HEXAGON

EDGECAM

Industry leading CAM system for NC part programming

EDGECAM

Industry leading CAM system for NC part programming

EDGECAM is a market leading computer aided manufacturing (CAM) system for NC part programming. With unparalleled ease of use and sophisticated toolpath generation, it's the only CAM system you'll need for milling, turning and mill-turn machining.

EDGECAM utilises your in-house knowledge and experience to drive the CAM process with automation tools to suit different applications - allowing you to maintain your competitive edge.

Mill / Turn

Multi task machining

The use of tail stocks, steady rests, sub spindles, twin turrets along with C Axis, CY Axis and B Axis are regular features on today's Multi-Task machine tools. In this collision rich environment, the programming of these machines is made simple and safe utilising EDGECAM's turning and milling combinations in a single environment. With the addition of 4 and 5 axis simultaneous milling, the simulation of these machine tools is made even easier.

Features at a glance

- One single machining environment

- Full machine tool and toolpath simulation

- Reduce machine tool prove out by graphically simulating the tool path

- Reduce component cycle time

- Full collision checking on component parts

- Support for Twin Spindle, Twin Turret, Pickoff spindle, C, Y & B axis machining

Mill / Turn Simulation

EDGECAM offers a full kinematic simulation package. All the cycles and movements are supported along with the full graphics of the machine, tails stocks and steadies allowing the programmer to visually simulate the machine motion and avoid collisions as they program. Looking for a complete digital twin? EDGECAM offers complete integration with NCSIMUL for true 1:1 G-Code verification.

4 / 5 Axis Simultaneous

Mill Turn machines have many uses and allow much more flexibility and capabilities not offered from other machine configurations.

EDGECAM uses the latest cutting technologies and machine cycles and with this offers 4/5 Axis simultaneous milling options.

Features include

- Axial Milling
  This mode allows the user to perform milling operations using the C axis with Rotary options allowing programming along the Z Axis

- Radial Milling
  Allows the user to machine features around the diameter giving flexibility to the user to turn the component, then create the mill features using driven tooling

- Y Axis Milling
  The Y axis milling capabilities allows the engineer more control over the toolpath creation and CNC code output. EDGECAM supports plane switching where available on the machine tool and keeps the CNC code to a minimum by outputting arcs as required.

B Axis Head Support

In a Mill Turn environment EDGECAM fully supports the use of B Axis head work whether working on single spindle or sub-spindle machines. Features include:

- B Axis positioning on upper turret allowing more precise and varied approaches to Mill/Turn parts giving the engineer a more flexible approach to programming complex components

- The B Axis can be programmed to tilt to any number of angles to allow the many milling options available to tackle any number of features such as Faces, Pockets and Holes

- B Axis machining on both the main and sub spindles maximises productivity allowing optimum machine performance. These features are also supported by our Full Kinematic Simulator with collision detection when using for 4/5 Axis Simultaneous work. This provides accurate feedback of the part being cut before release to the actual machine

Upper / Lower Turret 4 Axis TURNING

The Four Axis turning option in EDGECAM allows major advantages and functionality which is not always easily achieved at the machine control. EDGECAM programming techniques allow you to use more than one turret at the same time in the mill turn programming environment. This means you can use two fixed cutting tools in the same cycle by using a number of Four Axis commands from the Cycles menu.

EDGECAM supports the use of Upper and Lower Turret configurations and will support simulation of these including features such as:

- Mirrored Turning

- Balanced Turning

- Synchronised Turrets

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 with .

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 recognise 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 to 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

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.

With an extensive suite of advanced 3D cycles that are ideally suited for rapidly generating toolpaths for all surface and solids machining needs, EDGECAMs 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.

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.

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.

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 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 be 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

- Optimise the cutting process

- Reduce cycle time

- 'View Comparison' identifying areas of un-cut material
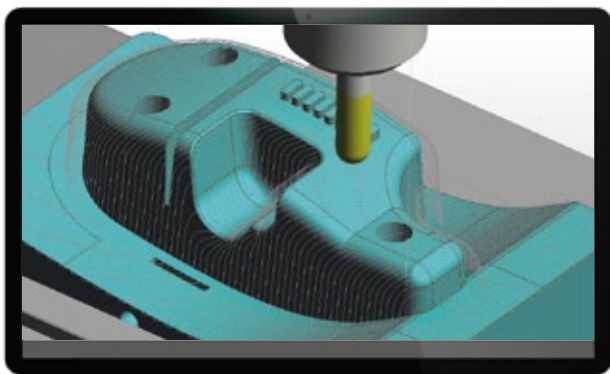
Pre-defined cutting strategy selection
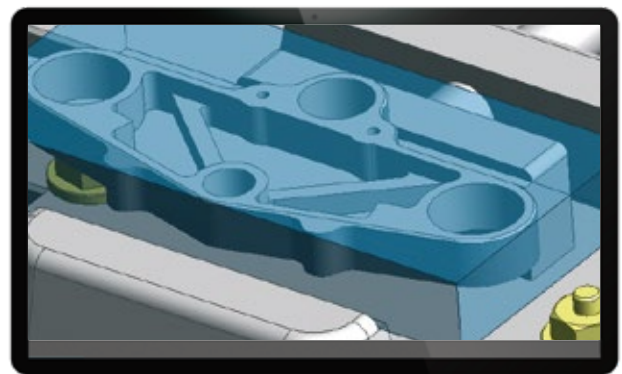
Pre-defined cutting strategy selection

Features at a glance

- Increase machine tool utilization

- Reduced programming & component cycle time

- Eliminate programming errors and reduce potential scrap

- Reduce machine tool prove out by graphically simulating the tool path

- Avoid collisions and expensive damage to machine tool

- Full support for canned cycles

- Reduce tooling inventory and stock

Turning

Production turning

EDGECAM Turning provides functionality for a wide range of machine tools, including 2-Axis lathes, multi-turret configurations, sub-spindle turning centers and mill/turn machines. EDGECAM fully supports turning centers including Bar Pull and Feed, Part Pick Off, Balanced and Mirrored Turning, and Turret Synchronization. EDGECAM produces advanced rough and finish turning cycles, together with support for facing, boring and drilling in either canned cycle or longhand format. Toolpath calculation takes into consideration the complete tooling insert and tool Holder including the "F" distance and previously machined material to avoid gouging and eliminate air cutting. Ease of use and an understanding that cycle times are critical, especially on multi-configuration mill/turn machines, underpin the development of EDGECAM's turning functionality. EDGECAM offers support for Sandvik Coromant Wiper inserts for turning tools, allowing these productivity enhancing inserts to be used reliably in all aspects of production machining.

Update stock

EDGECAM has the ability to keep the stock updated live within the sequence tree. The stock model is rest material, or material that hasn't been machined. Subsequent toolpaths will automatically detect the rest material resulting in 100% efficiency for any turning toolpath throughout the EDGECAM sequence. Update stock is supported from the most basic 2 axis turning center, right through to a CYB multi turret sub spindle Mill / Turn. When back turning into a recess or groove it's important that the back turning cycle knows the current condition of stock to avoid air cutting and potential collisions on the approach into the recessed area. On a sub spindle turning center, when a component is transferred from the main spindle to the sub spindle, the live stock transfers with it. Any subsequent machining on the sub spindle will detect the stock in the state that it left the main spindle which ultimately provides the most efficient machining sequence possible.

Collision Detection / Simulation

It is imperative that not just the tool in contact with material is checked for collisions, but also tools that are not in use on a turret. Most turning centers offer a relatively small working envelope that can be collision rich. A good example of this is on static turrets where tools such as boring bars, can extend out further than the tool in cut. EDGECAM will not only collision check the tool in cut but also all of the tools on the turret against the machine tool kinematics, fully supporting Mini Turrets, capto tooling and programmable steadies.

Swarf Clearance

When machining inside a bore, loose material can build up around the insert which can result either result in insert failure or severely decreased tool life. EDGECAM will allow you to retract tool out of a bore or away from a diameter after a set numbers of cuts. The user can retract the tool mid cycle, to a known position, after a set number of cuts to clear any loose material out of the bore.

Sub spindles

EDGECAM fully supports turning centers with a sub spindle & twin turrets, including:

- Bar pull & bar feed

- Part pick and return

- Running in conjunction with the main spindle

Twin turret support for both single and twin spindle lathes including:

- Balanced turning

- Z lag options improving metal removal

- Mirrored turning

- Turret synchronization and simulation

Cycle control

- Individual Element offsets
EDGECAMs turning cycles offer the ability to specify offsets to individual turned diameters, bores, grooves and faces. This function is useful where a turned component needs some elements to be finished turned, and others to be left a grinding allowance for subsequent machining or heat treatment.

    Most systems on the market today will only allow you to set a constant offset, where as EDGECAM gives the user full control over offsets for each individual element on the turn feature.

- Break Edges
Sub contract machinist do not always have the ability to go back to their client to ask them to revize the design to include the chamfers or break edges, even though they have been asked for on the engineering drawing. EDGECAM turning cycles offer the machinist the ability to specify a break edge where a chamfer hasn't been included on the model supplied to them by their client

- Down Cutting
This function within the finish turning cycle alternates the cut direction on the finish turn profile so the tool is always down cutting or it never drags up the face. This give the enhanced tool life and achieves a superior surface finish.

- Sequential Castellation grooving
TRADITIONAL grooving cycles wear the tool on one side after the initial full width cut. With Sequential Castellation grooving, the tool starts at one groove edge and moves to the other edge, producing full width cuts.

    It then goes back and removes the 'rings' left behind by the first cutting pass. This ensures that the load on the grooving tool is on the front of the tool, opposed to the sides. It also ensures even wear on the insert.

- Rough Turn Sectioning To keep the tool push off on a long diameter to a minimum, EDGECAM have developed a section strategy where the user can break the rough turn cycle into sections. The user sets a Z break distance and the roughing cuts are divided into short sections.

- Rough turn Variable cut Depth
This function is to Prevent notches wearing into the tool. Cuts are alternately 'ramped' then 'normal'. During the ramped cuts the cut depth gradually reduces to zero. The next cut (which will be 'normal' and starts at the same cut advance) then removes the leftover ramp. If a ramped cut is interrupted by the profile, it follows the profile until it re-joins its ramped path.
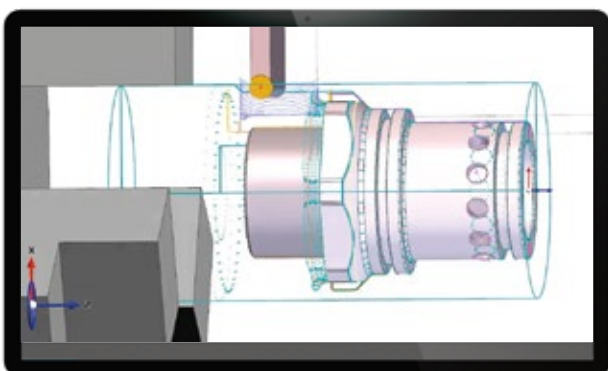
Waveform Turning
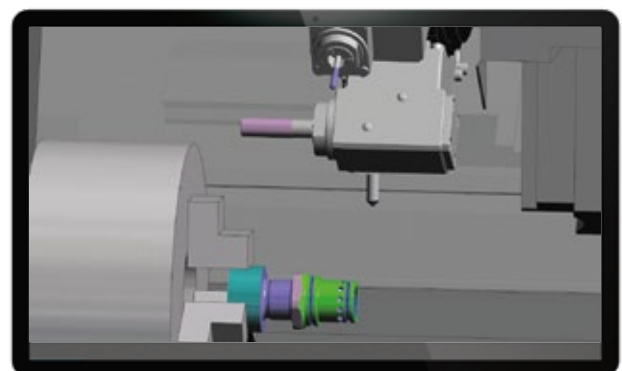
Simulation with collision view

The 3D stock model feature is a very useful option for us. Machining the component with smaller tools helps us to remove as much material as possible before finishing."

Jason West,
Astro Machine Works

Waveform

Waveform machining is included on EDGECAM Licensees at the Standard and higher product levels, no additional purchase necessary. Waveform maintains a constant chip load for high speed machining. The tool moves in a smooth path to avoid sharp changes in direction, maintaining a constant/high feedrate.

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.
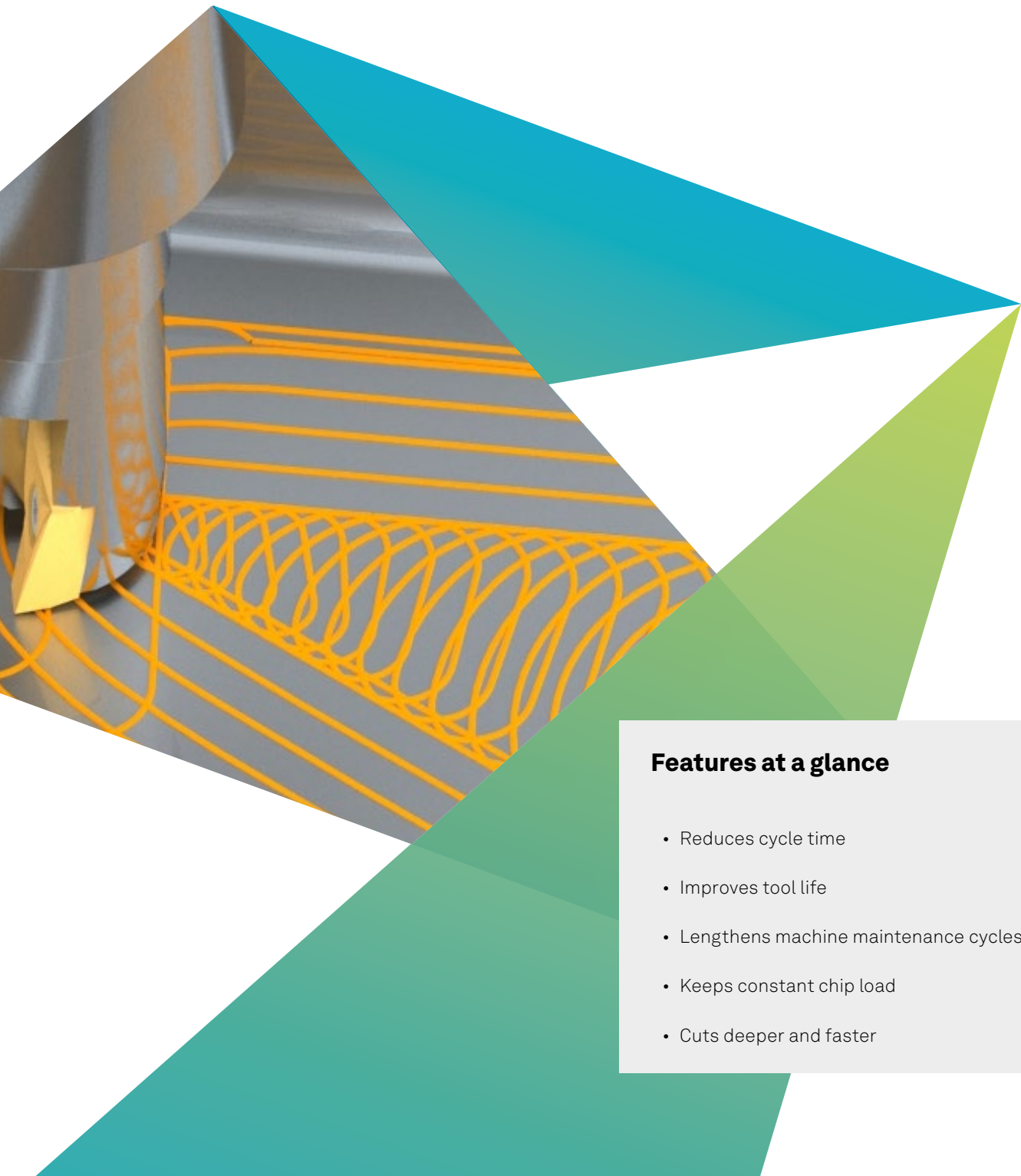
Features at a glance

- Reduces cycle time

- Improves tool life

- Lengthens machine maintenance cycles

- Keeps constant chip load

- Cuts deeper and faster

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 center 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.

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.

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.

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.

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.

Full Cut Depth 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. 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. Tool life is furtherer extended as most of the heat is removed in the chip.

Wire EDM

EDGECAM Wire EDM provides the flexibility and confidence to manufacture 2 and 4 axis parts accurately and efficiently. Advanced functionality enables manufacture of complex shapes with irregular tapers easily.

Intuitive operation user interface

The 2 and 4 axis machining operations give the user a choice of parameters such as machining direction, auto offsetting, lead on/off radius, tag distance, lead off distance, lead on/off technology to name just a few. Each parameter is accompanied by a bitmap giving the user additional information on how it will affect the resultant toolpath.

4-axis wire EDM

Multiple tagging and tag removal

In EDGECAM Wire it is possible to select from several different methods of unattended machining. If your machine is equipped with automatic wire threading, then you will most likely want to run unattended as long and as often as possible. Unattended machining is performed by leaving the slugs attached while all of the preliminary cuts are taken. Numerous strategies are available to cut the part; for instance, taking all of the rough cuts before finishing, in which case all rough cuts are taken while leaving the tags attached, then the tags are removed, and finally the finish cuts are taken. Alternatively, take the rough and finish cuts while leaving the component or waste material in place and then remove the tag and finish this area.

> **Roughing operations can be programmed in half the time and run off 70% faster.**
>
> **Roy Thomas,**
> Patterson Mold & Tool

Feature finding

Feature Finder will automatically identify regions which require Wire EDM machining on solid models. Multiple features can be created within a single transaction plus, the feature can either be a plane 2D shape or a 3D contour. Furthermore, the feature shape will automatically update if the solid model is modified.
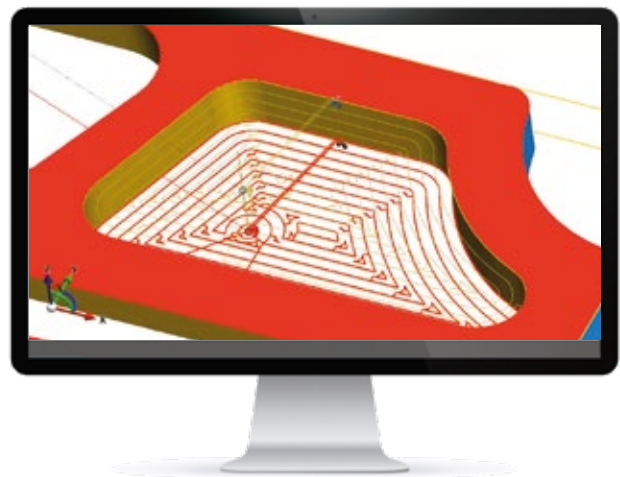

Automatic creation of cut destructions

Post Processor Database

EDGECAM Wire supports a comprehensive range of EDM machines from leading Machine Tool manufacturers including Agie, Charmilles, Brother, Fanuc, Hitachi, Makino, Ona, Sodick, Seibu and Mitsubishi. Machine Tool supplied technology data is also included for supported machine tools together with JOB/Script file output for Agie and CMD file output for Charmilles. The advanced post processors are easily configured to suit different machine models and configurations.

Automatic Cutting Strategies

EDGECAM Wire offers predefined cutting strategies for automatic ordering of rough, finish and tag removal passes to accommodate common shop floor needs such as 'attended day cutting' and 'unattended night cutting'. Other features include:

- 4 Axis 'null span' support including reliable offsetting where small spans are eliminated

- Square, conic and constant radius corner modes for variable taper

- Separate clearance for main cuts and tag removal available

- Additional M-Codes including 'Stop/Optional stop', 'Power on/off' and 'Wire cut/Thread'

- Automatic start hole file creation

- Incremental lead on points to help eliminate 'witness marks' on finished component

- Automatic approach and retract technology to allow power to be 'ramped' up and down gradually

Features at a glance

- Intuitive graphical user interface

- Extensive range of CAD interfaces for both import and export

- Comprehensive machine and postprocessor database

- Roughing and finishing cuts easily applied to multiple punches or dies

- Multiple tagging options with auto tag removal

- Reverse cutting on roughing, finishing and tag removal passes

- No-core pocket destruction of round, irregular and tapered apertures

Features at a glance

- Interactive, Intuitive and Informative

- Built-in machining strategies automatically create cutter paths

- User defined toolpath creation

- Ribbon bar interface

- User navigation and user guidance

Workflow

Designed for manufacturers to reduce costs, improve quality and hit shorter lead times. Operators will be able to apply toolpaths to prismatic parts in minutes.

Accelerated generation of CNC code

EDGECAM Workflow understands the component topology and the required manufacturing environment therefore accelerating toolpath generation. Workflow will have a significant impact on shortening programming time, and because it is so simple to operate, the learning curve for new users is considerably reduced.

Workflow - A Five Stage Process

1. File: Adopting the common Microsoft Office 2010 theme, this allows users to efficiently manage files and folders.

2. Set-Up: Dynamically sets up datum position, adds stock and fixtures through the interactive stock, fixture and machine manager functions.

3. Features: Using EDGECAM's powerful automatic feature recognition, all types of manufacturing features can be found.

4. Machining: The Planning board applies a suggested order of manufacture, which can be manipulated easily by drag and drop techniques.

5. NC Code: The toolpath is then simulated in the machine simulator, checking for collisions, gauges and limit over travel. The toolkit can be reviewed and edited before NC code is generated to complete the five-stage Workflow process.

Loading and positioning the component

Now fully automatic. In many CAM systems the user has to manually set the environment and use traditional commands to create a datum.

Importing fixtures

User defined fixtures, including vices, chuck and clamps can be applied using the fixture manager.

Manufacturing method and suitable machine tools

The user is presented with a list of suitable machine tools based on the component's geometry, ensuring machine limits are respected.

Adding user-defined stock, or stock from a database

Based on the components dimensions, a selection of defined stock materials are listed which allows a suitable stock to be easily applied.

Managing strategies to aid manufacture

EDGECAM engineers have derived suggested methods of manufacture to machine the part on a feature-by-feature basis, with logical cutters path.

These processes are used as 'toolpath accelerators' allowing users, if required to add additional inputs to produce the exact toolpath required for their processes. However, it is also possible with the addition of EDGECAM Strategy Manager module the user can update the manufacturing Strategies to match the companies' manufacturing processes, so that all manual intervention is removed.

> **The automatic feature recognition reliably detects all types of holes, whether it be a fit, a threaded or blind hole. This reduces the programming effort and saves time.**
>
> **Joseph Batz,**
> Sable Engineering

WORKXPLORE

Native CAD Viewer

EDGECAM WORKXPLORE is the ideal tool for directly displaying and evaluating 3D CAD files without the need for the original CAD application. The intuitive, easy-to-use interface enables novice and experienced users alike to explore any type of 2D/3D CAD file.

A wide range of CAD interfaces are available: Catia V5, Catia V4, Unigraphics, Parasolid, Pro/E, SolidWorks, Solid Edge, Cadds, IGES, STEP, Unisurf, STL (binary and ASCII), VRML, ISO toolpaths, DXF, DWG, HPGL. Please note that it is possible to load GD&T data from the CAD interfaces that support geometric tolerancing.

The software saves the native CAD data in its own lightweight format enabling users to calculate surface areas and volumes, measure thickness, dimensions and angles and much more without requiring the original CAD data. This lightweight format can be easily transmitted to other project members via email.

EDGECAM WORKXPLORE has been created for the efficient import and analysis of files of all sizes and types but it is the speed at which it imports large 3D CAD files which is particularly impressive, often taking less than half the time to open a file compared to the original CAD application.

Measurements

EDGECAM WORKXPLORE features a wide range of 2D and 3D measurement functions that are highly accurate due to the high precision B-rep 3D model employed. Even non-expert CAD users can quickly get to grips with the software's measurement functions and obtain good results instantly by using the software's predefined selection modes (points, 2D entities, planes, surfaces, etc). Measurements can be automatically included as measurement entities and can be anchored to characteristic points of the part. Entity labels automatically pivot to remain visible at all times.

EDGECAM WORKXPLORE's specialist measurement functions allow expert CAD users to recover point clusters from three-dimensional measuring equipment or machine probes and to quickly check any data revision against the original CAD geometry. The software also allows users to easily generate control point files for transmission to three-dimensional measuring equipment or NC machines.

Annotation

EDGECAM WORKXPLORE offers a wide range of annotation functions. Users can convey their ideas, observations, instructions and change requests easily and quickly. The need for 2D drawings is minimized as users can directly add dimensional and geometric measurements, annotations and labels to the 3D model.

Analyse

EDGECAM WORKXPLORE comes with a full range of specialist analysis tools which can be used to help make quotes, diagnostics, assembly notes or for 3D models production preparation. It also offers a range of analysis functions which are usually only available with more costly CAD solutions. As well as being extremely fast and efficient (even on very large models), the software's functions are very easy to use with results being displayed in color on the 3D model. In a single mouse click, labels are automatically inserted on surfaces displaying accurate values resulting from various calculations.

High performance dynamic sectioning allows users to explore inside a part or an assembly easily and accurately. The dynamic reference controller allows users to control the section plane with the mouse in rotational and panning directions or following a guide curve. The cross section can be made visible on the 3D model or as an isolated

entity and can be extracted and exported via the DXF, DWG, etc interfaces.

Curvature radius and plane face analysis is a valuable tool for users enabling fast cost and production time evaluation. The coloring of elements to be analyzed is automatic and users can insert measurement labels containing precize values of the elements selected. Generation of the bounding box of a part or a set of parts is instantaneous and provides information on the dimensions, volume and weight of the selected elements. This can also be used to determine the optimal stock model required for manufacturing.

Precize measurement information is available both on objects and surfaces and enables rapid calculation of volumes and areas. Complementary information such as the object name, encumbrance, number of faces, etc. is also available.

The calculation and display of drafts and undercuts are extremely quick, even on very big parts. EDGECAM WORKXPLORE automatically colors drafts and undercuts according to the mold stripping axis. Precize draft angle values are displayed dynamically as the mouse is dragged over the surfaces and can be inserted by default in the 3D model.

Automatic 3D part comparison allows real time 3D graphic display of the differences between two versions.
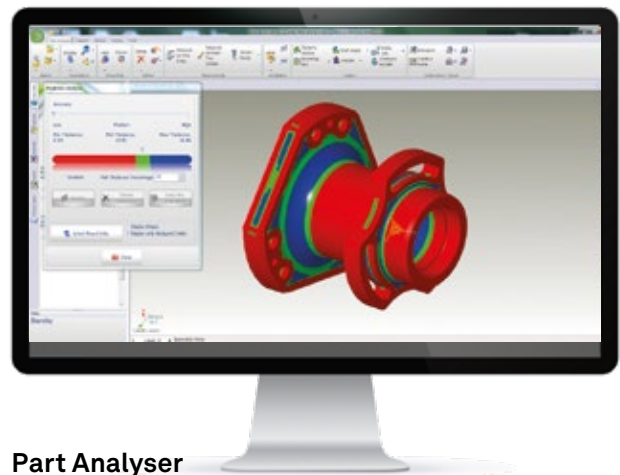
Part Analyser
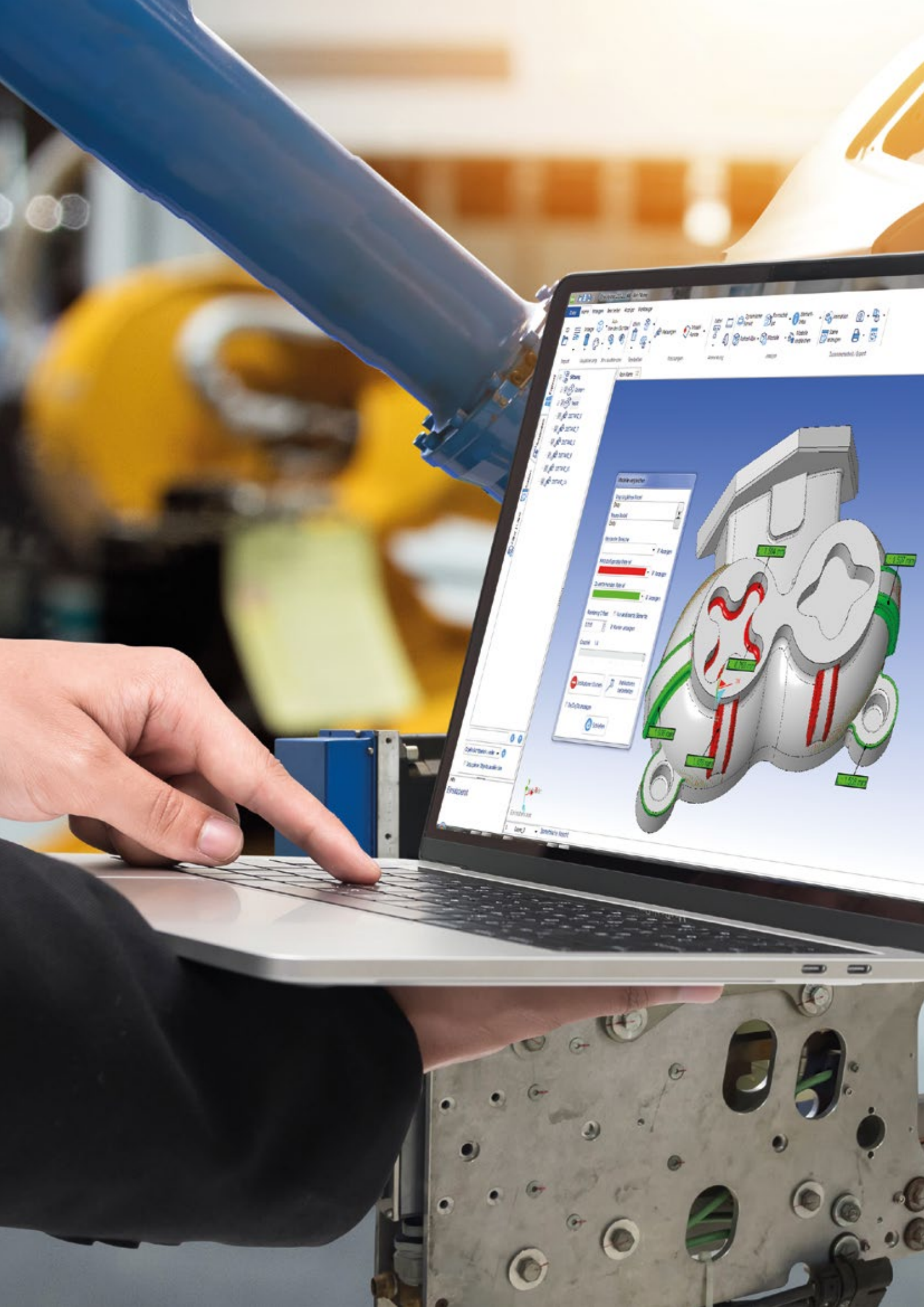
Animation

EDGECAM WORKXPLORE includes a fully functional animation kernel allowing users to generate exploded views or animated assembly movements. Setting up animations is achieved by simply initiating basic movements such as translation, rotation or following a guide curve.

Interference Detection

Dynamic collision analysis functions can be used during animations to carry out real time control of mechanism interoperability or process control. Users can also generate short videos directly from the animation menu that can then be shown to customers or other project members.

Documentation

With EDGECAM WORKXPLORE users can generate screen captures to illustrate technical documents and assembly sheets. In addition to the traditional screen capture functions, EDGECAM WORKXPLORE features an image collector which also allows users to easily manage and distribute large volumes of images. cludes a fully functional animation kernel allowing users to generate exploded views or animated assembly

Publication

EDGECAM WORKXPLORE allows users to easily share their CAD models throughout the entire design and manufacturing chain, with all project members whether they are product managers, marketing, sales, outside manufacturing consultants, customers or suppliers.

Creation of predefined scenes

Whatever their CAD software skill level, it is important for company staff to be able to use communication tools that capture their personal expertize and enable this to be exploited by other staff members.

Predefined scenes have therefore been created and stored in the software. The configurations, orientations and views of these scenes are saved in a fixed state along with related annotations, dimensions and labels. Users can then simply navigate from screen to screen to find the configuration preferred by the creator of the scenes:

Export

With EDGECAM WORKXPLORE users can quickly convert standard or native 3D models via the available export interfaces (IGES, STL, URML). B-rep models can also be saved under the IGES format.

Collaborate

Users need no longer worry about format compatibility or what software their partners use.

EDGECAM WORKXPLORE enables users to communicate 3D parts and assemblies to sub-contractors, customers or colleagues using a standalone, lightweight application that can be easily transmitted via the Internet.

The recipient can immediately display and work on the 3D model without requiring the original CAD data. Furthermore, user access protection can be applied to ensure that only the right people access the data.

INSPECT

EDGECAM Inspect combines our industry leading metrology tools and know-how, with our cutting edge manufacturing expertize. Is a full featured, easy to use solution for users wanting fast creation of on machine measurement cycles with quick, efficient tool path creation and generation of reliable, measurement results. EDGECAM Inspect offers unparalleled ease of use and sophisticated probe path generation for both in process and end item part inspections.

Process Validation

Complex CNC machining operations can now be easily validated with in process measurements. Manufacturers who are producing high value parts can quickly and easily develop probe path and integrate measurement cycles with machining cycles. Edgecam Inspect provides a comprehensive set of geometric features and constructions for measuring part characteristics such as webs, pockets, thickness and heights while the intuitive user interface and work flow allows the user to seamlessly transition from programming CAM cycles to measurement cycles.

Probe Calibration

Intelligent algorithms offer the user several choices when calibrating the probe for measurement tasks. Whether the application requires ultimate accuracy or speed and versatility, Edgecam Inspect has it all. Proprietary algorithms for managing probe radius compensation in every measurement scenario ensures the user of the integrity of the measurement results under multi axis measurement scenarios.

Programming Environment

Edgecam Inspect is the ideal environment for the CAM programmer to develop machining and probing cycles in one output file. Because the inspection workflow is tightly integrated into the CAM workspace, the transition from machining to measuring and back again is seamless. Whether stitching together probing and machining cycles for the purpose of automating part setup, managing stock for roughing operations or final part acceptance, Edgecam Inspect provides a single programming environment for managing all aspects of the machining process.

Interactive Results

Feedback is provided in the form of CMM style report templates along with graphical, color coded deviation information that is displayed on the CAD model. Edgecam Inspect also provides a useful results simulation capability that allows you to produce simulated results offline from the machine giving you the opportunity to tailor your reports without having to be connected to the machine. Flexible tolerancing options allow for full reporting of measured characteristics with a pass/fail evaluation.

Feature Set

Using our feature recognition tools, Edgecam Inspect helps optimize the programming process by allowing the user to program multiple features with a single mouse click. The Edgecam Inspect feature set provides the following features and part characteristics:

- Points and Point Arrays – evaluate simple axis deviation, material condition or profile with single or multiple point functions.

- Circular – Whether internal or external, Edgecam Inspect has the ability evaluate bores and/or bosses for size, location and/or form.

- Planar – Measure planar features with an option to report location and/or form.

- Web/Pocked – Measure thicknesses and widths with caliper like functions.

- Orientations – Report characteristics like angles between features or angle to an axis.
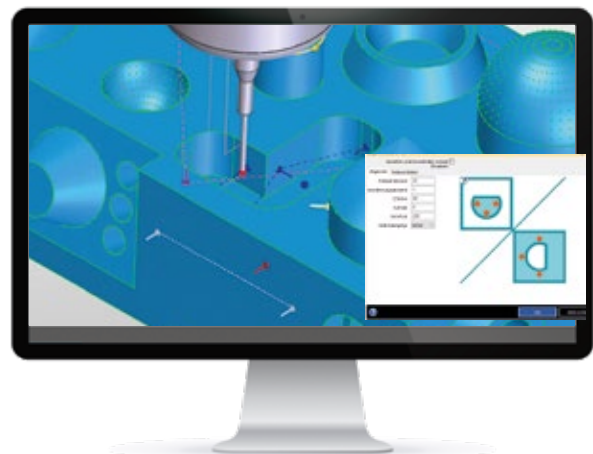
Component measurement on the machine

Features at a glance

- Robust probe calibration methods

- Support for any probe manufacturer

- Embedded probe cycles with machining cycles

- Informative results display

- Multiple report output formats

- Multiple machine control support

- Customizable

Features at a glance

- Robust probe calibration methods

- Support for any probe manufacturer

- Embedded probe cycles with machining cycles

- Informative results display

- Multiple report output formats

- Multiple machine control support

- Customizable

HEXAGON

Hexagon is a global leader in sensor, software and autonomous solutions.
We are putting data to work to boost efficiency, productivity,
and quality across industrial, manufacturing, infrastructure, safety,
and mobility applications.

Our technologies are shaping urban and production ecosystems
to become increasingly connected and autonomous – ensuring a scalable,
sustainable future.

Hexagon's Manufacturing Intelligence division provides solutions that
utilise data from design and engineering, production and metrology to make
manufacturing smarter. For more information, visit hexagonmi.com.

Learn more about Hexagon (Nasdaq Stockholm: HEXA B) at hexagon.com
and follow us @HexagonAB.