



Delcam  
FEATURE **CAM**



# welcome to delcam

Delcam is one of the world's leading suppliers of advanced CAD/CAM software for product development across a wide range of manufacturing industries. It has grown steadily since being founded in 1977 and now has 20,000 customers worldwide.

Delcam's FeatureCAM development group is based in Salt Lake City, Utah, where it has been developing the software for more than 20 years.

The FeatureCAM development, sales and marketing team joined Delcam in 2005. Since then, the team has grown rapidly. It now has sales and support organizations in 60 countries, helping customers across a broad cross section of industries to maximize their productivity with the software.

*"FeatureCAM did everything the competitors' products could, but was much easier to learn.*

*Even when I haven't used the software for awhile, I can sit down and start designing a custom fixture without a problem. Also, with FeatureCAM, we are able to lower our bids based on the ease and speed of programming.'*

*Ray Rubin,  
Future Fabricators*



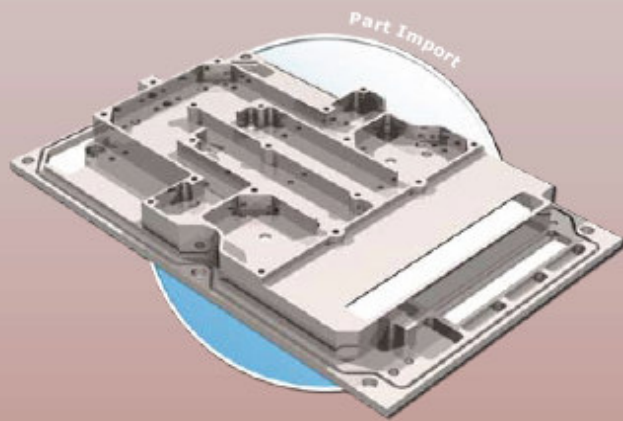
# what can FeatureCAM do for you?

FeatureCAM is the only CAM system designed from the start to meet two very important needs. Firstly, to provide a truly easy to use CAM system that is so intuitive that novice or occasional users can quickly create reliable and consistent toolpaths, reducing training costs and increasing your return on investment. Secondly, to ensure that you can program all your machine tools within a single user interface designed to achieve shorter programming times.

FeatureCAM offers solutions for 2D and 3D milling, 5-axis simultaneous, high-speed milling, turning, live tooling, and wire EDM. Each product incorporates advanced feature-based and knowledge-based technologies. Unlike operations-based CAM systems, FeatureCAM generates toolpaths based on the features of the part and then automatically selects appropriate tools, determines roughing and finishing passes, and calculates feeds and speeds. The selections made can be based on the built-in machining knowledge that Delcam supplies 'out of the box' within FeatureCAM, or from experience captured from your company, project or individual user's preferences.



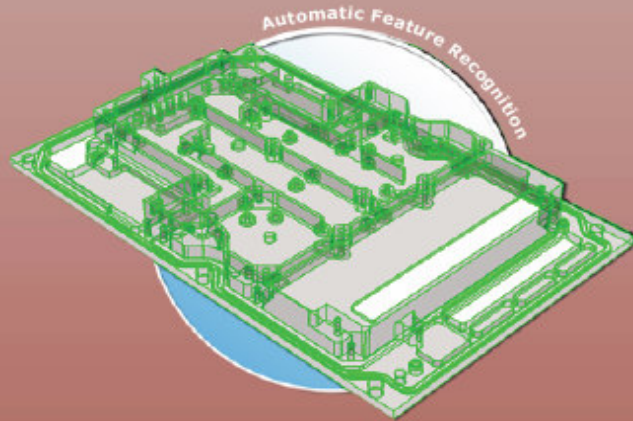
# automation & AFR



Feature recognition is a powerful way to simplify and accelerate programming. CAD models have all the information necessary to describe a part's shape, so there is no reason to spend time recreating that part data in a CAM system.

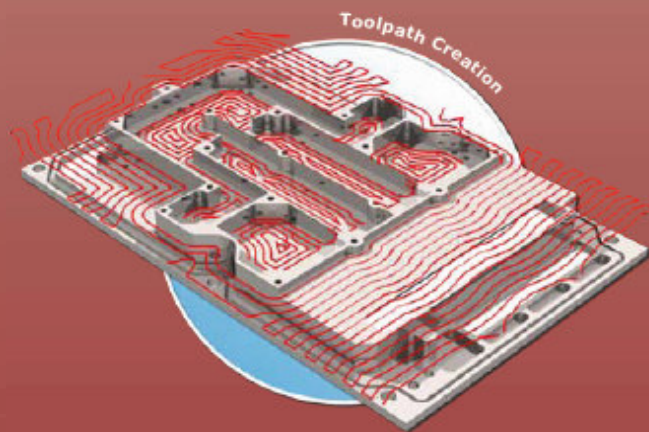
*'With about twenty years in the business, I've never programmed and cut parts this fast before. AFR is obviously where the industry will head and FeatureCAM is ahead of the pack.'*

*Mike Maendl,  
Prototfab*



Delcam's FeatureCAM analyzes imported CAD models and quickly recognizes the part features. It provides manual, interactive and fully automatic feature recognition (AFR). With AFR, all 2D features are recognized, including features that overlap or intersect.

Features are also fully associative with the imported model, so that design revisions do not cause problems. If a part is re-imported, FeatureCAM compares the initial model with the revised one and displays a list of the features, indicating which ones have been added, modified, removed or unchanged.



After the features are recognized, FeatureCAM applies its knowledge-based machining technology to select tools, calculate feeds/speeds and intelligently generate appropriate toolpaths.



# ease of use

Every CAD/CAM system claims to be easy to use, but FeatureCAM incorporates advanced technologies that truly make it unique. By programming parts based on features instead of operations, FeatureCAM streamlines machining and makes it easier to create and edit parts.

The knowledge-based technology in FeatureCAM gives you the power of automation, while still giving you control. All automation is produced using your preferences and specifications.

## Operations-Based CAM

Import or draw part  
Select a Rough operation  
Select rough boundary  
Select type of toolpath  
Select roughing tool  
Select feeds and speeds  
Select stepover and Z steps  
Select a Finish operation  
Select finish boundary  
Select type of toolpath  
Select a finishing tool  
Select stepover and Z steps  
Select feeds and speeds  
Click Simulation  
Create NC code

## Feature-Based CAM

Import or draw part  
Identify features  
Simulate operations

### FINISHED!

FeatureCAM automatically

- Determines all rough and finish operations for the entire part
- Selects the appropriate number of tools and tool sizes
- Calculates feeds and speeds
- Selects stepover and Z increments
- Generates leads and links
- Generates toolpaths
- Creates NC code

*'I really believe that no experience is necessary. The software is so intuitive that anyone with an engineering background can sit down with FeatureCAM for the first time and be creating code straight away. It is very fast and very easy to design and program parts in FeatureCAM. Creating features is just point and click, then the NC programming is done automatically. Within a few hours after opening FeatureCAM, I was able to create our first after-market bicycle suspension fork brace.'*

**Ted Tanouye,  
Chumba Wumba**



# 2D milling

FeatureMILL2.5D is the basis of the complete FeatureCAM range.

The software incorporates all the power and ease-of-use that are central to every FeatureCAM product.

Some of the functions included are:

Ability to import industry standard file types like IGES, DWG and DXF.

Step-by-step wizards to help even inexperienced or infrequent users to create reliable toolpaths quickly and easily.

Proven post-processors for all major machine tool types.

Parametric modeling and multiple fixture wizard.

Tombstone machining, with support for both horizontal and vertical machines, for easy multi-part machining.

Tool crib with thousands of tools, and the ability to create your own tool cribs and custom form tools.

Feed and speed tables with the ability to easily modify or extend.

Feed rate optimization that adjusts feed rates based on tool load.

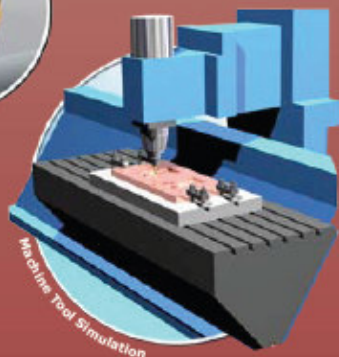
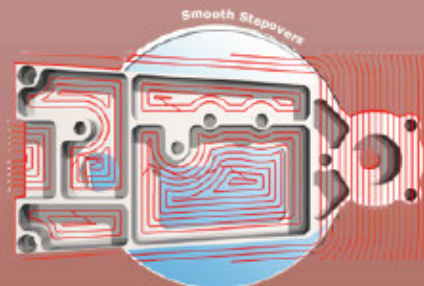
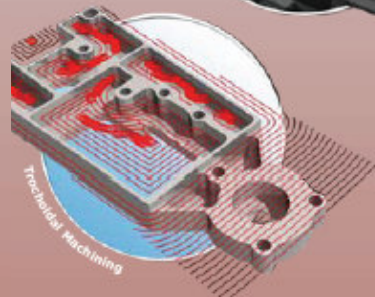
Customizing options such as user-definable features and an Application Programming Interface (API).

## Simulation Options

To ensure that toolpaths are optimized and no problems occur during the manufacturing process, it is important to be able to simulate the toolpaths you have produced. FeatureCAM provides centerline, 2D, 3D, RapidCut, part comparison and full machine simulation options. Machine models, with all the jigs, fixtures and machine specific features in place, can be built with FeatureCAM's Solid Modeling module or imported directly from the manufacturer, when available. Toolpaths also can be loaded onto different machine models to ensure you use the most appropriate machine to produce the part.

*'FeatureCAM is by far the easiest to use. I have taken people who have never programmed before and at the end of the week they were programming successfully.'*

*Eric Rosander,  
Innovative Precision*



# 3D milling & 5 axis

*'Anything I can imagine, I can program with FeatureCAM, and it takes me less time and effort than any other CAD/CAM package I've seen.'*

*James Wilkinson Jr,  
TeamRaptor Battlebot/CLK*

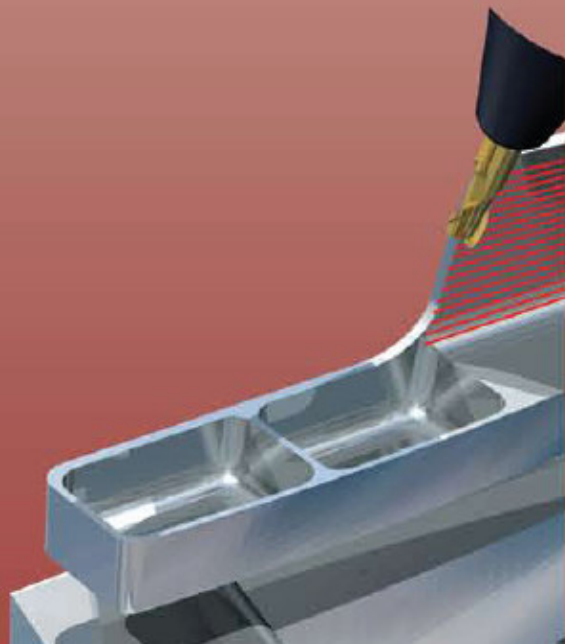
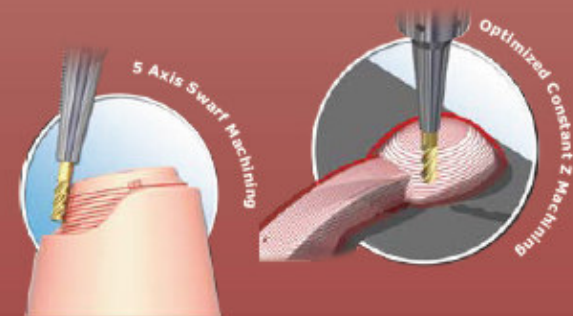
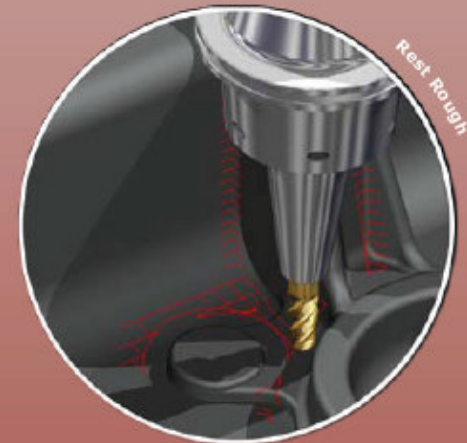
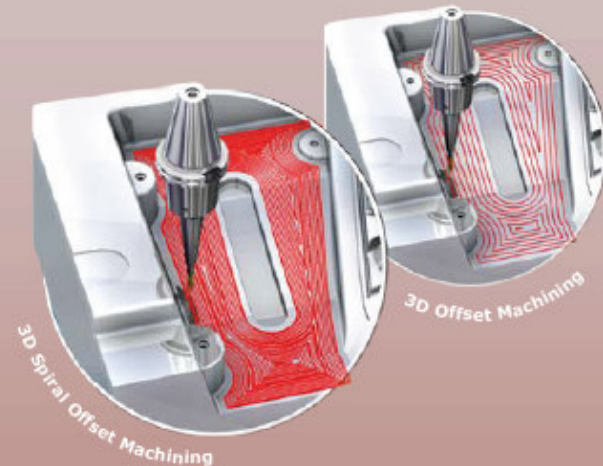
## 3D Milling

FeatureMILL3D offers a wide range of strategies for both conventional and high-speed 3D milling. They are all designed to give maximum metal removal rates, while keeping the load on the cutter as consistent as possible and minimizing any sudden changes in the cutting direction. Novel rest roughing and rest finishing options, including automatic multiple toolpath strategies, ensure the highest possible quality of surface finish is achieved, minimizing hand finishing both for parts with and without undercuts.

## 5 Axis Machining

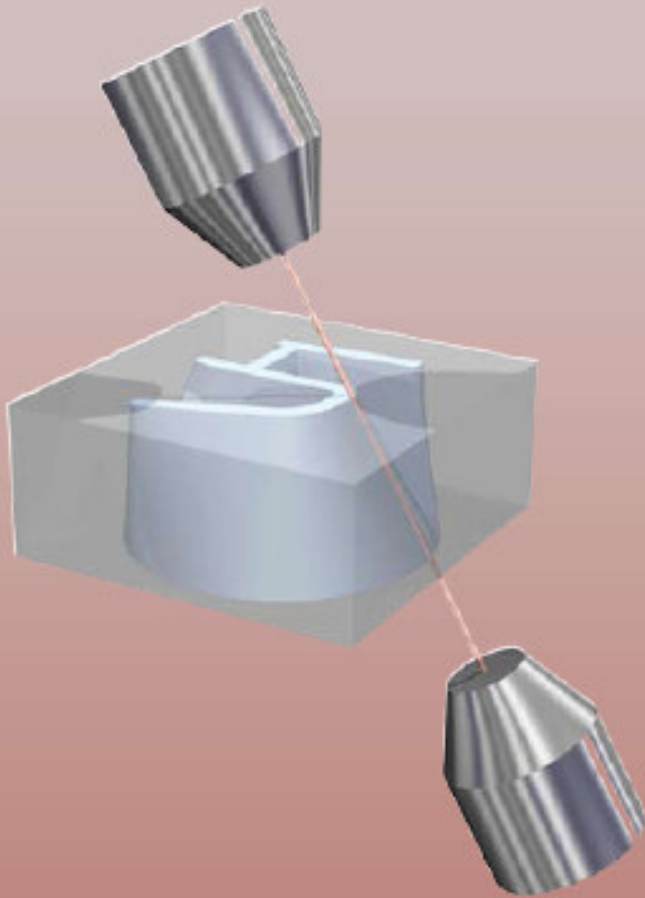
The five-axis simultaneous machining option in FeatureCAM offers a variety of methods for controlling the tool axis, either to access areas unable to be reached with three-axis machining or to give better cutting conditions. In addition, many three-axis toolpaths can be converted to a five-axis equivalent by using automatic collision avoidance to change the tool axis when collisions might occur. The module also supports five-axis drilling, five-axis trimming and swarf machining.

FeatureCAM's five-axis positional machining module allows companies with suitable equipment to produce complex parts with fewer set-ups, saving time and reducing possible errors. Accuracy is increased because all five sides of a part can be machined in a single set-up.



# wire

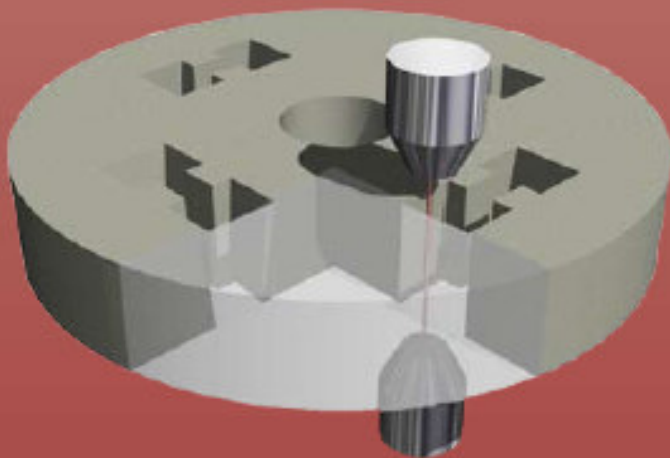
# EDM



## Wire EDM

FeatureWIRE offers the same automated approach utilized in other FeatureCAM products, for the full range of 2-axis and 4-axis wire EDM equipment.

The software automatically provides the operations for different cutting strategies, retrieves the required parameters from the cutting conditions database, generates the toolpaths, creates the NC code and provides a simulation of the complete process.



FeatureWIRE is supplied with a cutting conditions database that contains wire EDM generator settings, flushing settings and cutter compensation settings. The software also comes with post-processors for all industry-leading machines, including Agie, Charmilles, Fanuc, Mitsubishi and Sodick.

# turn & turn/mill

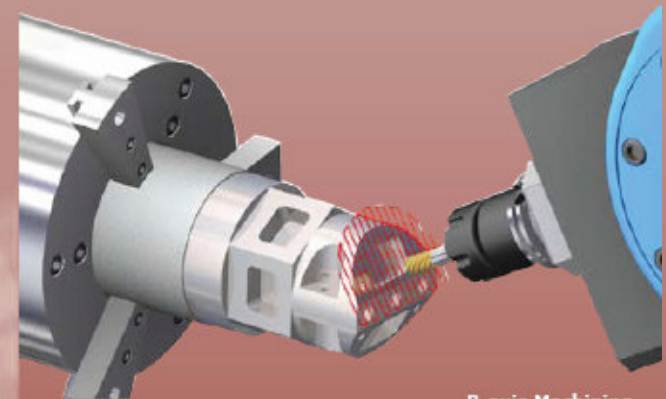
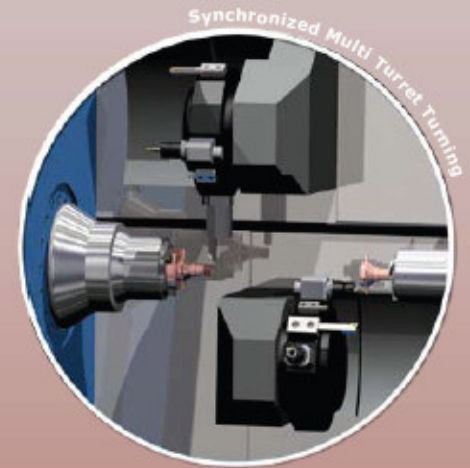
*'We love the automation in FeatureCAM; it increases our productivity and shortens programming time. We can import drawings from our AutoCAD software without having to worry about data transfer problems.'*

*Dale DeMay,  
The Stimpson Company*

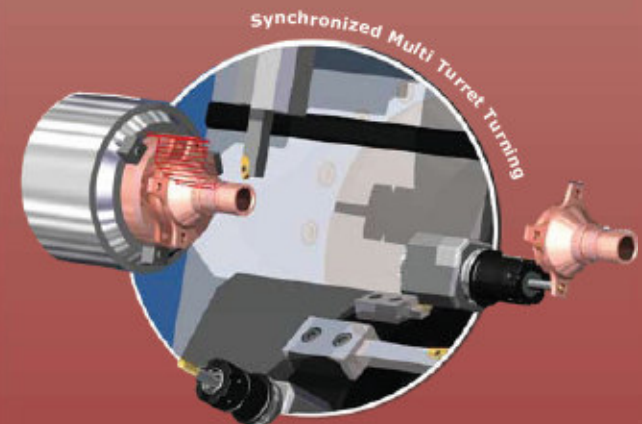
FeatureTURN provides quick and easy programming of all types of turned parts. Features that can be created include both common and specialist hole types, facing and cut-off features, and grooves and threads on the outside or inside diameter. A wide range of canned cycles is supported, for rough and finish profiling, threading, grooving, drilling and tapping.

FeatureTURN/MILL enables both turning and milling features to be created on a single machine in a single set-up. It incorporates all the options within FeatureTURN and FeatureMILL2.5D and can be used to program 5-axis positioning on B-axis lathes, and lathes with C- and Y-axis milling capabilities.

The Multi-Turret Turning module supports machines with up to four turrets, main spindle, sub-spindle and numerous axes. It can output NC code for all major controls, including Fanuc, Okuma and Heidenhain.



B-axis Machining



# industries

FeatureCAM's combination of ease of use and the ability to drive a complete cross section of different types of machine tools makes it ideal for a wide range of industries and applications. FeatureCAM's thousands of users worldwide are producing a huge variety of different types of prototypes, parts, fixtures, tools and molds.

FeatureCAM users are involved in a huge variety of industries including

automotive  
aerospace  
auto sport  
medical

military & defense  
consumer products  
mold & tool  
job shops

machine design  
production  
education  
research

Within these industries, customers use FeatureCAM in many different areas of the product design and manufacturing process from one off items, to mass production with thousands of parts.



# data translation

*'Delcam provides us with effective ways of importing our design data into FeatureCAM. In addition, the software provides the associativity to determine what feature or dimension has changed in the parts from one version to another.'*

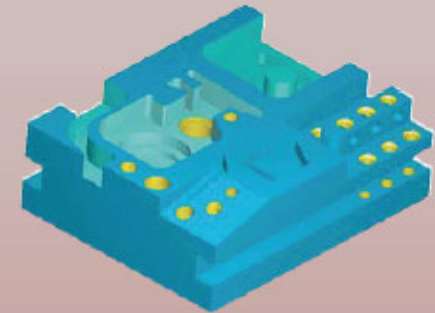
*Ger van den Engh,  
Cytopenia*

## Data Import or Creation

Whatever CAD software files you receive, the data can be imported into FeatureCAM quickly and accurately. Data can be read in 2D or 3D IGES, DWG and DXF formats; while models can be imported directly from AutoCAD, CATIA, Unigraphics, Pro/Engineer, SolidWorks and Solid Edge. FeatureCAM includes solid and surface modeling CAD tools, for cases where no CAD data is available.

## Solid Modeling

FeatureCAM's Solid Modeling module provides advanced tools for designing parts as well as adding fillets, blends and sweeps to imported data. Advanced tools enable core and cavity separation and well as the creation of split surfaces and other manufacturing features. Solid Modeling also allows you to model the manufacturing environment including the machine, clamps and fixtures.

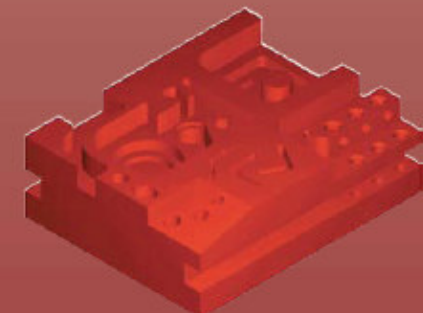


## FeatureCAM reads -

- ACIS
- IGES
- Parasolid
- STEP
- DXF and DWG

## Native data can be read directly from -

- SolidWorks
- SolidWorks Assemblies
- Autodesk Inventor
- SolidEdge
- CATIA
- Unigraphics
- Pro-Engineer



cy Machining. Reverse Engineering. 5-Ax  
Marine. Aerospace. CNC Routing. Total Mo  
Engraving. Product Design. Woodworking



**Small Heath Business Park  
Small Heath  
Birmingham  
B10 0HJ  
UK**

**+44-121-766-5544**

**3280 Electricity Drive  
Windsor  
Ontario  
N8W 5J1  
CANADA**

**+1-877-DELCAM1**

**275 E. South Temple  
Suite 305  
Salt Lake City  
Utah 84111  
USA**

**+1-877-DELCAM1**

[marketing@delcam.com](mailto:marketing@delcam.com)

Making. Subcontracting. Reverse Enginee  
rine. Inspection. Jewellery. Blades and Bl  
typing. Mould Making. Footwear. Medical