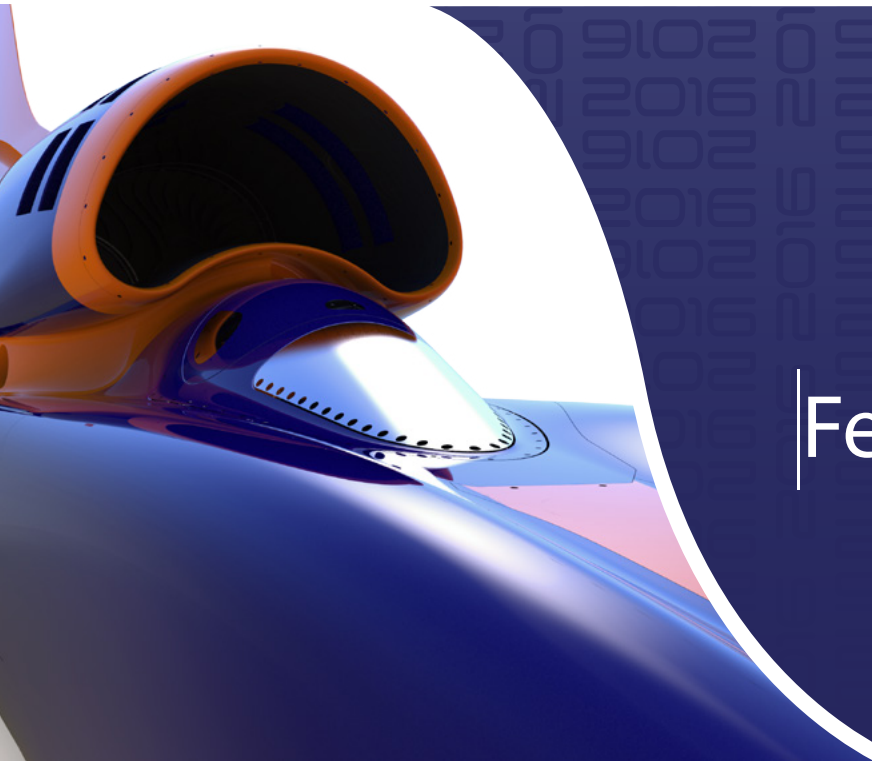


Delcam Advanced Manufacturing Solutions

FeatureCAM 2016

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

www.featurecam.com

What's New

FeatureCAM 2016 R1

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: 2016 R1 Date: 10 August 2015 10:19

Contents

What's New 1

What's new in FeatureCAM 2016 R1 2

User Interface and work-flow 5
- Reference help PDF 6
- Highlighting objects from the Part View 7
- Creating Setups during IFR 8
- Incrementing tool color in simulation 9
- Displaying NC Code 10
- Creating curves 11

Importing 12
- Solid names 13
- Importing AutoCAD files 13
- Spatial R25 SP2 Interop integrated 14
- Parasolid v27.1 14

Milling 15
- Vortex non-cutting moves 16
- Collision avoidance 17
- Swarf machining Z limits 18
- 5-axis 2D spiral toolpaths 20
- Side grooves 21
- Multiple coolant types 23
- User Defined Stock for NT toolpaths 24

Turning 25
- Turning tool orientation 26
- Roughing engage angle 28
- Negative leave allowance for grooves 29

Add-ins and extensions 30
- Setup Sheet add-in 31
- FeatureCAM to CAMplete 32
- FeatureCAM API improvements 33
- Nesting add-in 34
- Import Vise add-in 36
- Bar-fed mills support 38

XBUILD 39
- Intermediate CNC files 40
- Multi-code coolant 42
- 5-axis untransformed coordinates 44
- Formatting reserved words 45
- CNC documentation 46

What's new in FeatureCAM 2015 R3 47

User Interface 49
Improved icons 50
Selecting objects 50
Inserting Part Library features 51
Pasting features using polar coordinates 52
Deleting unused curves 53
Importing 54
Importing SolidWorks 2015 files 55
Importing SolidEdge ST7 files 56
Spatial R25 SP1 Interop integrated 56
Milling 57
Creating Face features using IFR 58
Automatic tool selection 59
Simulating tool holders 60
Machine movement limits 61
Turning and Turn/Mill 62
Indexing using a stock solid 63
Turnmilling program point 65
Turning head tool holder support 66
5-axis machining 70
C-axis indexing 71
5-axis engraving 73
Machine Design 75
Specifying machine movement limits 76
Testing machine movements 77
Simulating mini-turrets 79

What's new in FeatureCAM 2015 R2 80

User Interface 83
Changing the point size 84
Customizing the snapping cursor 85
Measuring curve length 86
Creating internal gears 87
Combining solids (SOLID) 88
Hiding rapid moves in centerline simulations 89
Chip recognition size 90
Recognizing features on large parts 90
Milling 91
Automatic tool selection 92
Vortex approaches flats from outside stock 94
Output options for 2D NT toolpaths 96
Previewing the toolpath point distribution 97
Tool pecking depths 98
Changing the Posting tolerance 99
Helical side finish 100
Turning 101
Removing undercuts in no-drag turning features 102

Controlling steady rests 103
Simulating bar stock 104
Custom turret names (MTT) 105
XBUILD 106
Generating Post documentation 107
Using macros in the post processor 107
Add-ins and extensions 108
Using Setup Sheets 109
FeatureCAM to CAMplete TruePath add-in 110
Probing update options 112
Support for Microsoft SQL server 2014 112
Machine Simulation Design 113
Multi-tool blocks 114
Selecting the tool block 116
Protecting solids in Machine Design documents 118

Index **119**

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 2016 R1 (see page 2)

What's New in FeatureCAM 2015 R3 (see page 47)

What's New in FeatureCAM 2015 R2 (see page 80)

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 2016 R1

FeatureCAM 2016 R1 contains the following new features and enhancements:

User interface

- **Reference help PDF** (see page 6) — The FeatureCAM Reference help is now available as a PDF file.
- **Highlighting objects from the Part View** (see page 7) — You can now highlight objects in the graphics window by moving the cursor over the object's name in the **Part View**.
- **Creating Setups during IFR** (see page 8) — You can now create Setups from within the **New Feature** wizard when using Interactive Feature Recognition.
- **Changing the simulation tool color** (see page 9) — You can now manually change the tool color at any time in a simulation.
- **Displaying NC Code** (see page 10) — NC code is now displayed using fixed width text, where all the characters line up vertically, to make it easier to read.
- **Creating curves** (see page 11) — The images in the curve creation dialogs have been improved to make them easier to understand.

Importing

- **Imported solid names** (see page 13) — When you import a solid into FeatureCAM the solid name is now retained from the original program.
- **Importing AutoCAD files** (see page 13) — You can now import ACIS solids from DWG files.
- **Spatial R25 SP2 Interop integrated** (see page 14) — Spatial R25 SP2 Interop is now integrated.
- **Parasolid v27.1** (see page 14) — FeatureCAM now supports parasolid v27.1.

Milling

- **Vortex non-cutting moves** (see page 16) — You now have more control over the non-cutting moves in vortex toolpaths.
- **Collision avoidance** (see page 17) — Toolpaths are now collision-checked between unmachined stock and the shank and holder.
- **Swarf machining Z limits** (see page 18) — You can now limit Swarf operations to within a specified Z range on the surface.
- **5-Axis 2D spiral toolpaths** (see page 20) — You can now specify axis smoothing options for 2D spiral toolpaths.
- **Side grooves** (see page 21) — The toolpath calculation for ID and OD Grooves has been improved with several new options and supported functions.
- **Multiple coolant types** (see page 23) — You can now use multiple coolant types for a single operation.
- **User Defined Stock for NT toolpaths** (see page 24) — When using a stock solid and you create a 2D NT toolpath, FeatureCAM now calculates the toolpaths using the stock solid boundary to prevent air-cutting.

Turning

- **Turning tool orientation** (see page 26) — You can now use a turning tool in multiple orientations without having to create a different tool in the database for each orientation.
- **Roughing engage angle** (see page 28) — You can now specify the engage angle, lead in angle, and lead out angle for Turning roughing operations.
- **Negative leave allowance for grooves** (see page 29) — You can now use a negative leave allowance for groove features created from curves.

Add-ins and extensions

- **Setup Sheet add-in** (see page 31) — There are new tags which you can use to create more useful setup sheets.
- **FeatureCAM to CAMplete** (see page 32) — In the **FeatureCAM to CAMplete** dialog (see page 110), there is a new button that enables you to mark solids that are selected in the part as clamps.
- **API improvements** (see page 32) — The API has been improved to support full machine architecture.

XBUILD

- **Intermediate CNC files** (see page 40) — You can now split the nc file into multiple intermediate files, and then combine these intermediate files in different ways to create multiple output files.
- **Multi-code coolant** (see page 42) — You can now use different on and off codes for a single coolant type, depending on which turret and tool is being used.
- **5-axis untransformed coordinates** (see page 44) — There are new milling reserved words that contain the untransformed X, Y, and Z coordinates in a 5-axis part.
- **Formatting reserved words** (see page 45) — When formatting numeric reserved words in XBUILD, you can now remove trailing zeros from integer values without removing the decimal point.
- **CNC documentation** (see page 46) — The HTML documentation has been improved to make the output more readable.

User Interface and work-flow

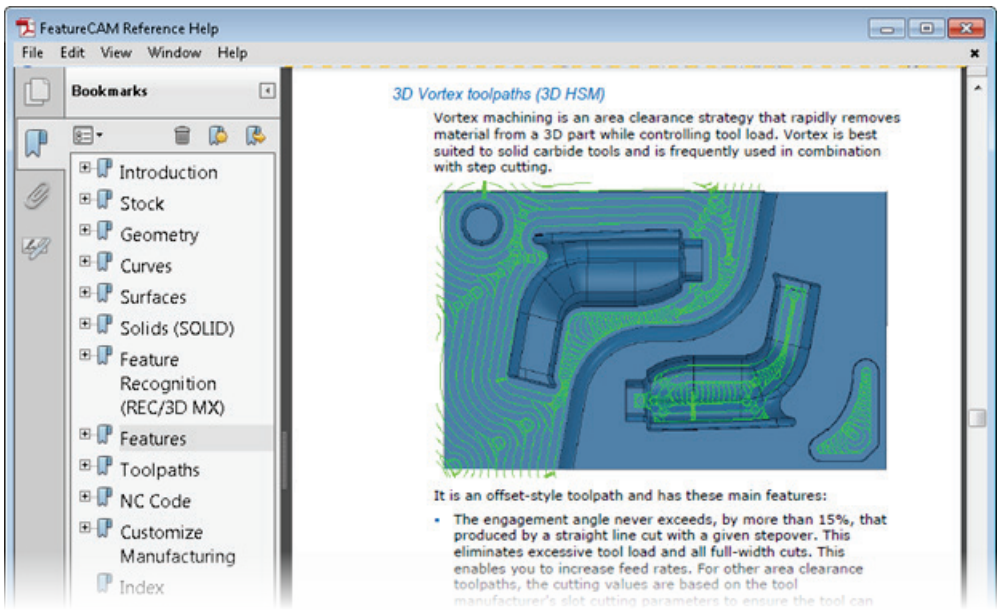
FeatureCAM 2016 R1 contains the following changes and improvements to the user interface and work-flow:

- **Reference help PDF** (see page 6) — The FeatureCAM Reference help is now available as a PDF file.
- **Highlighting objects from the Part View** (see page 7) — You can now highlight objects in the graphics window by moving the cursor over the object's name in the **Part View**.
- **Creating Setups during IFR** (see page 8) — You can now create Setups from within the **New Feature** wizard when using Interactive Feature Recognition.
- **Changing the simulation tool color** (see page 9) — You can now manually change the tool color at any time in a simulation.
- **Displaying NC Code** (see page 10) — NC code is now displayed using fixed width text, where all the characters line up vertically, to make it easier to read.
- **Creating curves** (see page 11) — The images in the curve creation dialogs have been improved to make them easier to understand.

Reference help PDF

The FeatureCAM Reference help is now available as a PDF file, which enables you to print it to use away from your computer.

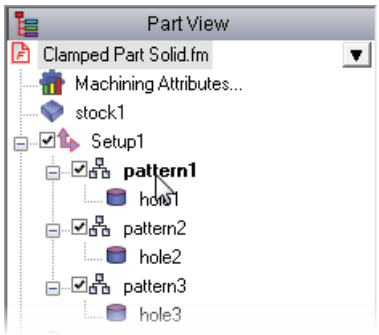
Select the new **Help > Reference (PDF)** to display the Reference help PDF.



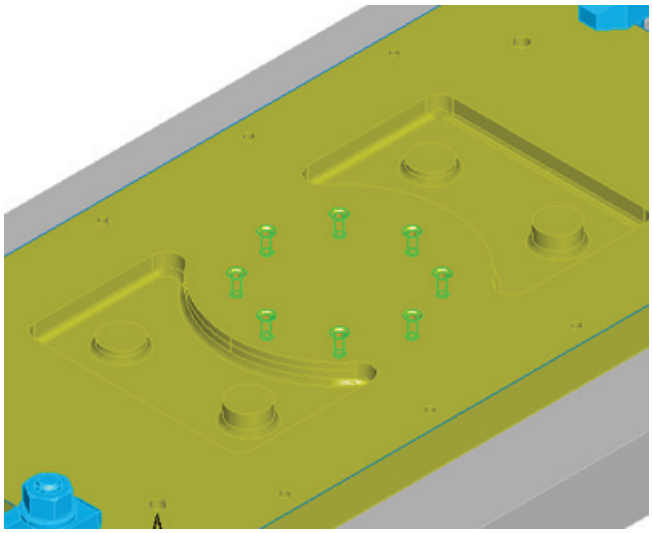
Highlighting objects from the Part View

You can now highlight objects in the graphics window by moving the cursor over the object's name in the **Part View**. This enables you to find features quickly and improves the work-flow in complex documents.

For example, move the cursor over an object in the **Part View**:



The object is highlighted in green in the graphics window:



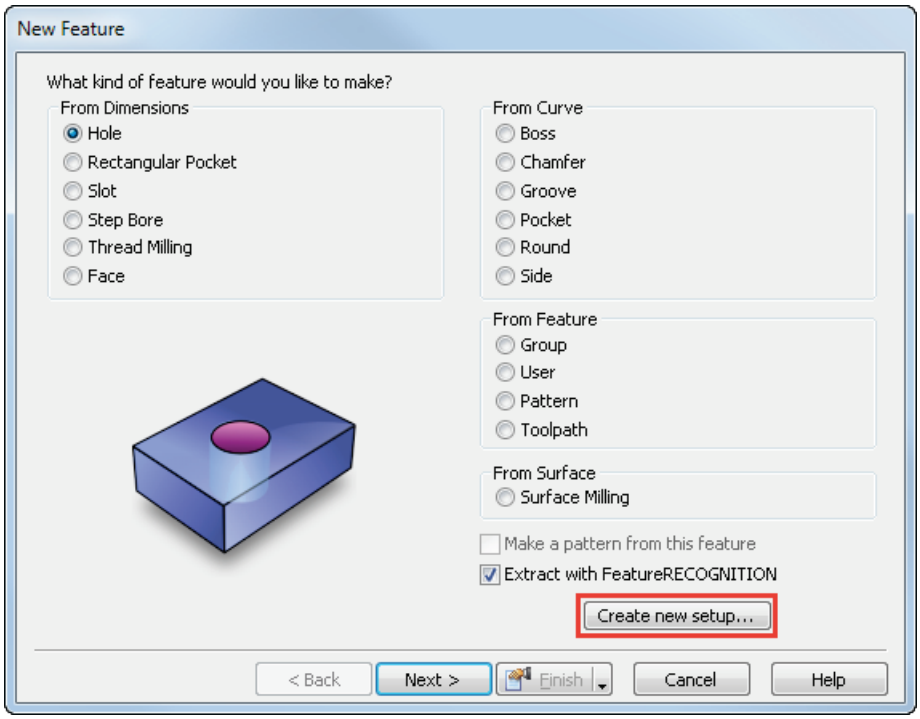
To turn off this option, deselect **Show feature dynamic highlight** on the **General** tab of the **Viewing Options** dialog.

Creating Setups during IFR

When creating Milling and Wire EDM features using Interactive Feature Recognition, you can now create Setups from within the **New Feature** wizard. This enables you to program parts with multiple setups more quickly.

To create a Setup using the **New Feature** wizard:

1. In the **New Feature** wizard, select the feature type you want to recognize.
2. Select **Extract with FeatureRECOGNITION**.
3. Click the new **Create new setup** button.



The **Setup** wizard is displayed.

4. Use the **Setup** wizard to create a new setup, then click **Finish**.

   The **New Feature** wizard is displayed.

   You can use the **Create new setup** button to create more setups, or click **Next** and use the **New Feature** wizard to create features.

Incrementing tool color in simulation

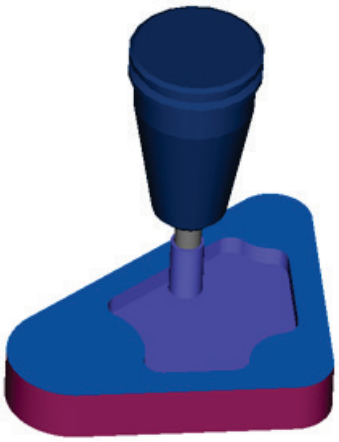
You can now manually change the tool color at any time in a simulation to make it easier to understand the toolpaths and see the tool simulation. For example you can use this to display different passes of an operation in different colors to check for air cutting.

This is only available for 3D, 3D rapid cut and machine simulations.

To change the tool color during a simulation:

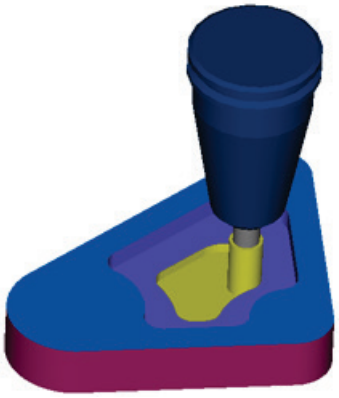
1. In the **Simulation Options** dialog, on the **General** tab, ensure **Tool Colors** is selected.
2. Run a simulation.

   The tool color is displayed during the simulation.



3. Click the new **Change the Simulation Tool Color** button.
4. Continue the 3D simulation.

   The tool color is changed in the simulation.

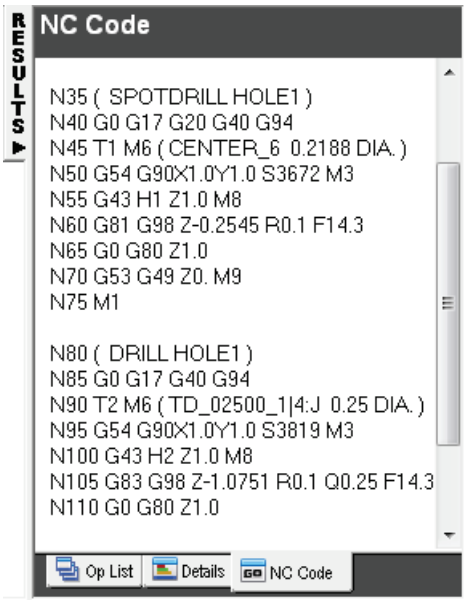


Displaying NC Code

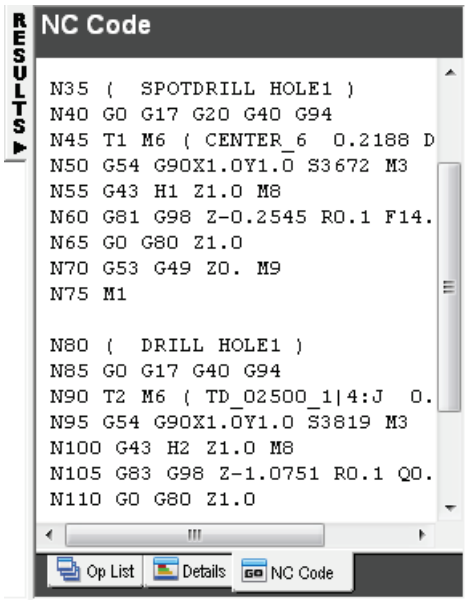
NC code is now displayed using fixed width text, where all the characters line up vertically, to make it easier to read.

You can display the NC Code in a variable-width font by right-clicking in the **NC Code** window and deselecting **Fixed Width Font**. Variable-width text requires less horizontal space.

Variable width



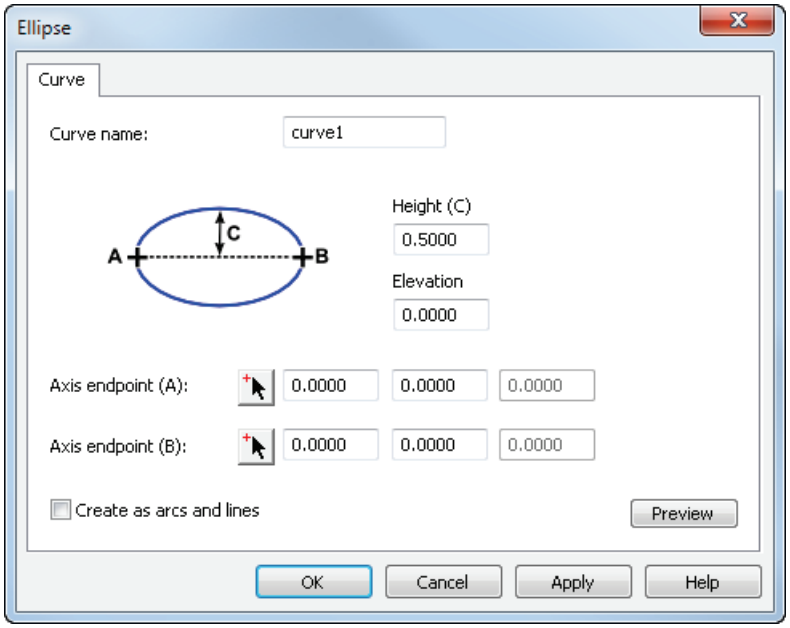
Fixed width



Creating curves

The images in the curve creation dialogs have been improved to make them easier to understand.

Letters are now used to show dimensions instead of arrows.



Importing

FeatureCAM 2016 R1 contains the following changes and improvements to importing:

- **Imported solid names** (see page 13) — When you import a solid into FeatureCAM the solid name is now retained from the original program.
- **Importing AutoCAD files** (see page 13) — You can now import ACIS solids from DWG files.
- **Spatial R25 SP2 Interop integrated** (see page 14) — Spatial R25 SP2 Interop is now integrated.
- **Parasolid v27.1** (see page 14) — FeatureCAM now supports parasolid v27.1.

Solid names

When you import a solid into FeatureCAM the solid name is now retained from the original program. Previously, imported solids were named **ps_solid** and numbered sequentially.

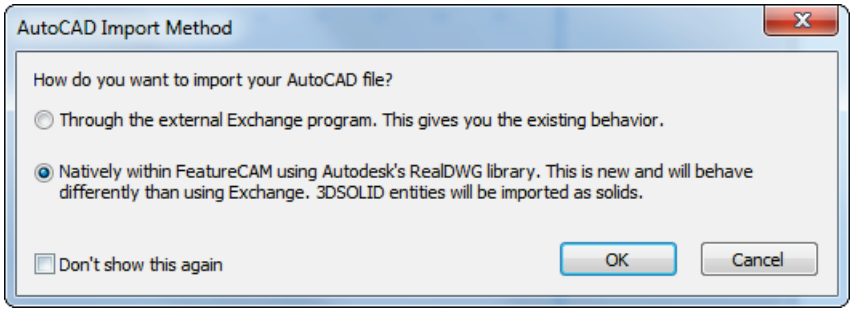
This applies when importing X_T, SolidWorks, SolidEdge files, when pasting from PowerSHAPE, and for any solids imported using Exchange.

You can turn this off with the `importUseParasolidName` variable in the INI file.

Importing AutoCAD files

You can now import AutoCAD files using the FeatureCAM native import. Previously you could only import using Exchange. The FeatureCAM native import is quicker, and enables you to import 3DSOLID, REGION, and BODY entities as solids, instead of as a collection of surfaces.

When you import an AutoCAD file that contains solids, the **AutoCAD Import Method** dialog is displayed.



Select **Natively within FeatureCAM...** and click **OK** to use FeatureCAM's native import.

For some files this may behave differently to the old method of importing using Exchange. To use the old method of importing, select the **File > Import Using Exchange** menu option.

Spatial R25 SP2 Interop integrated

Spatial R25 SP2 Interop is now integrated.

3D InterOp-enabled applications now support CATIA V5-6 R2015 and NX 10.

Parasolid v27.1

FeatureCAM now supports parasolid v27.1.

Milling

FeatureCAM 2016 R1 contains the following changes and improvements to milling:

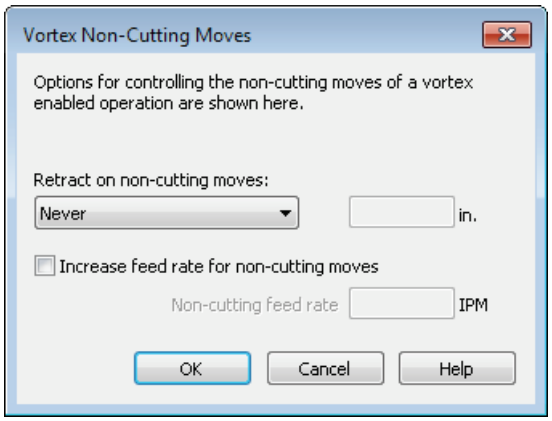
- **Vortex non-cutting moves** (see page 16) — You now have more control over the non-cutting moves in vortex toolpaths.
- **Collision avoidance** (see page 17) — Toolpaths are now collision-checked between unmachined stock and the shank and holder.
- **Swarf machining Z limits** (see page 18) — You can now limit Swarf operations to within a specified Z range on the surface.
- **5-Axis 2D spiral toolpaths** (see page 20) — You can now specify axis smoothing options for 2D spiral toolpaths.
- **Side grooves** (see page 21) — The toolpath calculation for ID and OD Grooves has been improved with several new options and supported functions.
- **Multiple coolant types** (see page 23) — You can now use multiple coolant types for a single operation.
- **User Defined Stock for NT toolpaths** (see page 24) — When using a stock solid and you create a 2D NT toolpath, FeatureCAM now calculates the toolpaths using the stock solid boundary to prevent air-cutting.

Vortex non-cutting moves

You now have more control over the non-cutting moves in vortex toolpaths, which enables you to reduce machining times. Use the new **Vortex Non-Cutting Moves** dialog to specify whether to retract and increase the feed rate on non-cutting moves.

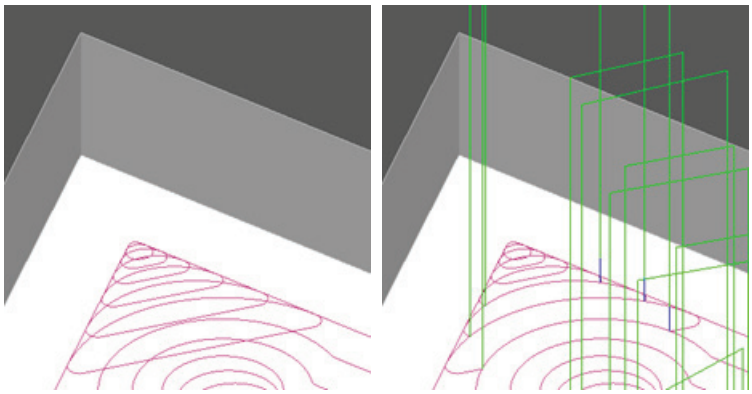
To display the **Vortex Non-Cutting Moves** dialog:

- for 2D toolpaths, click **Non-Cutting Moves** in the **Stepovers** tab of the **Feature Properties** dialog.
- for 3D toolpaths, click **Non-Cutting Moves** in the **Strategy** tab of the **Feature Properties** dialog.
- to change the default options, click **Non-Cutting Moves** in the **Milling** tab of the **Machining Attributes** dialog.



Under **Retract on non-cutting moves**, select whether you want the tool to retract on non-cutting moves:

- **Never** — the tool does not retract on non-cutting moves.
- **Automatic** — FeatureCAM will decide when the tool should retract on non-cutting moves.
- **Longer than** — the tool retracts instead of making non-cutting moves larger than the value you enter. For example, the left image is a standard vortex toolpath and in the right image the non-cutting moves are replaced with retracts.

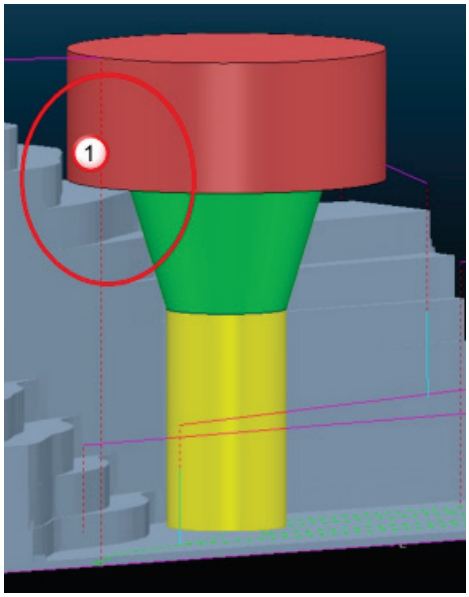


Select **Increase feed rate for non-cutting moves** to override the feed rate for non-cutting moves with the specified **Non-cutting feed rate**. If the **Non-cutting feed rate** is lower than the Feed rate specified on the **F/S** tab, the **Non-cutting feed rate** is ignored.

Collision avoidance

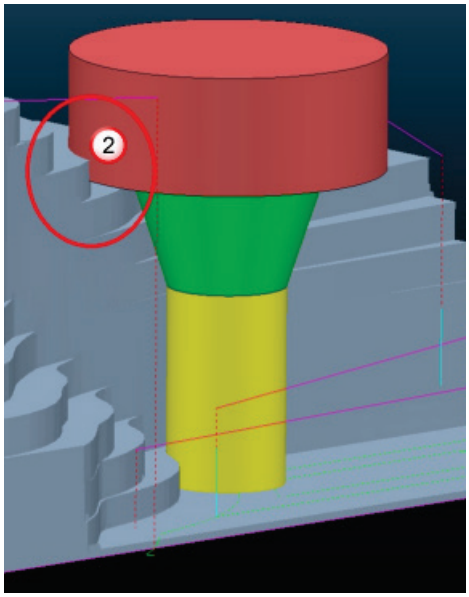
Toolpaths are now collision-checked between unmachined stock and the shank and holder. Previously, you could only collision-check the shank and holder against the model, not the stock.

2016 R1



1 No collision

2015 R3

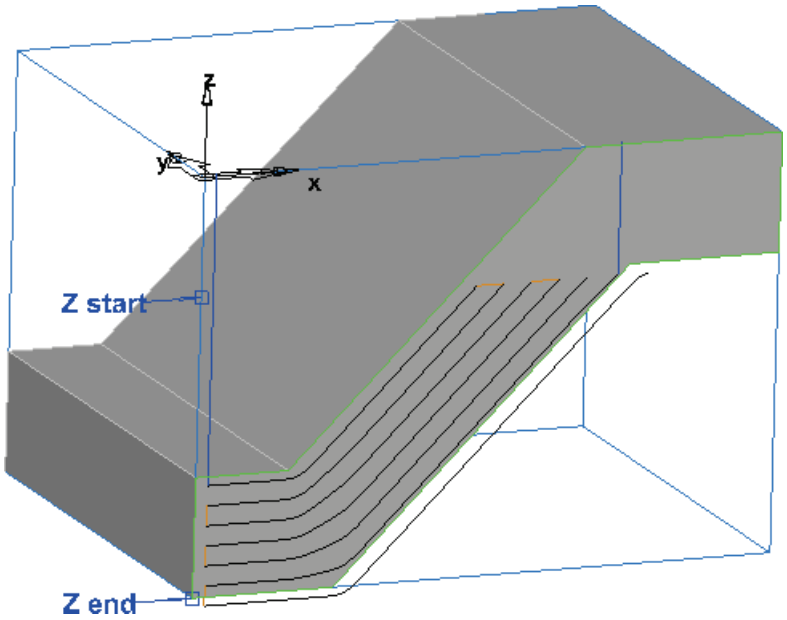


2 Collision

To enable collision-checking against the shank and holder, select **Holder collision clipping** on the **Strategy** tab of the **Feature Properties** dialog.

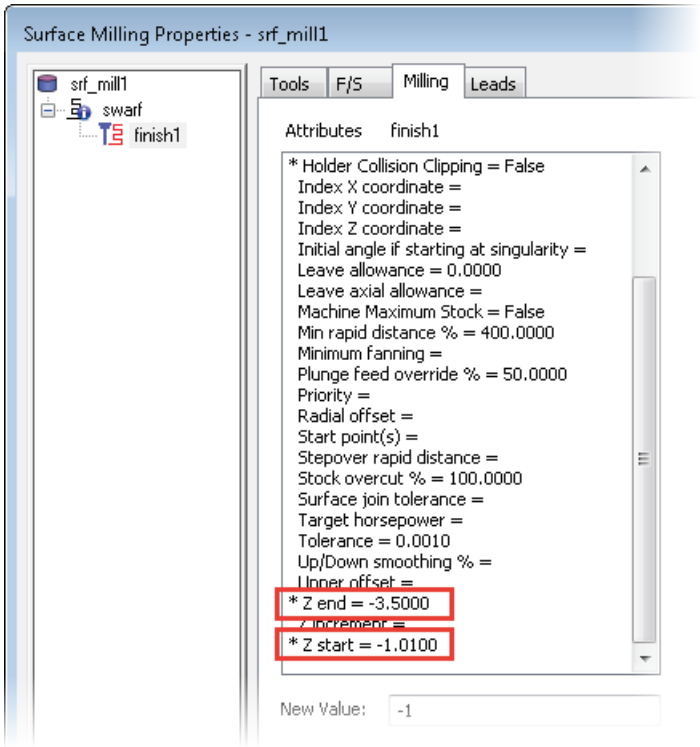
Swarf machining Z limits

You can now limit Swarf operations to within a specified Z range on the surface. This enables you to limit air cutting, such as by limiting the toolpath where part of a surface has already been machined, and gives you more control when using swarf machining with other operations.



This was previously available for most 3D finish operations except swarf.

To specify the Z limits, use the new **Z start** and **Z end** attributes in the **Milling** tab of the **Feature Properties** dialog.



Select **Z start** or **Z end** to display the location of the Z limits in the graphics window.

The toolpath follows the isolines of the surface, but the toolpath is clipped above and below the specified Z level.

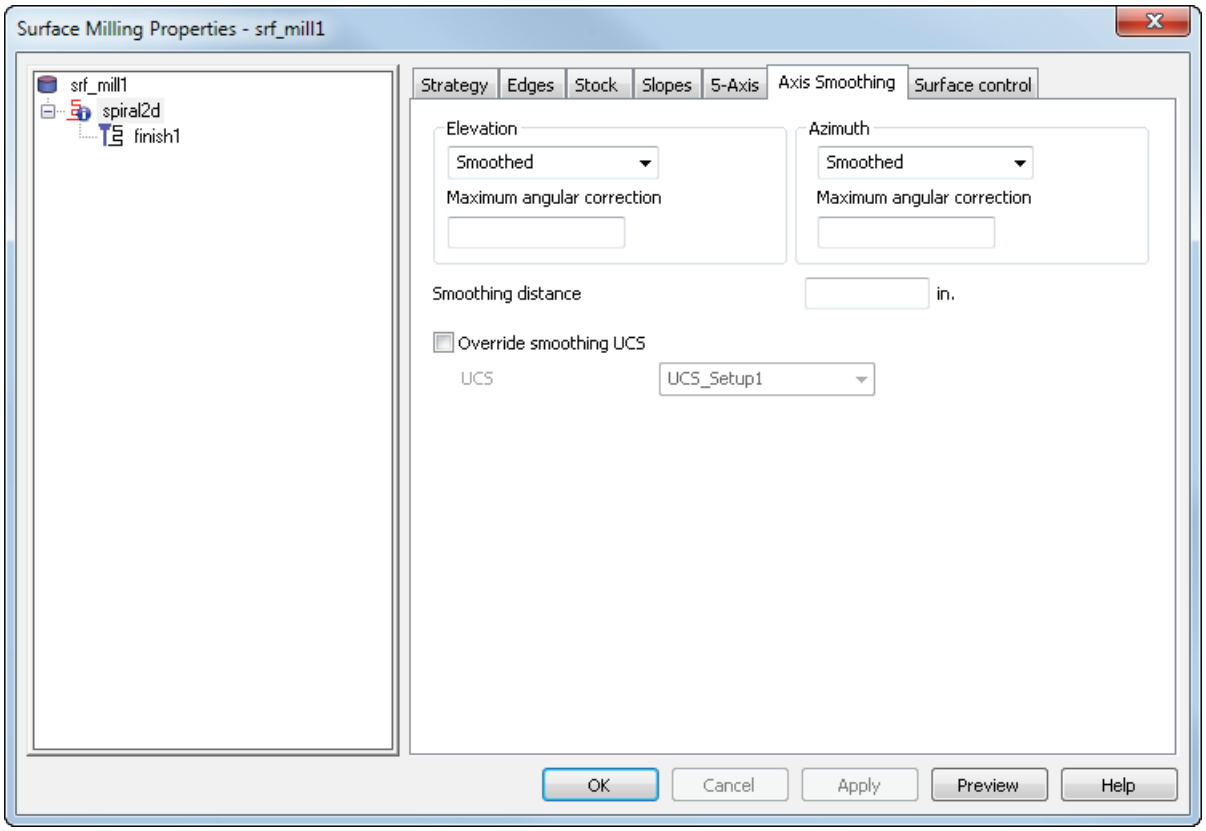
5 Axis simultaneous

For 5 axis simultaneous machining, you may not be able to enter values accurately for **Z start** and **Z end** depending on the rotation of the view and the axes, but you can use the **Pick Z location** button to pick the location in the graphics window and FeatureCAM will calculate the required value.

5-axis 2D spiral toolpaths

In FeatureCAM 2015 R3, the **5-Axis** tab was made available for 2D spiral toolpaths, which enables you to perform 5-axis engraving.

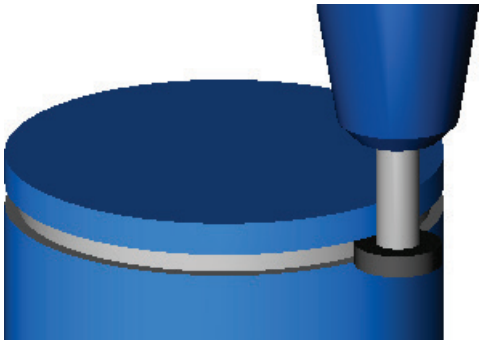
You can now specify axis smoothing options using the **Axis Smoothing** tab, which enables you to create smoother toolpaths which give a better surface finish and reduce tool wear.



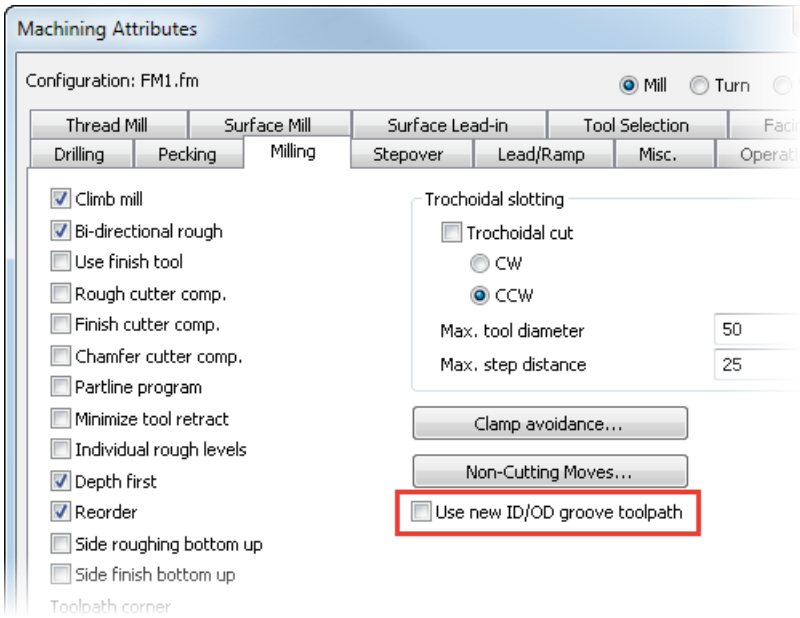
To display the **Axis Smoothing** tab, ensure **Tool axis smoothing** is selected on the **5-Axis** tab, then click **Apply**.

Side grooves

The toolpath calculation for ID and OD Grooves has been improved with several new options and supported functions. This gives you more control when creating side groove features, so you can create better and smoother toolpaths with better gouge checking.



The new toolpath calculation is enabled by default. To disable it, deselect **Use New ID/OD Groove** on the **Milling** tab of the **Machining Attributes** dialog.



When this option is selected, these changes are made to side groove toolpaths:

- The plunge and retract moves are checked for gouges.

  You can disable the gouge checking on plunge and retract moves by deselecting the **Plunge gouge check** option on the **Strategy** tab of the **Feature Properties** dialog.

- You can use wind fan finishing for the finish operation.

  To specify the wind fan settings, click **Wind Fan** on the **Strategy** tab of the **Feature Properties** dialog, and use the **Wind Fan Finish Options** dialog.

- You can use arc lead-in moves for the finish operations. Previously you could only use a linear lead-in move.

  To use an arc lead-in move, select **Arc Lead** on the **Stepovers** tab of the **Feature Properties** dialog.

- You can set a start point for the finish pass.

  To set a start point, use the **Start point** attribute on the **Plunge** tab of the **Feature Properties** dialog.

To create a side groove feature, create a Groove feature and select the **Inside/Outside** option on the **Dimensions** tab of the **Feature Properties** dialog.

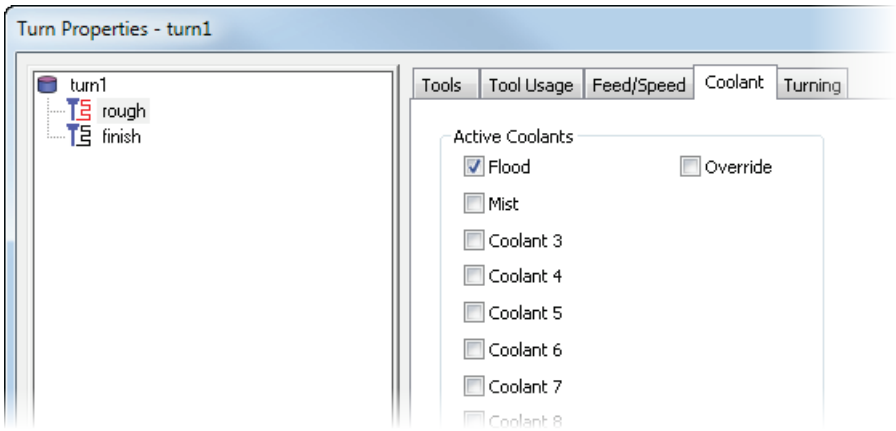
Multiple coolant types

You can now use multiple coolants for a single operation. Previously you could only use one type of coolant for an operation.

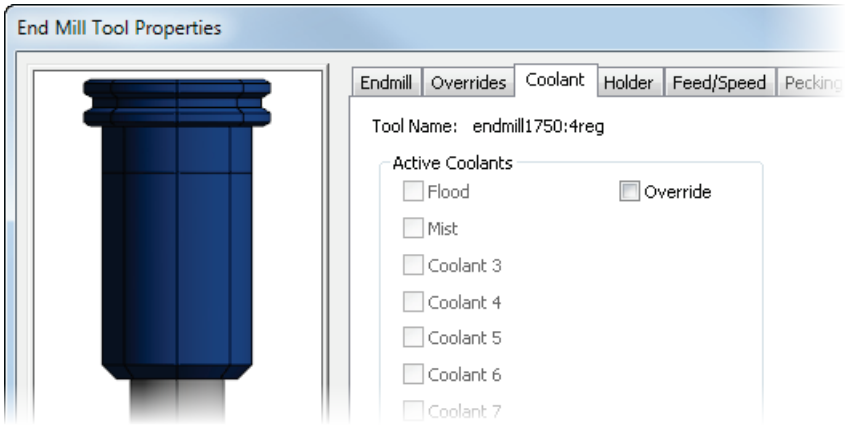
There is a new **Coolant** tab where you can specify the types of coolant for an operation. Previously, you selected the coolant type from a list on different tabs depending on the operation type.

The new **Coolant** tab is available in these places:

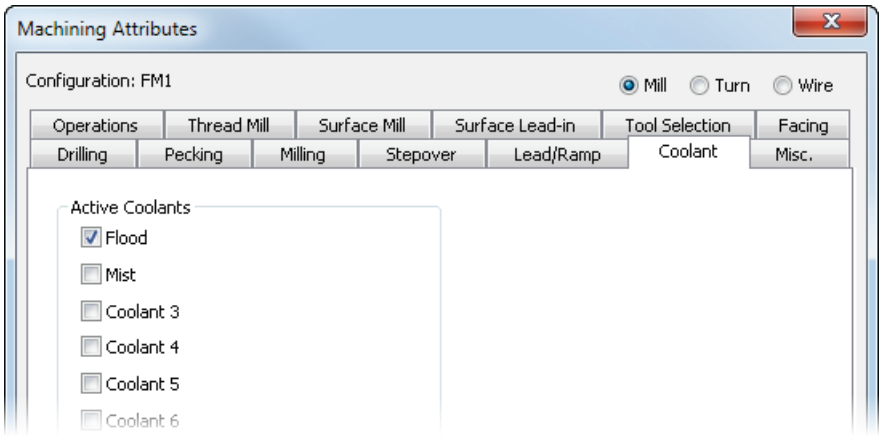
- Set the coolant for an operation in the **Feature Properties** dialog.



- Set the default coolant for a tool in the **Tool Properties** dialog.



- Set the default coolant for the document in the **Machining Attributes** dialog, for both **Mill** and **Turn**.

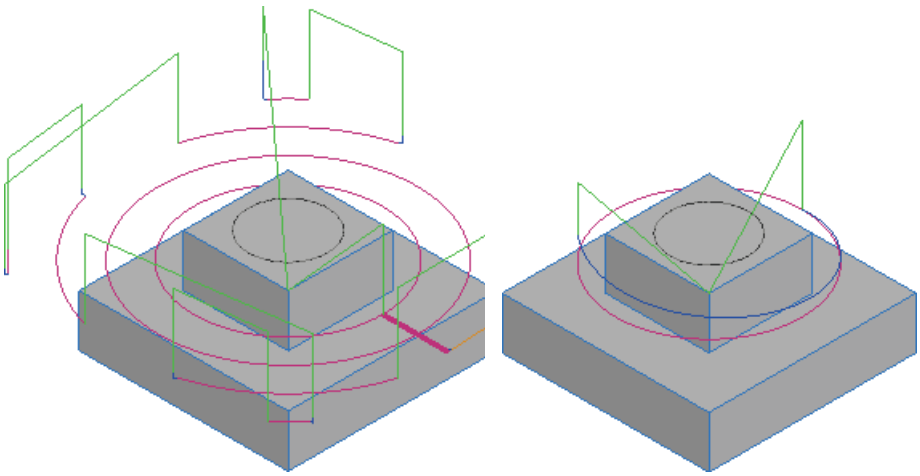


User Defined Stock for NT toolpaths

When using a stock solid and you create a 2D NT toolpath, FeatureCAM now calculates the toolpaths using the stock solid boundary to prevent air-cutting. Previously this was only available for OT toolpaths.

In this example, a Boss feature is cut using a stock solid.

2015 R3: 2016 R1



Turning

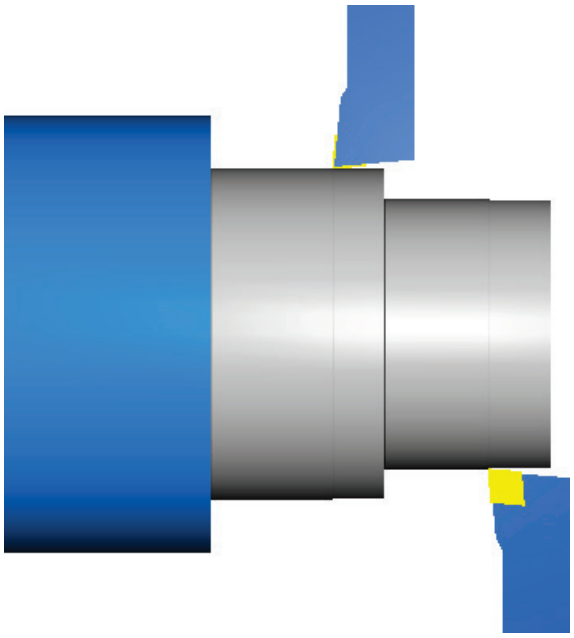
FeatureCAM 2016 R1 contains the following changes and improvements to turning:

- **Turning tool orientation** (see page 26) — You can now use a turning tool in multiple orientations without having to create a different tool in the database for each orientation.
- **Roughing engage angle** (see page 28) — You can now specify the engage angle, lead in angle, and lead out angle for Turning roughing operations.
- **Negative leave allowance for grooves** (see page 29) — You can now use a negative leave allowance for groove features created from curves.

Turning tool orientation

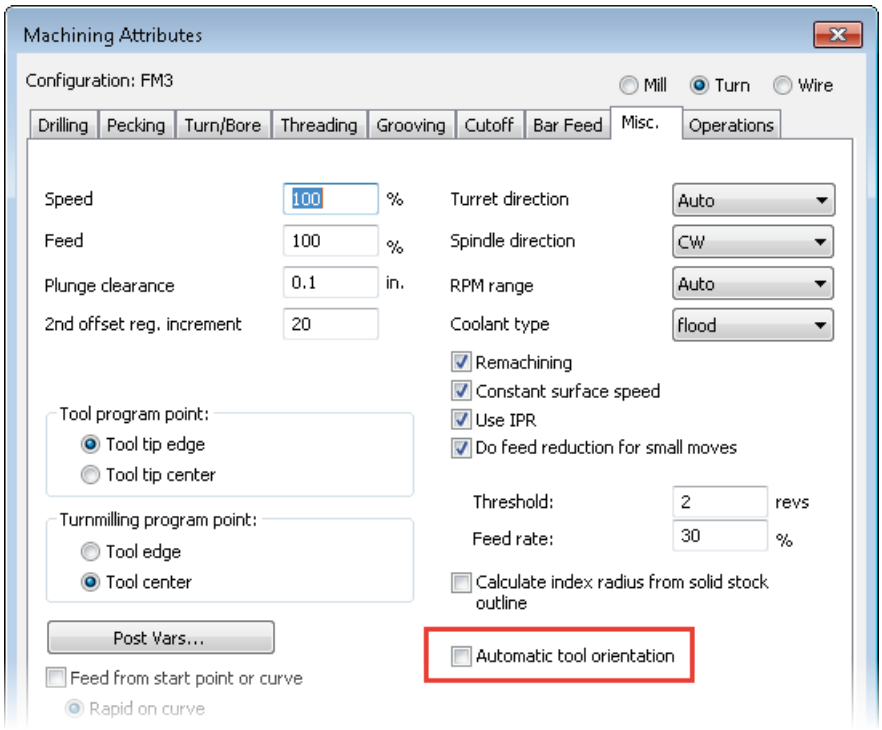
You can now use a turning tool in multiple orientations without having to create a different tool in the database for each orientation.

For example, this tool is used in two orientations, but only one tool is required in the tool database.



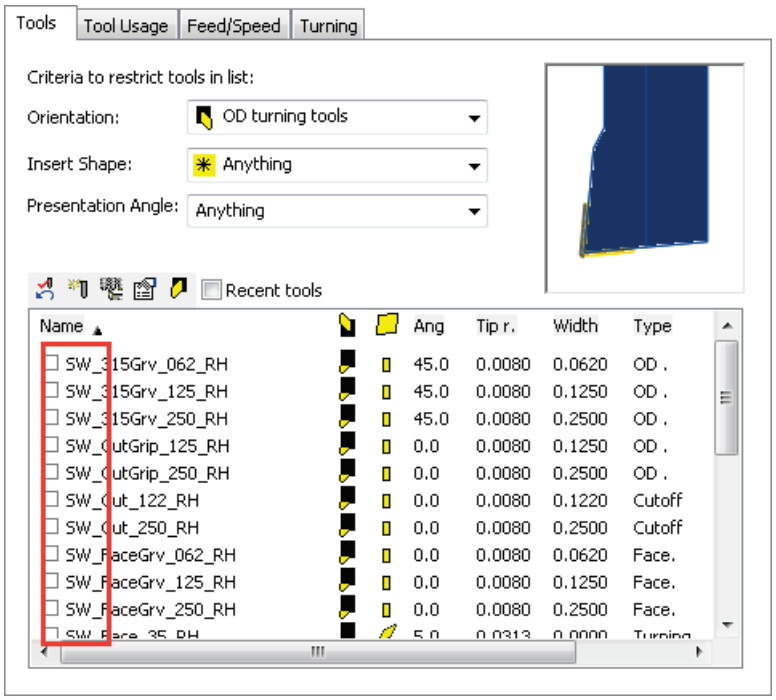
Previously, to use a tool in multiple orientations you had to create duplicates of the tool, and select the tool orientation on the **Orientation** tab of the **Tool Properties** dialog.

Select the new **Automatic tool orientation** option in the **Misc** tab of the **Machining Attributes** dialog.



When **Automatic tool orientation** is selected, only tools in the default orientation are available, but the specified tool orientation is ignored and a tool can be used in any orientation.

For example, for OD turning operations, only tools with a SW orientation are available:

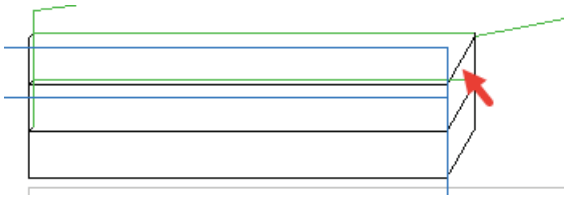


If you have any tools in the non-default orientation, specify the orientation as default and you can use it in any orientation, otherwise the tool will not be available.

Roughing engage angle

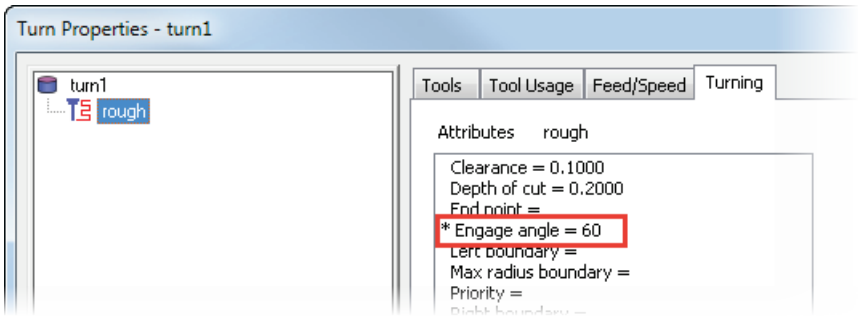
You can now specify the engage angle, lead in angle, and lead out angle for Turning roughing operations. Previously you could specify these options only for the finish pass.

This enables you to prevent the tool from plunging directly into the stock, reducing the load on the tool.



Engage angle

To specify the engage angle for a roughing operation, set the new **Engage angle** option on the **Turning**, **Facing**, or **Boring** tab of the **Feature Properties** dialog.



You can enter an **Engage angle** between 0 and 90. The tool engages the part at the specified angle for boundary moves, but a lower value may be used if the value you enter would be above the rapid level.

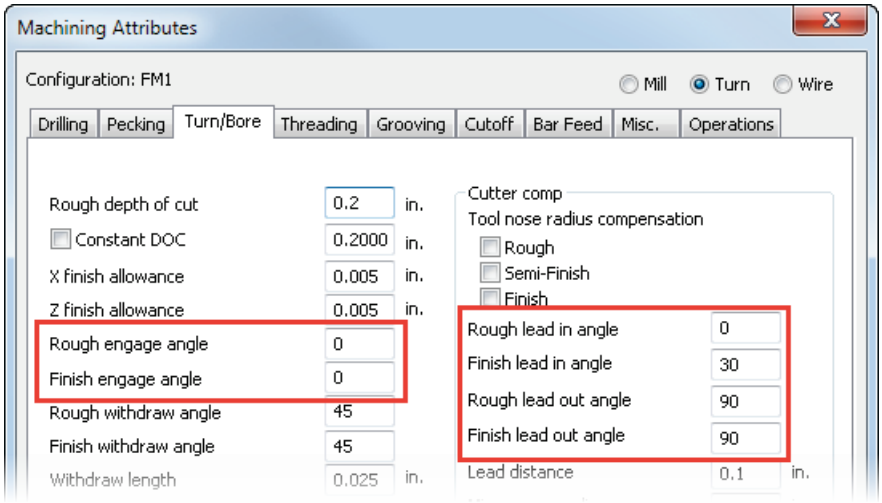
Lead in and lead out angle

To specify the lead in and lead out angles for a roughing operation, ensure the **TNR comp** option is selected on the **Strategy** tab of the **Feature Properties** dialog, and set a **Lead in angle** and **Lead out angle** on the **Turning**, **Boring**, or **Facing** tab.

The tool engages the part at the lead in angle for boundary moves and disengages the part at the lead out angle.

Default values

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



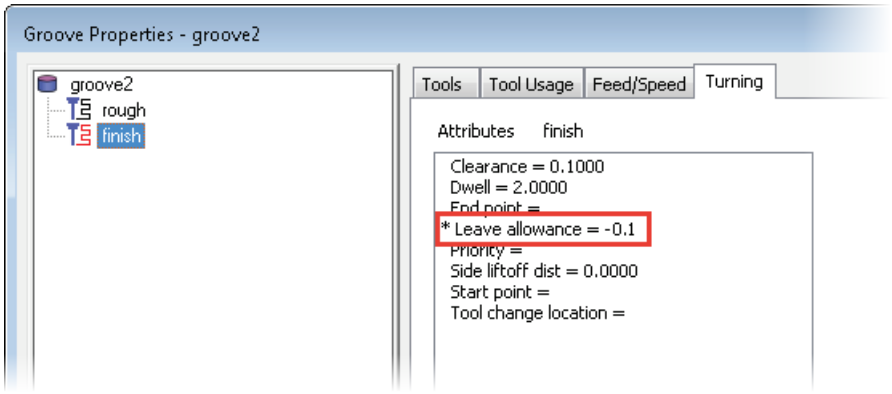
Previously you could set the default **Engage angle**, **Lead in angle**, and **Lead out angle**. You can now set these options separately for rough and finish operations.

Negative leave allowance for grooves

In FeatureCAM you can set a leave allowance to leave unmachined material after a finish pass, and you can set a negative leave allowance to machine past the feature boundary, such as to allow for shrinkage.

You can now use a negative leave allowance for Groove features created from curves. Previously, this was only available for Grooves created from dimensions.

You can edit the **Leave allowance** attribute for an operation on the **Turning** tab of the **Feature Properties** dialog.



To change the default value, enter a **Leave allowance** on the **Turning** tab of the **Machining Attributes** dialog.

Add-ins and extensions

FeatureCAM 2016 R1 contains the following changes and improvements to add-ins and extensions:

- **Setup Sheet add-in** (see page 31) — There are new tags which you can use to create more useful setup sheets.
- **FeatureCAM to CAMplete** (see page 32) — In the **FeatureCAM to CAMplete** dialog (see page 110), there is a new button that enables you to mark solids that are selected in the part as clamps.
- **API improvements** (see page 32) — The API has been improved to support full machine architecture.
- **Nesting add-in** (see page 34) — You can now use PowerSHAPE's nesting tool to nest parts in FeatureCAM.
- **Import Vise add-in** (see page 36) — You can now import vises into FeatureCAM for 3D simulation and gouge checking.
- **Bar-fed mills support** (see page 38) — Bar-fed mills are now supported, such as the Willemin-Macodel 408MT and Mazak integrex i-150.

Setup Sheet add-in

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:

- Turn tool orientation
  - `operation.tool_holder_orientation` — tool holder orientation (for lathe and thread tools)

    You can use this with or without the new **Automatic tool orientation** option (see page 26).
- Spot drill body diameter
  - `use_order_tool.body_diameter` — tool body diameter (User Order Tool Loop)
  - `operation.tool_body_diameter` — tool body diameter (Setup Loop)
  - `tool.body_diameter` — tool body diameter (Tool Loop)
- Drill cycle and first peck
  - `operation.drill_cycle_type` — drill cycle type of the operation
  - `operation.drill_peck_depth` — peck depth of a drill operation
  - `operation.drill_peck_depth2` — peck depth 2 of a drill operation
  - `operation.drill_min_peck` — min peck of a drill operation
- Operation number
  - `operation.number` — the operation number displayed in the **Details** tab of the **Results** window in FeatureCAM
- The CNC post processor file name
  - `fm.cnc_file_name` — The name of the cnc file used to generate the nc code.

- Tool list. See the Setup Sheet help for how to use these options.
  - Show a tool only once in the operation list, even when it is used in multiple operations.
  - List only the tools you have marked. You can mark tools by adding _SHOWTOOL to the **Comments** field in the **Tool Properties** dialog.

To see the Setup Sheet help, click the **Help** button in the **Setup Sheet Options** dialog.

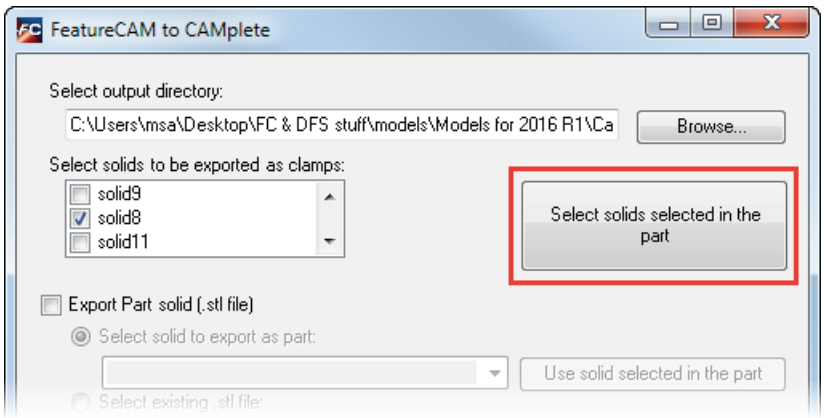
FeatureCAM to CAMplete

CAMplete TruePath is an application that you can use to analyze, modify, optimize, simulate and post 5-Axis toolpaths. You can use the FeatureCAM to CAMplete add-in to export a document to use with CAMplete TruePath.

In the **FeatureCAM to CAMplete** dialog (see page 110), there is a new button that enables you to mark solids that are selected in the part as clamps.

To mark selected solids as clamp solids, click **Select solids selected in the part**.

The solids are selected in the **Select solids to be exported as clamps** list.



FeatureCAM API improvements

The API has been improved to support full machine architecture:

- pallet changes are now supported.
- you can temporarily disable gouge checking between solids.
- you can temporarily remove a solid from the machine heirarchy.
- you can move the tool change location.
- you can query the current operation, feature, tool and setup.

See the FeatureCAM API help for more information. Select the **Help > FeatureCAM API help** menu option to display the FeatureCAM API help.

Nesting add-in

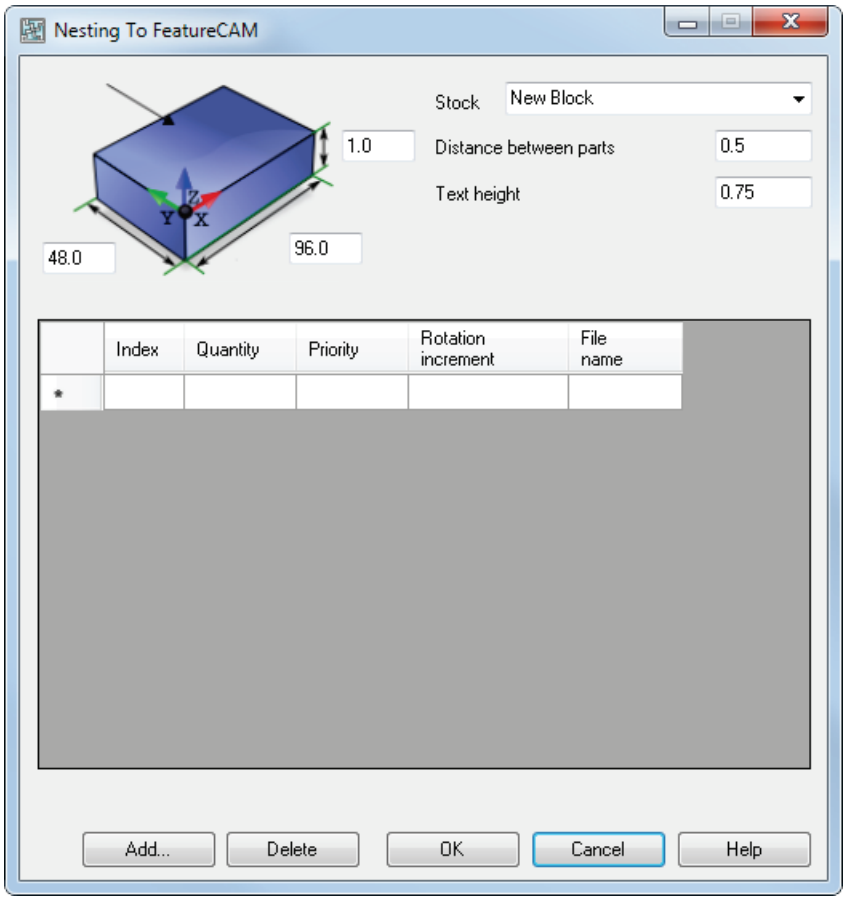
There is a new Nesting add-in that you can use to nest multiple parts and machine them from a single stock using PowerSHAPE's nesting tool.

You must have PowerSHAPE open to use the add-in, and you can only nest parts which have a solid model.

To use the Nesting add-in:

1. Load the **Nesting.bas** add-in.
2. On the **Utilities** toolbar, click **Nesting** .

   The **Nesting To FeatureCAM** dialog is displayed.



3. In the **Stock** list, select the block in which you want to nest the parts.

   Select an existing block to nest additional parts in a block you created previously, or select **New Block** to start a new block.
4. In the fields around the stock diagram, enter the dimensions of the block in which you want to nest the parts.
5. Enter the **Distance between parts** to specify the minimum distance you want to leave between the nested parts on the block.

6 Enter a **Text height** to specify the height of the label on each part.

7 Click **Add** and use the **Open** dialog to select the parts you want to nest.

8 Enter the **Quantity** to specify how many duplicates of each part you want to nest in the block.

9 Enter the **Priority** to determine which parts to nest first. The parts with lowest **Priority** values are nested first. For parts with the same **Priority** value, the largest parts are nested first.

10 Enter the **Rotation increment** to specify the smallest increment by which the part can be rotated when nesting. You may be able to fit more parts in a block by reducing the **Rotation increment**.

11 To remove a part from the dialog, select the row and click **Delete**.

12 Click **OK** to nest the parts.

FeatureCAM opens each file and puts all the features into a group, then adds the features to a new FM file. The solids are sent to PowerSHAPE and nested using PowerSHAPE's nesting tool.

When the nesting is completed in PowerSHAPE, a message dialog is displayed telling you to modify the positions of the parts in PowerSHAPE if necessary.

13 Modify the position of the parts in POWERSHAPE, then click **OK** to close the message dialog.

The positions of the parts are updated in FeatureCAM.

A message dialog is displayed asking if you want to save the stock.

14 Click **Yes** to run a 3D simulation and save the stock. This enables you to nest additional parts in this block later.

You can select saved stocks in the **Stock** list in the **Nesting To FeatureCAM** dialog to nest additional parts in a saved stock.

A message dialog is displayed saying the nesting is completed.

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

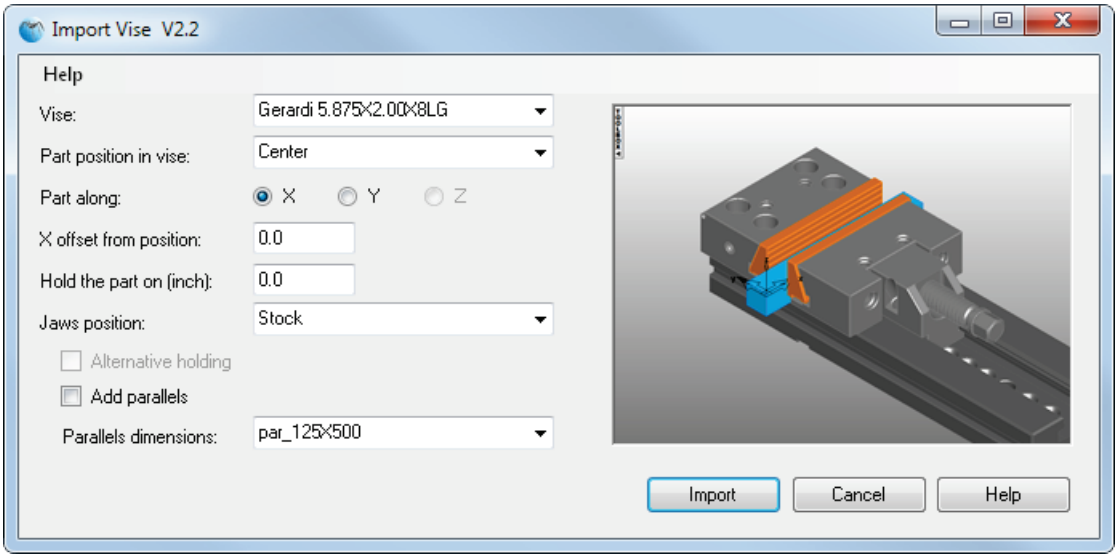
Import Vise add-in

There is a new Import Vise add-in that you can use to import vises into FeatureCAM for 3D simulation and gouge checking.

1 Load the **Import_Vise.bas** add-in.

2 On the **Utilities** toolbar, click **Import_Vise**.

The **Import Vise** dialog is displayed.



3 In the **Vise** list, select the vise you want to import.

A preview of the vise is displayed in the dialog.

Vises

1 Select an option in the **Part position in vise** list to specify how to align the part in the vise.

2 Under **Part along**, select the axis that you want to be perpendicular to the jaws to specify the orientation of the part in the vise.

3 Under **X offset from position**, enter the offset of the part along the axis parallel to the jaws. You can enter a negative value.

For example, select a **Part position in vise** of **Left** and enter a **X offset from position** of **-1** to extend the part by 1 inch past the left edge of the jaws.

4 Under **Hold the part on**, enter the height at which to hold the part in the vise.

5 Under **Jaws position**, select how you want to position the jaws to hold the part.

For **Selected Solid Faces**, **Selected Surfaces**, and **Selected geometry**, select items in the graphics window.

6 For vises with multiple holding positions, select **Alternative holding** to use the alternative holding position.

7 To add parallels to raise the part in the vise, select **Add parallels** and select the **Parallels dimensions** from the list.

8 Click **Import** to import the vise.

Turning chucks

1 Under **Part along**, select the axis about which the chuck rotates to specify the orientation of the part in the vise.

2 Under **Z offset from position**, enter an offset to raise or lower the part in the jaws.

3 Under **Part length from Jaws Faces**, enter the length of part you want to extend past the top of the jaws.

4 Under **Jaws position**, select how you want the jaws to hold the part.

For **Selected Solid Faces**, select solid faces in the graphics window.

5 Select the **Jaws types** you want to use.

6 Click **Import** to import the vise.

Bar-fed mills support

Bar-fed mills are now supported, such as the Willemin-Macodel 408MT and Mazak integrex i-150. Bar-fed mills enable you to machine multiple parts 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.

This requires a UDF add-in, and a customized MD file and post.

XBUILD

FeatureCAM 2016 R1 contains the following changes and improvements to XBUILD:

- **Intermediate CNC files** (see page 40) — You can now split the nc file into multiple intermediate files, and then combine these intermediate files in different ways to create multiple output files.
- **Multi-code coolant** (see page 42) — You can now use different on and off codes for a single coolant type, depending on which turret and tool is being used.
- **5-axis untransformed coordinates** (see page 44) — There are new milling reserved words that contain the untransformed X, Y, and Z coordinates in a 5-axis part.
- **Formatting reserved words** (see page 45) — When formatting numeric reserved words in XBUILD, you can now remove trailing zeros from integer values without removing the decimal point.
- **CNC documentation** (see page 46) — The HTML documentation has been improved to make the output more readable.

Intermediate CNC files

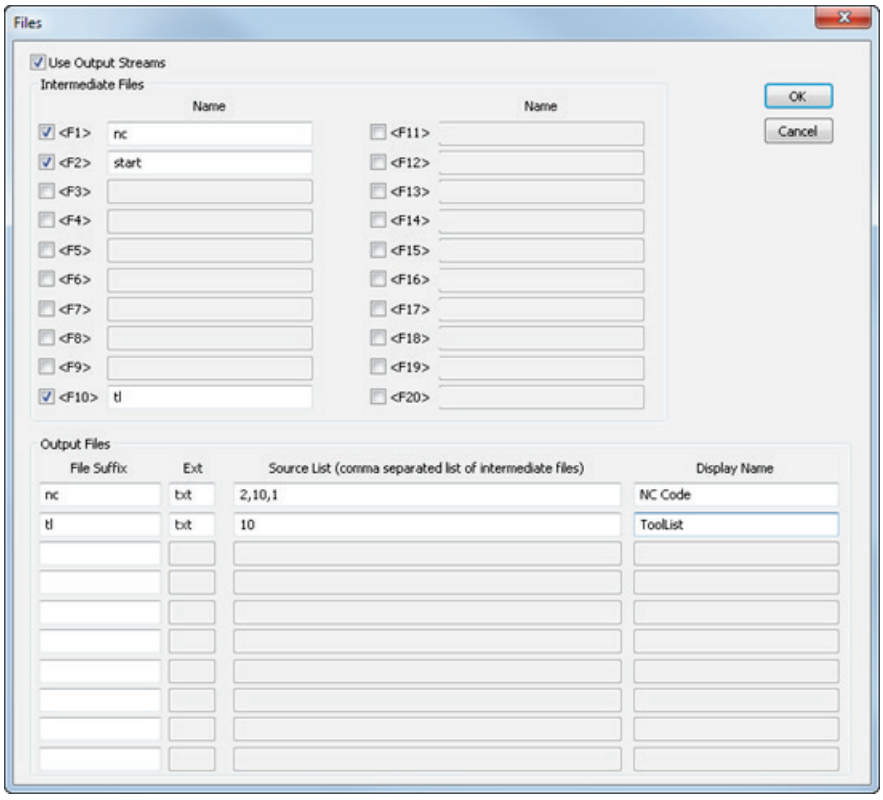
You can now split the nc file into multiple intermediate files, and then combine these intermediate files in different ways to create multiple output files. This gives you more control over the nc code files you create.

For example, you can use this to create a tool list at the start of the output file easily without needing to use a macro, by separating each line of tool information into an intermediate file, and placing it at the beginning of the output file.

You can also use this to group Wait codes together and output them at a single point in the post, and avoid mismatched wait codes when outputting to multiple channels.

To use multiple output streams for the nc code:

1. In XBUILD, select the **CNC-Info > Manage Files** menu option to display the new **Files** dialog.



2. Select **Use Output Streams**.
3. Under **Intermediate Files**, select the intermediate nc files you want to create and enter a **Name**.

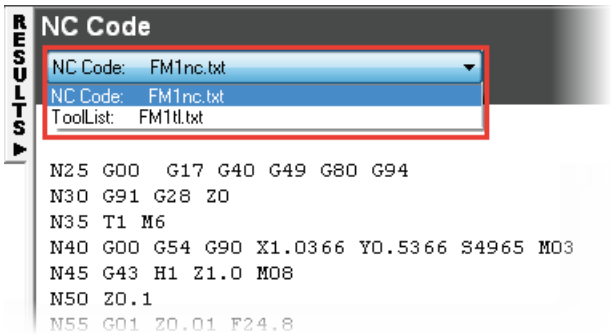
   Each intermediate file has a number, which you can use to send individual lines of the nc code into different intermediate files.

4. Under **Output Files**, specify the final nc files you want to create and output from FeatureCAM.
   a. Enter the **File Suffix** and **Ext** to specify the suffix and extension that will be applied to the file you output.
   b. Under **Source List**, enter the intermediate files you want to combine to create the output file.

      In the example above, the **NC Code** output file includes all the intermediate streams, and the **ToolList** output file includes only the tool list intermediate file.
   c. Enter the **Display Name**. This is the name of the nc file when displayed in the **NC Code** tab of the **Results** window in FeatureCAM.
5. Add the new reserved words to a format to separate the lines of nc code into the intermediate files.
   - `<F1>` to `<F20>` — Use this word at the start of a line to send the line to an intermediate file.
   - `<SET-F1>` to `<SET-F20>` — Use this word at the start of a line to send all proceeding information to an intermediate file, including in other formats.

   Alternatively you can use the intermediate file name, such as `<F:start>` and `<Set-F:start>`.

   For example, use `<F10>` before the tool name in the Tool Change format to send the tool name to the tenth intermediate file. Use `<SET-F1>` at the beginning of the Program Start format to send all nc code except the tool names to the first intermediate file.
6. Simulate the toolpaths in FeatureCAM.
7. In the **NC Code** tab of the **Results** window, select an output file in the list to display it.



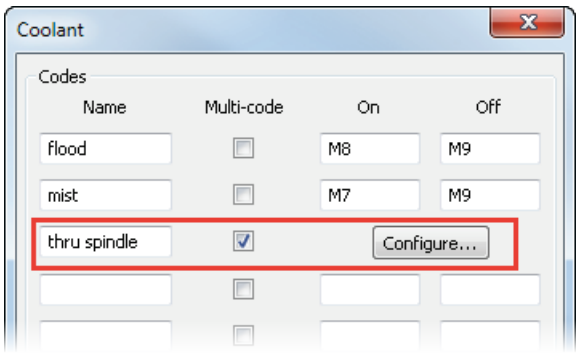
8. Use the **Save NC** dialog to save the nc code output files.

Multi-code coolant

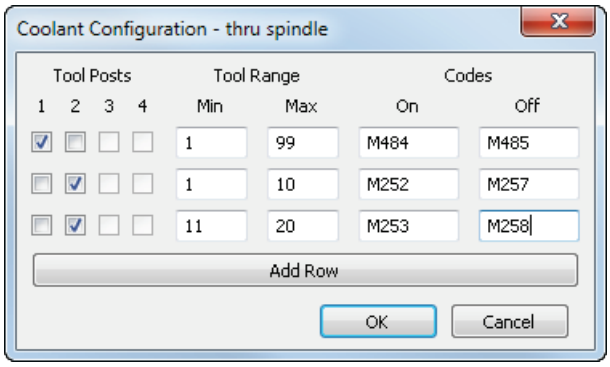
You can now use different on and off codes for a single coolant type, depending on which turret and tool is being used.

To use multi-code coolant:

1. In XBUILD, in the **Coolant** dialog, select the new **Multi-code** option to enable multi-code coolant for a coolant type.

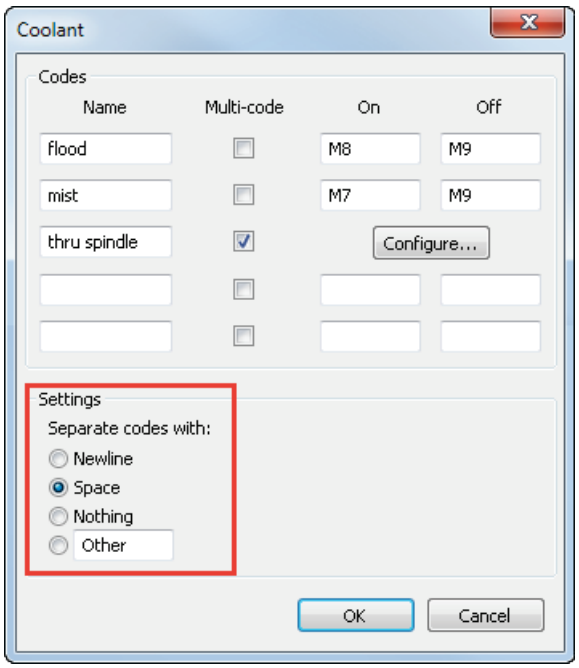


2. Click **Configure** to display the **Coolant Configuration** dialog.



3. For each row, select the **Tool Posts** for which you want to specify the **On** and **Off Codes**.
4. Enter the **Tool Range**, the range of tool numbers for which you want to specify the **On** and **Off Codes**.
5. Enter the **On** and **Off Codes**.
6. If needed, click **Add Row** to add more rows.
7. Click **OK** to close the dialog.

8 In the **Coolant** dialog, under **Separate codes with**, select an option to determine how to separate multiple codes when they are used together in the NC code.



Newline — Select this option to use a line break between coolant codes in the NC code.

Space — Select this option to use a space between coolant codes in the NC code.

Nothing — Select this option to not separate coolant codes in the NC code.

Other — Select this option and enter a string to use between coolant codes in the NC code.

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

5-axis untransformed coordinates

There are new milling reserved words that contain the untransformed X, Y, and Z coordinates in a 5-axis part. For example, you can use these in a post for positioning before turning on the RTCP transformation.

The new reserved words are:

- <UNXFORM-X-COORD>
- <UNXFORM-Y-COORD>
- <UNXFORM-Z-COORD>

These are only valid in the Program Start, Segment Start and Tool Change formats.

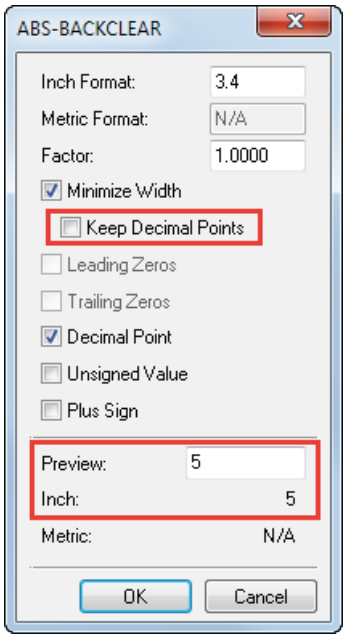
Formatting reserved words

When formatting numeric reserved words in XBUILD, you can now remove trailing zeros from integer values without removing the decimal point. Use this option to ensure numeric reserved words are interpreted correctly by your machine tool controller.

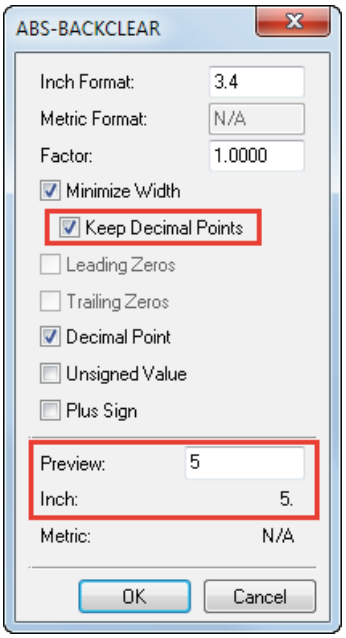
In the **Word Formatting** dialog, select **Minimize width** to remove the trailing zeros and select the new **Keep Decimal Points** option to keep the decimal point.

For example, when **Keep Decimal Points** is deselected, a value of **5** is output as `5`, when **Keep Decimal Points** is selected, it is output as `5.`.

Keep Decimal Points deselected:



Keep Decimal Points selected:



CNC documentation

In XBUILD, you can select the **File > Document CNC > HTML** menu option to create HTML documentation about the CNC file.

The HTML documentation has been improved to make the output more readable:

- The CSS and HTML style has been modified to better present the output.
- Sections of the documentation with no content are not displayed in the output.
- Unnamed post variables are not displayed in the output.
- Some sections of the documentation have been removed, such as Motion Commands, Spindle, Synchronize Spindles, Compensation Commands, Stop Commands, Airblast, and Circular Planes.
- Unnecessary data has been removed from the General Info, Machine Info, Turret Info, Feeds and Speeds Info, Milling Cycle, and Turning Cycle sections.
- Several section headers have been renamed, and table elements have been reworded to be more clear.

What's new in FeatureCAM 2015 R3

FeatureCAM 2015 R3 contains the following new features and enhancements:

User interface

- **Improved icons** (see page 50) — Several icon graphics across FeatureCAM have been updated.
- **Selecting objects** (see page 50) — There is a new **Select Partial** mode.
- **Inserting Part Library features** (see page 51) — You can now use the **New Feature** wizard to insert Part Library features into a document.
- **Pasting features using polar coordinates** (see page 52) — You can now use polar coordinates to locate features in the **Paste Special** wizard.
- **Deleting unused curves** (see page 53) — 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 55) — You can now import SolidWorks 2015 files into FeatureCAM.
- **Importing SolidEdge ST7 files** (see page 55) — You can now import SolidEdge ST7 files.
- **Spatial R25 SP1 Interop integrated** (see page 56) — Spatial R25 SP1 Interop is now integrated.

Milling

- **Creating Faces features using IFR** (see page 58) — You can now automatically round the corners of Face features that are created using Interactive Feature Recognition.
- **Automatic tool selection** (see page 59) — 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 60) — You can now simulate complicated tool holders more accurately.
- **Machine movement limits** (see page 60) — 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 63) — 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 65) — You can now specify the program point for turnmilling tools.
- **Turning head tool holders** (see page 66) — Turning head tool holders are now supported.

5-axis machining

- **C-axis indexing** (see page 71) — 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 73) — You can now create 5-axis engraving features with the tool axis normal to the surface.

Machine Design

- **Specifying machine movement limits** (see page 76) — You can now set the limits of movement for solids in a Machine Design file.
- **Testing machine movements** (see page 77) — 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 79) — 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 50) — Several icon graphics across FeatureCAM have been updated.
- **Selecting objects** (see page 50) — There is a new **Select Partial** mode.
- **Inserting Part Library features** (see page 51) — You can now use the **New Feature** wizard to insert Part Library features into a document.
- **Pasting features using polar coordinates** (see page 52) — You can now use polar coordinates to locate features in the **Paste Special** wizard.
- **Deleting unused curves** (see page 53) — 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:

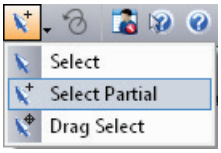
- **Feature Properties** dialog
- **Part View**
- **Part Library**
- **Tombstone Process Plan** dialog

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.

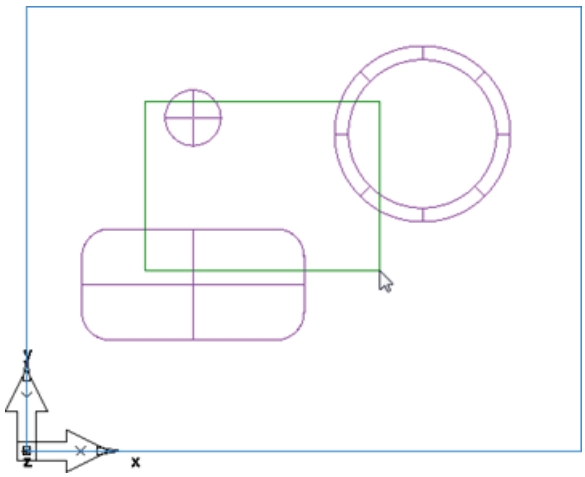
To enable the new **Select Partial** mode:

- 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.

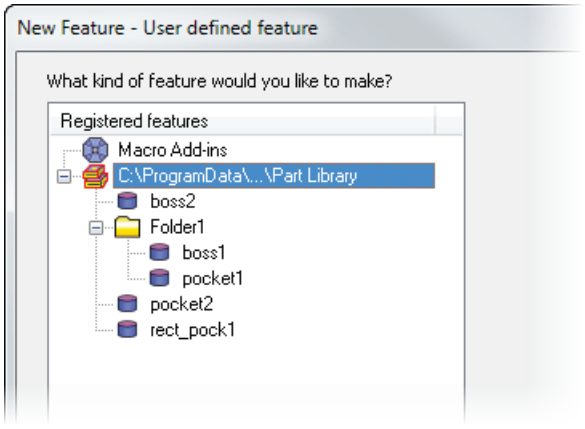
Inserting features from the Part Library into the document

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.

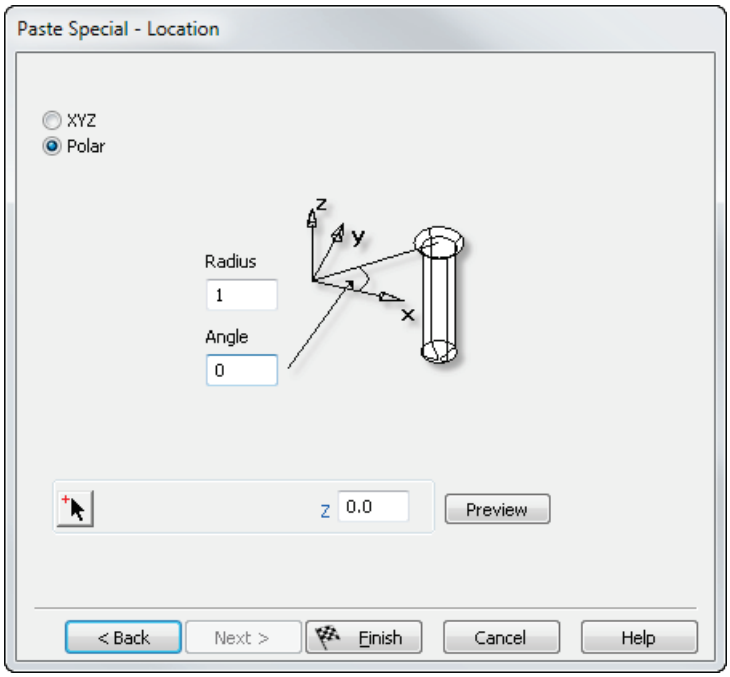
3 Select **Paste the clipboard contents. Select a new location** and click **Next**.

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 55) — You can now import SolidWorks 2015 files into FeatureCAM.
- **Importing SolidEdge ST7 files** (see page 55) — You can now import SolidEdge ST7 files.
- **Spatial R25 SP1 Interop integrated** (see page 56) — Spatial R25 SP1 Interop is now integrated.

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.

Spatial R25 SP1 Interop integrated

Spatial R25 SP1 Interop is now integrated.

InterOp now supports Creo 3.0 BREP, Assembly, PMI, and Graphical translation.

Milling

FeatureCAM 2015 R3 contains these changes and improvements to Milling:

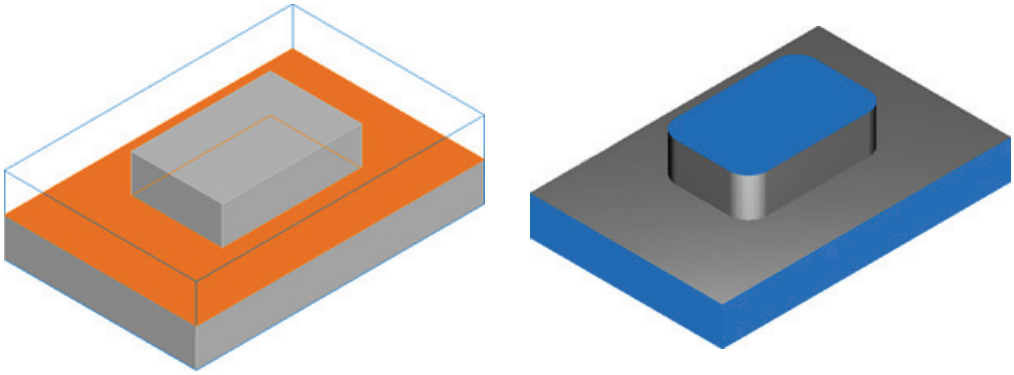
- **Creating Face features using IFR** (see page 58) — You can now automatically round the corners of Face features that are created using Interactive Feature Recognition.
- **Automatic tool selection** (see page 59) — You can now specify an additional clearance on the automatic tool selection to ensure the tool holder does not collide with the stock.
- **Simulating tool holders** (see page 60) — You can now simulate complicated tool holders more accurately.
- **Machine movement limits** (see page 60) — 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.

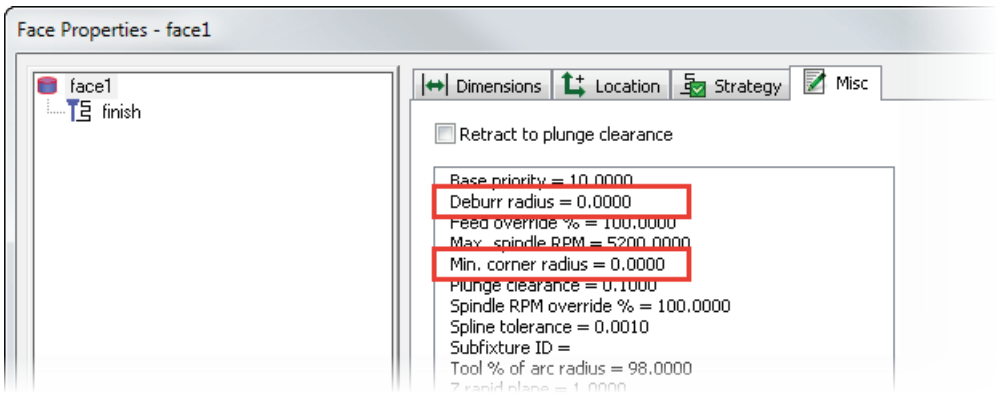
This enables you to use a Face feature instead of a Side feature in some situations, which can reduce the machining time by reducing air cutting. This is especially useful when using cutter compensation and partline programming.

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.

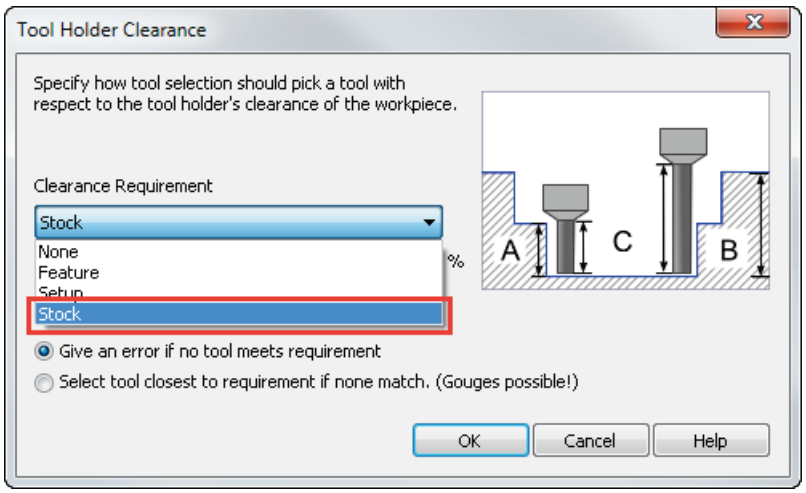
4 Click **OK** to accept your changes to the feature and close the dialog.

*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 92) 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**.

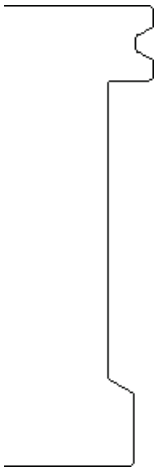
   The new **Tool Holder Clearance** dialog is displayed.
3. In the **Clearance Requirement** list, select **Stock**.
4. Click **OK** to close the dialog.

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.

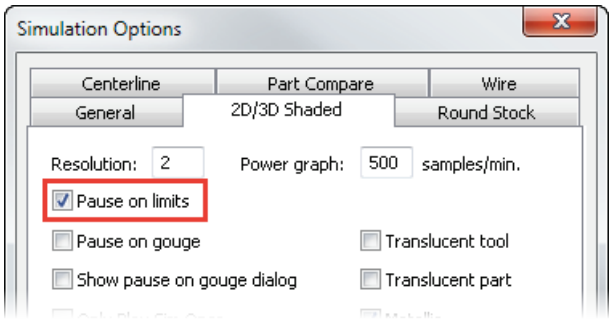
The tool holder shape does not take into account undercuts, for example:

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

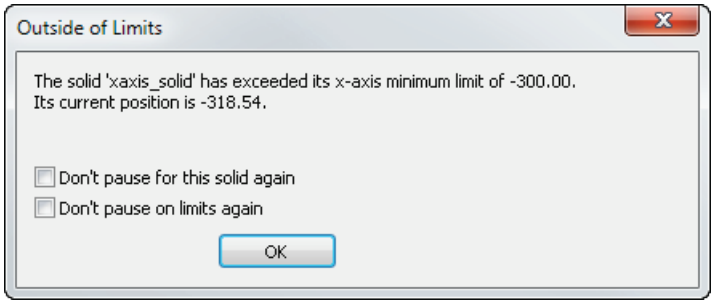
Machine movement limits

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 76), you need to edit the Machine Design file.

Turning and Turn/Mill

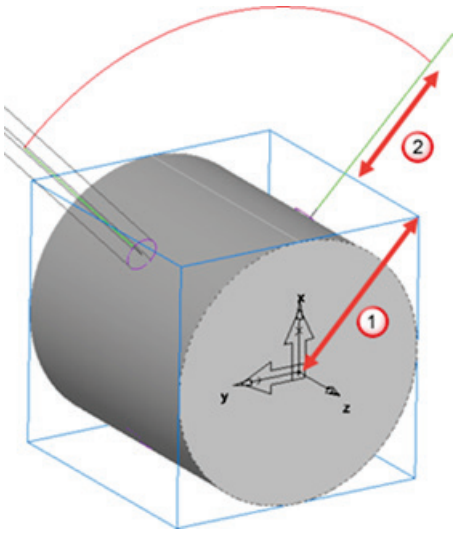
FeatureCAM 2015 R3 contains these changes and improvements to Turning and Turn/Mill:

- **Indexing using a stock solid** (see page 63) — 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 65) — You can now specify the program point for turnmilling tools.
- **Turning head tool holders** (see page 66) — Turning head tool holders are now supported.

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:

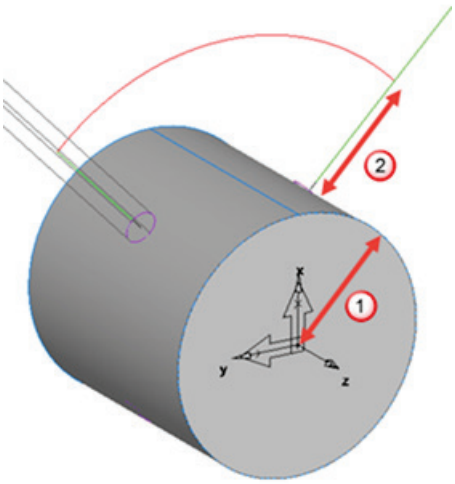
Calculate index radius from solid stock outline off:



1 — Distance to bounding box

2 — **Z rapid level** attribute value

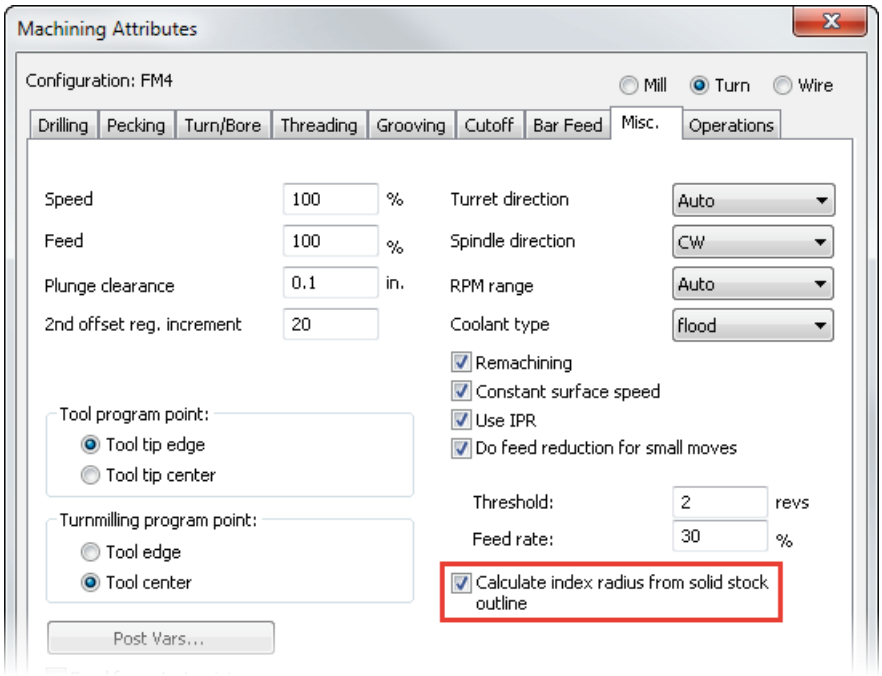
Calculate index radius from solid stock outline on:



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.

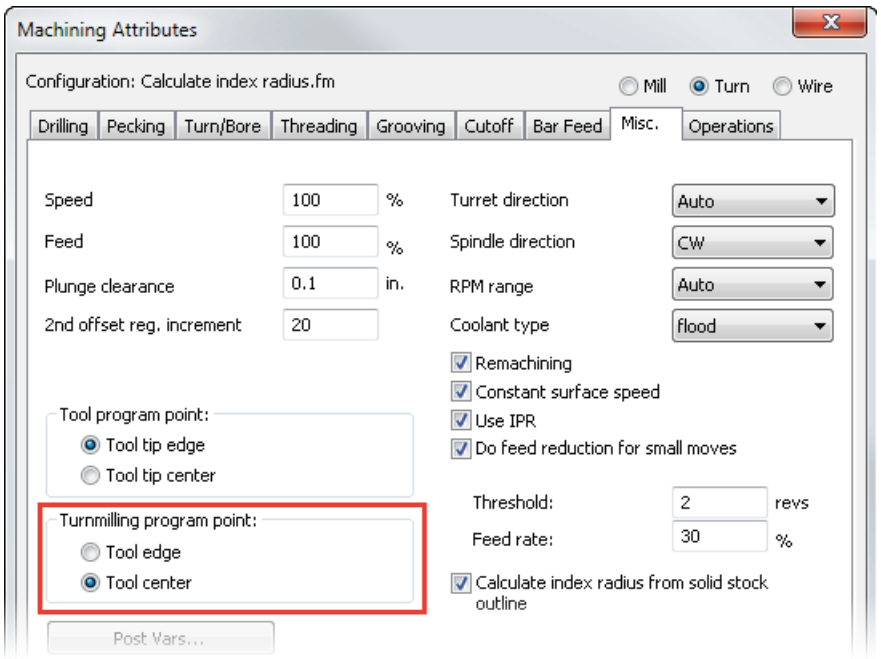


Turnmilling program point

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.



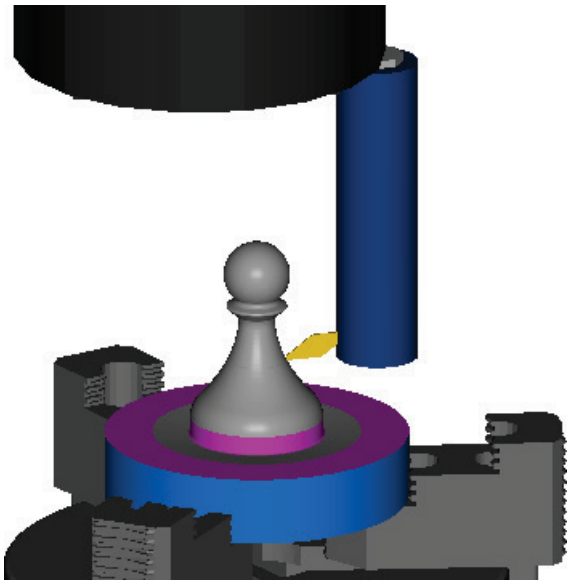
This option does not affect turnmilling tools that are used for milling.

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

Turning head tool holder support

Turning-head tool holders are supported in FeatureCAM, which enable you to perform turning and boring operations on a milling machine.

For example, in the image below the piece is machined by rotating the tool around the stock.



This requires the **Advanced Turn/Mill (MTT)** module, and it requires a modified post processor that provides the U coordinate.

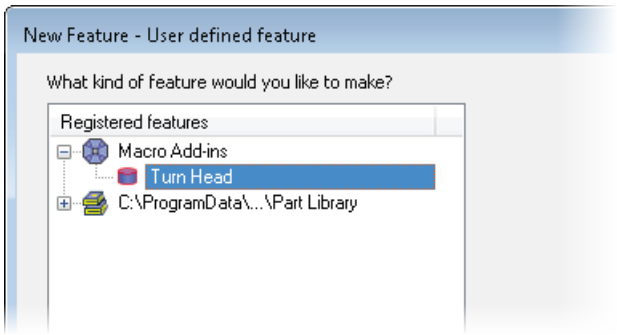
4-Axis indexing is supported, but 5-Axis positioning is not.

To create a turning head feature:

1. Create a Setup over the center of rotation of the turning head, so that the tool will rotate about the Setup Z axis.
2. Create a curve in the XZ plane that defines the profile of the turned shape.
3. Load the **TurningHeadCS.dll** add-in using the **Macro Add-ins** dialog.
4. In the **New Feature** wizard, under **From Feature**, select **User** and click **Next**.

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

5 In the **Registered features** list, under **Macro Add-ins**, select **Turn Head**, and click **Next**.

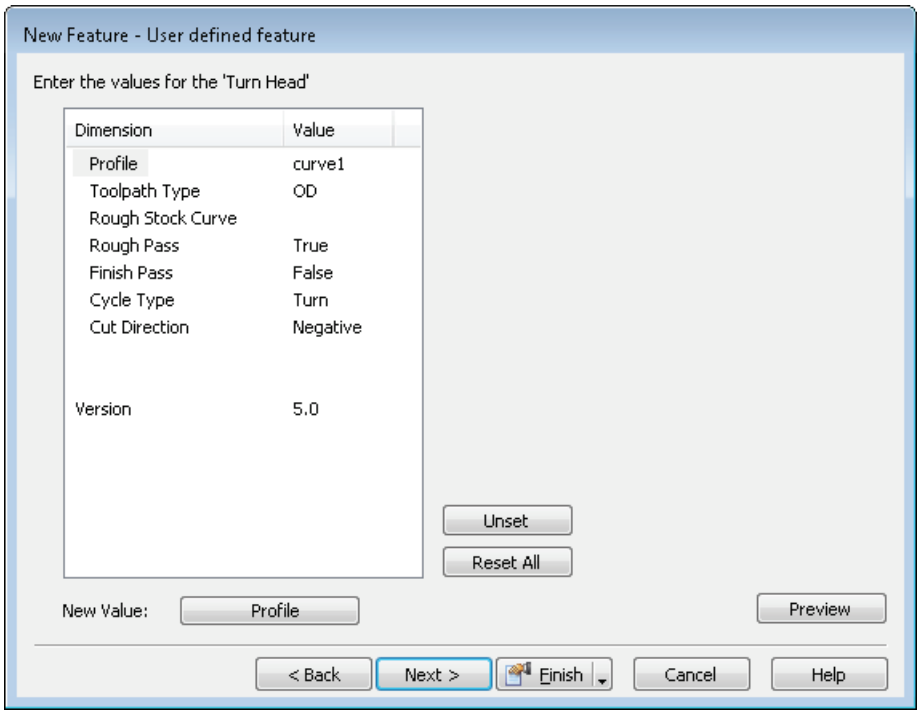


The **Curves** page is displayed.

6 Select the curve in the graphics window and click **Add from selected items** .

7 Click **Next** to display the **Location** page.

8 Click **Next** to display the **User defined feature** page.



9 Specify the parameters to define the feature. To change a parameter, select the parameter name, select an option in the **New Value** list, then click **Set**.

Profile — Select a curve to define the turning feature profile.

Toolpath Type — Select whether you want to create an Inside Diameter or Outside Diameter feature.

Rough Stock Curve — Select the curve that defines the toolpath boundary for the feature. Leave this unset to machine to the stock boundary.

Rough Pass — Select **True** to include a rough operation.

Finish Pass — Select **True** to include a finish operation.

Cycle Type — Select the cycle type.

In a **Turn** cycle, the roughing tool feeds along the Z axis while stepping down the X axis.

In a **Face** cycle, the roughing tool feeds from the outside of the part to the center while stepping down in the negative Z direction.

Cut Direction — Select the direction along the Z axis you want to cut the feature.

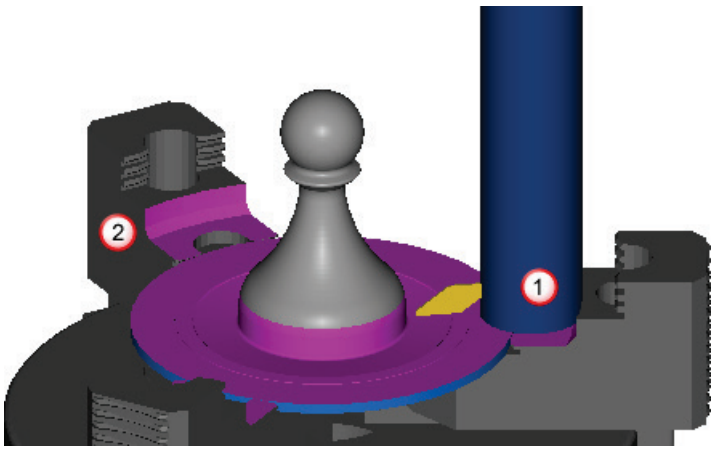
10 Click **Finish** to create the Turn Head feature and close the dialog.

11 Use the **Tool Block Selection** dialog to select the tool block solid as a tool block for the turning tools.

12 You may need to adjust the **Start point** and **End point** on the **Turning** tab of the **Turn Head Properties** dialog for each operation to ensure the tool does not collide with the part at the start and end of the toolpath.

13 Run a simulation to check for collisions.

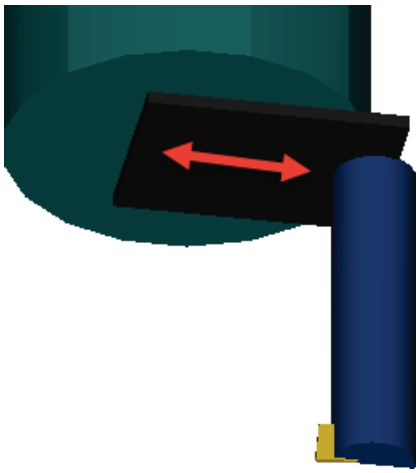
The rotation of the tool is not displayed in the simulation, but the tool is checked for collisions and gouges. For example, in this image the tool has gouged with the clamp at ①, and another gouge is displayed at ②.



Machine Design

To modify a Machine Design file to support turning head tool holders:

1. In the Machine Design document, ensure the **Machine Design > Enable Turn/Mill UI** is selected.
2. Create or import solids to represent the turning head tool holder and the plate that moves in the U axis that holds the turning tool.



3. Use the **Parent/Child Relationships** dialog to make the U plate solid a child of the turning head tool holder solid.
4. Use the **Specify Movement** dialog to enable the U plate solid to move in +delta X.
5. Create a UCS on the U plate solid to locate the tool.
6. Using the **Tool Block** dialog, make the turning head tool holder solid a tool block, and create a tool location on the tool block using the UCS you created on the U plate solid.

5-axis machining

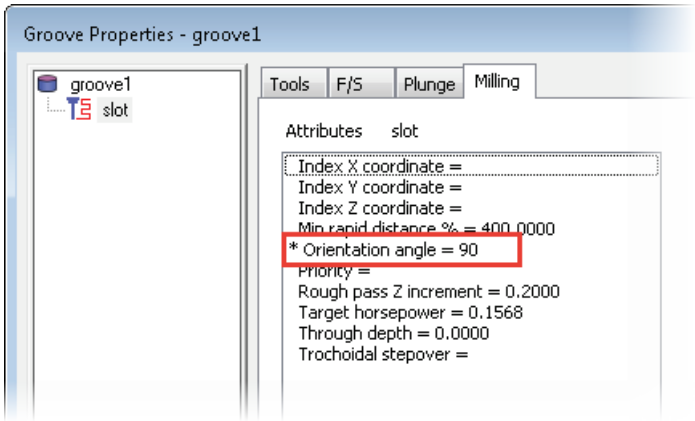
FeatureCAM 2015 R3 contains these changes and improvements to 5-axis machining:

- **C-axis indexing** (see page 71) — 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 73) — 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.



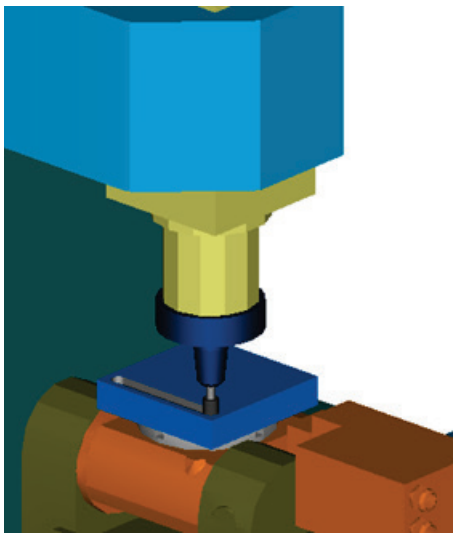
Use this option to specify the rotation of the X and Y axes about the Z-axis. This option only applies if the machine tool starts at the singularity (where the machine tool's Z-axis is aligned with the setup's Z-axis).

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

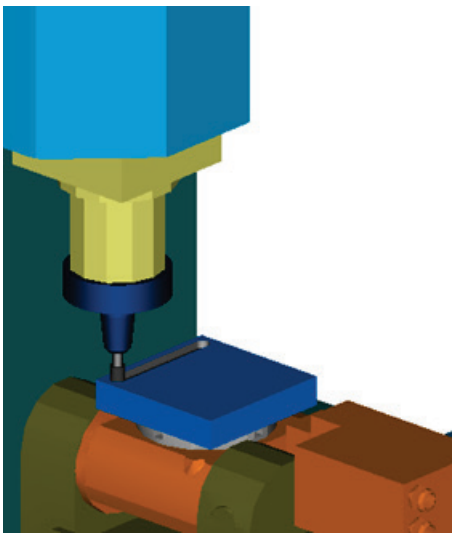
- Use the **Alternative 5-axis position** option to specify a C-axis orientation of either 0 or 180 degrees.
- Use the **Use Origin of this Setup as the Touch-off Point** option in the **5 Axis Fixture Location** dialog. This method applies the C-axis orientation to all setups in the part, instead of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

For example:

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

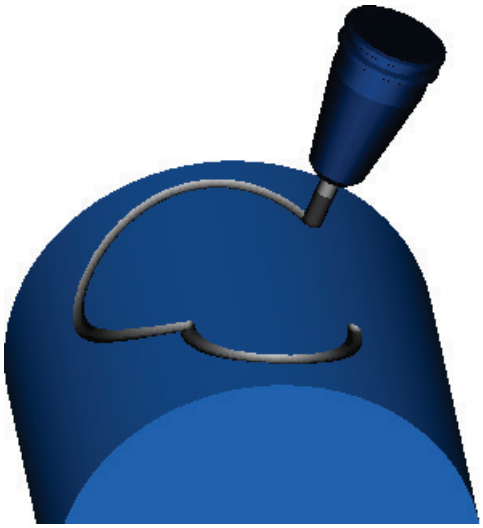


With an orientation angle of 90, the groove is cut in the machine's X direction.



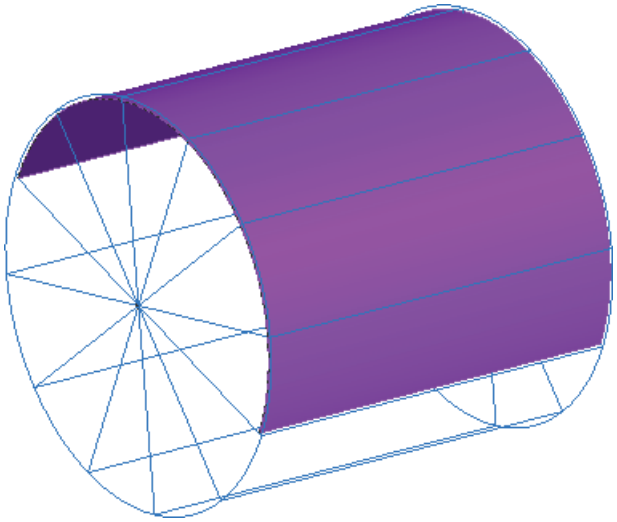
5-axis engraving

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.

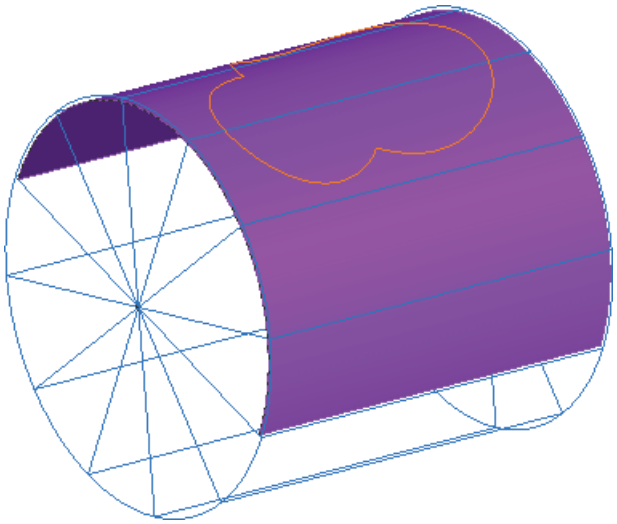


To create a 5-axis surface engraving feature:

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 76) — You can now set the limits of movement for solids in a Machine Design file.
- **Testing machine movements** (see page 77) — 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 79) — 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.

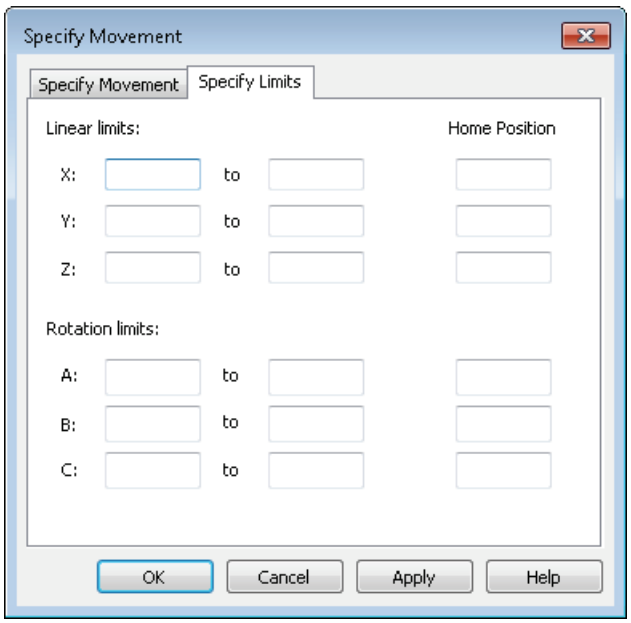
To specify the limits of movement for a solid:

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 60).

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.

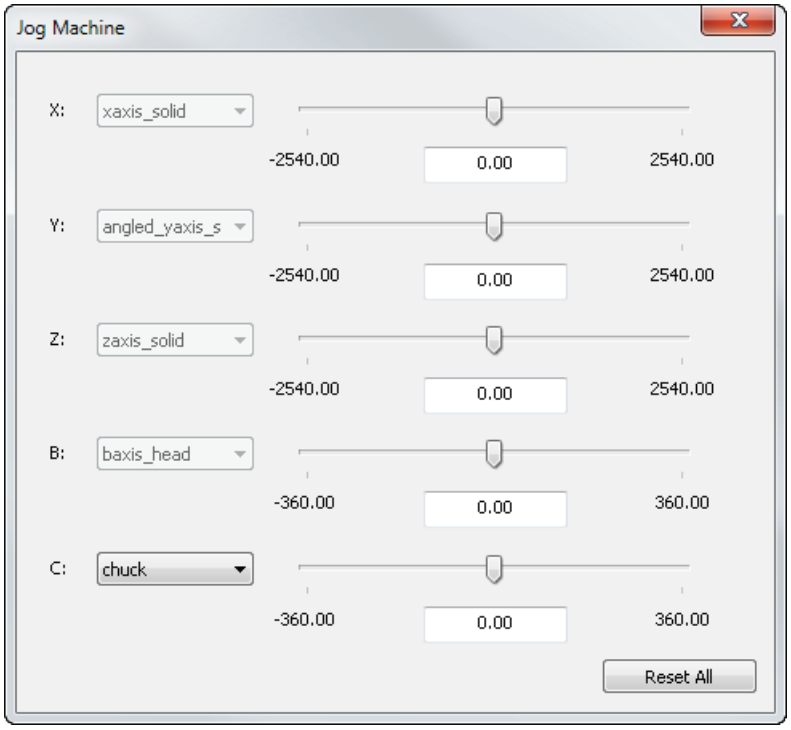
To use machine jogging to test machine movements:

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 76).*

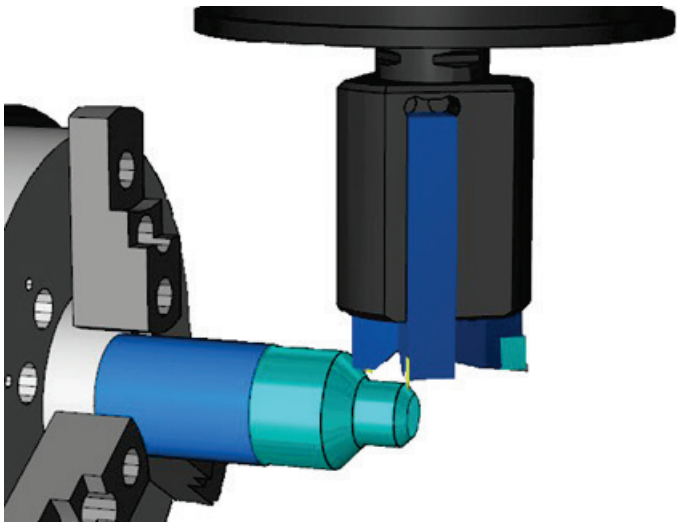
5 To hide solids in the jogging simulation, click **Select** in the **Standard** toolbar, then click solids in the graphics window.

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 84) — You can now change the size of geometry point objects to make them easier to see.
- **Customizing the snapping cursor** (see page 85) — You can now change the size and color of the snapping cursor.
- **Measuring curve length** (see page 86) — You can now find the length of curves and geometry segments.
- **Creating internal gears** (see page 87) — You can now perform analysis on internal gears curves.
- **Combining solids** (see page 88) — You can now combine multiple solids.
- **Hiding rapid moves in centerline simulations** (see page 89) — You can now hide rapid moves in centerline simulations.
- **Chip recognition size** (see page 90) — 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 90) — IFR now works more quickly for large parts.

Milling

- **Automatic tool selection** (see page 92) — 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 94) — Vortex toolpaths can approach flat areas from outside of stock instead of always ramping into it.
- **Output options for 2D NT toolpaths** (see page 96) — The **Output Options** dialog is now available for 2D toolpaths that use NT or Vortex stepover types.
- **Previewing the toolpath point distribution** (see page 97) — You can now preview the points of a surface milling toolpath to help with editing the point distribution.
- **Tool pecking depths** (see page 98) — You can now specify the pecking depths for individual tools.
- **Changing the posting tolerance** (see page 99) — You can now create more precise toolpaths, which is useful for machining small parts.
- **Helical side finish** (see page 100) — 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 102) — You can now remove undercuts in no-drag turning features to prevent gouges and simplify toolpaths.
- **Controlling steady rests** (see page 103) — You can now open and close the jaws of a steady rest without moving it to the home position.
- **Simulating bar stock** (see page 104) — You can now simulate bar stock in FeatureCAM by specifying the length of the stock displayed in simulation.
- **Custom turret names (MTT)** (see page 105) — You can now use customized turret names to make FeatureCAM more consistent with your machine.

XBUILD

- **Generating Post documentation** (see page 106) — You can now output your CNC data file as HTML or XML to make it understandable.
- **Using macros in the post processor** (see page 107) — The **Disable Macros** option in the **Post Options** dialog is now selected by default.

Add-ins and extensions

- **Using Setup Sheets** (see page 109) — There are new tags which you can use to create more detailed setup sheets.
- **FeatureCAM to CAMplete TruePath add-in** (see page 110) — 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 112) — 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 112) — FeatureCAM now supports Microsoft SQL server 2014.

Machine Simulation Design

- **Multi-tool blocks** (see page 114) — 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 116) — 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 118) — 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 84) — You can now change the size of geometry point objects to make them easier to see.
- **Customizing the snapping cursor** (see page 85) — You can now change the size and color of the snapping cursor.
- **Measuring curve length** (see page 86) — You can now find the length of curves and geometry segments.
- **Creating internal gears** (see page 87) — You can now perform analysis on internal gears curves.
- **Combining solids** (see page 88) — You can now combine multiple solids.
- **Hiding rapid moves in centerline simulations** (see page 89) — You can now hide rapid moves in centerline simulations.
- **Chip recognition size** (see page 90) — 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 90) — IFR now works more quickly for large parts.

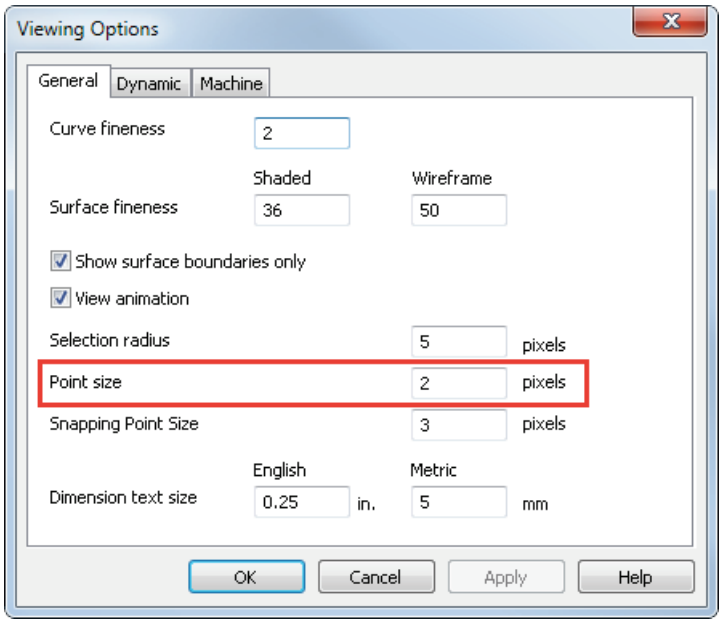
Changing the point size

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.

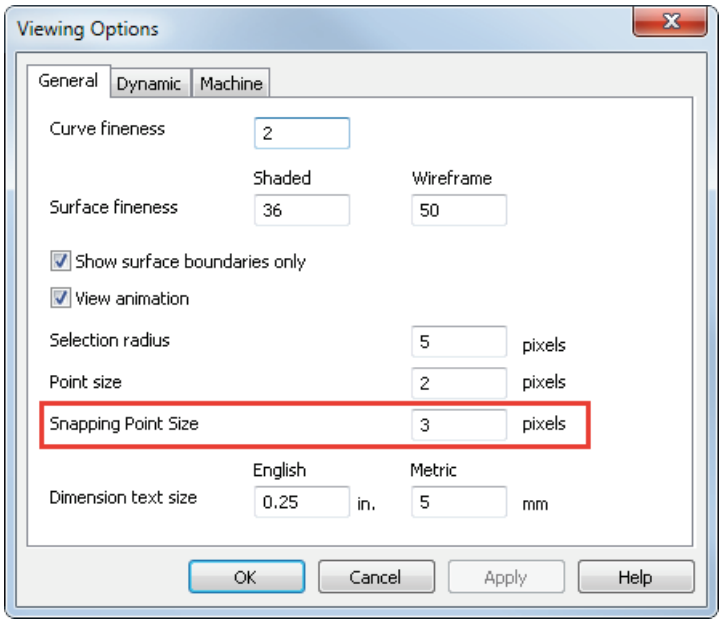
Customizing the snapping cursor

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.

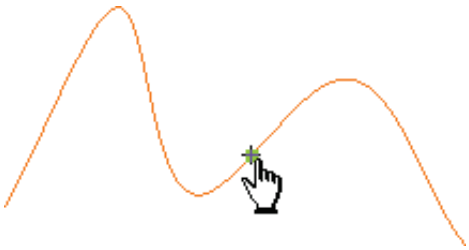
The selected color is displayed in the **Default Colors** dialog.

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

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

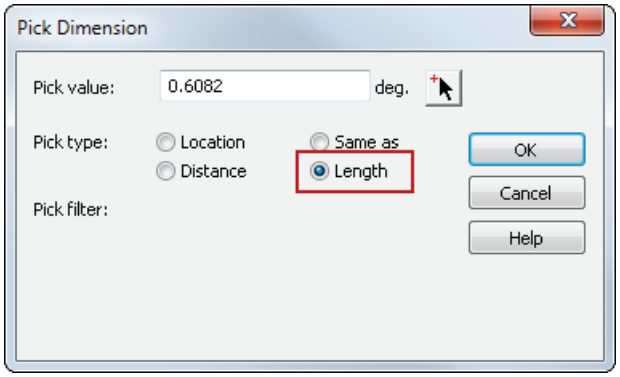
Measuring curve length

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.

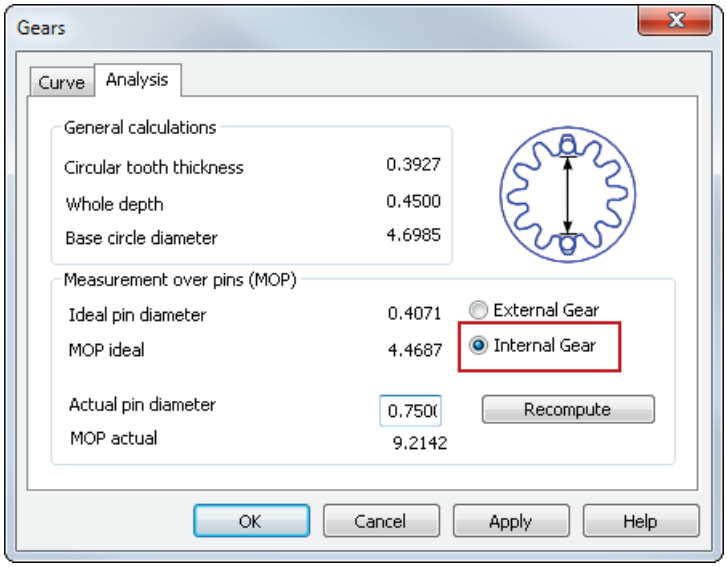


3. Click **Pick Location** and select the curve you want to measure in the graphics window.

   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, 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.

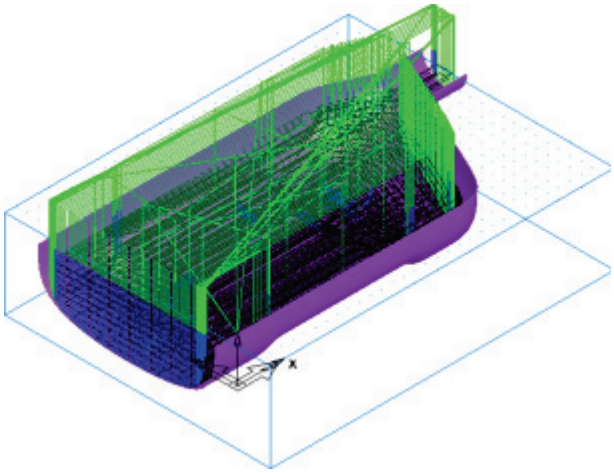
   *This dialog was previously called the **Boolean** dialog.*

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.

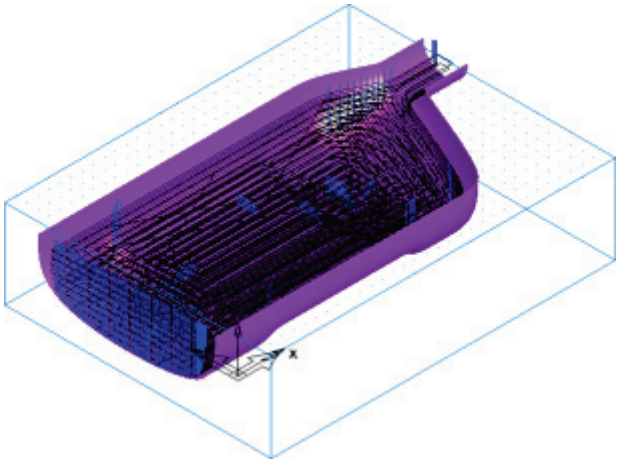
Hiding rapid moves in centerline simulations

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:

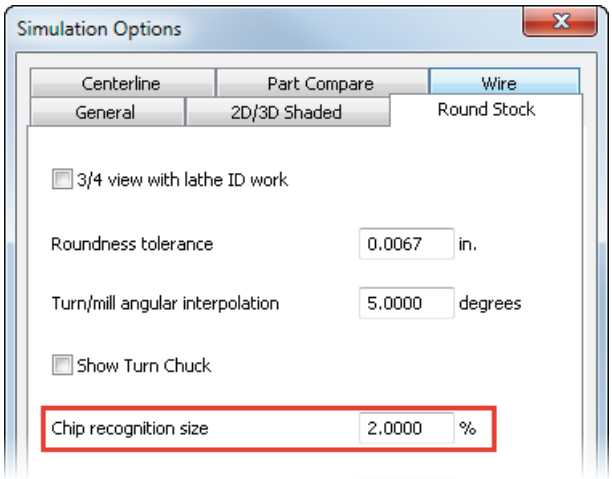


To hide rapid moves in centerline simulations, deselect the **View > Simulation > Show Centerline Rapids** menu option. The next time you run a centerline simulation, rapid moves are not displayed.

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 92) — 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 94) — Vortex toolpaths can approach flat areas from outside of stock instead of always ramping into it.
- **Output options for 2D NT toolpaths** (see page 96) — The **Output Options** dialog is now available for 2D toolpaths that use NT or Vortex stepover types.
- **Previewing the toolpath point distribution** (see page 97) — You can now preview the points of a surface milling toolpath to help with editing the point distribution.
- **Tool pecking depths** (see page 98) — You can now specify the pecking depths for individual tools.
- **Changing the posting tolerance** (see page 99) — You can now create more precise toolpaths, which is useful for machining small parts.
- **Helical side finish** (see page 100) — 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.

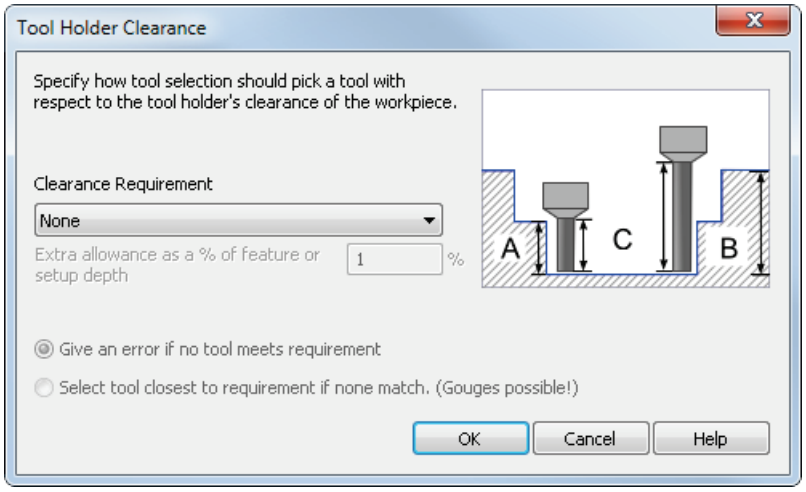
No additional clearance:

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.

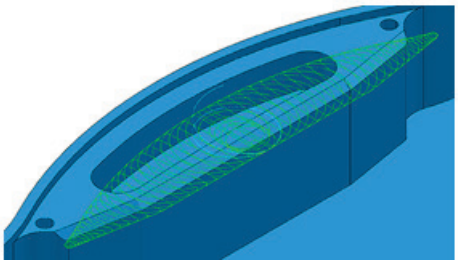


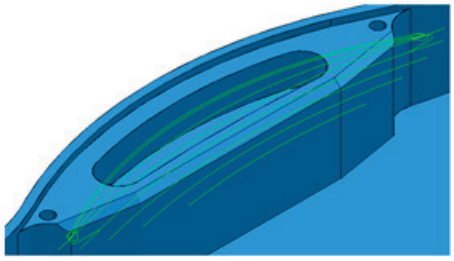
3. In the **Clearance Requirement** list, select the clearance you want between the tool holder and the part. Select from:
   - **None** — Select this option to leave no additional clearance. Old part files still select the same tools as before.
   - **Feature** — Select this option to ensure the tool 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 R3	FeatureCAM 2016 R1
The tool ramps into the part	The tool approaches from the outside

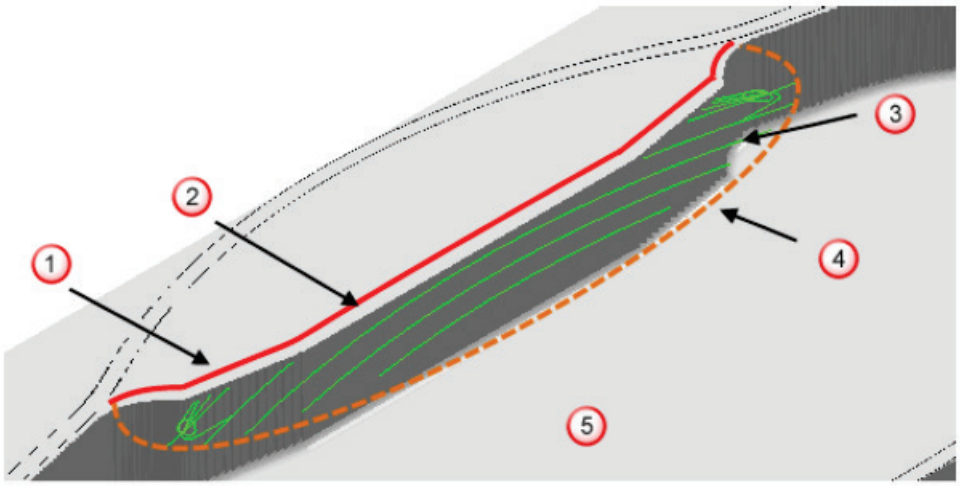




To approach from outside the stock, FeatureCAM:

- 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

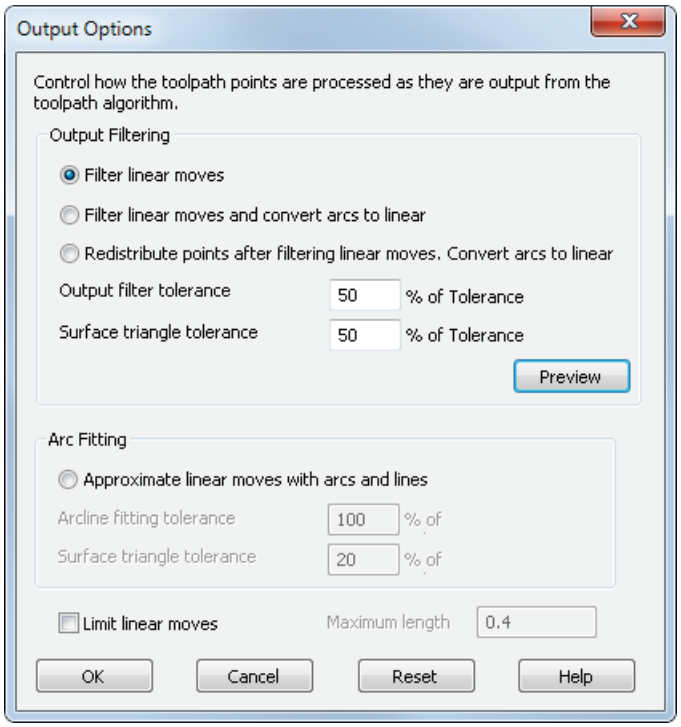
FeatureCAM only extends a section of the flat area if the extended section:

- 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

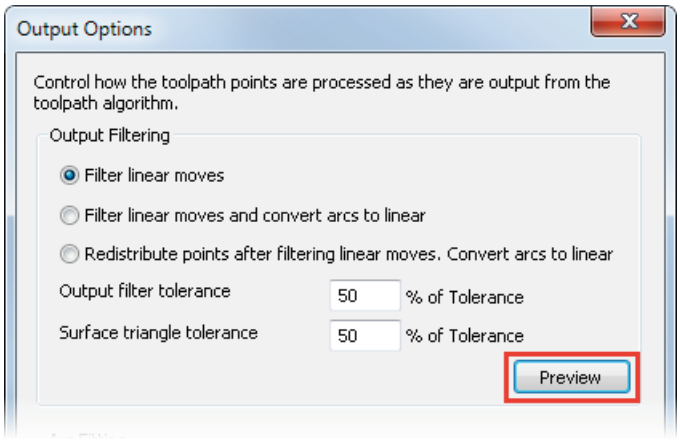
You can now preview the points of a surface milling toolpath to help with editing the point distribution.

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.

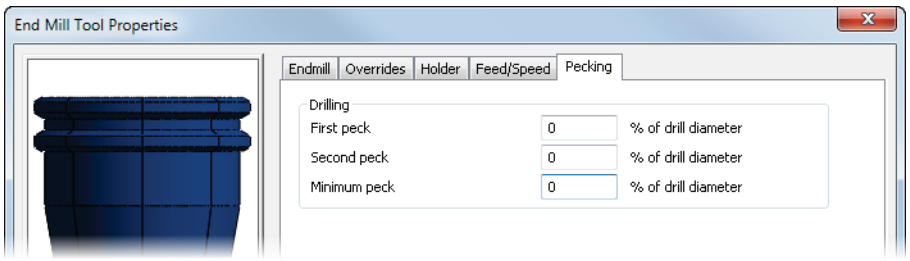
Tool pecking depths

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.

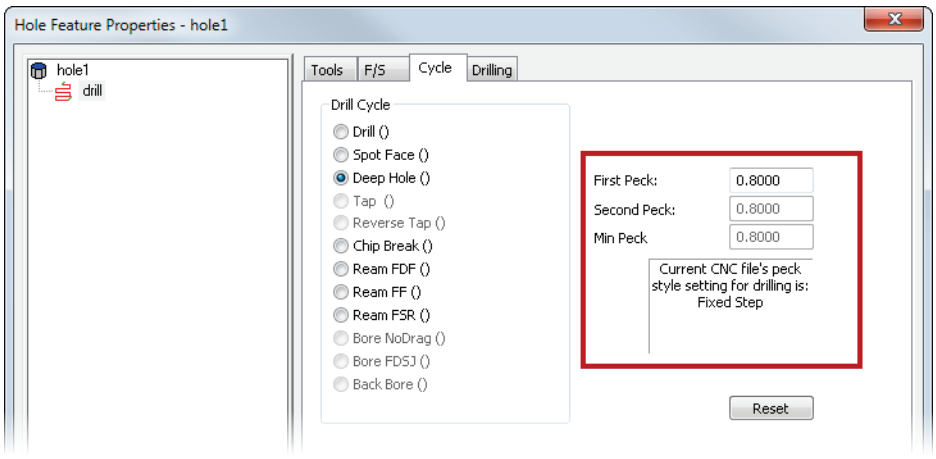
To specify the pecking depths at which the drill retracts:

- 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.

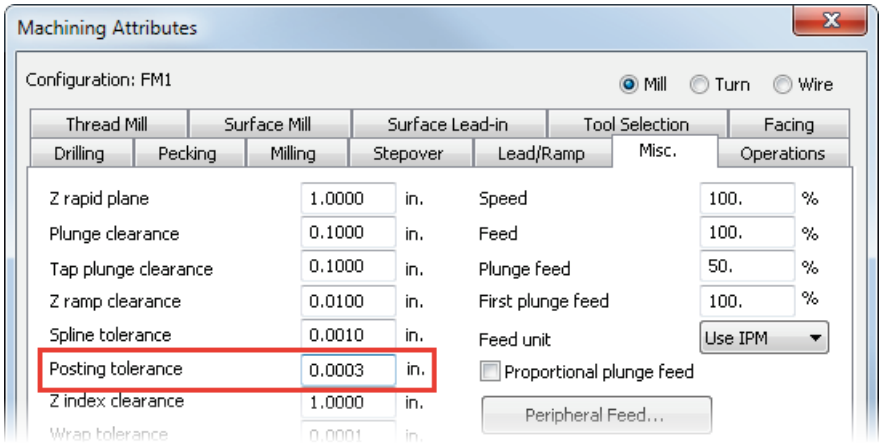


The peck style is specified in the .cnc file. This determines which of the tool's pecking values are used to calculate the absolute pecking depths for the operation.

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.

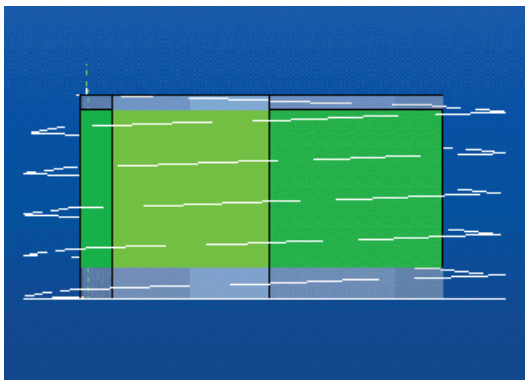


Reduce the **Posting tolerance** for small parts to create more precise toolpaths. You must also adjust your post processor to output more digits. For example, if you adjust the posting tolerance from 0.001 to 0.0001, then you must adjust the digit format in the post processor so that the extra decimal 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 102) — You can now remove undercuts in no-drag turning features to prevent gouges and simplify toolpaths.
- **Controlling steady rests** (see page 103) — You can now open and close the jaws of a steady rest without moving it to the home position.
- **Simulating bar stock** (see page 104) — You can now simulate bar stock in FeatureCAM by specifying the length of the stock displayed in simulation.
- **Custom turret names (MTT)** (see page 105) — You can now use customized turret names to make FeatureCAM more consistent with your machine.

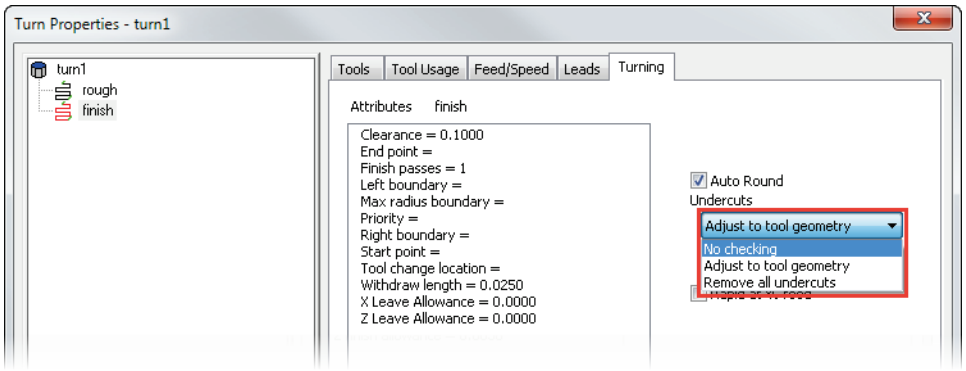
Removing undercuts in no-drag turning features

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.

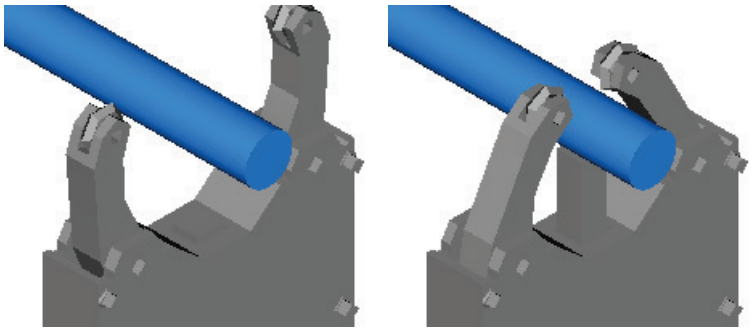


Controlling steady rests

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:



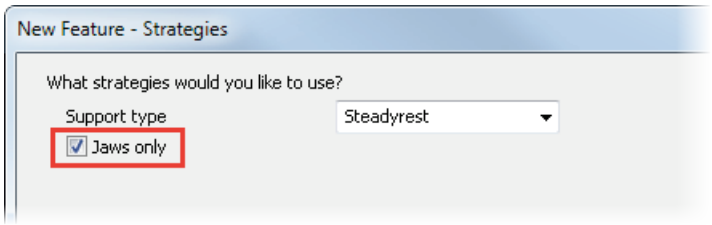
To control the jaws of a steady rest without moving it:

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**.

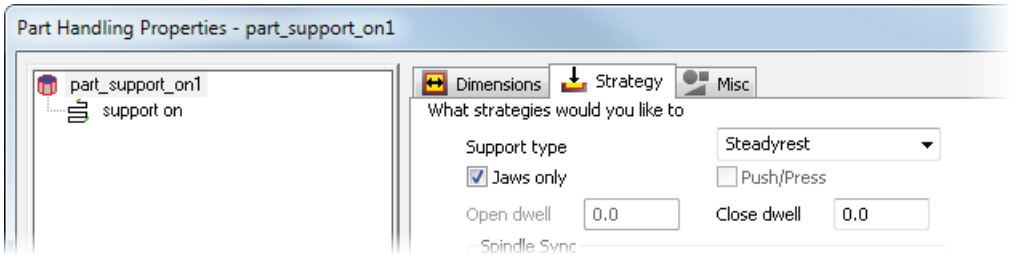
   To open a steady rest, select **Part Support Off**.
4. Click **Next**.

   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.

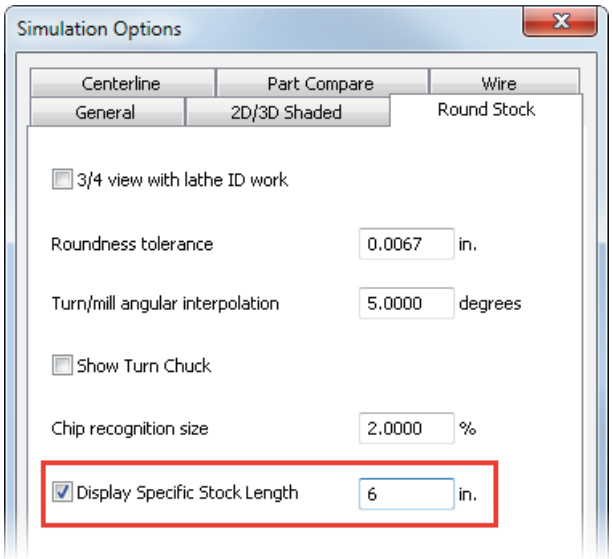


There is a new `<JAWS-ONLY>` reserved word in XBUILD. You must update your post Formats to use the reserved word to obtain the correct NC code that matches the new simulation.

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.

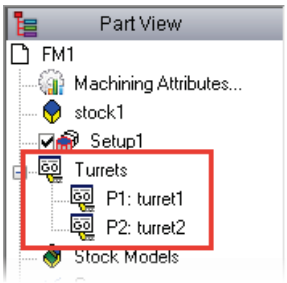


Custom turret names (MTT)

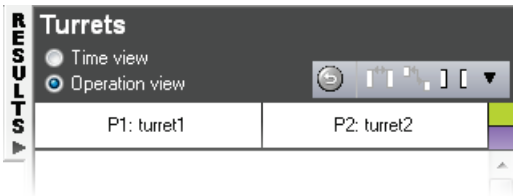
You can now use customized turret names to make FeatureCAM more consistent with your machine. Previously, all turrets were called upper and lower turrets.

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 106) — You can now output your CNC data file as HTML or XML to make it understandable.
- **Using macros in the post processor** (see page 107) — 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 109) — There are new tags which you can use to create more detailed setup sheets.
- **FeatureCAM to CAMplete TruePath add-in** (see page 110) — 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 112) — 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 112) — 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.

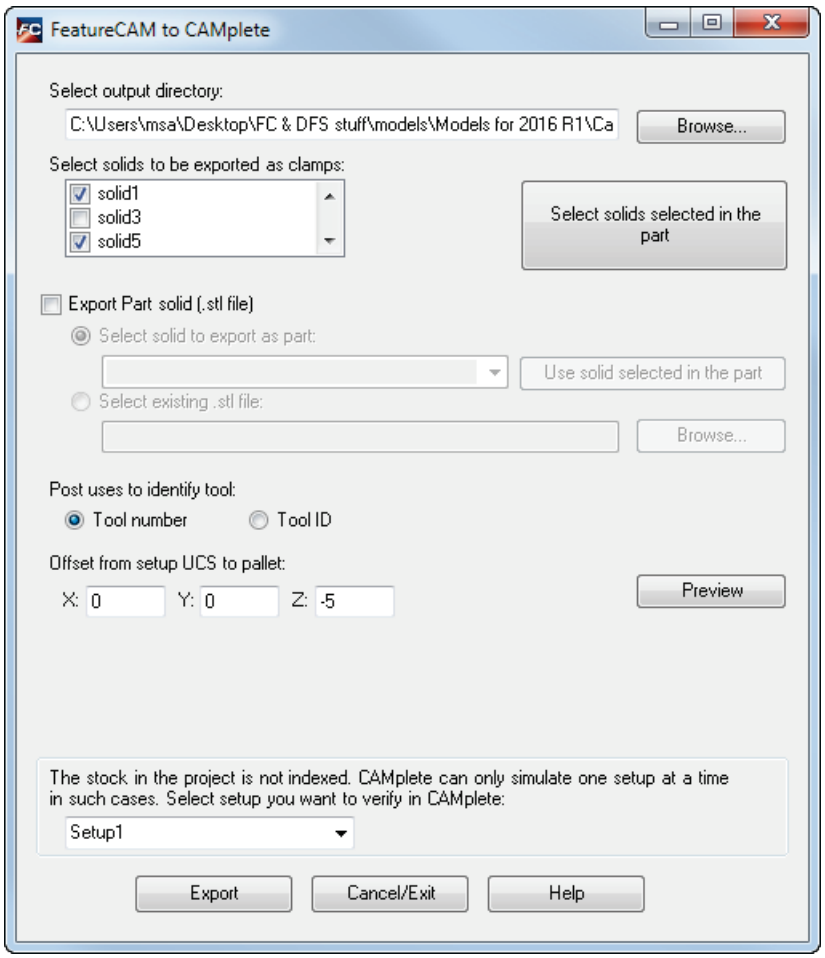
FeatureCAM to CAMplete TruePath add-in

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 **Select solids to be exported as clamps** list, select solids you want to export as clamps and fixtures.

   You can export multiple clamp and fixture solids.

To mark multiple solids as clamps, select them in the graphics window and click **Select solids selected in the part**.

7 Click **Export Part solid** to export a solid to use as the part in CAMplete. You can export only one part solid.

- 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.

If want to machine a part made of multiple solids, you can use FeatureCAM's solid modelling to combine them.

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.

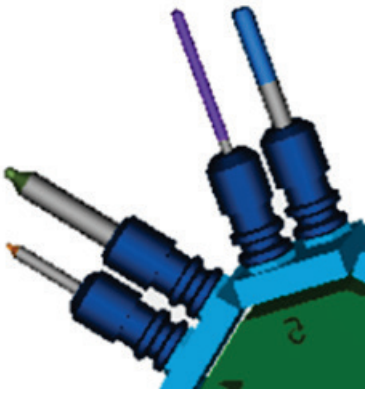
Machine Simulation Design

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

- **Multi-tool blocks** (see page 114) — 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 116) — 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 118) — 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

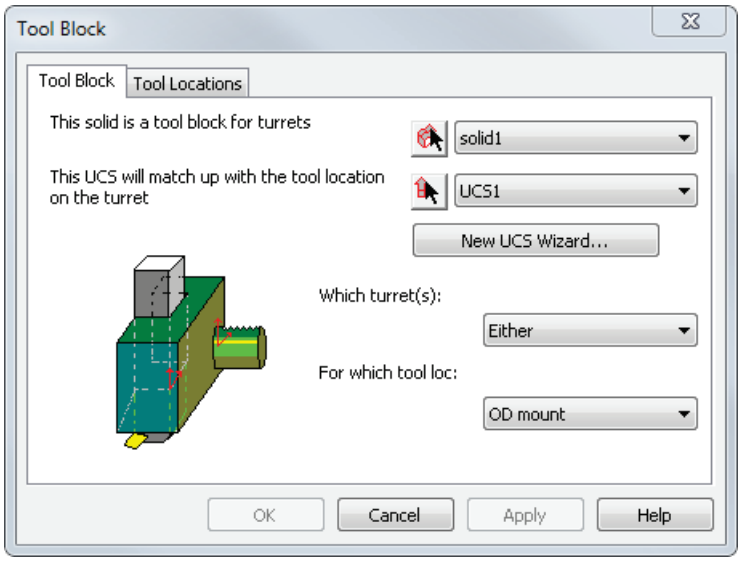
You can now create multi-tool blocks and double-sided tool blocks in Machine Design files and use them in machine simulation. Previously, each tool block could hold only one tool.



To create a multi-tool block:

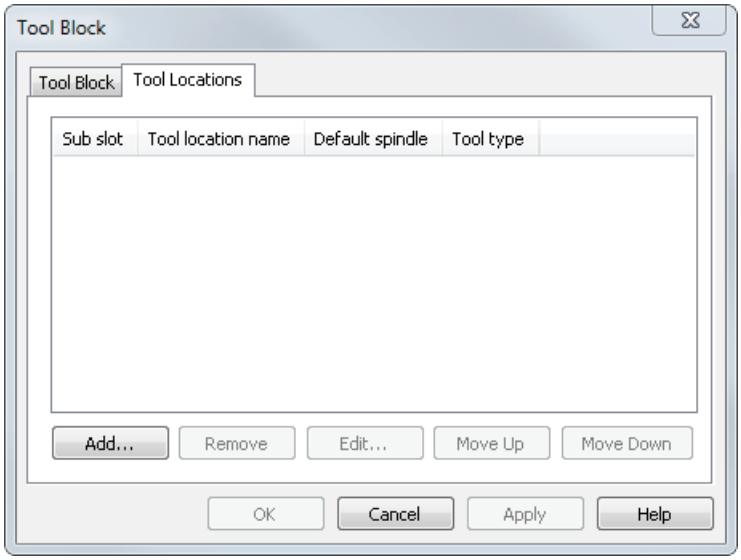
1. Ensure that the **Machine Design > Enable Millturn UI** menu option is selected to access the lathe design options.
2. Select the **Machine Design > Tool Block for Turret** menu option.

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



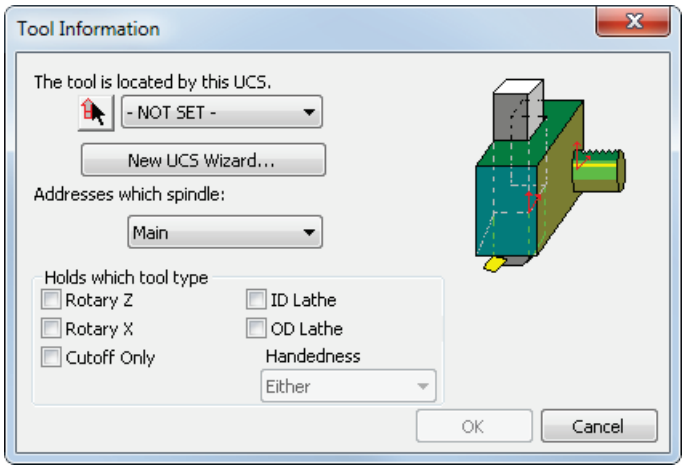
3. Select a solid to use as the tool block.
4. Select a UCS for the tool location on the turret, or click **New UCS Wizard** and create one.
5. Select which turrets the tool block can address.
6. Select which tool locations on the turret the tool block can attach to.

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.

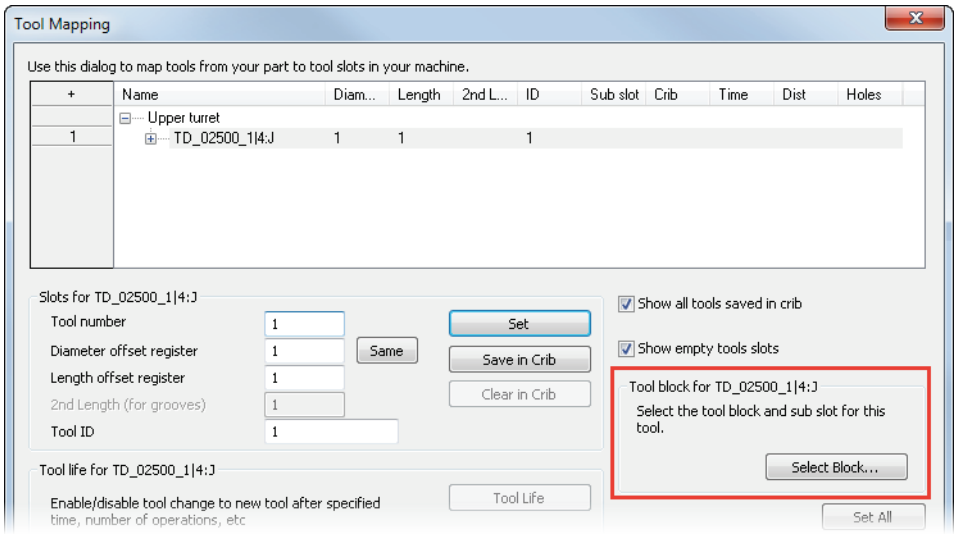
Selecting the tool block

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

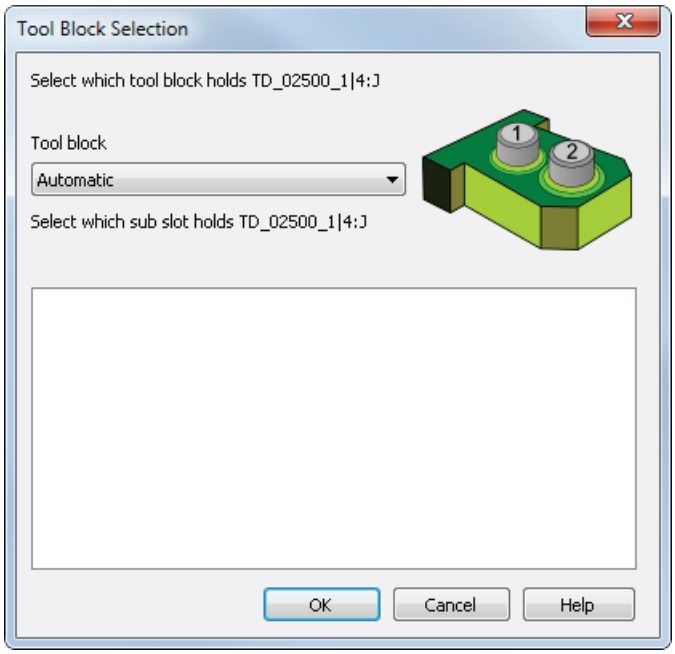
To select the tool block for a tool:

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.

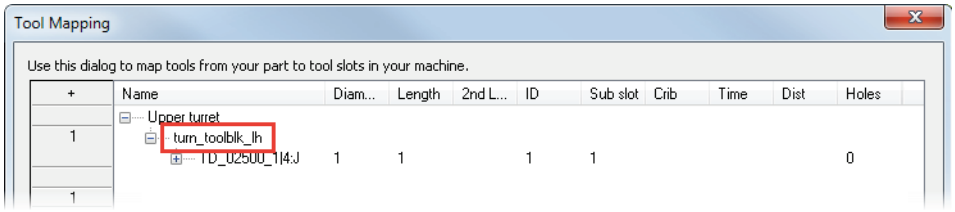


4 In the **Tool block** list, select the tool block solid from the Machine Design file that you want to hold the selected tool.

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.

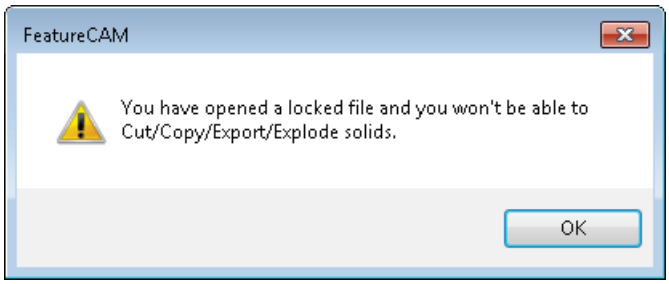


7 Click **OK** to close the **Tool Mapping** dialog and save your changes.

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.

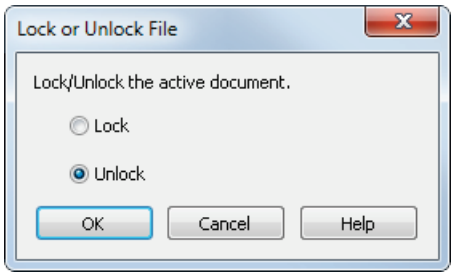


You can lock your files, but you cannot unlock them. You should not lock an original file, instead create a copy of it and lock the copy.

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.

Index

.

.cnx • 107
.html • 107
.md file protection • 118
.xml • 107

2

2D NT toolpaths • 96

5

5-axis • 20
5-axis engraving • 73
5-axis untransformed coordinates • 44

A

ACIS solids • 13
Analysis tab (Gears dialog) • 87
API • 33
AutoCAD • 13
Automatic tool orientation • 26
Automatic tool selection • 59, 92
Axis smoothing • 20

B

Bar stock • 104
Bar-fed mill • 38
Boolean dialog • 88

C

Calculate index radius from solid stock outline • 63
CAMplete • 32, 110
C-axis indexing • 71
Centerline simulation • 89
Changing the simulation tool color • 9
Chip recognition size • 90
Chuck • 36
Clamps • 32, 36
CNC documentation • 46
CNX • 107
Collision avoidance • 17
Combining solids • 88
Coolant • 23, 42
Cursor color • 85
Curve length • 86
Curves • 11
Customizing the snapping cursor • 85
Cutoffs • 90

D

Decimal points • 45
Deep Holes • 98
Deleting unused curves • 53
Disable Macros option • 107
Documentation for CNC files • 46
Drilling • 98
Duplicating features • 51, 52
DWG files • 13

E

Engage angle • 28
Engraving • 73

F

FeatureCAM to CAMplete • 32
FeatureCAM to CAMPlete • 110
Feed rate of non-cutting moves • 16
Fixed width text • 10
Flats • 94
FMA • 33
Formatiing reserved words • 45
Full machine architecture • 33

G

Gears • 87
Gouge-checking • 17
Grooves • 21, 29

H

Helical side finish • 100
Help • 6
Hide rapid moves in centerline simulations • 89
Highlighting objects from the Part View • 7
Holes • 98
HTML • 107
HTML documentaiton for CNC • 46

I

ID grooves • 21
IFR • 8
Imported solid names • 13
Importing SolidEdge files • 56
Importing SolidWorks files • 55
Increment tool color • 9
Index radius • 63
Indexing using a stock solid • 63
Interactive Feature Recognition • 8
Intermediate CNC files • 40
Internal gears • 87

J

Jogging • 77

L

Lead-in • 28
Lead-out angle • 28
Leave allowance • 29
Length of curves • 86
Lock Machine Design documents • 118

M

Machine Design document protection • 118
Machine jogging • 77
Machine movement limits
  Specifying limits • 76
  Testing limits • 77
  Using limits • 61
Machine simulation • 89, 90, 104, 110
Machining Attributes • 99
Macros • 107
Mazak integrex i-150 • 38
MD file protection • 118
Measuring curve length • 86
Microsoft SQL Server 2014 • 112
Mini-turrets • 79
Modifying solids • 88
Multi Turret Turning • 105
Multi-code coolant • 42
Multiple coolant types • 23
Multiple Setups • 8
Multi-tool blocks • 114

N

NC code • 10, 40
Negative leave allowance for grooves • 29
Nesting • 34
New Technology toolpaths • 96
No-drag turning • 102
NT toolpaths • 96

O

OD grooves • 21
Orientation angle • 71
Orientation of turning tools • 26
Output filtering • 97
Output options • 96, 97
Output streams • 40
Outside of Limits dialog • 61

P

Parasolid • 14
Part handling • 103
Part library • 51
Partial select • 50
Pasting features • 52
PDF • 6
Pecking • 98
Pick Dimension dialog • 86
Point distribution preview • 97
Point size • 84
Polar coordinates • 52
Post documentation • 107
Posting tolerance • 99
PowerSHAPE • 34
Preview toolpath points • 97
Probing • 112
Program point • 65
Protecting Machine Design files • 118

R

Rapid moves • 89
Reference help PDF • 6
Reserved words • 45
Retract on non-cutting moves • 16
Round stock • 90

S

Selecting objects • 50
Setup Sheet • 109
Setups • 8
Setup-sheet • 31
Show Centerline Rapids • 89
Side grooves • 21
Simulation • 89, 90, 104, 110
Simulation Options dialog • 90
Simulation tool color • 9
Small parts • 99
Snapping • 85
Solid modelling • 88
SolidEdge • 56
Solids • 13
SolidWorks • 55
Spatial R25 SP2 • 14
Specific stock length • 104
Spiral finishing • 100
SQL Server • 112
Steady rest • 103
Stock length • 104
Stock solid • 24
Stock solid indexing • 63
Support for Microsoft SQL server 2014 • 112
Swarf machining • 18

T

Tool block • 114, 116
Tool Block Selection dialog • 116
Tool color • 9
Tool holder • 60
Tool holder clearance • 92
Tool Holder Clearance dialog • 59
Tool Locations • 114
Tool Mapping dialog • 116
Tool orientation • 26
Tool pecking depths • 98
Tool selection • 59, 92
Toolpath points preview • 97
Tools with multiple inserts • 79
Turning chuck • 36
Turning tool orientation • 26
Turnmilling program point • 65
Turret names • 105

U

Undercuts • 102
Untransformed coordinates • 44
Unused curves • 53
Update options (probing) • 112
User-defined stock • 24

V

Variable width text • 10
Viewing Options dialog • 84
Vise • 36
Vortex • 94, 96
Vortex non-cutting moves • 16

W

Willemin-Macodel 480MT • 38

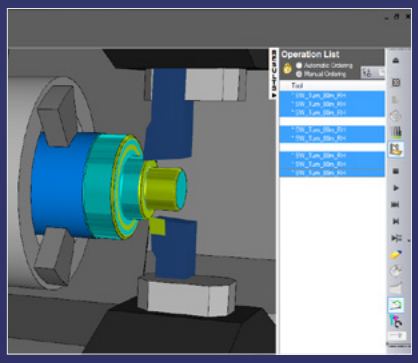
X

XML • 107

FeatureCAM 2016

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

www.featurecam.com



FeatureCAM 2016

www.featurecam.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 2015. All trademarks are the property of their respective owners.