoperability of the import process has been extended and you can now also load solids from JT files.

To import a JT file:

1. Open a new or existing part file. You must have a part open to import geometry.
2. Select the **File > Import Using Exchange** menu option. The **Import** dialog is displayed.
3. Select the JT file you want to use, then click **Open** to import the data.

You must install Delcam Exchange to use this feature. Contact your sales representative for more information.

Importing PDF files

The FeatureCAM license now includes the ability to import curves from Adobe Acrobat files using Delcam Exchange.

For example, to import curves from an acrobat file:

1. Create a document.
2. Select the **File > Import using exchange** menu option. The **Import** dialog is displayed.
3. In the **Files of type** list, select **PDF Files (*.pdf).**
4. Locate and select the acrobat file containing the curves you want to import, then click **Open**.
5. A message asks if you want to review the import log. Click **Yes** to view the log, or click **No** to continue.
6. If the **Import Units** dialog is displayed to inform you the file was defined using different measurement units to those used in your document, select an option then click **OK** to continue. The **Import Results** dialog is displayed.
7. Select **Use the wizard to establish the initial setup location and stock size** and click **Next** to create a part using the **Import Wizard**, or click **Accept the imported data 'as is' and exit the wizard** and click **Finish** to complete the import process.

   The curves are displayed in the Graphics view and listed in the Parts view.

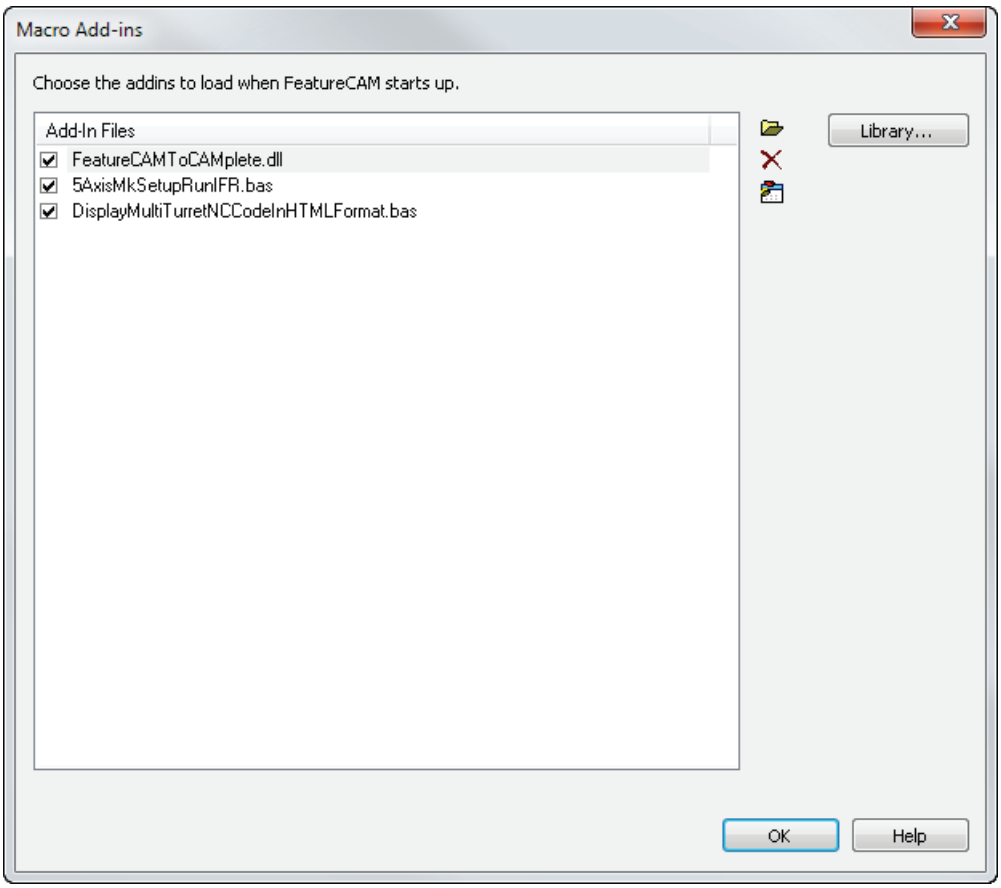
Add-ins and extensions

FeatureCAM 2015 R1 contains the following changes and improvements to its add-ins and extensions:

- **Using add-ins** (see page 130) — The **Macro Add-ins** dialog has been updated to improve the management of add-ins and macros.
- **The Add-in Library** (see page 132) — FeatureCAM now includes a catalog of add-ins that were previously only available on the website. The **Add-in Library** is a new feature that enables you to quickly view and select add-ins from the catalog.
- **Turn-curve tolerance** (see page 134) — A new TurnCurveTolerance.bas add-in enables you to tolerance multiple segments in turned or internal bore features.
- **Exporting NC programs to NCSIMUL** (see page 137) — The FeatureCAMToNCSIMUL.dll add-in can now export turn/mill programs as well as milling programs.
- **Extended machine support** (see page 138) — Add-in support for traveling steady rests, bar fed mills, and the Mori Seiki NTX1000 is available for FeatureCAM.

Using add-ins

FeatureCAM 2015 R1 now ships with most of the catalog of add-ins available for customizing and extending its abilities. This enables you to quickly view and select all the applications that have been developed for general and specific purposes without the need to locate and download them from the Internet. To accommodate this expansion, the **Macro Add-ins** dialog has been updated to make it easier to control and select the add-ins you want to use.



To choose the add-ins you want to use in FeatureCAM:

1. Select the **Options > Add-Ins** menu option. The **Macro Add-ins** dialog is displayed.
2. Select the check boxes of the add-ins you want to use. One or more buttons is added to the **Macro** toolbar for each add-in you can control manually.

   Alternatively, if the add-ins are not listed in the dialog, click **Library** (see page 132) to load them from the FeatureCAM library, or click to load them from file.

   *Use the **Customize Toolbars** dialog to create or change the toolbar buttons of add-ins.*

3 Deselect the check boxes of the add-ins you do not want to use.

To remove the selected add-in from the list, click .

4 If you want to view or edit an add-in, click . The IDE Editor is displayed.

5 Click **OK** to save your changes and close the dialog.

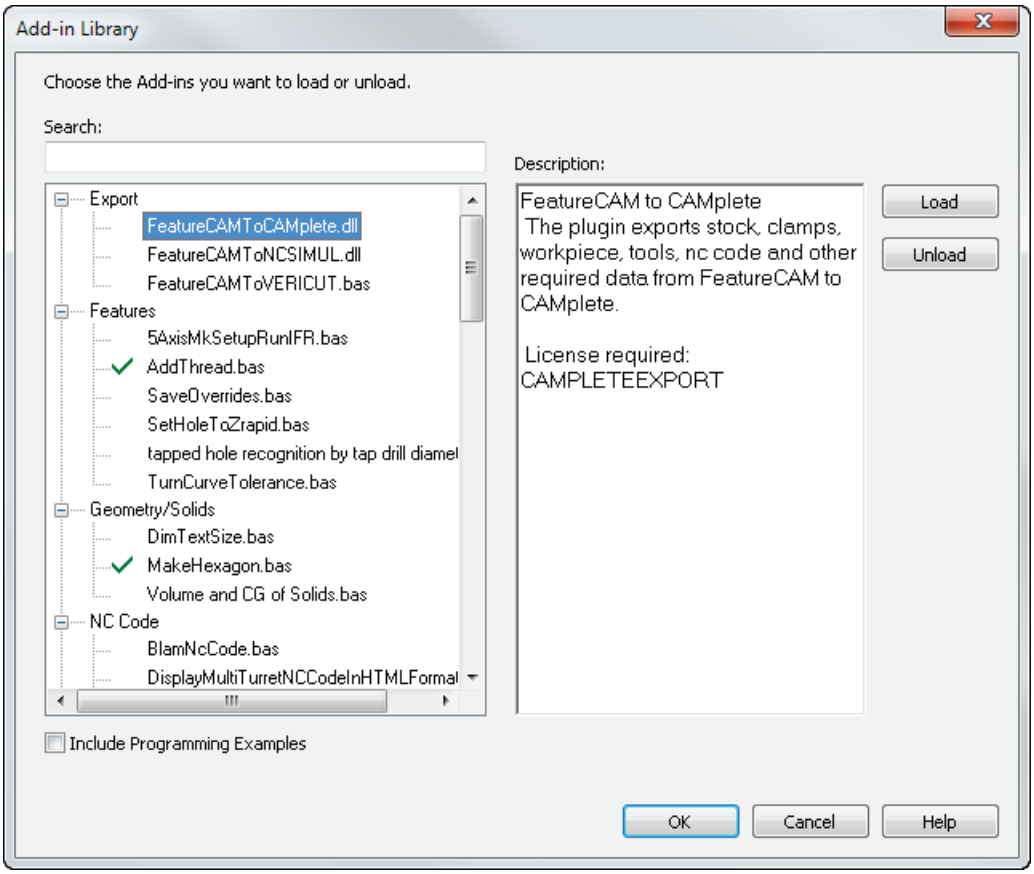
To run an add-in, click its button on the **Macro** toolbar. Alternatively, select the **View > Run Basic Macro** menu option, select the macro in the **Run Macro** dialog, and click **Run**.

The Add-in Library

The Add-in Library is a new feature for FeatureCAM 2015 R1. It contains most of the add-ins currently available for FeatureCAM together with descriptions of what they do and how they work. Use it to find out more about add-ins and to choose those you want to make available in the **Macro Add-ins** (see page 130) dialog.

To use the Add-in Library:

1 In the **Macro Add-ins** dialog, click the **Library** button. The **Add-In Library** dialog is displayed.



The left column lists the available add-ins and macros, categorized by type; add-ins already available in the **Macro Add-ins** dialog are marked by a check ✓ icon.

2 Choose one or more actions:

- To show programming examples in the list, select the **Include programming examples** check box.
- To search the list, type a string in the **Search** box. For example, to restrict the list to only those add-ins that include *NCCode* in their name, type **NCCode**.
- To display the **Description** of an add-in, select its entry in the list.

- To add an add-in to the **Macro Add-ins** dialog, select its entry in the list and click **Load**. The entry is marked by a plus + icon.
- To remove an add-in from the **Macro Add-ins** dialog, select its entry in the list and click **Unload**. The entry is marked by a cross ✕ icon.

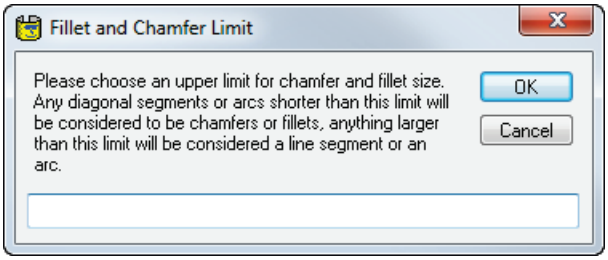
3 Click **OK** to save your changes and close the dialog. Loaded add-ins are listed in the **Macro Add-ins** dialog; unloaded add-ins are removed from it.

Turn-curve tolerance

TurnCurveTolerance.bas is a new add-in which enables you to tolerance turned or internal bore features according to ISO 282-2 standards or custom tolerances specified in design documents. Use it to quickly resize features without redrawing the curve.

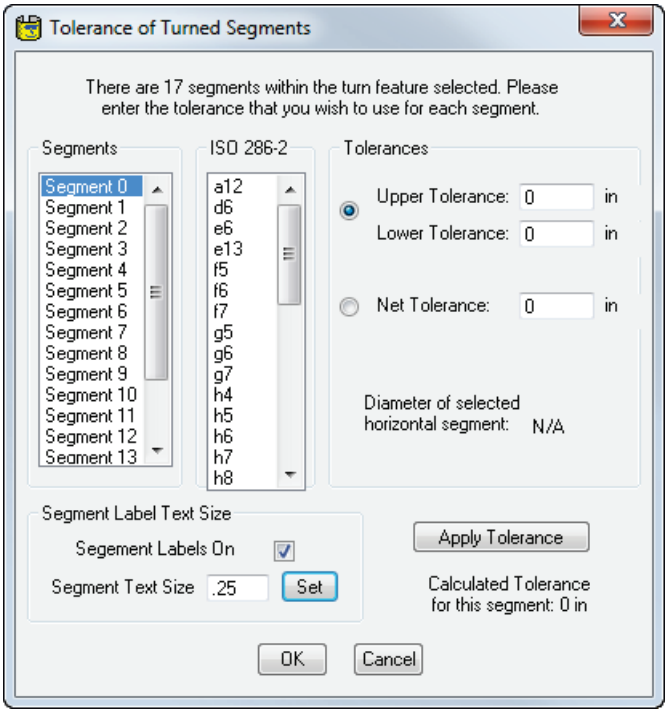
To adjust a curve:

1. Install the **TurnCurveTolerance.bas** add-in.
2. Select the turning or internal bore feature in the **Part View**.
3. In the **Macros** toolbar, click the add-in's button. The **Fillet and Chamfer Limit** dialog is displayed.



4. Enter the size of the largest fillet or chamfer of the feature:
   - Diagonals and arcs shorter than or equal to this value are treated as chamfers and fillets. When neighboring segments are adjusted, the add-in translates them without changing their size or orientation.
   - Diagonals and arcs longer than this value are treated as stationary parts of the curve. They affect the tolerance limits of adjacent segments and are not translated.

5 Click **OK**. The **Tolerance of Turned Segments** dialog is displayed, and the vertical and horizontal segments are labeled in the Graphic View.



6 Adjust the labels:

- To change the label size, enter a new value in the **Set text size** box, and click **Set**.
- To label only the currently selected segment, deselect the **Segment labels on** check box.

7 In the **Segments** list, select the segment you want to adjust. The segment and its label are displayed in red.

8 Specify the adjustment for the segment:

- To calculate the adjustment from specified tolerances, select the **Upper tolerance** - **Lower tolerance** option and enter the tolerance values.
- To calculate the adjustment from standard tolerances, select the **Upper tolerance** - **Lower tolerance** option, and select the tolerances in the **ISO 286-2** list.

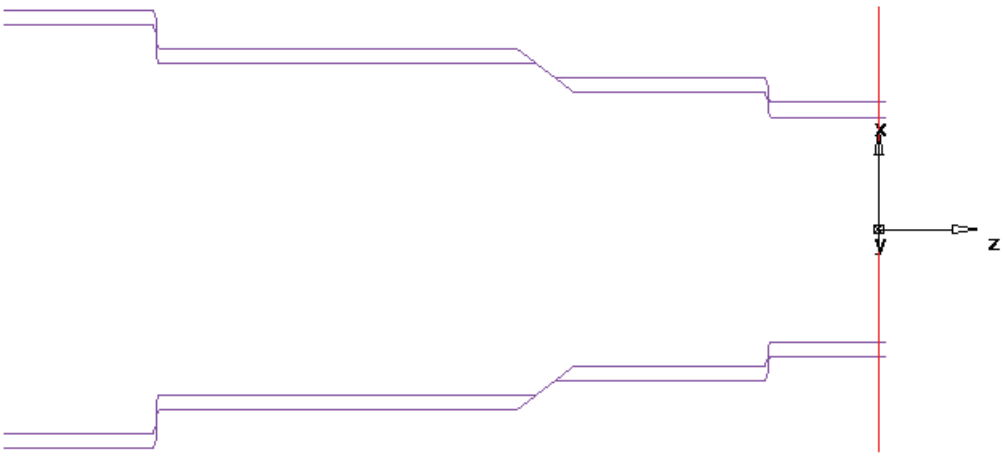
- To specify the adjustment, select **Net tolerance**, and enter the distance by which you want to move the segment.

9 Click **Apply Tolerance**. The adjustment for the segment is displayed below the button.

10 Repeat steps **7** through **9** for each segment you want to adjust.

11 Click **OK** to apply your changes and close the dialog.

FeatureCAM creates a new curve and feature, and displays the results in the Graphic window. For example:



Exporting NC programs to NCSIMUL

FeatureCAMToNCSIMUL.dll is an add-in, which provides the ability to export NC code for simulation and optimization in NCSIMUL. In FeatureCAM 2015 R1, the add-in has been extended to enable the export of turn/mill programs as well as the original milling programs.

To export a program to an NCSIMUL project:

1. Install NCSIMUL on your computer and the NCSIMUL tool for reading FeatureCAM files.
2. In FeatureCAM, open the milling or turn/mill document you want to export.
3. Run a simulation and generate the NC code, Ensure the code has no errors.
4. Select the **Options > Add-Ins** menu option. The **Macro Add-ins** dialog is displayed.
5. Select the **FeatureCAMToNCSIMUL.dll** check box.

   *If the add-in is not listed in the dialog, you can load it from the **Add-ins Library** dialog.*
6. Click **OK**. The **FeatureCAM to NCSIMUL** add-in button is displayed in the **Macros** toolbar.
7. Click the **FeatureCAM to NCSIMUL** button on the **Macros** toolbar. The **FeatureCAM to NCSIMUL** dialog is displayed.
8. In the **FeatureCAM to NCSIMUL** dialog, select the export options for the document.
9. Click **Export** to save the NC code to file. FeatureCAM simulates the part and exports the program files. When the export is finished, a message lists the files and their locations.

Extended machine support

FeatureCAM now includes add-in support for the following machines and features:

Mori Seiki NTX1000

The Mori Seiki NTX1000 is an integrated mill turn center designed for machining small, precision parts. A new machine design (.md) file and post are available for programming the machine together with a user defined feature for controlling the actions of the workpiece discharge unit.

Traveling steady rests

A new TravelingSteadyRest. bas add-in enables you to program a steady rest to follow behind the tool while roughing bar stocks. This add-in is not available in the library as it requires a customized post and machine model for each installation. For more information, contact the post-processor department of your local support office.

Bar-fed mills

Bar-fed mills, such as the Willemin-Macodel 408MT, enable you to machine multiple parts automatically from bar stock, and to automatically transfer the machined slugs to the sub-spindle to machine the cut-off face. This eliminates the need for manually positioning the part, which can be a source of inaccuracy.

XBUILD improvements

FeatureCAM 2015 R1 contains the following changes and improvements to the XBUILD module:

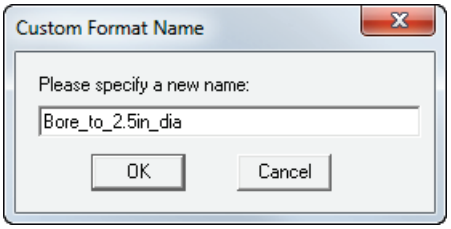
- **Custom formats** (see page 140) — The **Custom Format Name** dialog has an enhanced user interface to support the creation of naturalistic process names.
- **Customizing cutting data from Wire EDM** (see page 141) — You can now configure the headings of the cutting data table for use with Wire EDM.
- **Saving CNC files** (see page 144) — Post numbers can be used to identify individual output files for Fanuc-style multi-turret lathes.
- **Configuring reserved words** (see page 146) —The **Word Info** dialog includes a new option that enables you remove *all* non-significant zeros from numeric outputs
- **Using the Reserved Words dialog** (see page 147) — The **Reserved Words** dialog is easier and quicker to use.

Custom Formats

The **Custom Format Name** dialog now supports the use of underscores and periods in format names. This enables you to create more naturalistic process names and to accurately represent decimal measurements. In addition, space characters are automatically converted to underscores to ensure the names are unbroken.

To create a Custom Format:

1. In XBUILD, select the **Formats > Custom > New** menu option. The **Custom Format Name** dialog is displayed.
2. Enter a name for the custom format. You can use any upper- or lower-case character, periods, and underscores. For example:



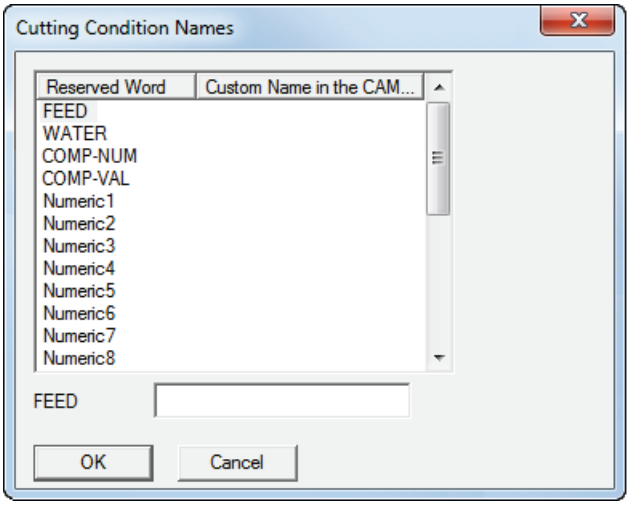
3. Click **OK** to create the format and close the dialog. The XBUILD Editor is displayed with the format name in the Title bar.
4. When you have finished creating the format, select the **File > Quit** menu option, and click **Yes** in the message dialog to save your changes.

Customizing cutting data for Wire EDM

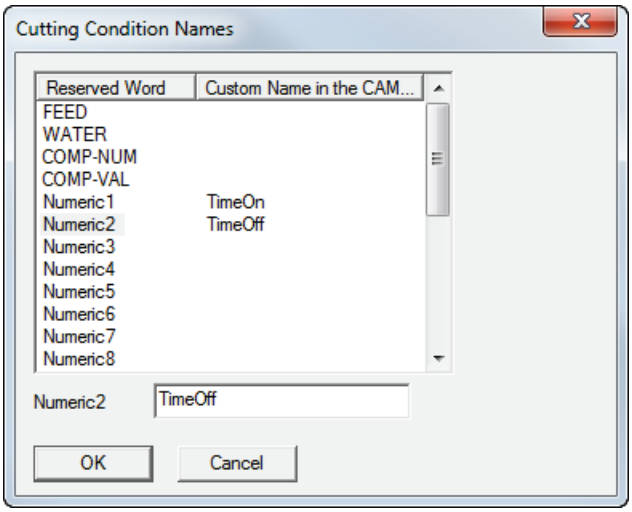
The **Wire EDM** tab (see page 123) in FeatureCAM has been extended to include customizable fields, which can be used to record machine-specific settings as well as the original **Feed**, **Water**, **Comp-num**, and **Comp-val** values. To enable you to manage and customize all the fields, the **Cutting Condition Names** dialog has been redesigned, and it now lists the original settings followed by 16 fields that can contain numeric information and 6 fields that can contain strings.

To specify the condition names:

1. In XBUILD, select the **CNC-Info > Machine** menu option. The **Machine Information** dialog is displayed.
2. In the **Machine type** list, select **Wire EDM** and click **OK** to close the dialog.
3. Select the **CNC-Info > Cutting Condition Names** menu option. The **Cutting Condition Names** dialog is displayed.



4 In the **Reserved Word** column, click the setting you want to customize, and type a name for it in the box at the bottom of the table. The name is displayed in the custom name column of the table. For example:

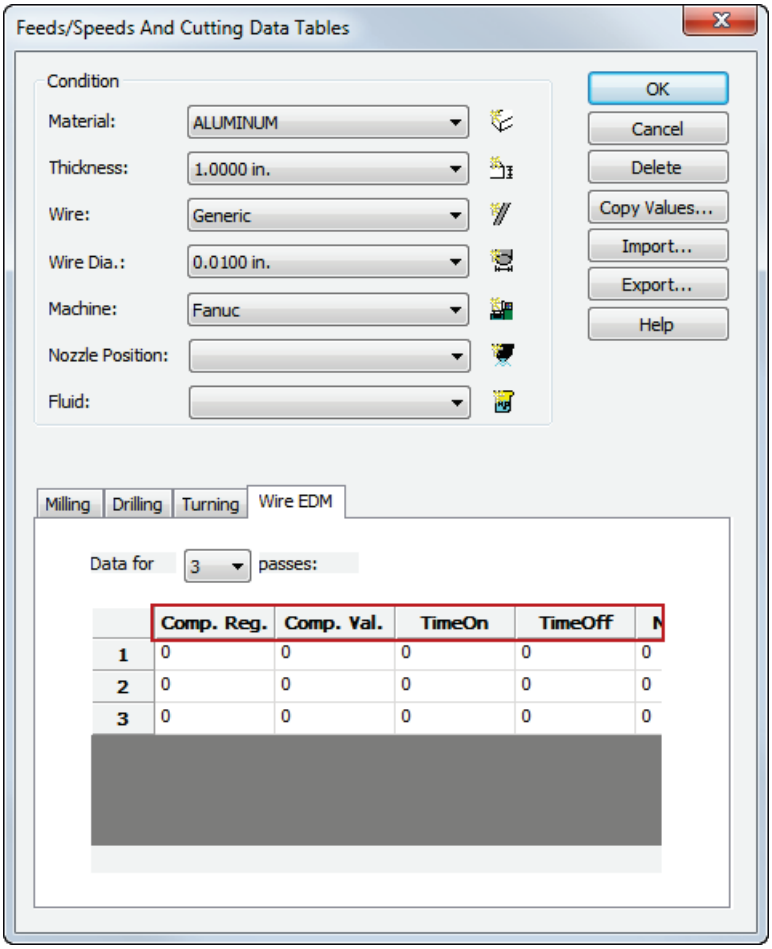


5 Repeat step **4** for each setting you want to customize.

6 Click **OK** to save your changes and close the dialog.

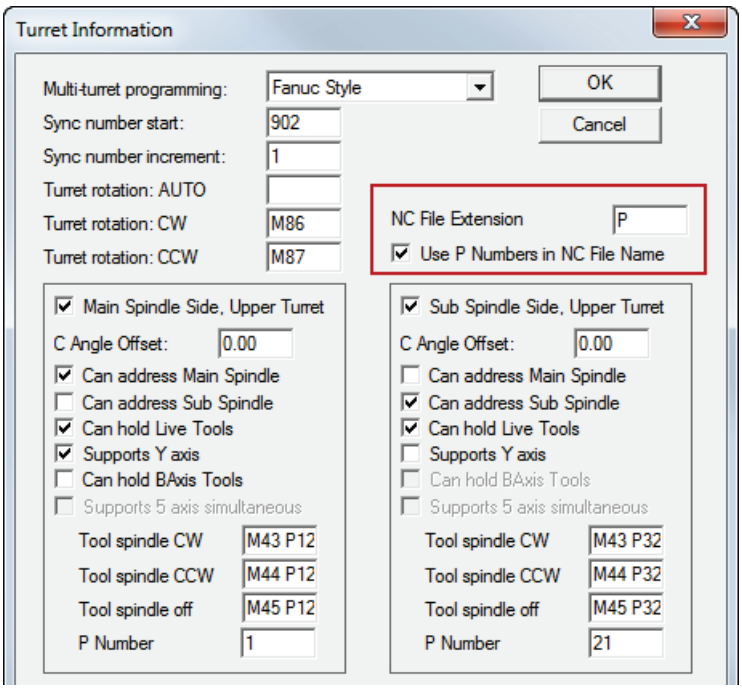
7 Select the **File > Save CNC** or **File > Save as CNC** menu option to save the post file.

When you next view the stock settings, the field names you specified are displayed on the **Wire EDM** tab of the **Feeds/Speeds And Cutting Data Tables** dialog. For example:



Saving CNC files

FeatureCAM 2014 R2 included the ability to display the post number of turrets throughout the user interface. In FeatureCAM 2015 R1, a new **Use P numbers in NC file name** option has been added to the **Turret Information** dialog that enables you to also include these post numbers in the names of saved NC files. In addition, a new **NC File Extension** box gives you the ability to customize the letter used as the post extension's prefix.



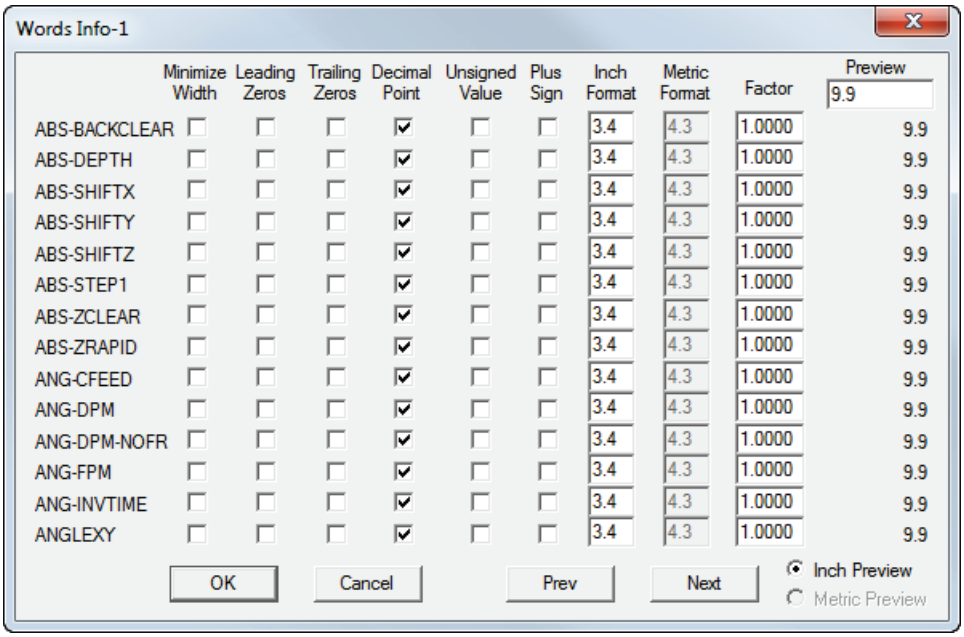
To specify the turret identifier for the CNC files:

1. In XBUILD, select the **CNC-Info > Turrets** menu option. The **Turret Information** dialog is displayed.
2. In the **Multi-turret programming** list, choose an option to specify the type of NC Program files you want to create. Select:
   - **None** for lathes with one turret.
   - **Okuma Style** for multi-turret lathes that require one program file for all turrets.
   - **Fanuc Style** for multi-turret lathes that require a separate program file for each turret.
3. By default, the turret identifier starts with **P**. If you want to use a different prefix, enter a new character in the **NC file extension** box.

4. Select the **Use P numbers in NC file name** check box to use the **P number** of each turret as the suffix of the **NC file extension**. Deselect the check box to use the default post numbers for the file extensions.
5. Complete the settings, and click **OK** to save your changes and close the dialog.
6. Save the post file.
7. In FeatureCAM, generate the NC code for your part, and select the **File > Save CNC** menu option. The **Save NC** dialog is displayed.
8. Enter the output settings and click **OK** to create the NC program files:
   - With **Fanuc-style** and **Use P numbers in NC file name** selected; the P numbers are **1**, **3**, **11**, **20**; and an **NC file extension** of **T**, FeatureCAM creates files named: %DocumentName%.T1.TXT, %DocumentName%.T3.TXT, %DocumentName%.T11.TXT, and %DocumentName%.T20.TXT.
   - With **Fanuc -style** selected and **Use P numbers in NC file name** deselected; P numbers of **1**, **3**, **11**, **20**; and an **NC file extension** of **T**, FeatureCAM creates files named: %DocumentName%.TXT, %DocumentName%.T2.TXT, %DocumentName%.T3.TXT, and %DocumentName%.T4.TXT.
   - With **Okuma-style** or **None** selected, only one file is created.

Configuring reserved words

The **Word Info** dialog enables you to view and change the output format for reserved-word values. In FeatureCAM 2015 R1, the dialog has been redesigned, and now includes a **Minimize width** option, which allows you to omit all non-significant zeros from numeric outputs, and a **Preview** column, which displays the effect of the settings for each reserved word.

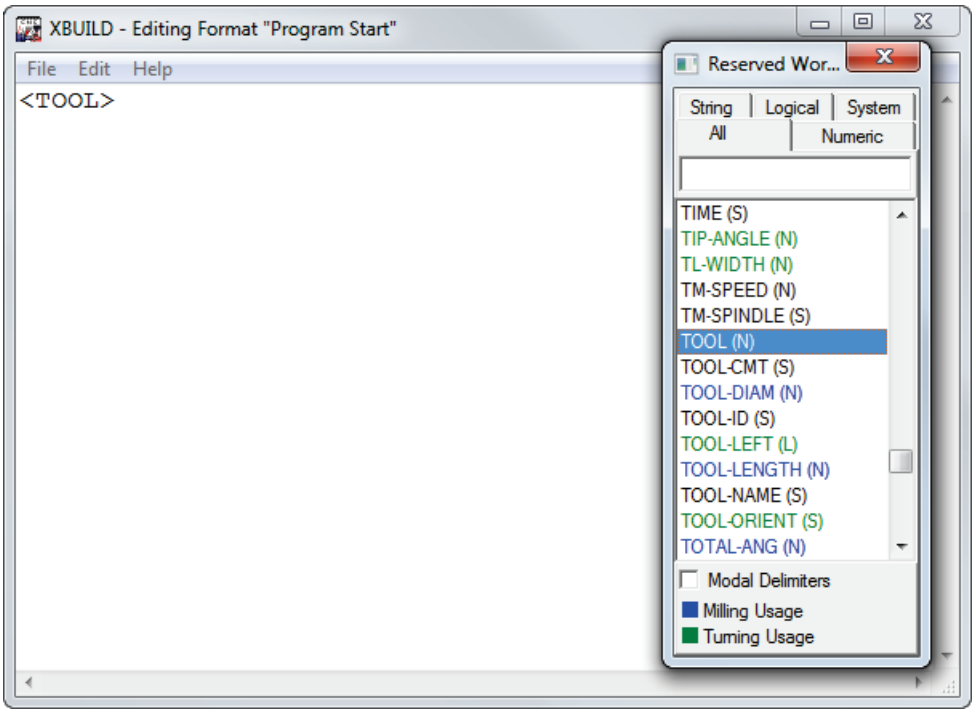


To configure the output of reserved words:

1. In XBUILD, select a **Words** option in the **CNC-Info** menu. The **Words Info** dialog displays the sub-set of reserved words for the option you selected.
2. For each reserve word you want to update, select the **Minimize width** check box. This removes leading zeros from decimal-only outputs and trailing zeros from integer-only outputs. For example, when the check box is selected, the reserved word outputs **4** as `4`, and **0.2** as `.2`; when the check box is deselected, **4** is output as `4.0`, and **0.2** is output as `0.2`.
3. Check the effect of your changes using the **Preview** column:
   - To test the effect of the settings on different outputs, type a new value in the box above the column.
   - To test the effects of the **Inch format** and **Metric format** settings, select the preview options below the column.
4. Click **OK** to close the dialog and save your changes.

Using the Reserved Word dialog

The **Reserved Word** dialog of the **Formats** editor enables you to quickly add variables to the format you are editing.



In FeatureCAM 2015 R1, several new features have been added to the dialog that make it easier for you to find and add words to a format:

- Words are now identified by category: string words are followed by **(S)**; logical words are followed by **(L)**: and numeric words are followed by **(N)**; system words have no identifier.
- Tabs across the top of the dialog enable you to restrict the list to a specific category of word. For example, to list only system words, select the **System** tab; to list all reserved words again, select the **All** tab.
- In the **All** tab, words are listed in alphabetical order instead of by alphabetical order within category. This enables you to look for words by name without knowing the category it belongs to.
- A new search box below the tabs enables you to list only those words that contain a specified string. For example, type **spindle** in the box to list only those words that contain the string "spindle". When you select a word, the box is automatically cleared and the list scrolls to display the word's entry.
- In addition to being selectable, the **Modal delimiters** check box can now be toggled on and off using the **Control** key. To insert a word in modality brackets, press the **Control** key; select the word in the list, and release the key.

Index

.

.cnx • 66
.html • 66
.md file protection • 79
.xml • 66

2

2D NT toolpaths • 54

4

408MT • 139

5

5-axis engraving • 30
5-axis patterns • 112

A

Acrobat files • 129
Addendum • 97
Add-ins • 99, 131, 133, 135, 138, 139
Analysis tab • 97
Analysis tab (Gears dialog) • 44
Apply button • 93
Authentication mode • 86
Auto-chamfer • 109
Autodesk Inventor • 127
Automatic Feature Recognition • 90
Automatic tool selection • 18, 50

B

Bar stock • 64
Bar-fed mills • 139
Boolean dialog • 45
Bore depth • 107
Boundaries • 110

C

Calculate index radius from solid stock outline • 24
CAMplete • 70
C-axis indexing • 28
Centerline simulation • 46
Chamfer • 109
Chamfer limit • 135
Chip recognition size • 47
Clamps • 90
Clipping curves • 110
CNC files • 145
CNC-Info menu • 147
CNX • 66
Collision avoidance • 102
Combining solids • 45
Command line options • 99
Condition dialog • 124
Counterbore operation • 107
Cursor color • 42
Curve length • 43
Curves • 95, 97
Custom Format Name dialog • 141
Customizing the snapping cursor • 42
Cutoff Properties dialog • 121
Cutoffs • 47

Cutting Condition Names dialog • 142
Cutting data • 124
Cylinder snapping • 92

D

Database • 86
Debug options • 99
Dedendum • 97
Deep Holes • 57
Deleting unused curves • 11
Depth of cut • 115
Diametric pitch • 97
Die feature • 123
Dips • 115
Disable Macros option • 67
Drilling • 57
Duplicating featuress • 7, 9

E

Engraving • 30
Exchange • 128
Exporting program files • 138
ezfm.exe • 99

F

Face features • 91
Fanuc-style programming • 145
Feature recognition • 90, 91
FeatureCAM to CAMPlete • 70
FeatureCAMToNCSIMUL • 138
Feed/Speeds and Cutting Data Tables dialog • 89, 124, 142
File Options • 86
Fillet limit • 135
Finish button • 93
Finishing strategies • 105
Flats • 52
Flowline • 105
Fluid • 124
Follow turning • 118
Formats • 141
Formats editor • 148

G

Gears • 44, 97
Gouge avoidance • 102

H

Helical side finish • 59
Hide rapid moves in centerline simulations • 46
Holder collision checking • 102, 104
Hole Feature Properties dialog • 107
Hole patterns • 112
Holes • 57
HTML • 66

I

IDE Editor • 131
IFR • 91
Importing files • 127, 128, 129
Importing SolidEdge files • 14
Importing SolidWorks files • 13
Index axis • 112
Index radius • 24
Indexing usinga stock solid • 24
ini files • 99
Installation message • 85
Interactive Feature Recognition • 91
Internal gears • 44
Inventor files • 127
Islands • 123
ISO 282-2 standard • 135
Isoline • 105

J

Jogging • 34
JT files • 128

L

Length of curves • 43
License checks • 99
Lock Machine Design documents • 79

M

Machine Design document protection • 79
Machine jogging • 34
Machine maximum stock • 104
Machine movement limits
  Specifying limits • 33
  Testing limits • 34
  Using limits • 21
Machine simulation • 46, 47, 64, 70
Machining attributes • 87, 109
Machining Attributes • 58
Macro Add-ins dialog • 131
Macros • 67
MD file protection • 79
Measuring curve length • 43
Measurment over pins • 97
Microsoft SQL Server 2014 • 72
Minimize width check box • 147
Mini-turrets • 36
Modal delimiters • 148
Modifying solids • 45
Module • 97
Multi Turret Turning • 65
Multi-tool blocks • 74
Multi-turret machining • 116, 118, 120

N

NC file extension • 145
NCSIMUL • 138
New Feature Wizard • 112
New Part Document wizard • 127
New Stock Material dialog • 89
New Technology toolpaths • 54
No-core machining • 123
No-drag turning • 61
Nozzle position • 124
NT toolpaths • 54
NTX1000 • 139

O

Okuma-style programming • 145
Operating systems • 85
Operations • 116, 118
Orientation angle • 28
Output filtering • 56
Output options • 54, 56
Outside of Limits dialog • 21
Overrides • 120

P

P numbers • 145
Panels • 87
Parameters • 99
Part catchers • 121
Part handling • 62
Part library • 7
Part view • 87
Partial select • 6
Pasting features • 9
Patterns • 112
PDF files • 129
Pecking • 57
Pick Dimension dialog • 43
Pilot length • 107
Pin gauge • 97
Pinch turning • 118
Pitch diameter • 97
Pocketing (Wire) • 123
Point distribution preview • 56
Point size • 41
Polar coordinates • 9
Polygonal curve • 95
Post documentation • 66
Post Options dialog • 141
Posting tolerance • 58
Pressure angle • 97
Preview options • 147
Preview toolpath points • 56
Pro/E files • 128
Probing • 72
Program files • 138
Program point • 26
Protecting Machine Design files • 79

R

Rapid moves • 46
Reserved words • 147, 148
Rests • 139
Round stock • 47

S

Scripts, running • 99
Segments • 135
Selecting objects • 6
Sequence • 105
Server authentication • 86
Setup Sheet • 69
Show Centerline Rapids • 46
Simulation • 46, 47, 64, 70
Simulation Options dialog • 47
Small parts • 58
Snapping • 42, 92
Solid modelling • 45
SolidEdge • 14
Solids • 93, 128
SolidWorks • 13
Specific stock length • 64
Spiral finishing • 59
SQL server • 86
SQL Server • 72
Steady rest • 62
Stock length • 64
Stock materials • 89, 104
Stock models • 110
Stock solid indexing • 24
Strategy tab • 118
Support for Microsoft SQL server 2014 • 72
Surface Milling Properties dialog • 104, 105, 110
Surface selection • 100
Synchronizing operations • 116

T

Tool block • 74, 77
Tool Block Selection dialog • 77
Tool holder • 20
Tool holder clearance • 50
Tool Holder Clearance dialog • 18
Tool Locations • 74
Tool Mapping dialog • 77
Tool pecking depths • 57
Tool Properties dialog • 120
Tool selection • 18, 50
Tool selection tab • 109
Toolbox • 87
Toolpath boundaries • 110
Toolpath points preview • 56
Toolpaths • 104, 105
Tools with multiple inserts • 36
Touch off at the shoulder check box • 107
Traveling steady rest • 139
Turn Properties dialog • 118
Turn-curve tolerance add-in • 135
Turnmilling program point • 26
Turret Information dialog • 145
Turret names • 65
Turret selection • 120
Turrets tab • 116

U

Undercuts • 61
Unused curves • 11
Update options (probing) • 72
Use solids as clamp option • 90

V

Viewing Options dialog • 41
Vortex • 52, 54

W

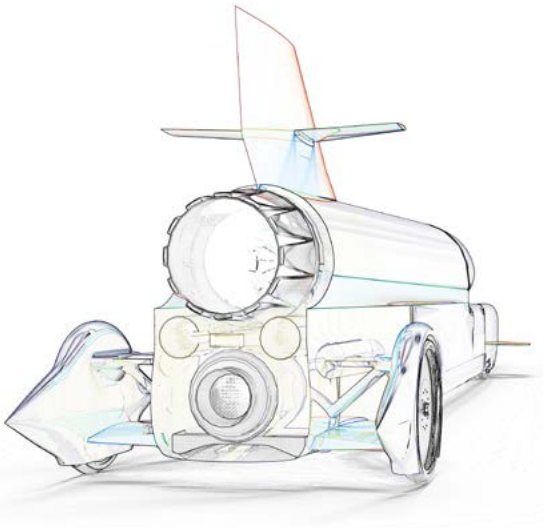
Windows 7 • 85
Wire EDM • 123, 124, 142
Words Info dialog • 147

X

XBUILD • 141, 142, 145, 147
XML • 66

Z

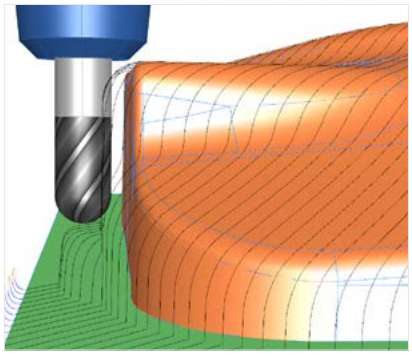
Zigzag (Wire) • 123
Z-zero position • 107



FeatureCAM 2015 R3

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

www.featurecam.com

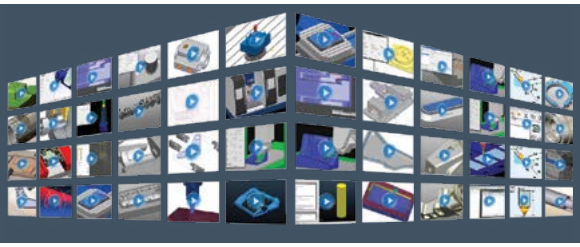


FeatureCAM 2015

www.featuecam.com

www.delcam.tv

Unlock the power of Delcam software



LEARNING ZONE

Visit **www.delcam.tv/lz** and see how Delcam software could benefit your business!

Delcam Advanced Manufacturing Solutions

Powering your productivity

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

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

AUTODESK.

© Copyright Delcam Ltd 2014. All trademarks are the property of their respective owners.