ENG

CAM-TOOL

CAD/CAM System for Molds & Dies

System Requirements

OS	Windows 7 Professinal 64bit
	Windows 8.1 Pro 64bit
	Windows 10 Pro 64bit
CPU	Multi - Core Processor
MEM	8GB or more
HDD	80GB or more
Graphics	3D Acceleration OpenGL board (NVIDIA series)
	Memory 1GB or more

CGS C&G SYSTEMS INC.

www.cgsys.co.jp
✉ overseas@cgsys.co.jp

Facebook http://www.facebook.com/CGSYS
Youtube http://www.youtube.com/user/CGSYS0777

TOKYO HEAD OFFICE
  2-2-24 Higashishinagawa Shinagawa-ku, Tokyo, 140-0002, JAPAN
  TEL.: +81-3-6864-0781 FAX.: +81-3-6864-0786

KITAKYUSHU HEAD OFFICE
  1-5-15 Hikino,Yahatanishi-ku,Kitakyushu-City, Fukuoka, 806-0067, JAPAN
  TEL.: +81-93-642-4508 FAX.: +81-93-641-3615

INDONESIA TECHNICAL CENTER
  MENARA MULIA 27th Floor Jl.Jend.Gatot Subroto Kav.9-11 Karet Semanggi Setiabudi Jakarta 12930
  TEL.: +62-21-2953-9512 FAX.: +62-21-2953-9501

CGS NORTH AMERICA INC.(CANADA) www.camtool.com
  2160 Fasan Drive Oldcastle (Windsor) Ontario, Canada N0R 1L0
  TEL.: +1-519-737-6009 FAX.: +1-519-737-1647

CGS ASIA CO.,LTD.
  11th Fl., CTI Tower, 191/83 Ratchadapisek Rd., Klong Toey, Bangkok, 10100, Thailand
  TEL.: +66-661-9620 FAX.: +66-661-9621

Reseller

CAM-TOOL-E/V12/2017.03

CAD/CAM SYSTEM for Molds & Dies

"World's Reputable CAM Program creates Preeminent Molds & Dies"

CAM-TOOL is a five-axis-control-machining-center compatible,

high-end CAD/CAM system with a hybrid CAM engine (Polygon and Surface Calculation).

This Hybrid capability enables the direct machining to be applied to materials with high degree of hardness.

In addition, it allows high quality and highly efficient machining in other situations.
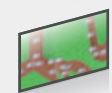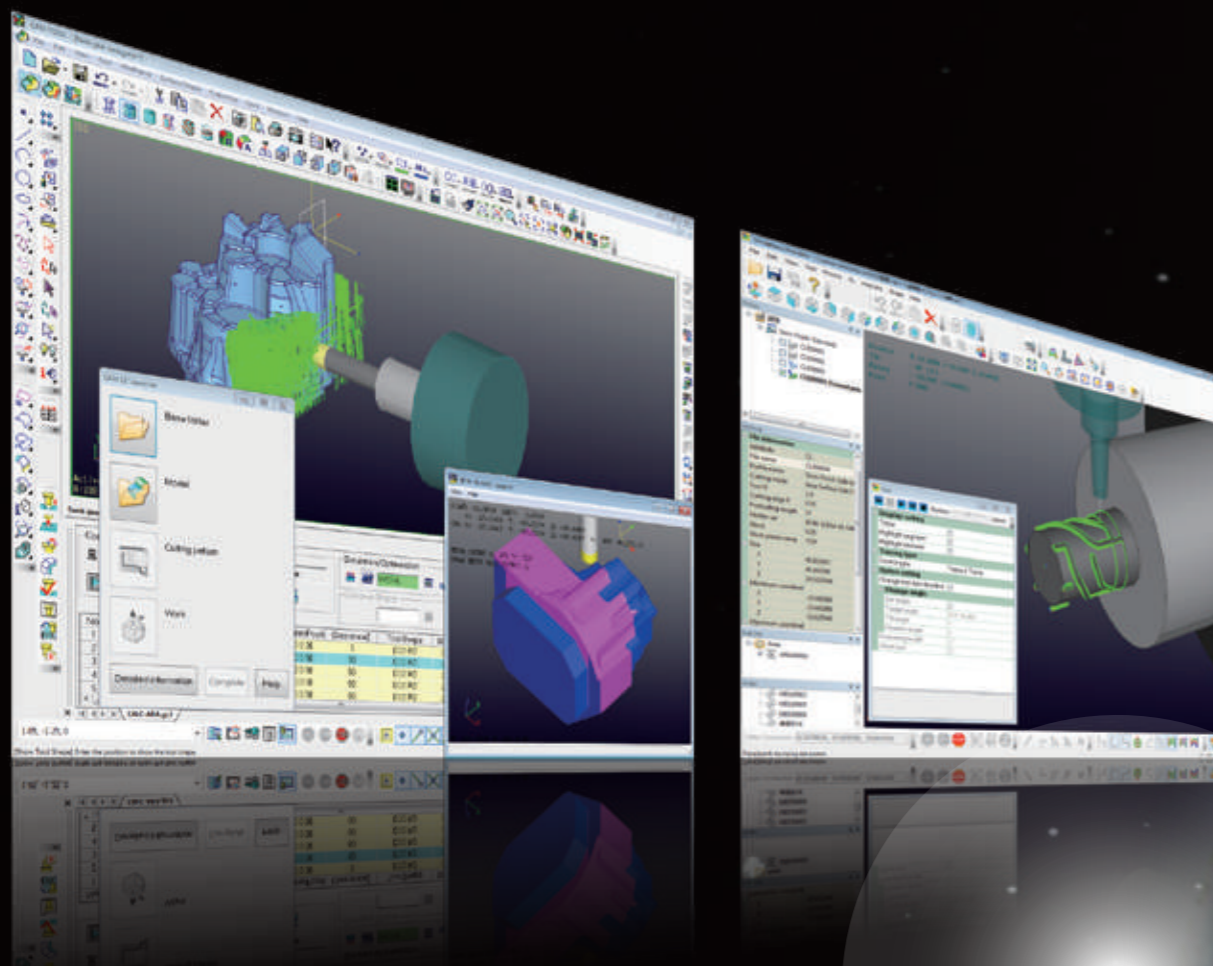
MODELING
Original Kernal surface modeler is excellent for modeling for Mold & Die work

HOLE & 2.5D CAM
Operation integrated into 3D CAM achieves efficient and safe machining.

3D CAM
Suppress Tool load fluctuation even in High speed direct cutting on high hardness steel.

5-AXIS CAM
Provides the precise 5-axis machining by the same operation as 3-axis.

SPEED & SAFETY
Additional Functions for productivity and safety.

OUTPUT & SOLUTION
Additional Solutions for automation and labor saving.

QUALITY

From Modeling to Simultaneous 5-Axis Machining,
CAM-TOOL strongly supports the Mold & Die Manufacturing Process
with tools that make your life easier and more profitable.

3D CAM

MODELING

2.5D CAM   HOLE

5 AXIS CAM

FUNCTIONS

From Modeling to Simultaneous 5-Axis Machining,
CAM-TOOL strongly supports the Mold & Die Manufacturing Process
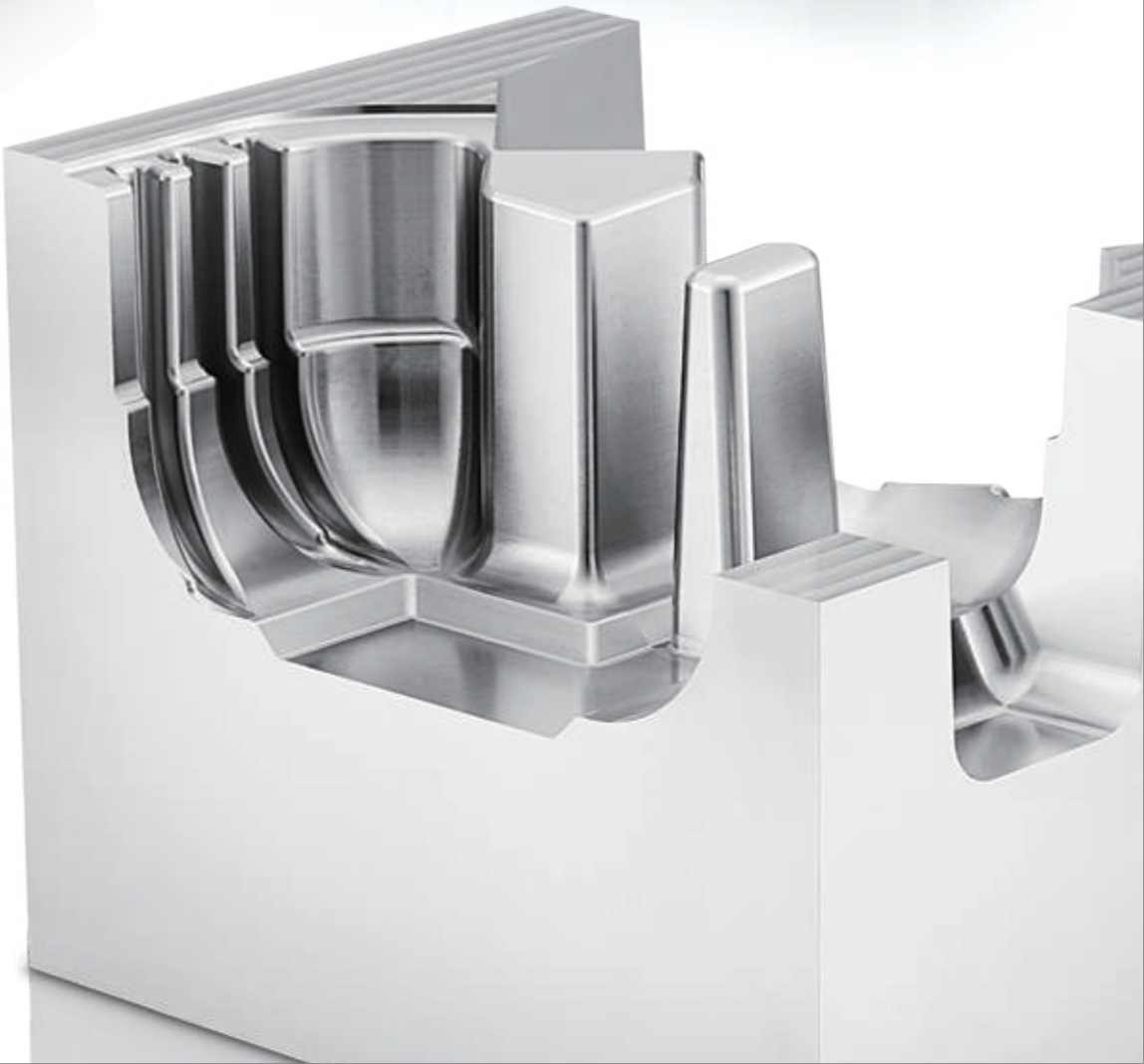with tools that make your life easier and more profitable.
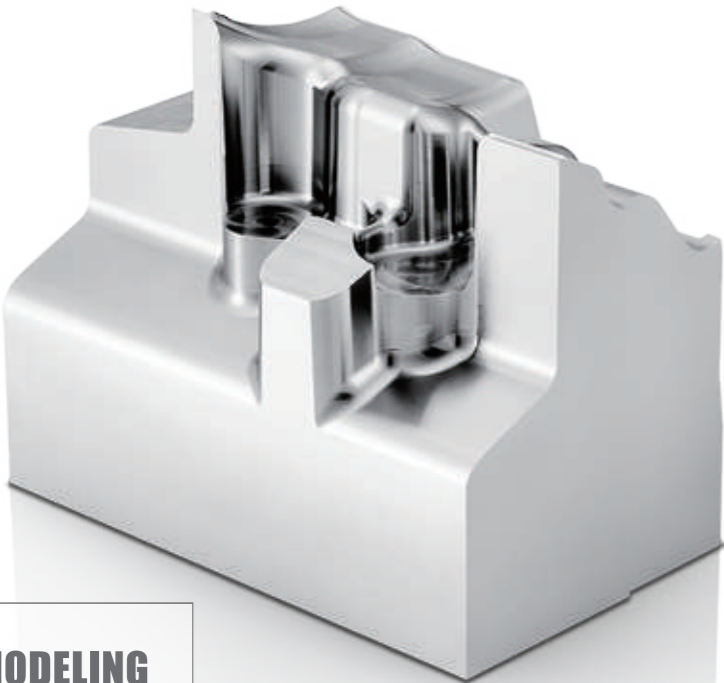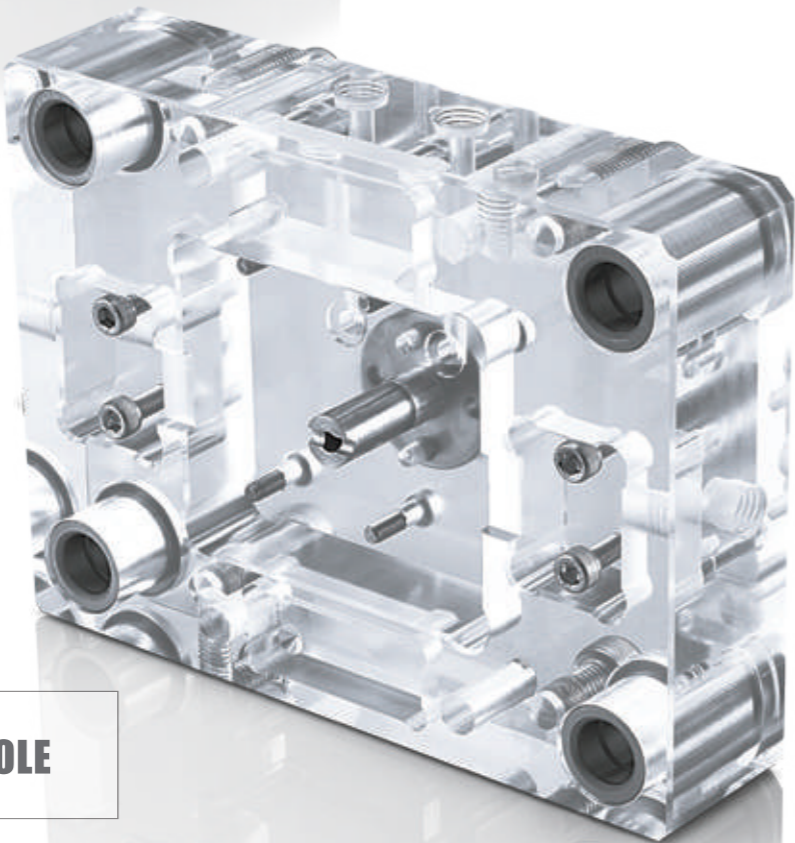
MODELING
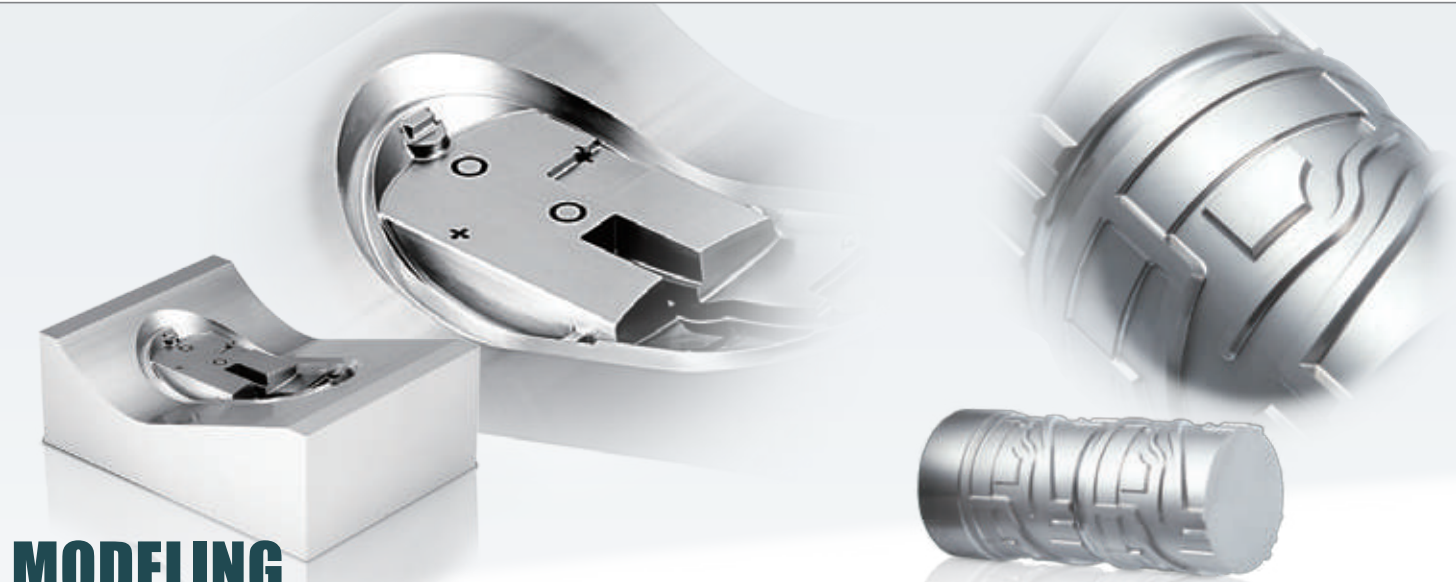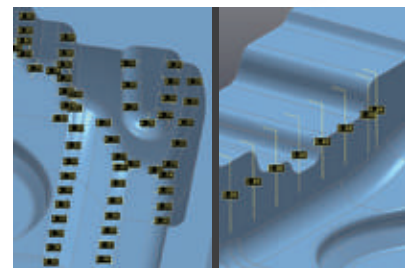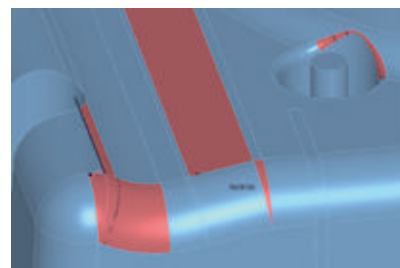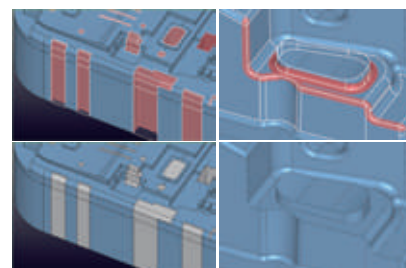### Original kernel surface modeler is excellent for modeling for Mold & Die work

Fillet Surface Radius Info / Draft Info
Checking easily for fillet radius of the shape & draft angle from the machining direction helps to decide the shank and protruding length of the cutting tool.

Check Surface
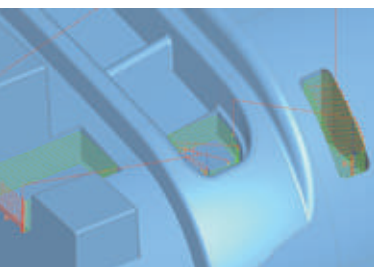Detects defective parts of a surface.
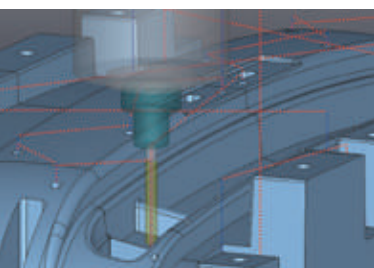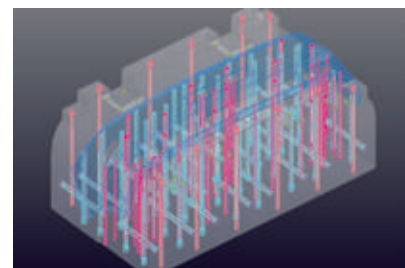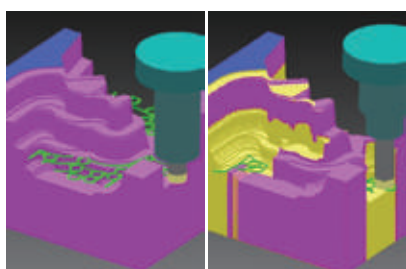
Surface Plus (Option)
Like a solid modeler, the blank spaces created after [Delete] or [Move] of the surfaces, can be automatically filled by [Extend/Trim]. This function contributes to reduce the modeling work process for mold design.

HOLE & 2.5D CAM
Operation integrated into 3D CAM achieves efficient and safe machining.

2.5D CAM
It's possible to set multiple sections at the contour of wire frame and define a pocket and an island shape. Section and plane parts which are machined by 2-axis/2.5-axis in the surface model are detected and cutter paths are generated.

[Cutting Modes]
Contour Cutting / Round of Cavity / Round of Core / Flat Cutting / 2-Dimensional Cutting
2.5D Rough Cutting / 2.5D Side Cutting / 2.5D Plane Cutting / 2.5D Re-machining / 2.5D Chamfering

HOLE
Hole feature can be defined by Entity, based not only on Point Circle but Arc/Curve/Surface.
Hole processes are controlled on the "Calculation Process List" the same as 2.5D/3D.
This provides safe Hole cutting because of interference checking for the remaining stock model after machining. It is also possible to improve efficiency of Hole cutting thanks to optimized CL.

[Cutting Modes]
Canned Cycle Drilling / Chamfering / G01 Drilling / Helical Tapping
Circular Hole-wall Cutting / Circular Hole-step Cutting
Circular Hole Rough Cutting / Circular Hole-bottom Finishing

3D CAM
Suppress Tool load fluctuation even in High speed direct cutting on high hardness steel.
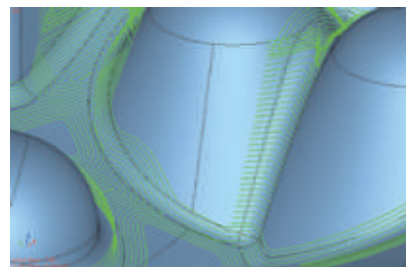
Roughing
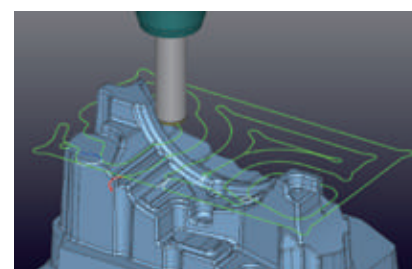
Z-level Rough Cutting with Multiple Tools
This cutting mode performs efficient Z-level offset roughing, recognizing user-defined work or stock model from the previous process, along with "Arc processing" and "Z step down for gentle slope", the "High Feed Tools" also can be used. Tool-paths recognizing the "Tip Shape" of the "High Feed" tool allows an accurate stock model to be generated, which enables the achievement of efficient machining with a constant stock remaining everywhere.
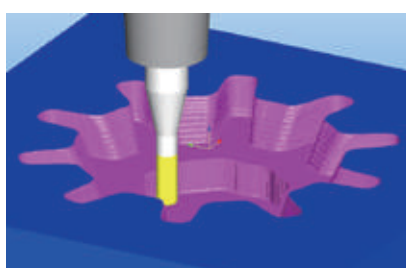
Z-level Re-machining
This is a function that performs Z-level offset cutting for the remainders detected by specifying the tool of the previous process. Ball end mill, radius end mill and flat end mill tools are available.
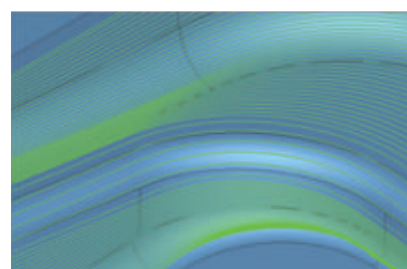
Corner Processing (Polygon)
This mode outputs "Along surface offset path" and "Contour line offset path" in the remainder regions such as groove parts or ridgeline parts where the tool of the previous process could not access. The type of the output cutter paths differs depending on the angle of inclination in the remainder region. "Along surface offset path" is output in a gently sloping area and "Contour line offset path" is output in a steeply sloping area.

Z-level Scanning-line Rough Cutting
This cutting strategy creates Z-level & bidirectional tool paths for roughing. Bidirectional scanning-line reduces the connecting move, which makes it possible to shorten the cutting time. Moreover, both core and cavity shapes can be cut, and for a composite shape, the system automatically detects cutting areas and outputs cutter paths.

Climb-up Roughing (Z-level High Efficiency Rough Cutting)
High-performance roughing with constant cutting-load can be achieved by using blade length of the tool efficiently. After roughing by specified Z pitch, climb-up roughing by smaller Z pitch is processed with variable XY step-over considering cutting-load.

Partial Cutting

Avoid Tooling Interference
The system automatically specifies non-interference areas with tooling shapes, then generates cutter paths. Because each path mills only uncut-area, machining data for each protrusion length can be created. This function corresponds to "Z-level Rough Cutting with Multiple Tools" and other cutting strategies for finishing.

Curve Control Along Surface
This mode cuts along 3 dimensional surfaces within the closed area enclosed by contours. U/V of surfaces don't affect the cutter paths. Air-cut can be reduced especially for a large metal mold.
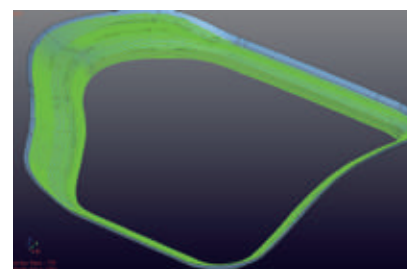
3D Offset Cutting
This is a finishing mode that outputs offset cutter paths along surfaces with a constant step. This mode outputs cutter paths for the entire shape in the specified area with a constant step and leaves uniform stock
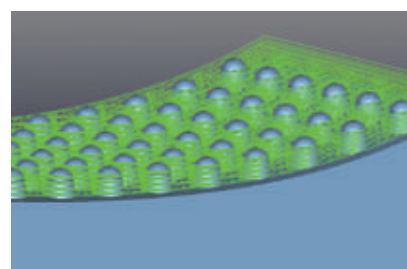
Base surface modes
"Base surface modes" in simultaneous 5axis CAM are new powerful cutting modes to create tool path on specified base surface.It achieves extreme quality surfac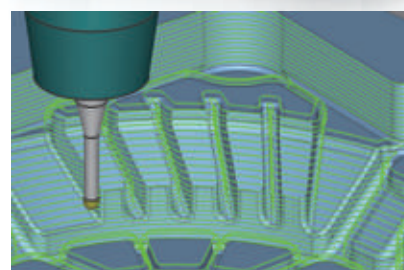e finish by the surface calculation characterizing CAM-TOOL. It can eliminate retracts substantially, so it will also prevents the deterioration of surface finishing, and furthermore, it reduces cutting time!
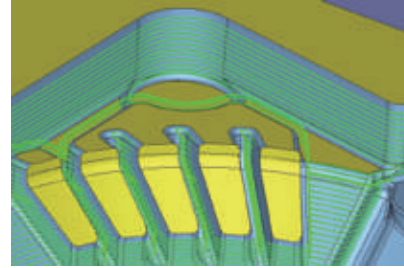
Finishing

Z-level Finishing
Performing Z-level cutting, which uses climb cutting, suitable for high-speed and high-precision machining, enables a high-quality surface finish required for mold manufacturing.
It is also possible to cut near-horizontal areas by adding "Offset cutting", or by using "Low Lying Processing" or "Scanning-Line area" together. With a variety of useful parameters and functions like "Spiral cutting", Z-level Finishing has been widely used from the semi-finishing process to the finishing process.
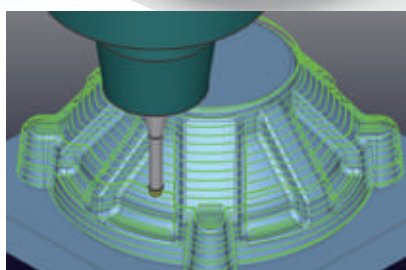
Spiral cutting
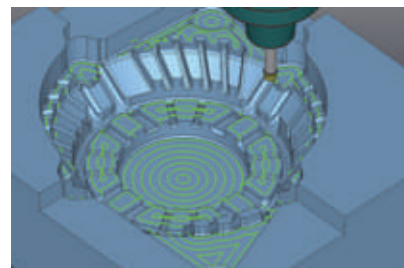
Horizontal Area Cutting
Horizontal areas of the specified shape are automatically detected and machined. The climbing cut is always performed and cutter paths traveling around (contour line) are output. The output NC data consists of G01 with simultaneous two axes (Simultaneous three axes for slant cutting). This mode is useful for the shape, containing many horizontal areas and requiring bottom machining with a flatnose tool.
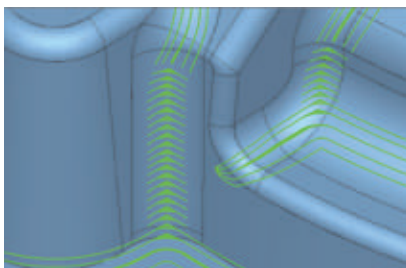
Re-machining
This is a function for re-processing the regions that were left uncut, using a smaller tool. Based on the radius of the previous tool, remainder regions are automatically detected and cutter paths are generated for those regions only. Cutting of remainder regions can be performed along ridgelines or perpendicularly to ridgelines, depending on the angle of the ridgeline.

Face not to machine / Different stock surfaces
"Surfaces avoiding to machine" or "Surfaces with different stock from others" can be specified at "Shape setting". It allows more efficient tool path for machining to be created easily.
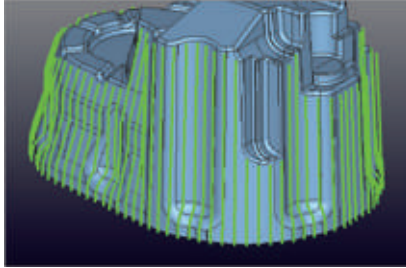
Box Cycle Cutting
Considering slide direction of the mold, this cutting mode creates vertical directional tool path on particular walls. Remaining cusps crossed slide direction can be reduced by vertical milling, which enables to shorten polishing time. Improving of dimensional accuracy also can be expected by repressing of cutting tool incline.

[Cutting Modes (Surface)]
- Z-level Rough Cutting
- Z-level High Efficiency Rough Cutting
- Z-level Finishing
- Scanning-line Area
- Low Lying Processing
- Surface Finishing
- Aiming Check
- CL + Check
- Re-machining
- Pencil Cutting
- Curve Cutting
- Curve Control Along Surface
- Horizontal Area Cutting
- Z-level Low Angle Finishing
- Box Cycle Cutting
- Base surface cutting
 Rough Cutting
  Side Finishing
  Bottom Finishing

[Cutting Modes (Polygon)]
- Z-level Rough Cutting with Multiple Tools
- Z-level Re-machining with Multiple Tools
- Scanning-line Cutting
- Z-level Scanning-line Rough Cutting
- Z-level Re-machining
- Corner Processing (Polygon)
- 3D Offset Cutting
- Z-level Finishing (Polygon)

5-AXIS CAM
Provides the precise 5-axis machining by the same operation as 3-axis.

3 + 2 Axis
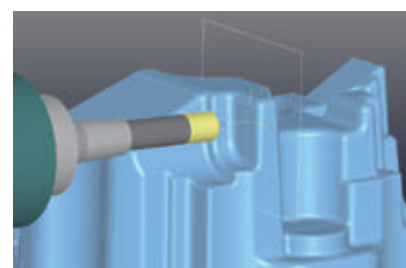
**Determination of Machining Direction**
Provides a variety of commands such as "Undercut Check", "Extract FlatPart", "Angle Shading", for determining machining direction that is needed for 3+2 axis machining. Specifying work plane for determining machining direction, plus saving and returning to the work plane setting can be done with a simple operation.
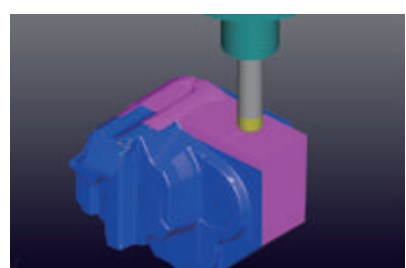
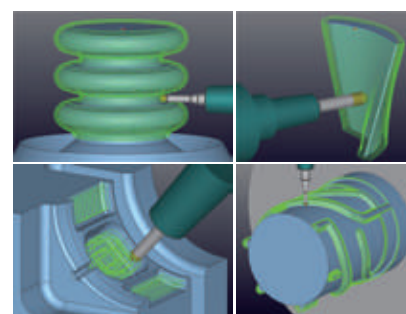**Optimization from Multiple Directions**
Optimization for 3+2 machining can be performed effectively after CAM calculation. Optimizations such as "Simulation" that includes displaying undercut areas and " Delete air-cut" by machining from multiple directions, make it possible to provide efficient and highly reliable machining data.
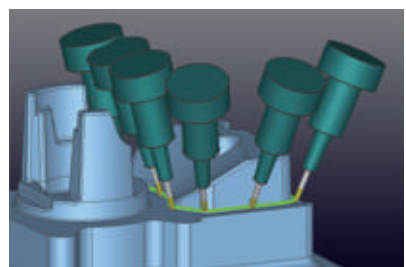
Simultaneous 5-Axis
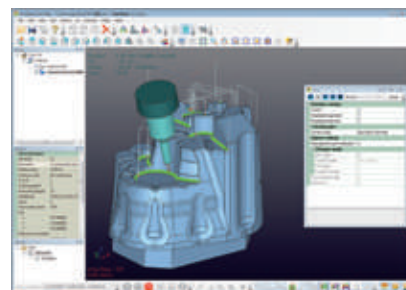
5-Axis conversion function
Creates 5-axis Simultaneous tool paths, by adding the information about tool direction to 3-axis tool paths. In converting tool paths, "smoothing control" adjusts the tilt axis and rotation axis to make the Machine Tool move more smoothly. It is also possible to have the tool tilt gradually before the point where the tool direction changes rapidly. These functions provide high-quality finished surfaces.

Automatic interference avoidance
Collision of Tool, Chuck and Holder with the shape can be avoided by controlling tilting axis automatically.
Change tool direction gradually to avoid drastic change of tool direction.
It minimizes the cutting mark.

Specialized cutting mode for simultaneous 5-axis machining.
To generate 5-axis tool-paths for undercut portions, a specialized cutting mode is also available, corresponding to various types of machining.
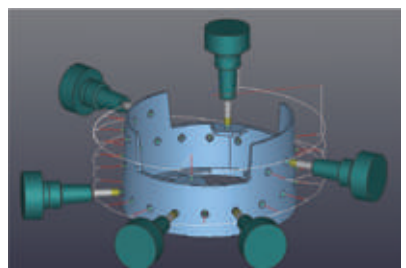
Multi-Axis Drilling
G01 cycled drilling data for multi- directional holes can be processed by single profile.
It is allowed to create safe and efficient drilling data, corresponding to "Check Interference", "Cutting Simulation" and " Machine Simulation",
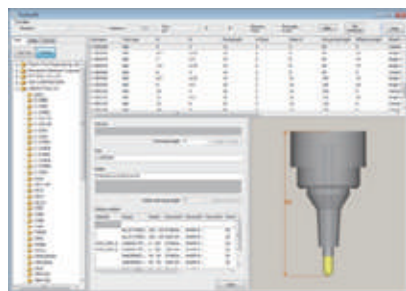
[Simultaneous 5-Axis Cutting Modes]
- Swarf Cutting
- Z-level Undercut Finishing
- Undercut Curve Control Along Surface
- Undercut Re-machining
- G01 Drilling - 5X

SPEED & SAFETY
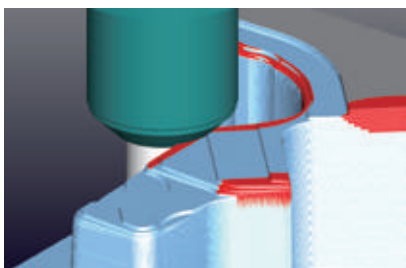### Additional Functions for productivity and safety.

Tooling Data base
Tooling DB manages Tool, Holder and its cutting conditions. It can define Tool shank shapes flexibly, such as multi-tapered or radius shank. Furthermore, users can download the catalog data of tool and holder information from our WEB site. Users can filter the tool with effective length and materials easily, and define a tool set which is combined with tool and holder.
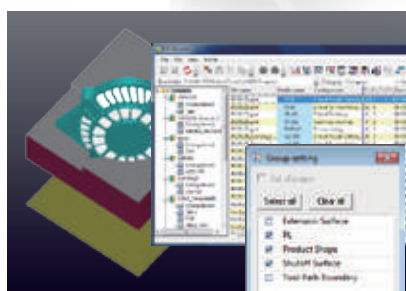
Surface Group / Template
Registering each machining surfaces (ex. PL Surface, Product Surface) as specified groups makes the shape setting operation for CAM easier.
It also automatically set shapes at once for plural machining processes when using "Template".

CAM EZ Launcher
"CAM EZ Launcher" accelerates to create machining processes, and contributes to prevent human errors, achieving simplification and standardization of CAM operation.
For 2.5D machining, "2.5 EZ Launcher" is also available.

Optimization / Cutting Animation
Optimization can be performed with Tool registered in DB and simulates the milling to show the condition of the work after machining and to also check the interference point. It's also possible to generate tool path more efficiently and safely because of "Delete air-cut". "Auto clearance", "Variable feed rate", etc. It is now possible to see the animated machining process after optimization.
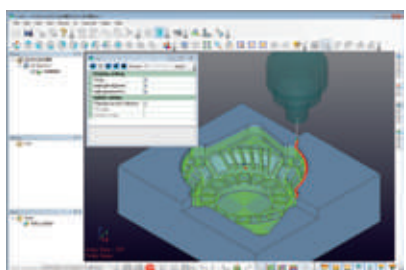
CL Editor
With a User Friendly GUI, it's possible to edit and check various types of CL.

[Main Function]
CL information / Trace / Move / Copy
Delete CL partially
Modify Approach
Change Approach Position

Machine Simulation (Option)
Possible to detect CL interference against the component parts of machine-tool, and verify stroke-range for each axis. Utilizing the same GUI as "Simultaneous 5-axis Editor" provides the consecutive and user-friendly environment for easy operation.
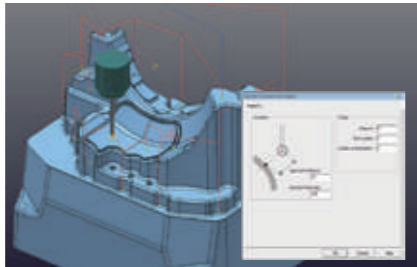
OUTPUT & SOLUTION
Additional Solutions for automation and labor saving.
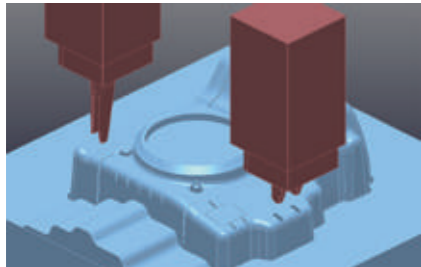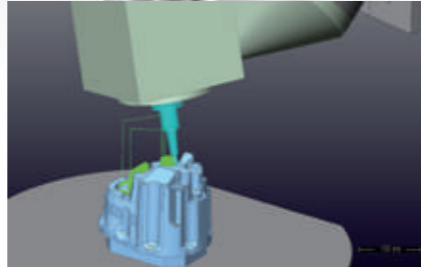
Create Electrode (Option)

This extracts electrode shape from the work model, and has a wide variety of functions, such as "Fill in Blank", "Move Electrode", "Fluctuation Offset", etc. It also corresponds to EPX format for EDM.

Vericut Interface (Option)

Vericut interface provides seamless operability from CAM-TOOL to Vericut. It starts from Vericut Icon in CAM-TOOL then exports data of tooling (tool and holder), NC program name, tool number and stock model information into Vericut automatically.

OM Inspect (Option)

This is the command for inspecting the machining precision directly on a machine. Accurate inspection is possible by using the same system for both creating machining data and setting points to inspect. Efficient inspection is possible, and outputting an inspection result into a file is also available.
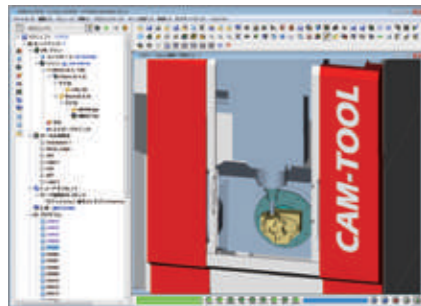
Process Sheet

NC Process Sheet can be output automatically when NC data is created. It is output as EXCEL format or WEB format.

Post processor

Using the NC Machine Parameter File format,the major CNC controller are available. NC data can be output by simple settings.

Various type of 5 axis machine tools such as Head rotation type, Table rotation type and Mixed type, are available.
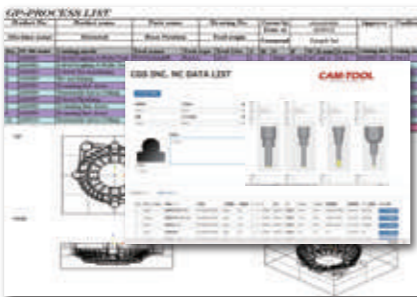
[5-axis positioning compliant]
Tool Center Positioning Control
Tilted working plane command

[5-axis Simultaneous compliant]
Tool Center Positioning Control

Calculate Machining Time

NC data is analyzed and machining time prediction which made the error span of time less than ±10% is performed.
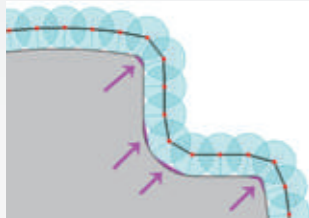
3D Mouse (Option)

3Dconnexion's 3D Mouse is supported. Operationality for view control, such as Zoom-in/out and Pan, will be improved significantly by less clicking.
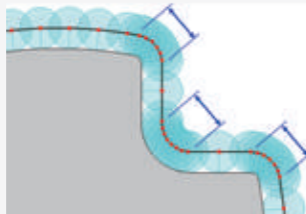
High-precision

"Gpcam" is the Heart and Soul of CAM-TOOL with its "Tool Path directly from the Surface Calculation" using its proprietary algorithm which allows it to achieve high-precision machining

In the case of "Triangulation Mesh Calculation" that many CAM systems adopt, the positional accuracy is degraded since the tool-path is generated along the Triangular shapes that are approximated from surfaces of the CAD model. In the case of the "Surface Calculated Tool Paths" of CAM-TOOL, there is no approximation of the surface shape, so it is possible to keep the accuracy of tool-path. Since there is no gap from CAD data, CAM-TOOL's distinguished algorithm calculates the tool contact points on the original surfaces.
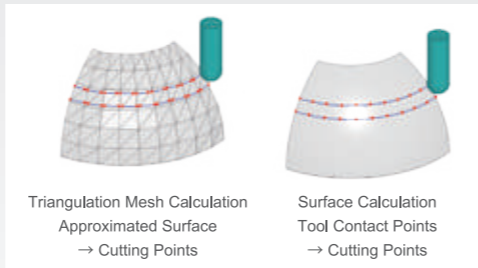
Triangulation Mesh Calculation
Approximated Surface
→ Cutting Points

Surface Calculation
Tool Contact Points
→ Cutting Points

When machining by the tool-path with component points created using "chordal deviation" (tolerance in some CAM Systems), there is the possibility of considerable over-cutting (gouging) and leaving remains (stock) at particular areas with large or discontinuous curvature, and that may cause deterioration in the accuracy of the net shape.

Considerable Over-cuttings and Remains

CAM-TOOL calculates relocation of component points of the tool-path using its special algorithm, corresponding to curvature of the shape. This calculation enables the proper relocation of component points along the shape. This also achieves an accurate tool-path that is a tremendous advantage over the other CAM Software for an accurate reproduction of the model shape.

Relocation of Component Points of Tool-path

Data Translate

High quality translation for most famous native CAD systems.

Import		
IGES	CATIA V5*	CADmeister*
STEP	Parasolid*	
DXF / DWG	NX*	
STL	SOLIDWORKS*	*Option

Export	
IGES	Parasolid*
STEP	SOLIDWORKS*
DXF	
CATIA V5*	*Option

System Module

It is the method of adding a module and change to the system configuration united with business is possible.

Use / Main Module	Viewer	Modeling	HOLE	2D Milling	3D Milling	3+2 Axis Milling	5 Axis Milling
Base	●	●	●	●	●	●	●
Modeler	—	●	○	○	○	●	●
Postprocessor	—	—	●	●	●	●	●
HOLE	—	—	●	○	○	○	○
2.5D CAM	—	—	—	●	○	○	○
3D CAM	—	—	—	—	●	●	●
CL Editor	—	—	—	○	○	○	●
3+2 Axis	—	—	—	—	—	●	●
Simultaneous 5 Axis	—	—	—	—	—	—	●

● Indispensable   ○ Selectable

Option Module	Surface Plus (Modeler Option)
	Create Electrode
	OM Inspect
	Machine Simulation
	Vericut Interface
	Translater
	CAM Processor
	Calculate Machining Time
Pakage / BaseZero	This is the functional limited package products of "BaseModule".

Modern Architecture

Native correspondence is carried out at the 64-bit OS environment. It was released from memory restrictions and comfortable work environment is realized also to large scale.

Native 64-bit Software
Multi-Core & Multi-Processor
Hyper-Threading

MachinigTechnology Center

CGS's Machining Technology Center is managed by technical experts, to support R&D activities. The Technology Center is engaged in the testing and verification of the CAM System's developments. In addition, evaluation of the latest cutting-materials, cutting-tools and bench marks for our customers are responsibilities of the Tech Center.It is also aggressively involved in technology exchanges with cutting-tool manufacturers,machine-tool manufacturers and educational institutions.

PRODUCTS

SOLIDWORKS Add-in CAM system

CG CAM-TOOL for SOLIDWORKS®

CERTIFIED Gold Product
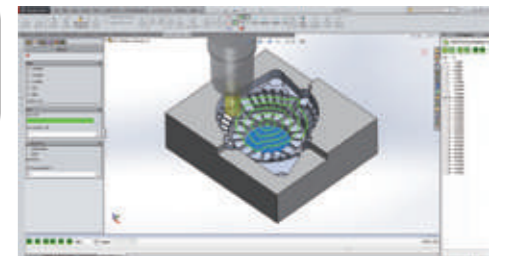
CG CAM-TOOL is a totally new CAM System. Its main strengths are an easy to use Interface and a simple Template capability that allows high precision/quality results for the Mold & Die Industry. It is also excellent for the machining of small to medium sized parts. It benefits from the fact, that at the heart is the CAM Engine of CAM-TOOL. Using CG CAM-TOOL will expedite your business startup even if your operators are inexperienced, as the Tool Database and Template capabilities will create a consistent output. *This software is a certificated Gold Partner Products of SOLIDWORKS.