G ZERO FIVE

SOLID MACHINIST

Edgecam Solid Machinist is a powerful and seamlessly integrated CAM system for generating milling and turning machining strategies and NC code from solid models.

Automated Feature Recognition

Within Edgecam, the integrity of your design is maintained because the solid model is imported without translation. Edgecam Solid Machinist uses automatic feature recognition to interrogate the solid model and quickly identify machinable features. Edgecam then offers the user the most appropriate tooling and machining strategy to generate accurate toolpaths.

Intelligent manufacturing on a solid foundation

The associative link between Edgecam Solid Machinist and the original model ensures that even late design changes won't affect lead times. Anyone interested in tighter integration between engineering and manufacturing should consider Edgecam Solid Machinist.

Edgecam Strategy Manager

Used in conjunction with Edgecam Solid Machinist, Edgecam Strategy Manager provides fast and reliable machining of solid models, resulting in increased productivity. The application streamlines programming by capturing the knowledge from previously machined parts and applying it to new components – eliminating programming errors and delivering an unprecedented level of consistency and automation

Edgecam Part Modeler

Edgecam Part Modeler is a 3D modeling tool specifically designed for rapid construction or editing of solid models. It offers a cost effective solid modeling solution, complete with fully associative drafting capability. It is also the perfect complement to Edgecam Solid Machinist and the ideal choice for customers who are modelling components for production machining, or need to construct workholding systems such as chuck jaws or milling jigs and fixtures.

Solid Machinist has a wide range of innovative features which offer a fast and reliable route to manufacture.

KEY FEATURES:

Open files from all major CAD systems including :
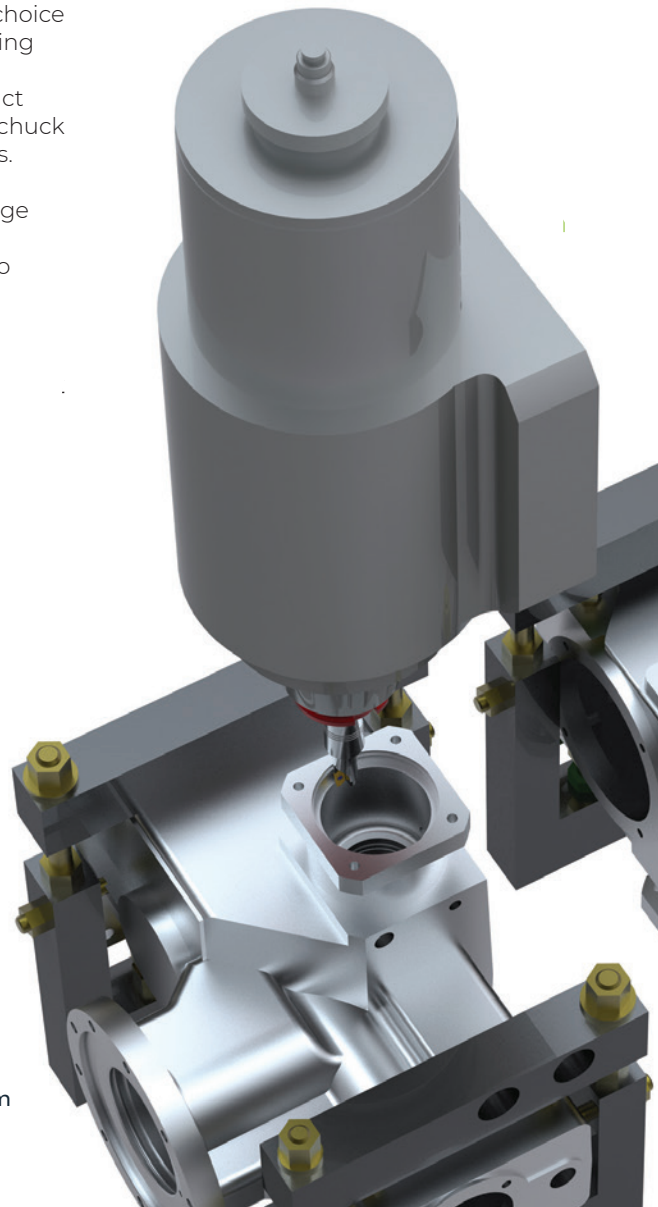
Autodesk Inventor

SolidWorks

Solid Edge

Unigraphics NX

CATIA V5

Pro/ENGINEER

G ZERO FIVE

4 & 5 AXIS MACHINING

Edgecam 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 a wide range of 4 and 5 axis strategies applicable to solid and surface geometry. Edgecam has now made this easier to use with the operational style interface yet still have all the control required for the higher demands such as:

- SWARF cutting for machining of variable taper walls

- 5 axis finishing across multiple surfaces with control over lead/ lag and side tilt angles 5 axis profile machining for slotting, de-flashing and trimming of sheet forms

- Full support for all common tool profiles, including lollipop cutters

- Easy-to-use machiningstrategies are geared to maximize productivity and quality

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

Intuitive, easy-to-use graphical user interface

Ideal for the rotary machining of automotive and aerospace components

Wide range of advanced options providing complete control of the tool

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

Interactive machining simulation
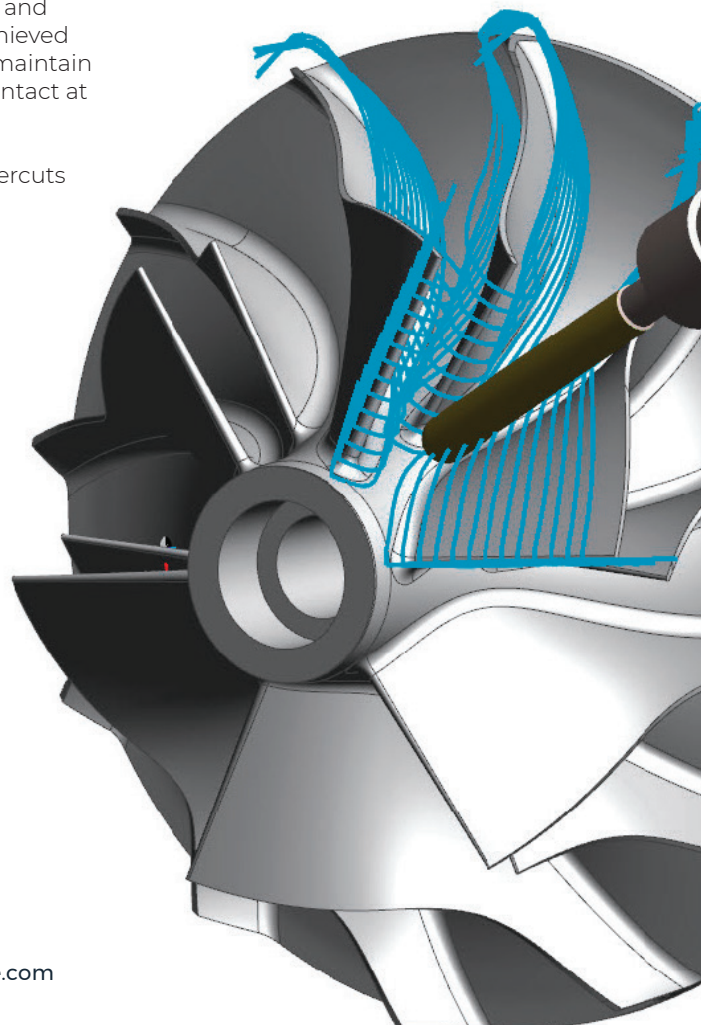
4 & 5 AXIS MACHINING

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
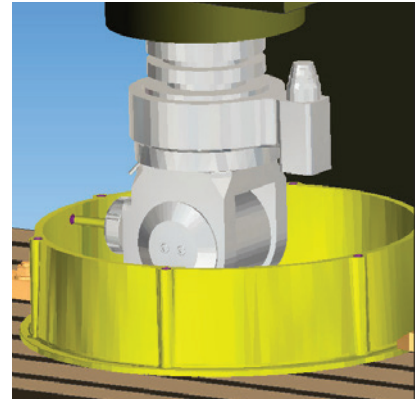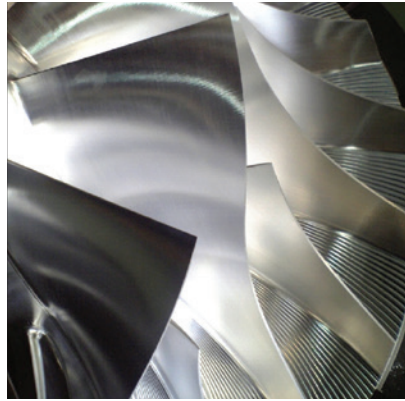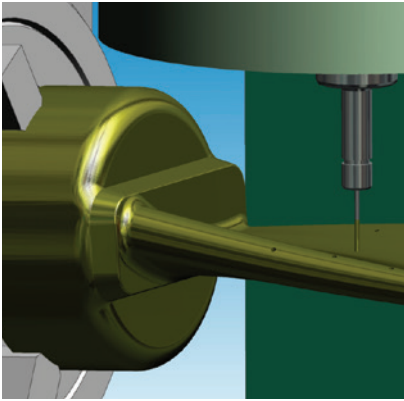
3 to 5 axis Tool Path Conversion

Using the knowledge of 3 axis machining methods, the standard Edgecam milling cycles and operations can be used on a component then apply the 5 axis tool path conversion. This produce 5 axis movement where required, ensure tool lengths are kept to a minimum and the cutting tool and holder tilted away from the component avoiding any collision. This methodology is an easy way to move into 5 axis programming technology.

Turn Milling

Use the 4th axis rotary attachment on the milling machine to produce a turned shaft using milling cutters rather than using a lathe for a partial operation. This process relies on the percentage engagement of the milling cutter while rotating the component which is made simple using Edgecam. The same principal is also used to produce cam forms.

Five Axis Finishing

Five axis finishing across multiple faces is similar to a parallel lace or scanning tool path but controls the tilt relative the surface which is driving the cycle.

SWARF Milling

Side Wall Axial Relief Feed. This is common practice when driving the side of the tool along a surface which tilts from side to side, this is in common practice on many aerospace parts. The tilt is control by the surface wall and the tool lift controlled by the base surface or bounding curve.

5 Axis Curve

The 5 axis curve option runs the tool once along a curve, with the tool axis parallel to the drive surface. This technique is very useful for de-flashing.

5 Axis Blend Between

An ideal method for machining fillet surfaces where a cut needs to start parallel to one curve but finish parallel to another.

Advanced 4 & 5 Axis Milling

The advanced 5 axis module provides total control of 4 and 5 axis tool paths using solids, surfaces or wireframe geometries. Advance 5 Axis has additional functionality beyond the standard 5 axis and is ideal for more complicated components such as blings, blisks and port machining where tool path plus entry and exit path control with is of a very high importance.

5 Axis Positioning

5 axis machines are also capable of 5 axis positioning, also called 3+2. This is where the component can be positioned using a combination of 3 axis linear movement with 2 axis rotary movement. A standard 3 Axis machining method can then be applied on to the component face orientatedtowards the spindle. These tool paths may also have the 3 to 5 axis conversion applied.

Tool Path control

5 axis tool paths can result in large movements of the machine tool from what can be a very small cut on the component. These movements can cause severe damage the part and machine. Edgecam provides methods for collision avoidance where the cutter and holder are checked for collision and the necessary tilts applied to move away from the potential collision area.

Inverse Time Feed is control

implemented to ensure the feed rate at the cutting tip is does not slow or dwell when small movements of the cutter produces a large movement of the machine tool. Inverse time feed allows a specified distance to move the tool during a specified time, this ensures the tool tip motion is correct and the machine tool movement will compensation to suit.

G ZERO FIVE

INSPECT

INTELLIGENT SOFTWARE FOR ON MACHINE PROBING

Edgecam Inspect combines industry leading metrology tools and know-how from Hexagon Manufacturing Intelligence with cutting edge manufacturing expertise from Vero Software.

Edgecam Inspect 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.
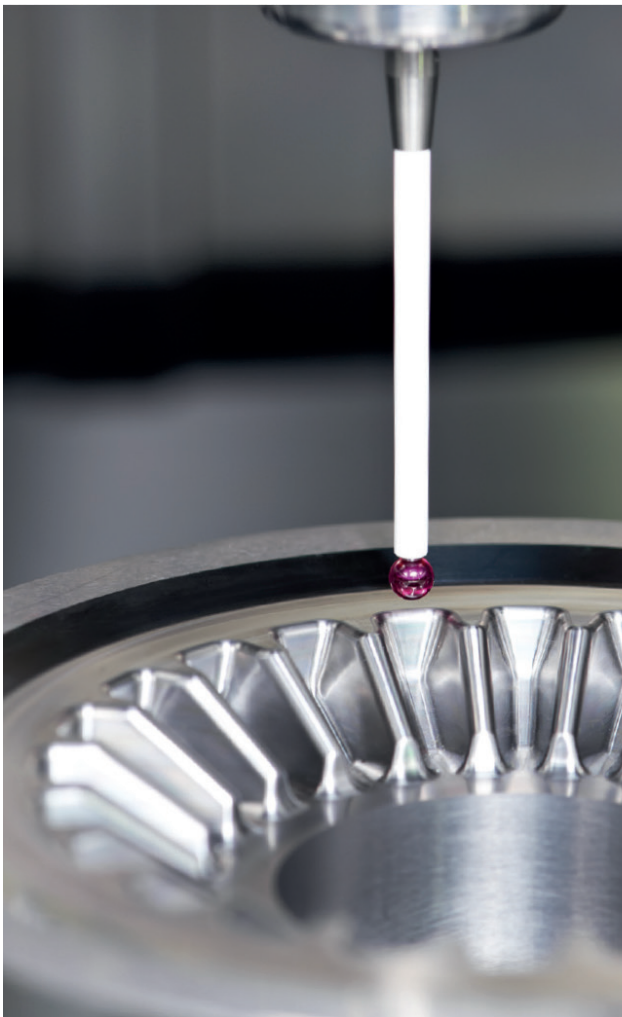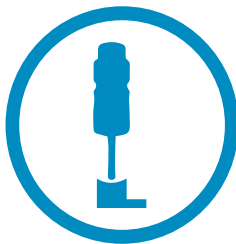
Edgecam Inspect offers the CAM user a robust, user friendly set of tools for creating measurement cycles, which can greatly reduce setup time and error and provide reliable in-process measurement results.
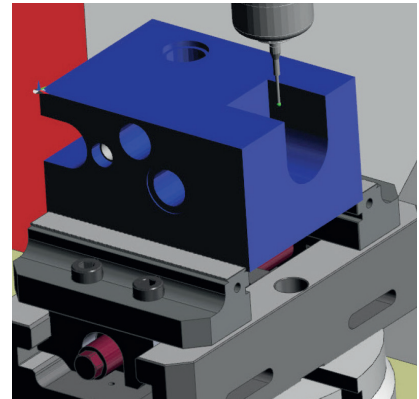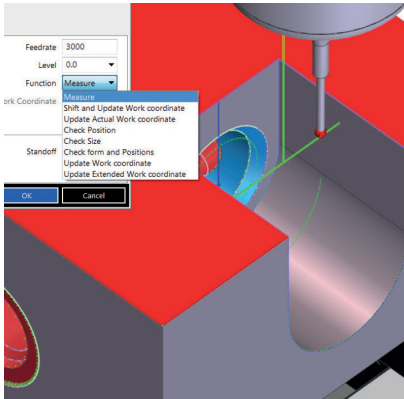
Process Validation

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.