

G ZERØ FIVE

PRODUCTION MILLING

INTELLIGENT MACHINING FROM NATIVE CAD DATA: Edgcam 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.

The ability to use Solid models and the design data from all the major CAD system maintaining associative links allows quick and efficient programming, especially when a Design modification is made. Edgcam will inform the user of model updates and show what the changes are and where tool paths need to be corrected. A simple update of the tool paths is all that is required rather than a reprogram of the part. Tombstone and multi part fixture is supported, along with extended datum shifts. Edgcam offers ease of use operational programming with intuitive dialogues making programming simple for the new user and comprehensive tool path control for the more advanced requirements.

Edgcam 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 with . Face Milling, Roughing, Profiling, Hole Cycles, Thread Milling, Chamfering, Slot Milling are some of the standard operations available and recognise the active stock

In Process stock

Tool paths can be controlled using the current stock which ensures tool path approach is secure and eliminates fresh air cutting. The stock can originate from automatically created stock or from a forging or casting model produced in CAD.

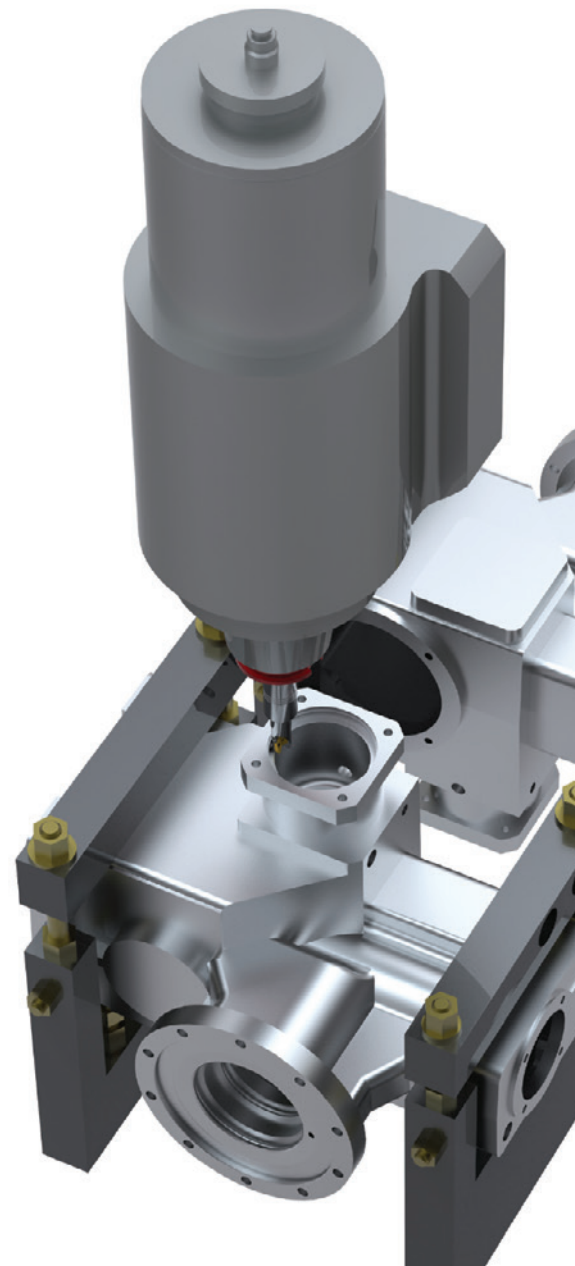
Face Milling

Creates a series of straight cuts on a horizontal plane. Face milling will recognise the boundary shape and remove air cuts where applicable, links moves for change in cut direction can be controlled to give smooth transition producing and even flow tool path which is kinder to the machine tool and cutter engagement.

Hole Cycles

Edgcam includes all the standard drilling, tapping and rigid tapping routines with canned cycle and sub routine output. Back boring is also available where conventional machining cannot be achieved on the part. When using solid models the hole size , thread data, depths etc is extracted and the necessary tooling suggested from the tool libraries

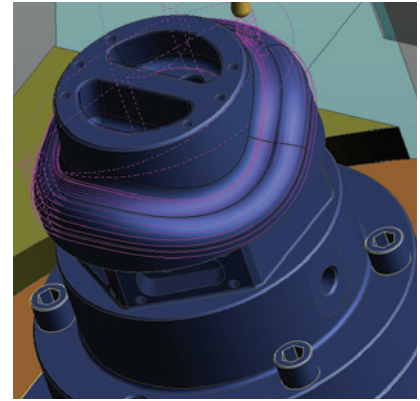
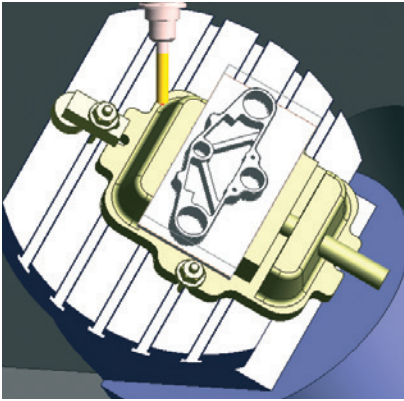
In process stock
Rest material removal
Easy operation interface
Total tool path control
Angle Heads
Probing
Tool Libraries
Shop Floor Documentation



vero
Software



Edgecam offers intuitive dialogues making programming simple for the new user and comprehensive tool path control for advanced requirements



Rough Milling

Has a variety of tool path control and methods, from face, concentric, spiral and the wave form tool path. Edgecam produces the required tool path and tool entry into the material. Shape recognition allows sub routines to be applied with a simple check box. Edgecam will recognise the change in geometry walls and create sub routines as appropriate.

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. Wave form toolpath has been developed to remove tool load spikes and maintain an even chip thickness and 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.

Thread Milling

Thread milling is a popular technique when machining threaded

holes on larger components, especially in the oil and gas, power generation and other heavy industries. Edgecam's thread milling cycle will automatically suggest entry points, and lead in lead out paths. Single pass or multi-pass helical move tool paths can be output.

Automation

Strategy Manager Automation is a flow chart decision making process using your manufacturing methods/knowledge. Edgecam recognises manufacturing features and data from a 3D solid model and applies your proven manufacturing tool paths and techniques. This creates tool path cycles automatically with your own tooling to your requirements. This greatly reduces the offline programming time maximising on your investment plus aids learning curves for new engineers.

Probing

Edgecam supports Renishaw probing part setup cycles. The probing cycles can be integrated by simply adding an additional toolbar to the user interface. This supports all datum offset part positional requirements which can be a pre-requisite to actual machining whether on 3 axis mill or multi-pallet tombstone fixture.

Indexing and Part Positioning

Support for single or compound indexing with a using A, B or C axis combination using safety clearance zones to give 4 or 5 axis positioning. Datum shifting and extended offsets can be output for each new position.

Angle Heads

The support for Angle Head attachments is available in Edgecam at all levels. The holder and tool can be saved in the comprehensive tool libraries and called up in the tool change along with material feeds and speeds. The use of angle heads will use plane switching if supported by the machine tool control. When used, the holder body along with the tool is collision check in the machine simulator.

Shop Floor Documentation

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. Machine tool set up information along with digital images can also be added along with stock and fixture requirements. Documents and pre set tool drawing can be attached. This module is standard with all systems and is very useful solution for pre setting tooling areas.

G ZERO FIVE

WAVEFORM ROUGHING

CONSISTENT MATERIAL ENGAGEMENT

Waveform roughing strategy is a high speed machining technique that maintains a constant tool cutting load by ensuring the tool engagement into the material is consistent. The tool path moves in a smooth path to avoid sharp changes in direction which maintains the machine tool's velocity.

Waveform Roughing

- Reduces cycle time
- Improves tool life
- Lengthens machine maintenance cycles
- Keeps constant chip load
- Cuts deeper and faster

Constant Engagement With Material

Although the Concentric pattern looks much simpler at the first glance the problem is that the tool "digs" into each corner causing the tool to overload, leading to reduced tool life or tool breakage. In reality the machine tool operator may have to reduce the cycle feed rate to compensate and thus, increase the manufacturing time.

As Waveform maintains a constant engagement the feed rate can remain at the optimal value throughout the cycle. This will improve the tool life and greatly reduce the risk of tool breakage.

The Waveform Pattern

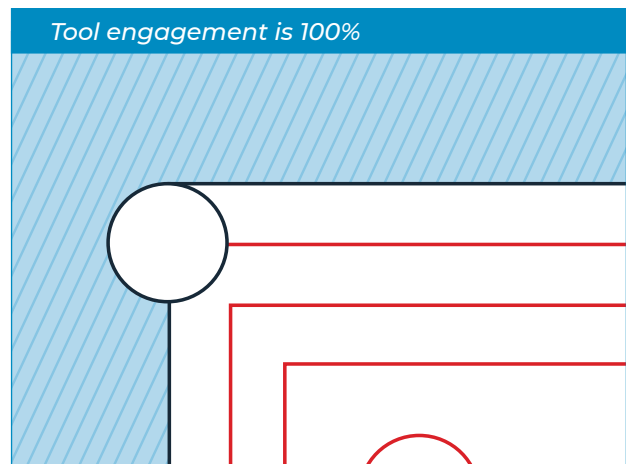
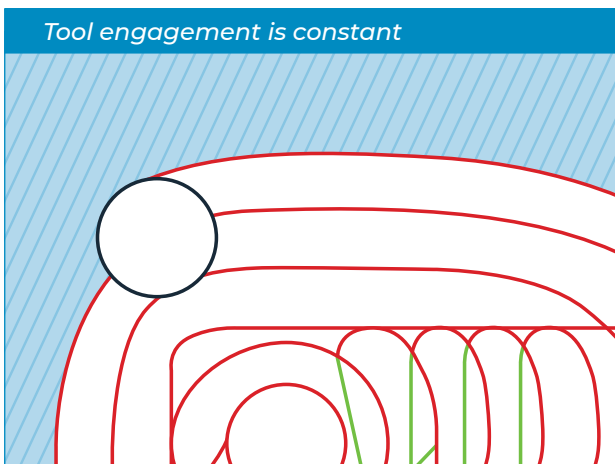
To maintain a constant chip load the cycle uses the philosophy that we machine from "Stock to part". This reduces the amount of intermittent cuts, particularly on external regions, which means the tool is engaged with the material for longer without lifting clear. Traditionally, cycles generally offset the component until they meet the stock. This can lead to the generation of sharp corners and discontinuous tool paths.

For pocket regions the tool will helical in to depth at the centre and open the pocket up so that it can create a continuous spiral cut until the edge of the pocket is reached. Any remaining corners are then removed.

Automatic Adjustment for Tool Engagement

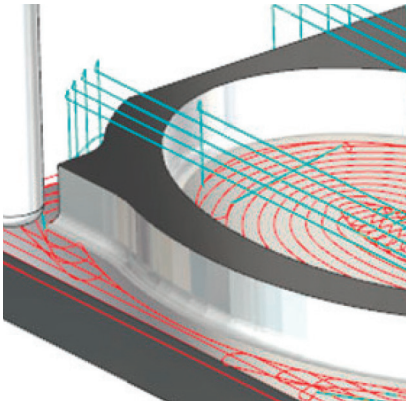
To maintain the tool engagement and the chip load the tool path is automatically adjusted to compensate. When cutting into a concave area tool engagement is increased. The cycle adjusts the step over between the passes to compensate and maintain the desired engagement.

When cutting a convex area the opposite affect occurs. As the material falls away the tool path step over is increased to maintain the desired engagement.



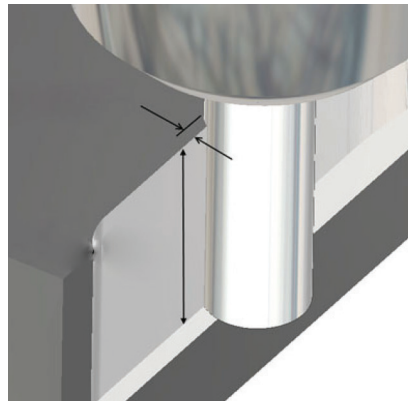


Waveform machining is standard with Edgcam, no additional purchase necessary



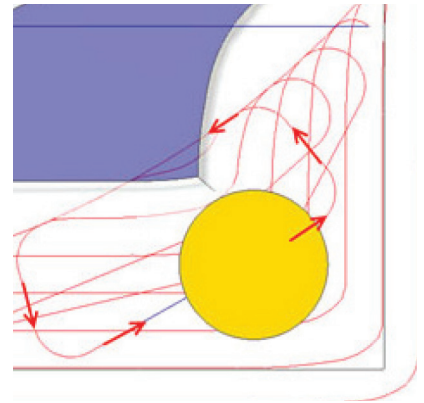
Smooth Tool Path

By ensuring the cycle produces a smooth tangent tool path, the velocity of the machine can be maintained and the desired feed rates achieved. This also has the benefit of reducing shaking and vibration on the machine and component.



Full Cut Depth Machining (High Speed Machining)

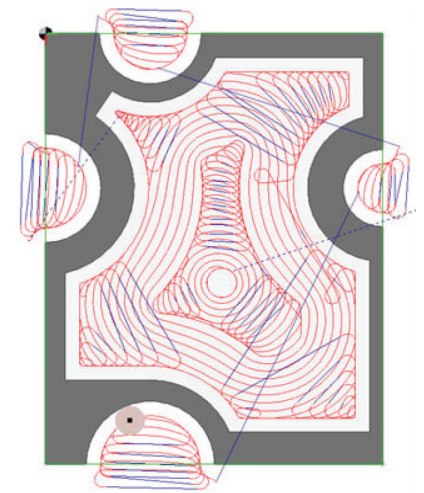
Waveform Roughing greatly improves standard roughing by ensuring a constant volume of material is removed. In addition, this also opens up the way to use high speed machining, particularly for hard materials.



Linking the Tool Path

The links within the cycle are aware of the rapid and High Feed rate settings for the machine tool. When moving to the next cut the cycle will automatically choose the fastest method to get to that point. In localised areas the tool will stay at depth, but on long moves the tool retracts and rapids to position.

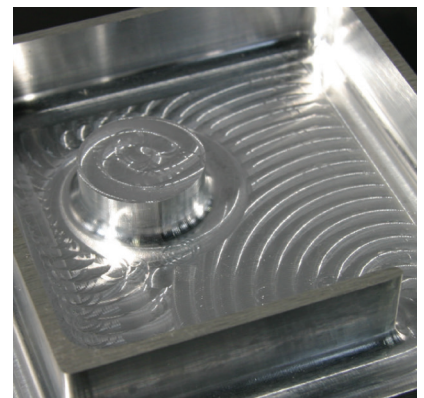
Cutting along as much of the flute length as possible distributes wear evenly along the entire flute length, rather than just the tip. The radial cut depth is reduced to ensure consistent cutting force allowing cut material to escape from the flutes.



Stay at Depth

When the tool stays at depth the path will automatically move around the stock when required. The moves at depth can be at high Feed and allows the user to specify a small retract to stop the tool rubbing on the floor of the part.

Tool life is furtherer extended as most of the heat is removed in the chip. An example of the feed rate and depth of cut that can be achieved in hard materials, both in Metric and Imperial measurements, is listed below.



Simple Interface

We have ensured that the cycle uses the information in the part and Code Generator where possible and kept the interface to only 3 modifiers that the user can adjust for the waveform pattern. This ensures the cycle is easy to apply and is integrated into the main Roughing cycle.



MATERIAL	SS1650 Carbon Steel	6AL4V Titanium
TOOL	10 mm endmill	1/2 inch endmill
DEPTH OF CUT	20mm	3/4 inch
STEPOVER	10%	10%
FEED RATE	5700mm/min	50 in/min
SPEED	9500 rpm	3128 rpm

G ZERØ FIVE

3D MILLING

EXTENSIVE RANGE OF 3D MILLING CYCLES

Edgecam'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

Industries involved in the manufacturing of 3D forms such as Aerospace, Prototyping, Mould Tools and General Engineering will all benefit from Edgecam's 3D machining cycles. Edgecam offers not only best in class prismatic machining capabilities, you also have powerful 3D solid and surface machining strategies, all in one solution.

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 optimised toolpath control, reduced cycle times and an overall higher level of machining efficiency. Unlike many CAM systems, Edgecam is 'CAD neutral', 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 independent 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 utilising 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.

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. Wave form toolpath has been developed to remove tool load spikes and maintain and even chip thickness and 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.

One single machining environment

Full machine tool and toolpath simulation

Full collision checking

Full boundary control – tool contact, inside, outside etc

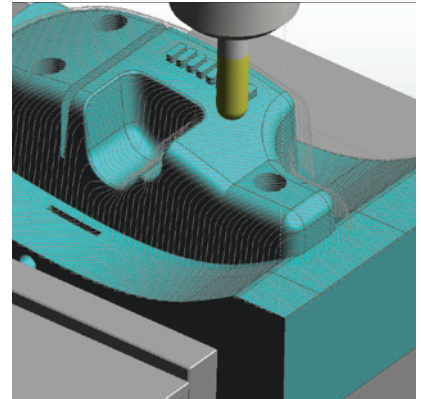
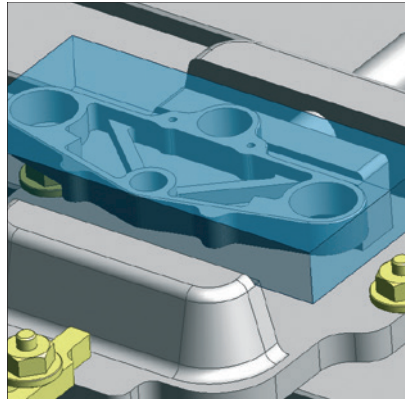
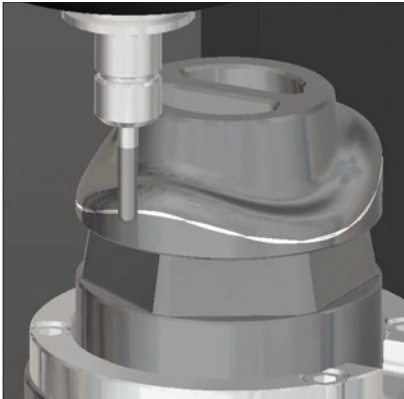
Efficient link moves for High Speed Machining

In process stock, reducing air cutting





Edgecam offers not only best in class prismatic machining capabilities, you also have powerful 3D solid and surface machining strategies



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.

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 allow 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.

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 logo's and text.

Finishing

Along with projection methods, Constant Cusp machining can be employed to produce an even surface finish on large areas of a mould 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 a 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. When combined our 3D Machining cycles with our 5 axis module, the capability of converting 3 axis tool paths to 5 axis toolpaths provides the ability to get better reach and cutting conditions.

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
- Optimise the cutting process
- Avoid costly prove-outs
- Reduce cycle time
- 'View Comparison' identifying areas of un-cut material