Milling

Intelligent Machining From Native CAD Data

EDGECAM Milling provides functionality to program wire frame geometry or solid model component parts on a variety of machine tool configurations, from 2.5 axis milling to complex surface tool paths on 3 to 5 axis milling machines.

It seamlessly integrates 4 and 5 axis simultaneous machining within its milling and mill / turn environment to allow a range of multiaxis cutting strategies to be applied to the most complex tooling or components.

EDGECAM offers ease of use operational programming with intuitive dialogs making programming simple for the new user and comprehensive tool path control for the more advanced requirements. EDGECAM offers range of milling commands for the production engineer which can be used on both milling machines with W axis and Quills plus lathes with driven tools.

Features at a glance:

- Wide range of advanced options providing complete control of the tool.

- In process stock

- Rest material removal

- Easy operation interface

- Total tool path control

- Intuitive, easy-to-use graphical user interface.

- Ideal for the rotary machining of automotive and aerospace components.

- Angle Heads

- Probing

- Tool Libraries

- Shop Floor Documentation

- 5 axis modules include full machine simulation to aid visualization of the machining process.

Face Milling, Roughing, Profiling, Hole Cycles, Thread Milling, Chamfering, Slot Milling are some of the standard operations available and recognize the active stock. Other operations include:

- Update stock – tool paths can be controlled using the current stock which ensures tool path approach is secure and eliminates fresh air cutting.

- Waveform – Waveform cycle is superior to the traditional roughing cycle where machinable geometry is offset inward or outward by % step over.

- Automation Strategy Manager is a flow chart decision making process using your manufacturing methods/ knowledge.

- Probing

- Indexing and Part Positioning

- Angle Heads

- Shop Floor Documentation of the operation process is automatically created along with tooling kit/list, operation breakdown and can be stored centrally on a server so all production staff can access the data.

- Introduction to 5 axis is made easier with the 3 to 5 axis tool conversion and the peace of mind that the program is correct using the machine tool simulator.

- EDGECAM's 4 axis strategies are ideal for the rotary machining of automotive and aerospace components such as camshafts, crankshafts and blades, as well as the production of rotary dies and components for the oil & gas industry.

- 4 and 5 axis simultaneous machining offer key advantages over conventional indexed 3 axis machining.

- Reduced cycle time by machining complex components in a single setup. In addition, dimensional accuracy can be significantly improved through the elimination of positioning errors between setups.

- Improved surface finish and extended tool life are achieved by orienting the tool to maintain optimum tool to-part contact at all times.

- Improved access to undercuts and deep pockets - through tilting the tool or component allows shorter series tooling to be employed, eliminating the need for secondary setups.

- Reduced fixturing, as the cutter can be presented to the component at any required angle.

- 5 axis machining is now common place in all areas of manufacturing as high technology machines have become more affordable along with design demands requiring more complicated tool paths.

- 3 to 5 axis Tool Path Conversion & Tool Path Control

- Turn Milling & SWARF Milling Side Wall Axial Relief Feed

- 5 axis Positioning

- Tool Path Control

3D Milling

EDEGCAM's 3D capability offers a complete solution for generating high quality, gouge protected toolpaths that meet the demands of manufacturers tasked with the programming and machining of complex parts and free form shapes.

With an extensive suite of advanced 3D cycles that are ideally suited for rapidly generating toolpaths for all surface and solids machining needs, EDGECAM's advanced machining cycles bring optimized toolpath control, reduced cycle times and an overall higher level of machining efficiency.

Unlike many CAM systems, EDGECAM is 'CAD natural', so whatever CAD system you use, interoperability between CAD and CAM is seamless, with no data translation. This means that you machine exactly what the designer intended and toolpaths remain associative to the master model.

EDGECAM can directly load: Autodesk Inventor®, Solid Edge®, SolidWorks®, Pro/ENGINEER®, Pro/DESKTOP® Unigraphics files up to and including NX5, and CATIA V5. EDGECAM also accepts files in the following independant formats: IGES, DXF, VDA, Parasolid®, STEP AP203 and AP214 files and ACIS.

Extensive range of 3D cycles

3D machining technology is embedded in all EDGECAM's milling cycles and applies 2D or 3D toolpaths based upon the cycle being used and the interrogation of the geometry to be machined.

Roughing

EDGECAM applies the most efficient approach move for each region of the model utilizing waveform cycle and trochoidal cutter paths to avoid full width cuts, automatically adjusting the toolpath for efficient and safe machining, improving cutting conditions and allowing higher machining speeds to be maintained.

Parallel lace

This command is sometimes referred to as scanning. A series of parallel toolpaths are applied to the model to produce a finish part or used with depths of cut to produce a roughing cycle.

Profiling

Profiling commands are essential not only for 2.5D machining, but also for 3D freeform machining. You use this cycle to finish surfaces in a series of XY profiles down the Z axis on surfaces and solids. Using cusp height control, this will adjust the depths of cut to maintain a constant surface finish. Steep and shallow areas can also be controlled allowing the shallow areas to be finished using alternate techniques. 3D profiling can be applied to follow profiles in XY and Z moves, reducing air cut time by following the components 3D form. Lead in and lead out moves are calculated to avoid any gouging of the component or stock.

Waveform roughing

Waveform cycle is superior to the traditional roughing cycle where machinable geometry is offset inward or outward by % step over. Traditional tool paths have to run slower feeds and speeds due to the variable widths of cut condition when encountering corners and material entry.
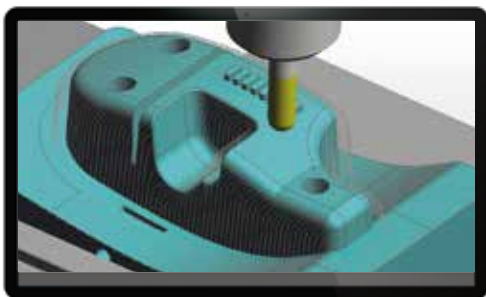
Pre-defined cutting strategy selection
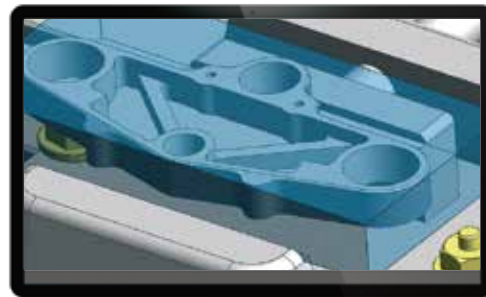
Pre-defined cutting strategy selection

Waveform toolpath has been developed to remove tool load spikes and maintain an even chip thickness, while generating a fluid tool path throughout the machinable elements using a flowing motion. Consistent tool loads generated from the Waveform tool path offers the user the opportunity to rethink speeds, feeds and depths of cut. The Waveform tool path increases tool life and is also kinder to the machine tool.

Rest machining

This intelligent cycle can automatically remove areas of residual material left behind by the tool size and depths of cut. Intermediate slices may be used to reduce the size of the step left by the roughing cycle. Only the step region is machined for intermediate slices. Rest roughing allows the use of large tools to clear away the bulk of the material for the main roughing then select a smaller tool to remove residual material, thus optimising cycle times.

Projection

When machining 3D forms, specific tool path patterns may be required such as circular, radial or spiral on a boss, possibly following flow curves. Controlling these paths is made easy by creating 2D tool paths then projecting the patterns onto the part surface. This technique is very useful for engraving logos and text.

Finishing

Along with projection methods, Constant Cusp machining can be employed to produce an even surface finish on large areas of a mold tool or component. The toolpath constantly adjusts to follow to part surface shape resulting in a uniform cusp.

The use of Rest Finishing and Pencil Milling cycles allow the user to machine the internal corners and radii of the model, ensuring the finished component is fully machined with little or no hand dressing.

Many complex parts are not all freeform shapes where full XYZ movements of the machine tool are required, but have many flat areas where standard end mills would produce a faster toolpath and superior finish. EDGECAM's Flat Land command will automatically seek out and machine these flat regions.

Simulator for 3D

The Machine Simulator offers full simulation of the machine tool and machining process, as well as detecting collisions between machine, holders, tooling, helping you to :

- Avoid expensive collisions and costly prove-outs

- Optimize the cutting process

- Reduce cycle time

- 'View Comparison' identifying areas of un-cut material