FeatureCAM Product Descriptions

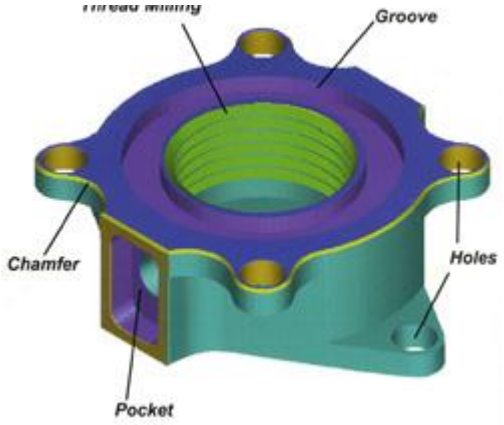
FeatureMILL 2.5D

FeatureMILL 2.5D is FeatureCAM's flagship product. It incorporates all the power and ease of use that are a part of every FeatureCAM product. Included are advanced 2.5D milling, production milling, 4th axis indexing, 4th axis wrapping, engraving, IGES/DXF/DWG file import, feedrate optimization and advanced 3D simulation verification.

CREATING FeatureMILL2.5D FEATURES

FeatureMILL2.5D includes the following features for part creation:-

- Holes - Counter-bore, counter-sink, tapped or other types of holes
- Pockets - With an unlimited number of islands at a variety of heights
- Grooves - Engraving, OD, ID and face grooves
- Bosses - Unlimited number of bosses at a variety of heights
- Thread Milling - OD or ID threads



You have the option to apply draft angles, chamfers and bottom radii to the walls of any milled features. These can be manufactured using flat end, ball end or tapered tools. Also you can quickly create radial, linear, rectangular and point list patterns of features by entering a few additional dimensions.

Manufacturing Details

Feed Rate Optimization

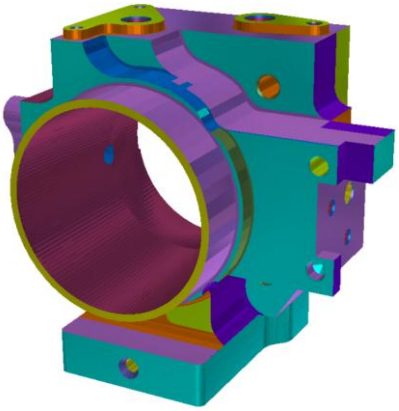
FeatureMILL2.5D's feed rate optimization minimizes and evens tool wear, and provides faster running programs.

Numerous milling and drilling options are included with FeatureMILL2.5D :-

- Ability to rough using cutter comp
- Multiple roughing and finishing diameters can be set for face features
- Bottom leave allowance and bottom finish allowance on semi-finish pass
- Multiple region support for solid stock to eliminate cutting air
- Chamfered, counter bore, counter-sink, tapped, counter-drilled, reamed holes
- Multiple roughing tools (larger to smaller) for fast material removal
- User definable milling tool holders
- Minimize retracts or tool changes
- Tool life management
- Precise tolerances
- Extensive drilling and milling macros
- Part line programming
- A variety of toolpath options
- Trochoidal toolpath for simple slots and grooves
- Climb / conventional milling
- Pre-drilling, helical or zig-zag approach
- Peck drilling
- Pilot drilling
- Arc / line approximation for toolpaths
- Multiple stepover types
- Cutter comp or part line paths
- Machining time estimates
- Inch and metric dimensions

FeatureMILL 3D Lite

FeatureMILL 3D Lite includes FeatureMILL 2.5D and provides additional functionality for simple surface modeling and surface milling.



FeatureMILL 3D Lite will be useful to primarily-2D shops that from time to time require basic 3D machining of single surfaces or STL files.

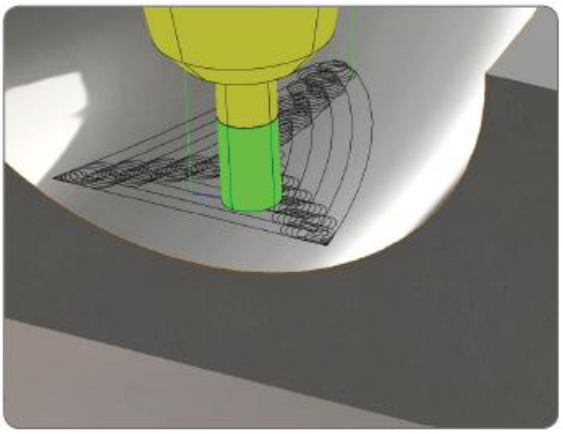
For surface machining the supported toolpaths are Finishing: Parallel, Isoline, 2D Spiral, roughing: Z level, Parallel

Other notes about this product: -

- 3D shops are unlikely to want this product -- they'll want the full 3D product.
- No solid modeling is available in this product. Only surface modeling.
- No flowline guide surface is available
- A single check surface can be used

FeatureMILL 3D MX

FeatureMILL 3D MX is the mid-level 3D machining module in FeatureCAM that includes FeatureMILL 2.5D, FeatureCAM 3D Lite and FeatureRECOGNITION. It provides additional functionality for programming multiple surfaces with a collection of important techniques such as 3-axis swarf milling, 3D chamfering, and Z-level roughing and finishing.



Supported toolpaths include:

- Parallel roughing & finishing
- Parallel roughing & finishing with perpendicular remachining pass
- Z-level roughing with both offset/spiral and zig-zag
- Z-level finishing
- Z-level finishing with bottom-up
- Z-level finishing with continuous 4 and 5-axes
- Isoline finishing with undercut handling
- Isoline finishing with continuous 4 and 5-axes
- 2D spiral finishing in and out
- Radial finishing
- Flowline finishing fixed in 3-axes
- Flowline finishing with continuous 4 and 5-axes
- Flowline finishing with undercut support
- Between 2 curves finishing
- Horizontal & Vertical fixed in 3-axes
- Horizontal & Vertical with continuous 4 and 5-axes
- 4-axis rotary milling (linear, circular, spiral)
- Swarf milling fixed in 3-axes
- 3D chamfer
- Part compare
- Stock model support
- Vortex (for 2.5D applications - Pockets and Boses)

FeatureMILL 3D HSM

Includes FeatureMILL2.5D, FeatureMILL 3D Lite, FeatureMILL 3D MX and FeatureRECOGNITION, plus a unique 3D surface wizard for 3D surface modeling, 3D feature wizard for easy creation of 3D toolpaths, automatic ball-end tool selection, and a variety of advanced 3D gouge-free toolpath techniques for supporting traditional and high speed machining and stock model support.

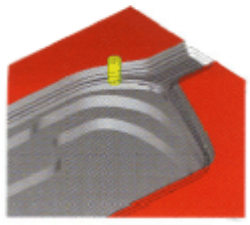
3D HSM (High Speed Machining) is for organisations that need advanced 3-axis toolpaths with renowned calculation performance and surface finish. This adds several leading technologies to 3D MX including High Speed Machining strategies, remachining, 3D spiral options, and additional state-of-the-art toolpath technologies proprietary to Delcam.

The New Vortex roughing toolpath will also be included in FeatureCAM 3D HSM at no extra cost to customers on maintenance

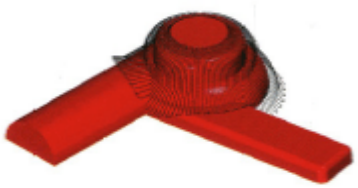
Allows native file import from SolidWorks, SolidEdge and Inventor. 3D models in IGES, DXF/DWG, Parasolid & ACIS are supported.

Manufacturing Details

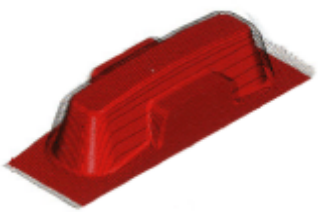
- Z Level Rough - slices the feature at various depths and mills the corresponding 2.5D contours as a pocket or boss shape at each depth. Toolpaths are parallel to the XY plane.
- Plunge - a specialized cutting tool is used to remove large amounts of material through a series of vertical plunging movements. The advantage of this technique is parts can be roughed quickly since the force of the operation is directly up the spindle.
- Parallel - creates a raster pattern within a boundary and projects it onto the model. The part is machined in a series of parallel passes at a specified angle and stepover. A second perpendicular pass can be applied to improve surface finish in steep areas. Toolpaths are parallel to the X or Y axes.
- Z Level - creates a toolpath by slicing the model at specific Z heights. Toolpaths are parallel to the XY plane. This technique works well for steep sloped walls.
- Isoline - uses the isoline curves of a surface to mill the surface. Toolpaths follow the rows or columns of individual surfaces. This gives a great level of control over the surface finish and tool loading.
- 3D Spiral - creates a toolpath by offsetting the model and block. Toolpaths move in a spiral toward or away from the center of the part. Stepover is constant in 3D. This technique is best suited to machining areas which require a constant stepover and works well on near vertical surfaces.
- Radial - creates a radial pattern within a boundary and projects it onto the model. Toolpaths move out radially from the center of the feature.
- Between 2 Curves - limits machining to between two curves. The toolpath direction can be set to along or across.
- Horizontal + Vertical - this strategy combines two different toolpath operations, one for finishing shallow portions of the part and a Z-level for finishing the steep regions.
- Steep and shallow machining
- Steep and shallow cusp control
- Corner Re-machining - a re-machining technique used to clean up corners that occur between non-tangential surfaces.
- Pencil milling - a single clean-up pass for corners.
- 4-axis rotary milling (linear, circular, spiral)
- Swarf milling fixed in 3-axes
- Swarf milling with continuous 4 and 5-axes
- 5-axis trim with continuous 4 and 5-axes
- 3D chamfer
- Flowline finishing – fixed in 3-axes, with continuous 4 and 5-axes, and with undercut support
- Part compare



Z-Level Semi-Finish



Radial Finishing

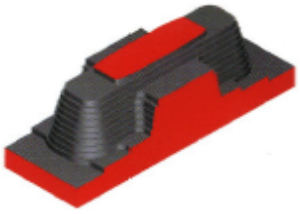


Parallel Finishing

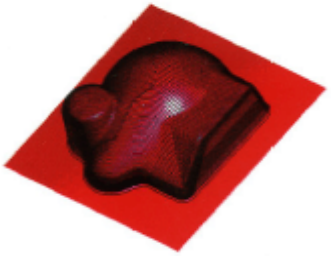
Cambridge Numerical Control

Units 8 & 9 Royce Court Burrel Road St. Ives Cambs PE27 3NE Tel : +44 (0)1480-468639 Email sales@cnc.uk.com

FeatureCAM Product Details



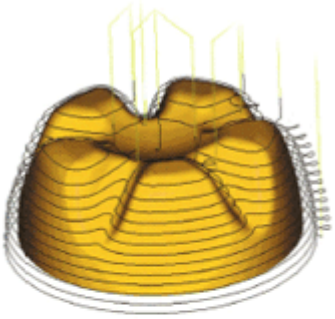
Z-Level Slice and Pocket Roughing



Spiral Finishing

3D Pockets and Bosses

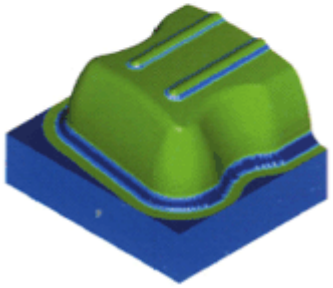
Some toolpaths used in FeatureMILL.



3D Toolpath Transitions
FeatureMILL3D includes a variety of options for controlling transitions, which provide excellent support for high speed or hard metal cutting.

Simulation
FeatureMILL3D provides additional simulation options to make verifying toolpaths quicker and easier.

RapidCut Simulation - With FeatureMILL3D's RapidCut simulation, you can view the results of 3D toolpaths in a matter of seconds, without waiting for the tool to animate. You can also use this feature to fast forward to specific toolpaths and then animate the tool to study certain toolpaths more closely.



Part Comparison - You can meet part tolerance requirements in less time with FeatureMILL3D's part comparison. FeatureMILL3D instantaneously compares the original part with the simulation. Areas with remaining material are shown in shades of blue and areas with gouging are shown in yellow, orange or red, according to the extent of the gouge. Green indicates areas that are cut within tolerance. You can change tolerance amounts to meet you specific requirements

Application

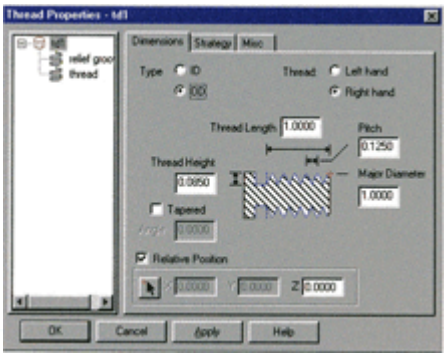
3D milling is used in a wide variety of applications. Some of the most common would be mold making, tool making and prototyping. However some aerospace and automotive components require 3D multi-surface machining.

Cambridge Numerical Control
Units 8 & 9 Royce Court Burrel Road St. Ives Cambs PE27 3NE Tel : +44 (0)1480-468639 Email sales@cnc.uk.com

FeatureTURN

The fastest turning program on the market. FeatureTURN includes undercut detection and prevention, re-machining to eliminate cutting air, boundary limits, shaped stock, and extensive canned cycle support.

Drawing



In addition to standard drawing tools, FeatureTURN includes a thread wizard that allows you to identify industry-standard thread types, such as 1-8-UNC, and then automatically create the thread to those specifications.

Creating FeatureTURN Features

FeatureTURN includes the following features for part creation.

Holes - You can create a variety of hole types aligned with the part's Z-axis. Any common hole type can be created including counter-sink, counter-bore, reamed or tapped.

Facing Feature - To face off the part, you simply enter the cut thickness and two diameter values.

OD Turning Feature - You can create outer turning features from any two-dimensional geometry or curves. Round tool roughing is supported and you have precise control of entry and exit.

ID Boring Feature - Bores are created from simple two-dimensional geometry or curves.

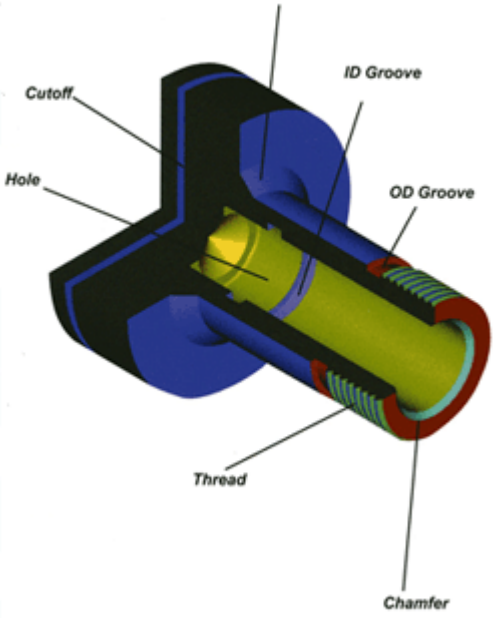
Groove Feature - OD, ID and face grooves are created from curves or by entering dimensions that describe the groove's shape.

Thread Feature - Both OD and ID threading features can easily be created from dimensional data. Tapered threads are supported and relief grooves can be conveniently generated within the thread feature. 2006 You can also thread along a form shape.

Cutoff Feature - Cutoff feature includes optional chamfer and part catcher control.

Bar feed feature - FeatureTURN provides support for both bar feeders and bar pullers..

Sub-spindle Feature - FeatureTURN supports machines with sub-spindles.



Canned Cycle Support

FeatureTURN gives you the ability to create part programs on your PC and edit them on your machine's controller. In addition, canned cycles will minimize the size of your NC programs

Canned cycle support includes:

rough profiling	threading	grooving
finish profiling	drilling	tapping

Postprocessors

FeatureTURN posts to virtually any CNC controller, including strong support for Fanuc controllers.

Manufacturing Details

Numerous manufacturing options are available, such as:

- Complete control of feed direction in rough semi-finish and finish passes
- Spindle direction is automatically determined by tool orientation
- Shoulder stroking groove finishing techniques eliminate up cutting
- Drilling operations include spot drilling, pilot drilling, drilling, reaming and tapping
- Complete support of threading - including control of depth of cut, feed direction, spindle rotation and tool orientation
- Support for cut-grip tooling
- Cutoff including chamfer and part catcher control
- Inch and metric dimensions
- Spring passes

Cambridge Numerical Control
Units 8 & 9 Royce Court Burrel Road St. Ives Cambs PE27 3NE Tel : +44 (0)1480-468639 Email sales@cnc.uk.com

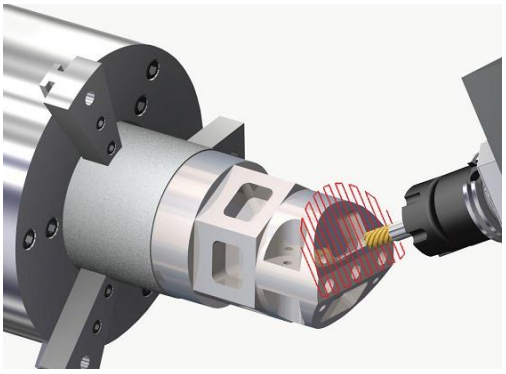
Application FeatureTURN is used on turning centers including lathes with sub-spindles. Applications are numerous including automotive, aerospace, medical, fasteners, pins, bushings etc.

FeatureTURN/MILL

Description

FeatureTURN/MILL allows turning and milling features to be conveniently combined in a single setup. It supports machines with C & Y axis milling, and machines with or without polar coordinate interpolation. Supports bar-pullers and sub-spindles.

FeatureTURN/MILL provides several capabilities, including:

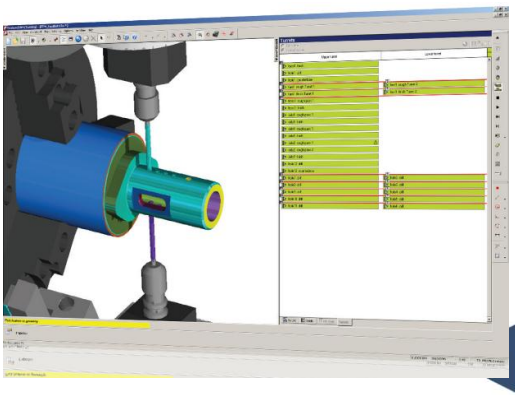


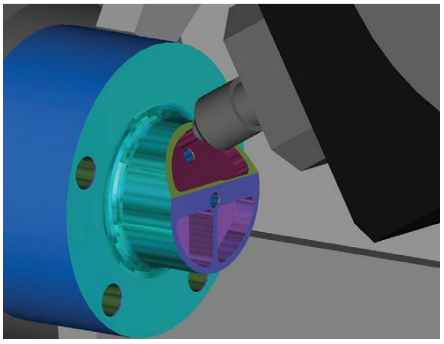
- Support for lathes with C and Y axis milling capabilities and sub-spindles.
- Drilling or milling features can be positioned on the face or on the OD of the part, and are easily programmed with the FeatureTURN/MILL feature wizard.
- FeatureTURN/MILL contains all the milling features of FeatureMILL2.5D, including holes, slots, pockets, grooves, sides and patterns.
- Milling features can be created on the OD and then wrapped around the lathe's Z-axis. This process takes the Y moves of a milling feature and translates them into C-axis rotations.
- Simulations accurately reflect the rotation of the part during machining so that you can verify precise toolpaths on your computer.
- Polar interpolation can be performed by FeatureTURN/MILL or by the control.
- FeatureTURN/MILL includes the same post processing capabilities as FeatureMILL2.5D and FeatureTURN. You simply create your features and let FeatureTURN/MILL take care of the NC code details.
- Turnmilling allows you to rough out turned parts. Users can cut with a live milling tool whilst rotating the workpiece in the turning spindle. Provides better handling of interrupted cuts and improves chip breaking for tough materials such as 302 stainless steel compared to conventional turning.
- One turn/mill Library post is supplied FOC with this module. Additional turn/mill posts are chargeable using the CUSTFEATPOST product code

Multi-Turret Turning

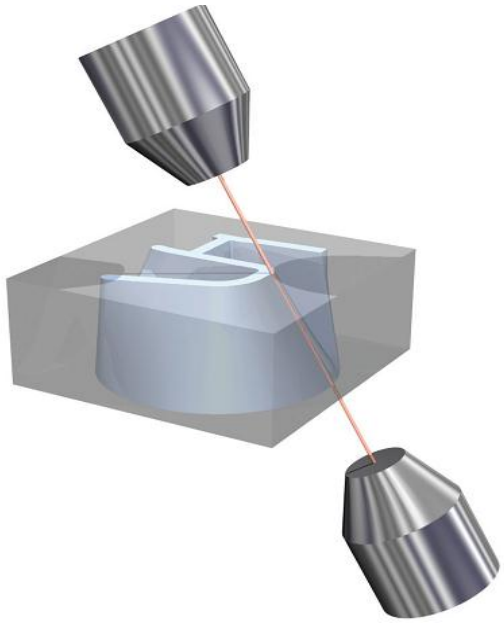
Optional add-on module to FeatureTURN/MILL that provides an excellent solution for production machining, making thousands of precision parts with reduced cycle time. Seamlessly integrated into FeatureCAM's existing turning programs. Supports B-axis milling, synchronizing of up to 4 turrets and 2 spindles. This module also incorporates 5-axis Positioning.

- Support for 5-axis positioning on B-axis lathes.
- Outputs NC code for all major controls, including Fanuc, Okuma, Heidenhain and more.
- Seamlessly integrated into FeatureTURN and FeatureTURN/MILL.
- Supports 4 turrets, main spindle, sub-spindle and numerous axes.
- Ability to move operations between spindles and turrets to achieve shortest cycle time.
- Provides a graphical, time-based view for moving around operations, eliminating the need to piece together several small NC programs





FeatureWIRE



Description
Programming for 2-axis, 2-axis taper, and full 4-axis wire EDM. It utilizes feature-based technology with features such as dies and punches. Functionality includes support for skim cuts, no-core cutting, and a cutting conditions database.

In addition to standard drawing tools, FeatureWIRE includes wizards that allow you to create industry-standard gears and cams by simply specifying the design parameters.

FeatureWIRE includes the following features for part creation.

- Dies - Creates dies for stamping or female metal parts, with no-core cutting supported
- Punches - Creates punches for stamping or male metal parts
- Sides - Creates parts with open curves
- User Definable Features

Taper

- Constant taper using conical corners or cylindrical (ISO) corners
- Variable - conveniently specify a separate taper angle for each segment of the feature

Full XYUV 4-axis contouring

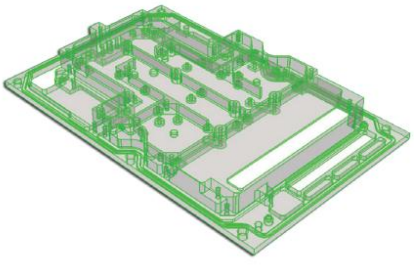
- Automatic synchronization of upper and lower curves
- Graphical picking of upper and lower curve Synchronization points for manual specification

Post Processing
FeatureWIRE contains post processors for industry-leading machines, including Agie; Charmille, Fanuc, Mitsubishi and Sodick. FeatureWIRE supports unification of wire EDM postprocessor files info the XBUILD post processor files.

Automation
FeatureWIRE automatically provides the operations for different cutting strategies, retrieves the cutting conditions database, generates toolpaths, creates NC code and provides simulation.
Simulation
With FeatureWIRE, slugs can be removed during 3D simulation.

Customizable Databases
FeatureWIRE's cutting conditions database contains wire EDM generator settings, flushing settings and cutter compensation settings. You can easily modify or extend the database to match the requirements of your wire EDM machine.

FeatureRECOGNITION (Standard with FeatureMILL 3D MX and FeatureMILL 3D HSM)



Optional module for FeatureMILL2.5D, FeatureMILL 3D Lite, FeatureTURN, and FeatureWIRE included with **FeatureMILL 3D MX and FeatureMILL 3D HSM**. FeatureRECOGNITION automatically and intelligently recognizes and programs features from solid or surface models. Allows native file import from SolidWorks, SolidEdge and Inventor. 3D models in IGES, DXF/DWG, Parasolid & ACIS are supported.

Also includes the New Vortex roughing technology for 2.5D applications (Pockets and Boses) if used with FeatureMILL 2.5D or FeatureMILL 3D MX

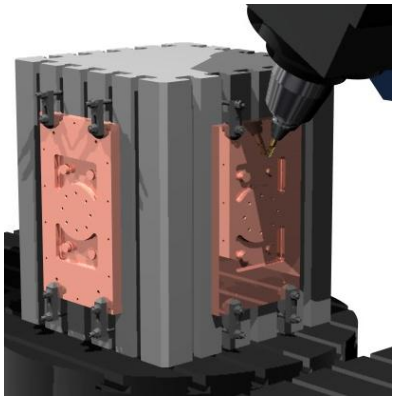
Solid Modelling

Optional module allows you to create parts in either solids or surfaces and define clamps and fixtures for simulation purposes. Solids simplify advanced modeling and provide exceptional filleting, blending, and sweeping. Advanced tools include core/cavity separation and the creation of parting surfaces.

Cambridge Numerical Control
Units 8 & 9 Royce Court Burrel Road St. Ives Cambs PE27 3NE Tel : +44 (0)1480-468639 Email sales@cnc.uk.com

FeatureCAM Product Details

Tombstone Machining



Optional module provides full tombstone support for both horizontal and vertical machines. Full 3D graphic simulation of the tombstone is included.

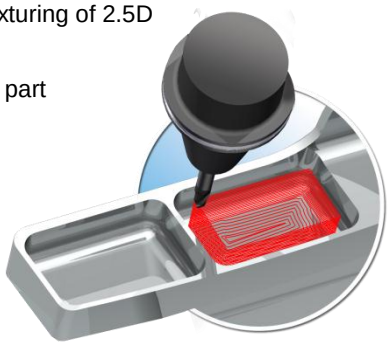
Tombstone Machining provides several capabilities, including:

- Supports both horizontal and vertical machines
- Manages the placement of the parts on the tombstone
- Helps you determine the milling coordinate systems
- Sequences the operations so that tool changes are minimized
- Generates NC code
- Simulates the code while displaying the tombstone, parts, tool, tool holder and spindle
- Detects any gouges during simulation

5-Axis Positioning

Description
Optional module provides accuracy and high precision for parts with multiple setups. It increases productivity and decreases errors by reducing the number of times the part must be re-clamped. This module is included with Multi Turret Turning.

If you have a 5-axis machining center, the new 5-axis Positioning Option makes minimizing fixturing of 2.5D parts easy.



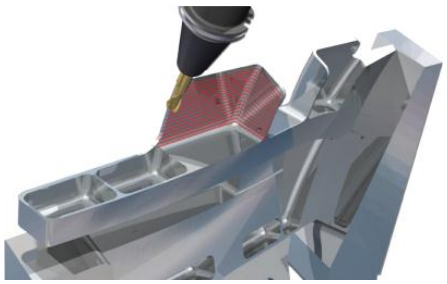
- Create the 2.5D part and then let FeatureCAM calculate the rotations and transform the part coordinates for you.
- Both horizontal and vertical machines are supported.
- Generate the code with respect to one coordinate system or use a coordinate system for each face.
- Supports 5-face machining.

5-Axis Simultaneous

Description

Optional module provides continuous 5 axis machining of parts, including swarfing, profiling and contouring. It increases productivity and decreases errors by reducing the number of times the part must be re-clamped.

5-Axis Simultaneous Machining



Complex 5-axis milling with FeatureCAM's ease-of-use Includes FeatureCAM's signature straightforward user interface, simplified and consistent with other FeatureCAM modules
Accelerates toolpath creation, while still providing control
Support for 5-axis milling machines as well as TURN/MILL machines
New toolpath techniques, including 5-axis trim and swarf milling

5-Axis Simultaneous provides several capabilities, including:

- Automatic tool axis control
- Available for parallel, z-level finish, isoline, 3D spiral, flowline, swarf and 5-axis trim
- Ability to specify simultaneous tool axis with vertical, lead/lean and others
- Additional level of tool axis control includes tilt axis for gouge avoidance, which permits deeper cuts with shorter tools
- 5-axis hole feature RECOGNITION identifies holes at any index angle
- 5-axis machine support in Xbuild

Cambridge Numerical Control
Units 8 & 9 Royce Court Burrel Road St. Ives Cambs PE27 3NE Tel : +44 (0)1480-468639 Email sales@cnc.uk.com