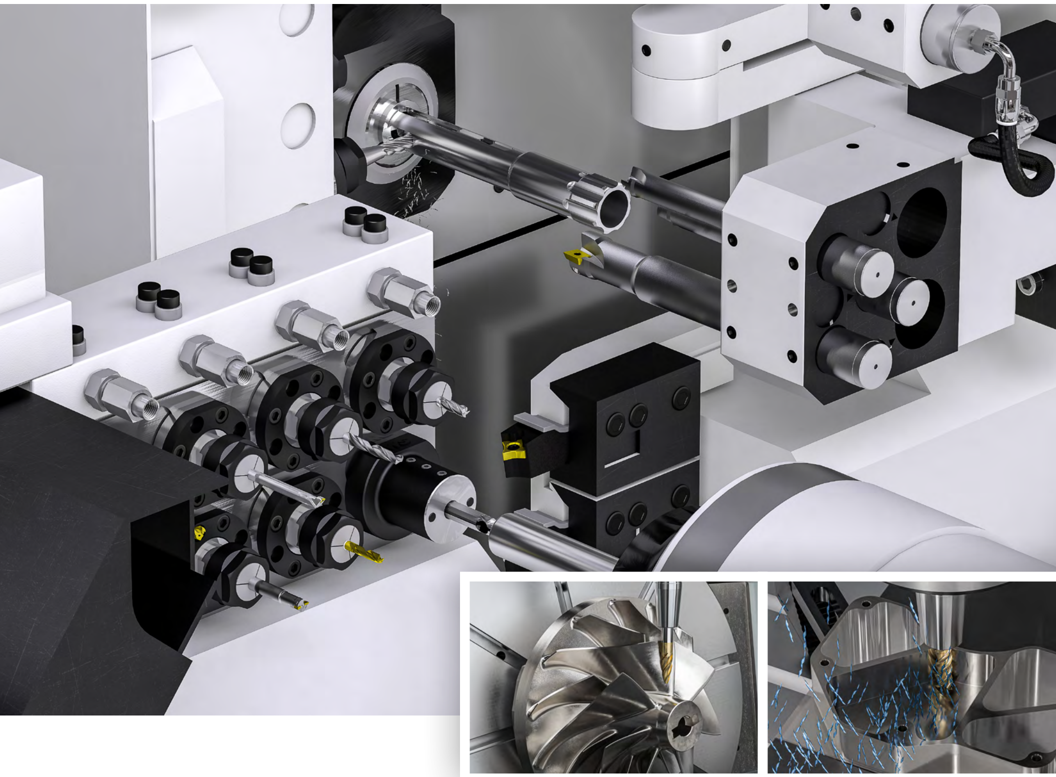
THE FUTURE OF CAM

InventorCAM 2022

InventorCAM + Inventor

The Complete Integrated Manufacturing Solution



Modules Overview

iMachining 2D & 3D | 2.5D Milling | HSS | HSM | Indexial Multi-Sided | Simultaneous 5-Axis | Turning | Advanced Mill-Turn | Swiss-Type | Solid Probe

iMachining® — The unique, revolutionary Milling technology — patent by SolidCAM

MILLTURN — SolidCAM Power for Multi-Channel Mill-Turn & Swiss-Type

InventorCAM — The Solid Platform for Manufacturing

Table of Contents

Document number: IVMOPENG1_2022

About this booklet

This Modules Overview booklet is designed to provide you with an overview of the available modules offered by InventorCAM. Following the information about each module, there are various InventorCAM parts that demonstrate the use and features of the available modules.

The parts are included with the installation of InventorCAM and are located on the hard drive of your computer (full path: **C:\Users\Public\Documents\SolidCAM\InventorCAM2022\User\Getting_Started_Examples\IV**).

Using the interactive features in this booklet

The various parts can be automatically opened using the interactive features in this booklet. To enable the correct loading of the parts, make sure they are always stored in the default installation location.

To use the interactive features in this booklet:

- Click the icon to load the example part.

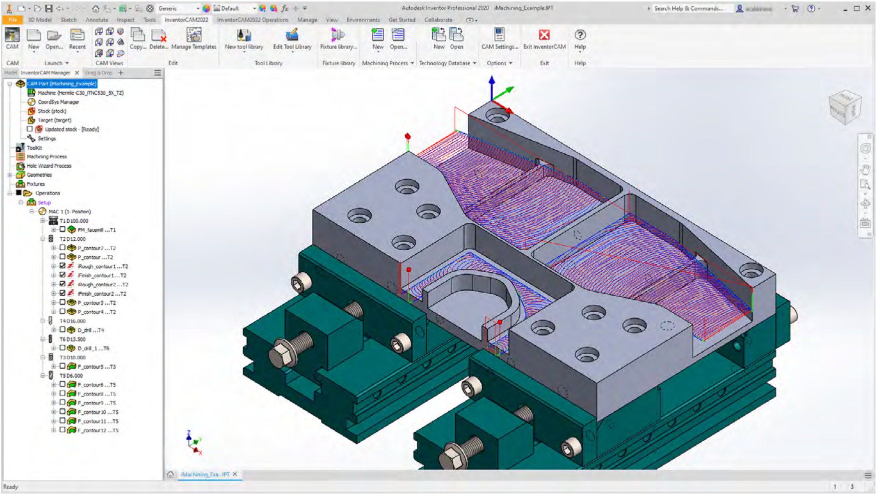
Adobe Reader

It is highly recommended to use Adobe Reader when viewing the PDF version of this booklet. If you do not already have Adobe Reader, it is a free PDF viewer that can be downloaded from Adobe's website via this link – https://get.adobe.com/reader/. By default, Adobe Reader allows the launching of external files and connecting to URLs, which is required for the interactive features to function properly. If you prefer using a different PDF viewer, it is important to make sure that your preferred Reader allows the executing of external commands.

To see Professor videos that cover all the features and uses of InventorCAM, visit our website at www.solidcam.com/professor.

For the most up-to-date PDF version of this Modules Overview booklet, visit our website at www.solidcam.com/subscription/documentation.

InventorCAM is the Complete Manufacturing Solution Integrated in Autodesk Inventor



InventorCAM is the Leading, ‘Best-in-Class’ CAM Suite for Profitable CNC Programming in Autodesk Inventor

InventorCAM, including the revolutionary iMachining, is seamlessly integrated in Autodesk Inventor with full tool path associativity to the Inventor model.

InventorCAM + Autodesk Inventor provides a powerful, easy-to-use integrated CAD/CAM solution that support the complete range of major manufacturing applications including iMachining 2D, iMachining 3D, 2.5D Milling, High Speed Surface Machining (HSS), 3D Mill High Speed Machining (HSR/HSM), Indexial Multiaxis Machining, Simultaneous 5-Axis Machining, Turning, Mill-Turn and even Solid Probe.

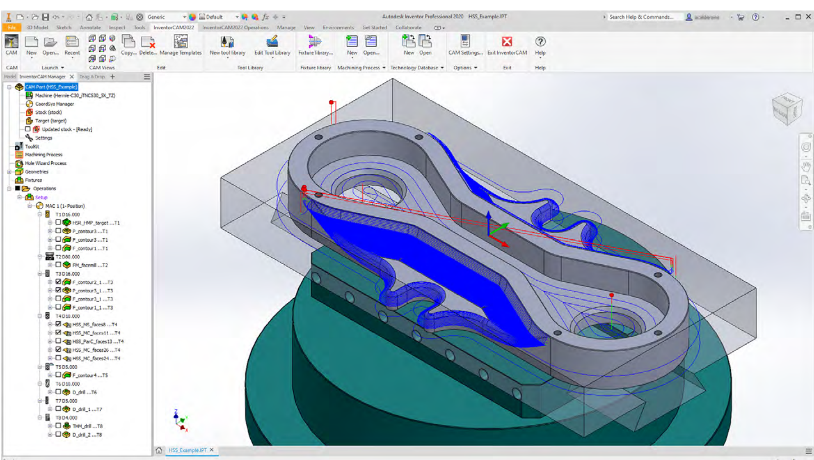
Major Benefits of Using InventorCAM in Autodesk Inventor:

- Autodesk Inventor look and feel through InventorCAM's totally seamless single window integration
- Full associativity means that tool paths automatically update when Inventor model changes
- InventorCAM is Autodesk Inventor certified since 2005
- InventorCAM works in the Autodesk Inventor assembly mode to define fixtures, tooling and vices
- InventorCAM + Autodesk Inventor is scalable with various packages for all your CNC-Machine types and applications

You Never Have to Leave the Autodesk Inventor Window!

With the single-window integration in Autodesk Inventor, all machining operations can be defined, calculated and verified without leaving the parametric Inventor assembly environment.

All 2D and 3D geometries used for machining are fully associative to the Autodesk Inventor design model. If you make any changes to your Inventor model, all of your CAM operations are updated automatically.

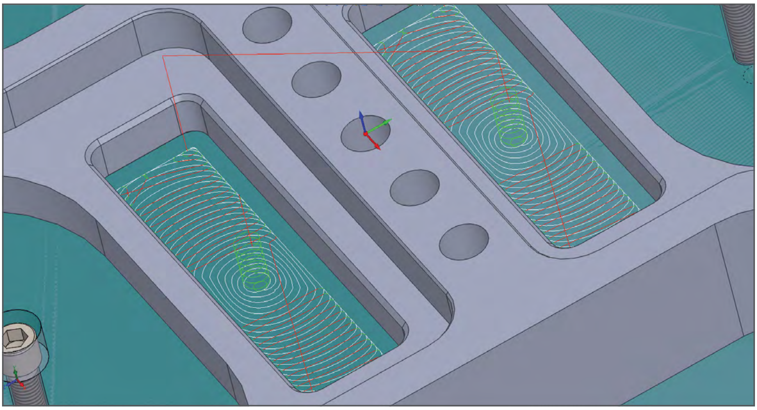


iMachining 2D

Imagine Putting the Knowledge and Expertise of Hundreds of CAM and CNC Masters Right in the Palm of your Hand – Experience the iMachining Technology Wizard & the iMachining Tool Path!

Patented iMachining: "Simply Amazing"

This is what customers, machine tool manufacturers and tooling companies alike say about iMachining. The revolutionary iMachining CAM module, fully integrated in Autodesk Inventor, will make you and your CNC-Machines more profitable and more competitive than ever before.



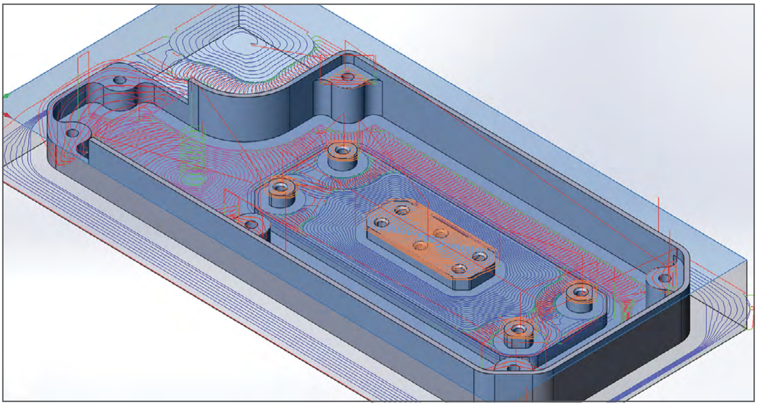
The Revolution in CNC Machining

- Saves 70% or more in CNC machining time
- Extends tool life dramatically
- Provides optimal feeds and speeds, taking into account many factors such as the tool path, stock and tool material as well as the machine specifications and limitations

iMachining provides unbelievable savings and increased efficiency in the programming of your CNC milling operations, translating into profits and success. All InventorCAM customers worldwide, who bought iMachining, are enjoying these immense savings!

Unique Technology Wizard

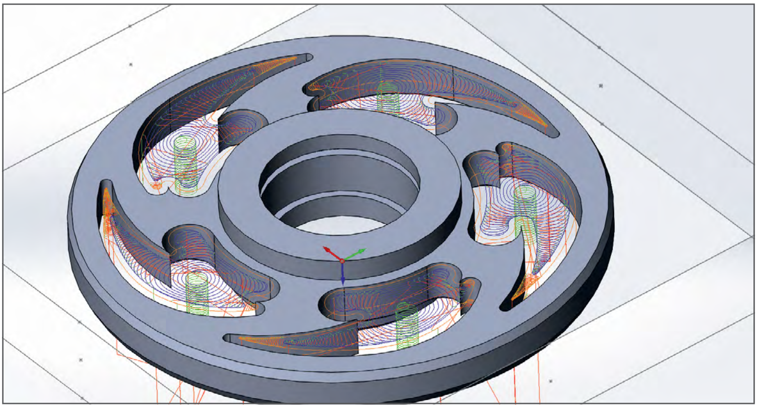
InventorCAM's iMachining has the exclusive patented iMachining Technology Wizard, the industry's first and only Wizard that automatically calculates the Cutting conditions for the iMachining tool path.



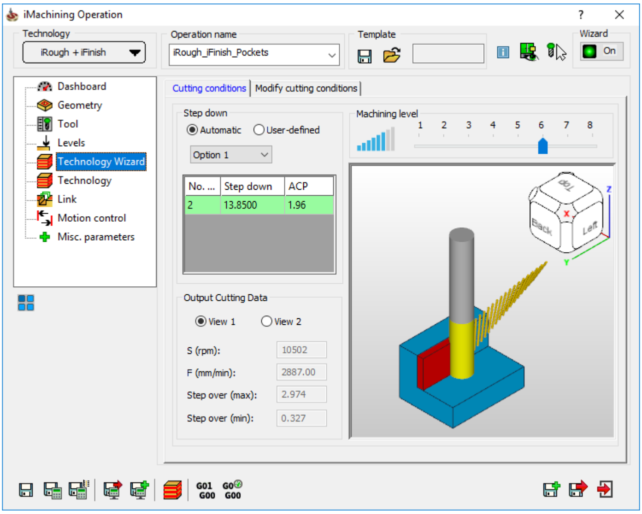
The Wizard provides optimal feeds and speeds, taking into account many factors such as the tool path, stock and tool material as well as the machine specifications and limitations.

Using "Controlled Step Over" technology, the iMachining technology strictly maintains the Cutting conditions set by the Wizard.

InventorCAM with iMachining is the only CAM system that takes out the guesswork from defining the Cutting conditions and automatically provides the optimum values for milling.



iMachining Technology Wizard + iMachining Tool Path is the Ultimate Solution!



InventorCAM's iMachining highlights:

- Increased productivity due to shorter cycles – time savings 70% and more!
- Dramatically increased tool life
- Unmatched hard materials machining
- Outstanding small tool performance
- 4-Axis and Mill-Turn iMachining
- Automatic, optimal feeds and speeds provided by the unique Technology Wizard
- High programming productivity
- Best user interface
- Shortest learning curve in the industry

“We have found all claims for iMachining to hold true in Dixon Surgical – incredible tool life, faster cycles, lighter cutting loads and protection of small cutters. The user interface is very clear and programming iMachining is faster than traditional machining strategies.”

Jay Dixon, Dixons Surgical, UK

“Every day we don‘t use InventorCAM iMachining, we are undoubtedly losing money!”

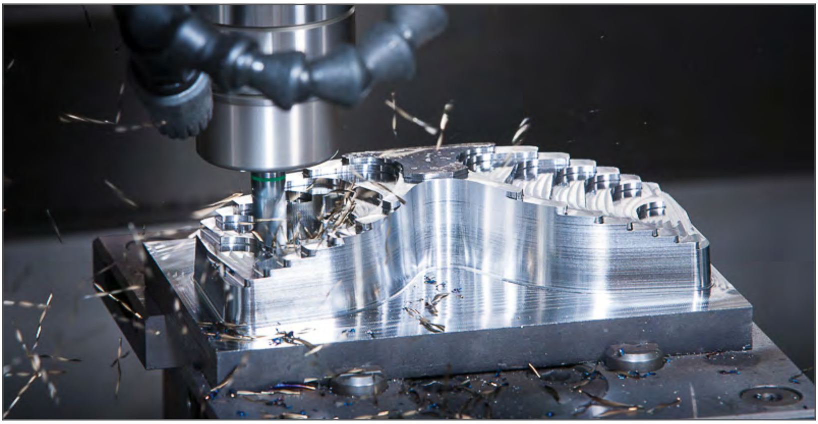
Jason Near, Rotary Airlock, USA

“With iMachining, even on low-performance machines, we can reach very high Material Removal Rates.”

Dreiling Maschinenbau GmbH, Germany

iMachining 3D

Uses Proven iMachining 2D & Technology Wizard Algorithms for Roughing, Rest Machining and Semi-Finishing of Molds, Complex 3D Parts and 3D Prismatic Parts



iMachining 3D provides amazing 3D machining results, regularly saving 70% and more in machining time, even reaching up to 90% savings.

iMachining 3D produces a complete, ready to run CNC program, with optimal cutting conditions achieved by the expert knowledge-based Technology Wizard, to rough and rest rough a entire 3D part, all in a single operation.

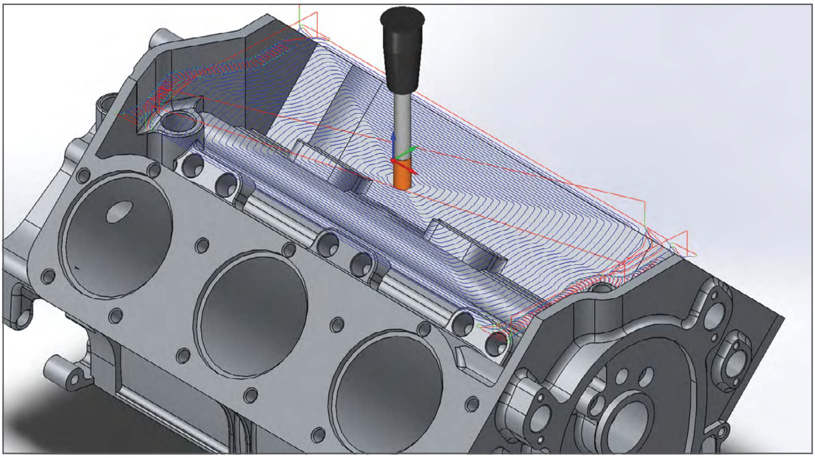
All the volumes requiring removal are subdivided into milling regions. By default, sophisticated analysis algorithms are then used to determine the optimal order of Step down (roughing) and Step-up (rest roughing) tool path passes. By achieving its unique local machining feature, iMachining 3D is able to eliminate almost all retracts and long position moves.

With its full-depth Step down, intelligent Step-up and its smart positioning, almost all retracts, long positioning and air cutting are eliminated to achieve the shortest optimal cycle times in the industry for the roughing, rest roughing and semi-finishing of molds, complex 3D parts and 3D prismatic parts.

iMachining 3D provides a complete machining solution for your 3D parts when combined with other InventorCAM Milling technologies for finishing.

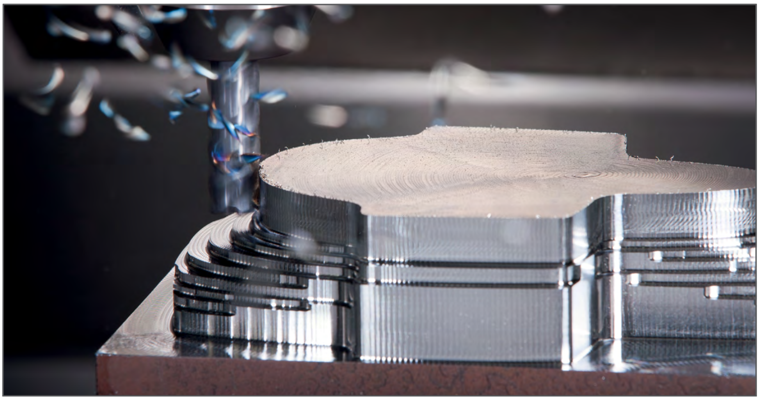
Exclusive iMachining 3D Features:

- Quick solid model geometry selection, eliminating the need to define chain geometries
- Optimized machining of each Z-level, using proven iMachining 2D tool path
- Deep roughing utilizes the whole length of the flute, shortening cycle time and increasing tool life
- Rest material machining in small upward steps, optimized for constant scallop height, further shortens cycle time
- Intelligent localized machining and optimal ordering eliminates almost all retracts and long positioning moves, producing the shortest cycle times in the industry
- A dynamically updated 3D Stock model eliminates all air cutting
- Tool path automatically adjusts to avoid contact between the holder and updated stock at every stage of the machining process



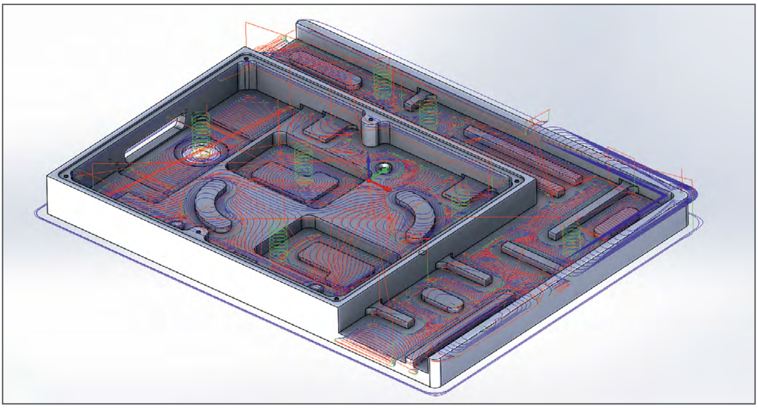
iMachining 3D for General Shaped Parts

iMachining 3D provides a complete machining solution for your general shaped 3D parts when combined with 3D HSM for finishing molds and complex 3D parts.



iMachining 3D for Prismatic Parts

You can also completely rough your 3D prismatic parts that include multiple pockets and islands on different Z-levels, all in a single operation. In this case, iMachining 3D provides a complete machining solution when combined with the 2.5D Milling and AFRM modules for finishing your 3D prismatic parts. You can also use iMachining 3D to rough your 3D prismatic parts entirely and then use iMachining 2D to finish the floors and walls.



iMachining 3D Saves Menes 85% in Cycle Time with Steel:

“I hope we get such results every day – we will have great savings in costs with iMachining 3D!”

Igor, Chief Programmer, Menes

iMachining 3D Cuts Machining Time on a Makino by 75% Against Another CAM System:

“iMachining 3D cut machining time from 4 hours, with a competitor’s CAM system, down to 58 minutes ... a 75% time savings!”

Galtronics, China

InventorCAM Customer A.P.A. on iMachining 2D & 3D, Mainly for Aluminum Machining:

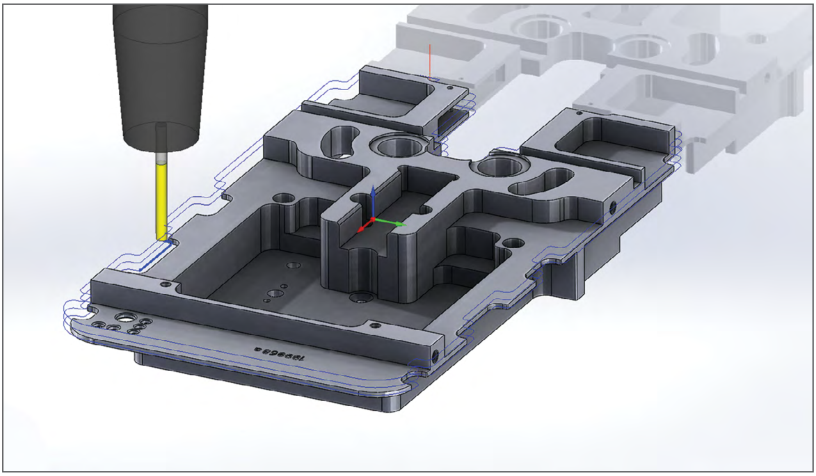
“Amazing, I have no words to describe my satisfaction with iMachining. I can’t even express how much time it saves us in Aluminum!”

David Franko, Owner, A.P.A.

2.5D Milling

The Most Powerful and Easiest to Create 2.5D CNC Milling Tool Paths: Interactive Control + Feature Recognition!

The most straightforward, easy-to-use interface, seamlessly integrated in Autodesk Inventor, combined with the latest tool path technology, provides the fastest, most powerful and easiest to create 2.5D CNC Milling tool paths.



Easily work on parts, assemblies and sketch geometry to define your machining operations. Quickly place fixtures and components for full visualization.

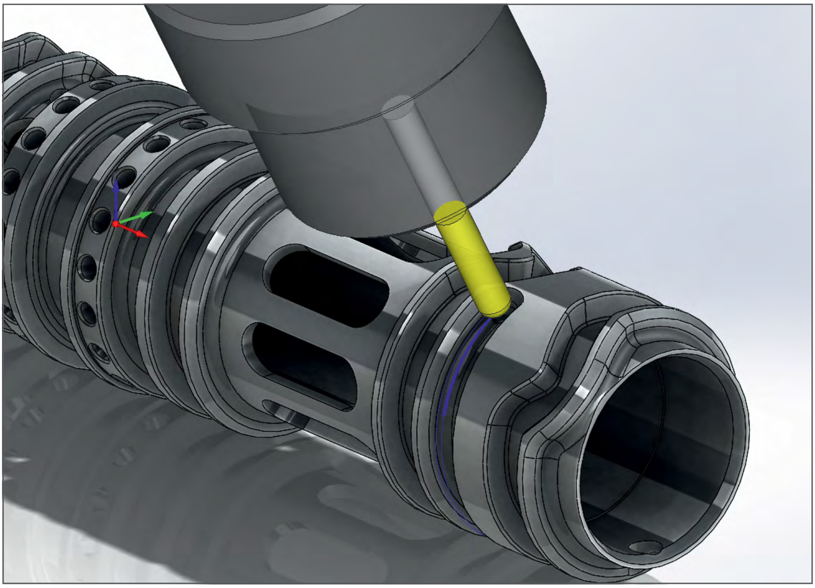
Best of Both Worlds: Complete Interactive Control + Feature Recognition

InventorCAM provides both interactive and automated 2.5D Milling operations on Autodesk Inventor models. Designed for both the novice and advanced user, InventorCAM offers the best of both worlds, with your choice of fully controlled selection of geometry, parameters and CNC programming strategies or with the Automatic Feature Recognition and Machining module for Pocket, Chamfer and Drill geometries.

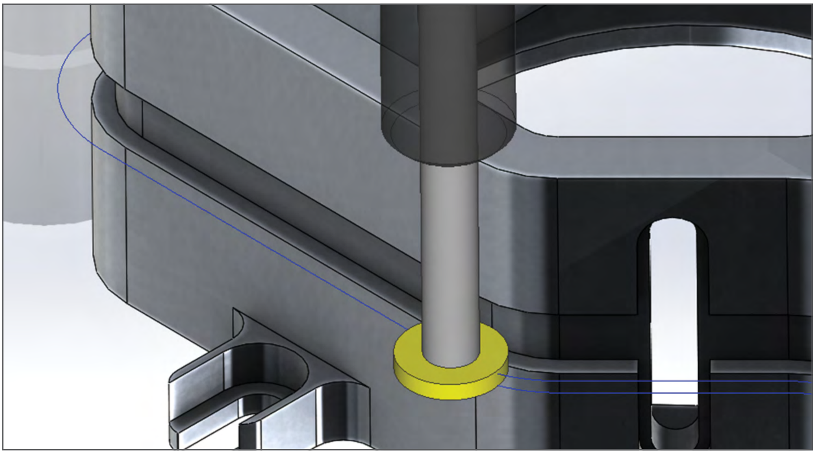
Interactive 2.5D Milling Operations

Aside from the standard 2.5D Milling strategies such as Profile, Pocket and Drilling operations, InventorCAM offers:

- Chain modification options (offsetting, trimming, extending etc.), enabling changes to geometry without changing the CAD model
- Automatic rest material machining to cut the material that remains after using larger tools
- Chamfer machining using the same geometry defined in Profile and Pocket operations
- Thread Milling operation for machining of standard internal and external threads
- Engraving of text on flat and wrapped faces and middle line engraving of multi-line text
- Contour 3D operation drives the tool along a 3D curve, cutting the model at different depths
- Machining of geometry wrapped around rotation axes, by transforming linear movements to rotary movements

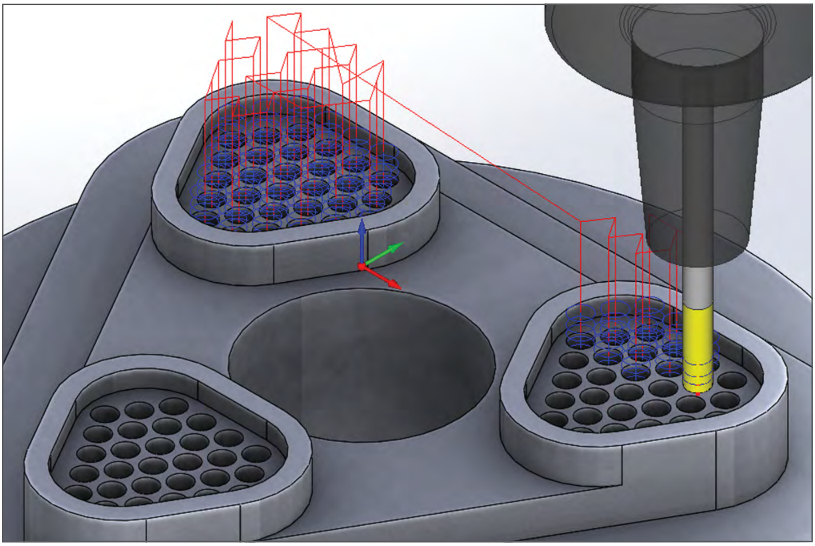


- Special operation for machining of side slots with undercut using a T-slot tool



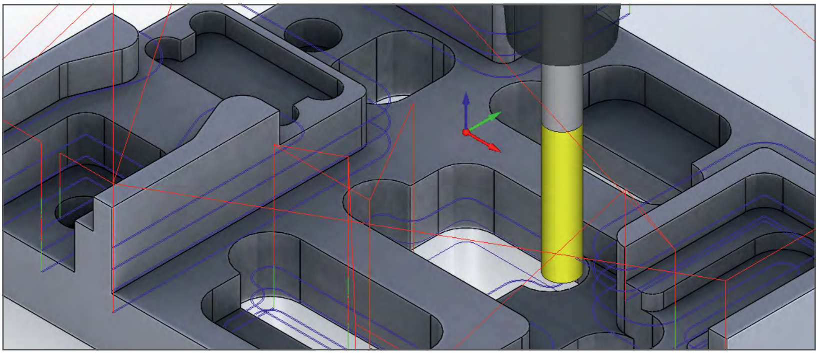
Drill Recognition

Automatic recognition and grouping of holes from the CAD model with the option to modify resulting geometry. A single Drill Recognition operation can machine groups of holes that have various levels and cutting depths.



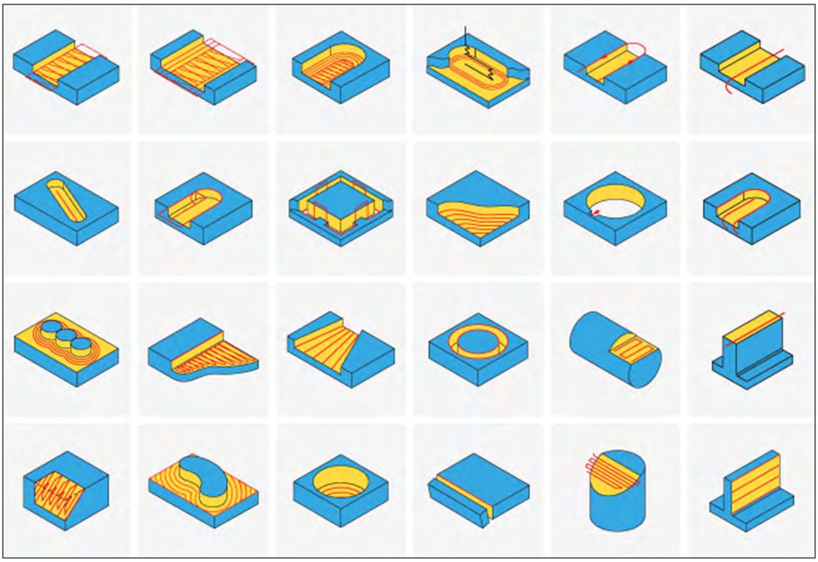
Pocket Recognition

InventorCAM's powerful pocketing strategy is taken to the next level with the automatic recognition of all pockets from the CAD model. All strategies and options of the standard Pocket operation are available, with the capability to machine pockets that have various levels and cutting depths.



Toolbox Cycles

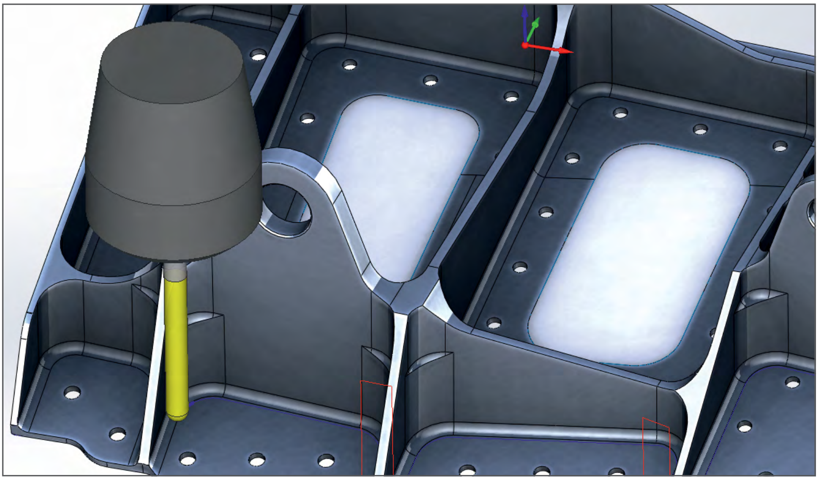
The useful and convenient Toolbox Cycles provide specialized sub-operations for slots, corners, bosses, ruled surfaces, etc.



High Speed Surface Machining (HSS)

High Speed Surface Machining of Localized Surfaces is an Important Module for Every Machine Shop!

InventorCAM HSS is the High Speed Surface Machining module for smooth and powerful machining of localized surface areas, which even includes undercuts. It provides easy selection of the surfaces to be machined, with no need to define boundaries. It supports both standard and shaped tools.



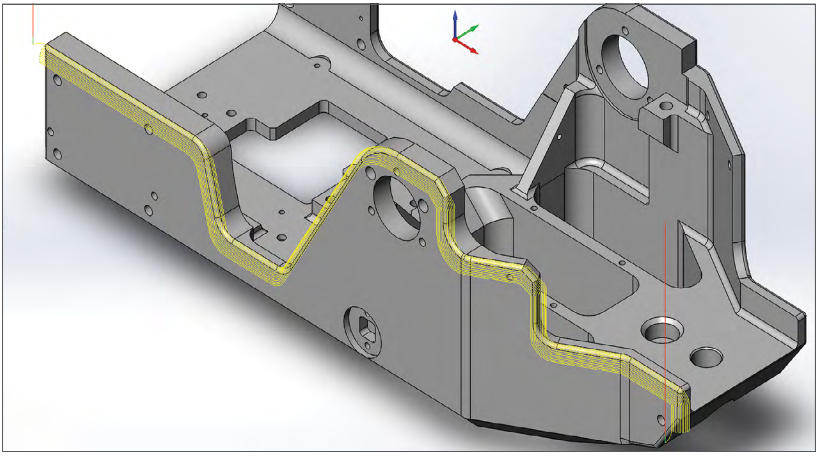
Powerful Surface Machining Strategies for Smooth, Gouge-Free and Optimal Tool Paths

The InventorCAM HSS module provides numerous surface machining strategies, that produce an efficient, smooth, gouge-free and optimal tool path to finish the selected surfaces.

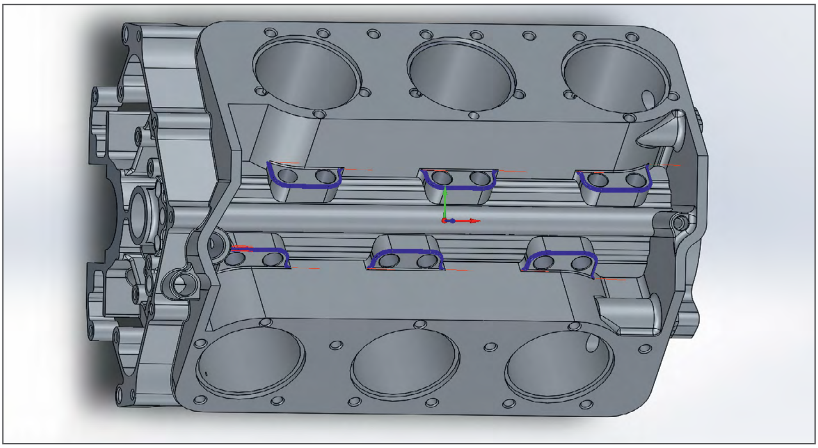
HSS provides special tool path linking options, generating smooth and tangential lead-ins and lead-outs. The linking moves between the tool paths can be controlled by the user to avoid holes and slots, without the need for modifying the surface of the model. Retracts can also be performed to any major plane.

Total Tool Control to Machine Only the Areas You Choose

HSS is the CAM module that takes your 2.5D Milling way beyond profiles, pockets and faces, providing a 3D machining capability by driving along specific surfaces on both general shaped 3D parts and 3D prismatic parts.



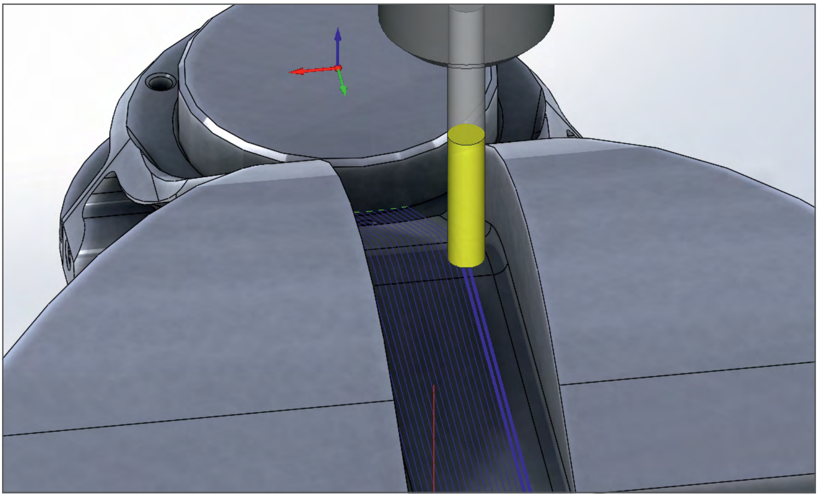
The HSS tool path is focused on single or multiple surfaces and excels in creating a flowing tool path on a group of surfaces that make up a complex 3D shape such as a fillet.



Experience total tool control to machine only the areas you choose, without the need of constraint boundaries or construction geometry.

Advanced Gouge Control for Holder, Arbor and Tool

Complete gouge control is available for holders, arbors and tools. Adjoining check surfaces that are to be avoided can be selected. Several retract strategies are available with full user control.



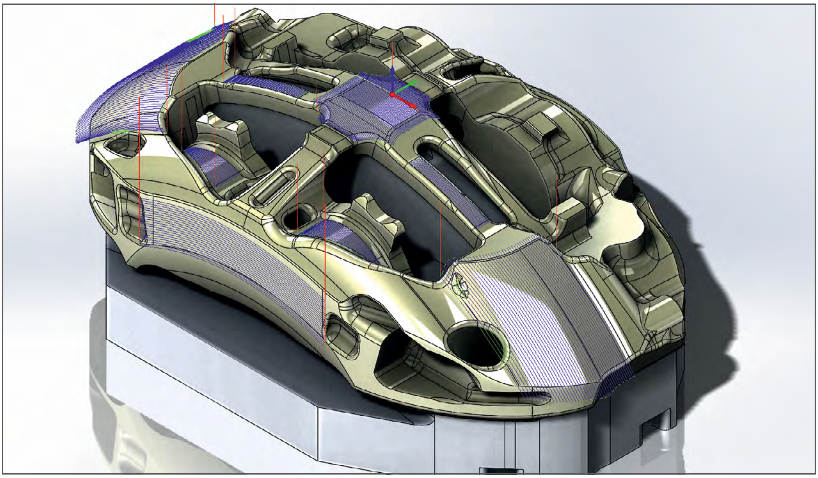
Advanced Linking

Total freedom to control tool entry and tool exit motion for avoiding surface modifications. Tool paths can be extended or trimmed, gaps and holes can be jumped, and you can choose from multiple lead-in and lead-out options.

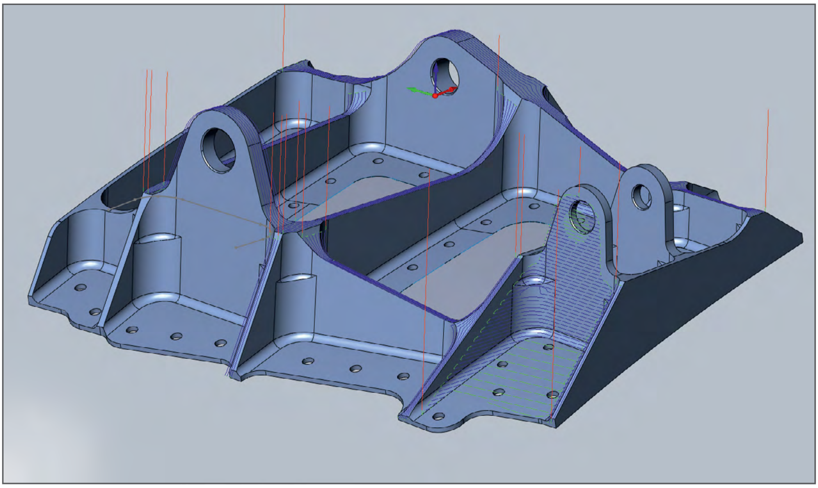
Handling Undercuts in HSS

Use Tapered, Lollipop, or T-Slot tools for undercuts or other types of geometry that are difficult to machine.

Important Module for Every Machine Shop



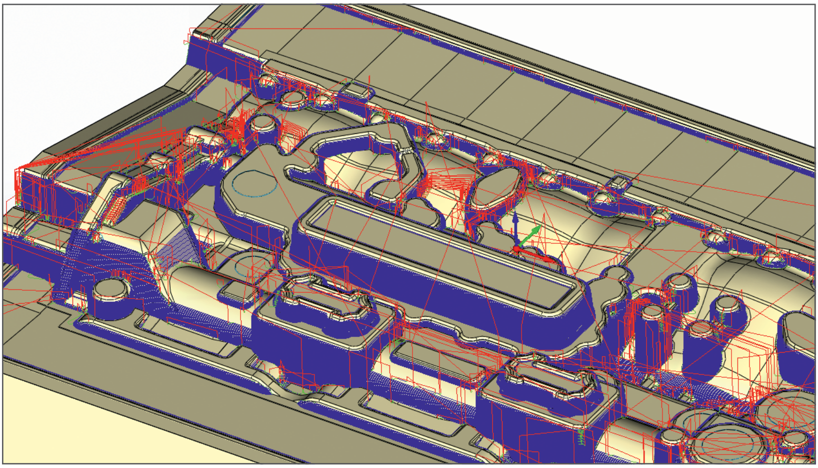
The advantages of the InventorCAM HSS module translates to significantly increased surface quality. It is a critical add-on for every machine shop for the machining of all types of parts.



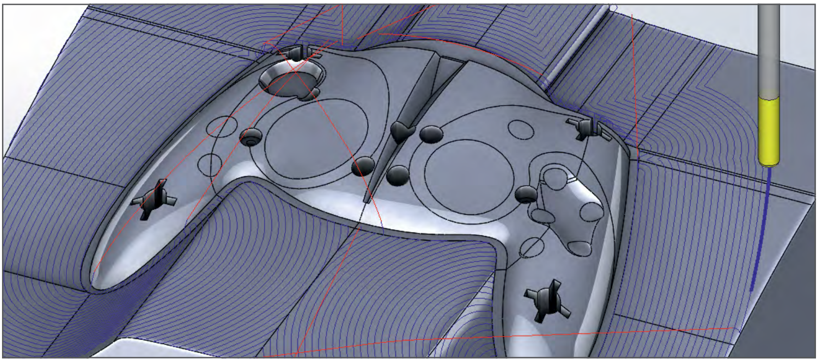
3D Mill High Speed Machining (HSR/HSM)

Powerful Roughing and Finest Finish Tool Paths Available for 3D Machining!

Experience 3D machining taken to an entirely new level of smoothness, efficiency and smart machining, with the finest finish tool paths available anywhere for 3D machining.



The InventorCAM HSR/HSM module is the very powerful and market-proven 3D High Speed Machining module for complex 3D parts, aerospace parts, molds, tools and dies.



HSR/HSM offers unique machining and linking strategies for generating 3D high speed tool paths. It smoothes the paths of both cutting moves and retracts, wherever possible, to maintain a continuous machine tool motion, which is an absolute essential requirement for maintaining higher feed rates and for eliminating dwelling.

HSR – High Speed Roughing

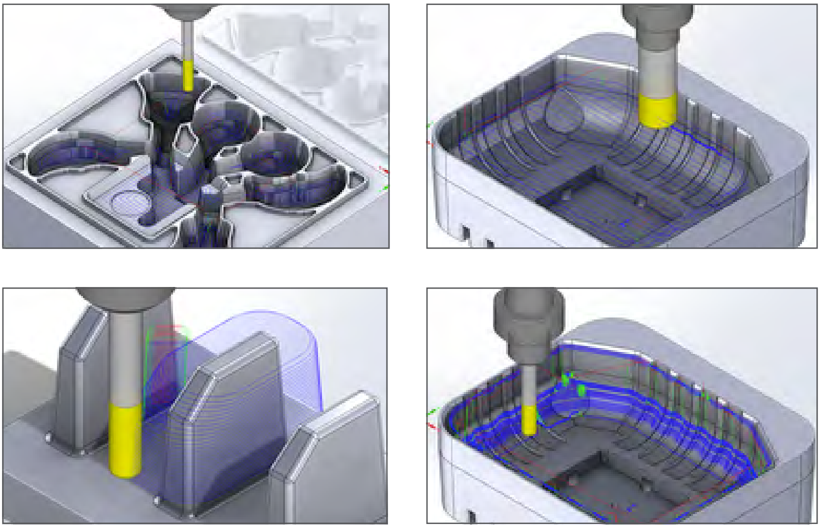
InventorCAM HSR provides powerful high speed roughing strategies such as HM, Contour, Hatch, Hybrid Rib roughing and Rest roughing.

HSM – High Speed Finishing

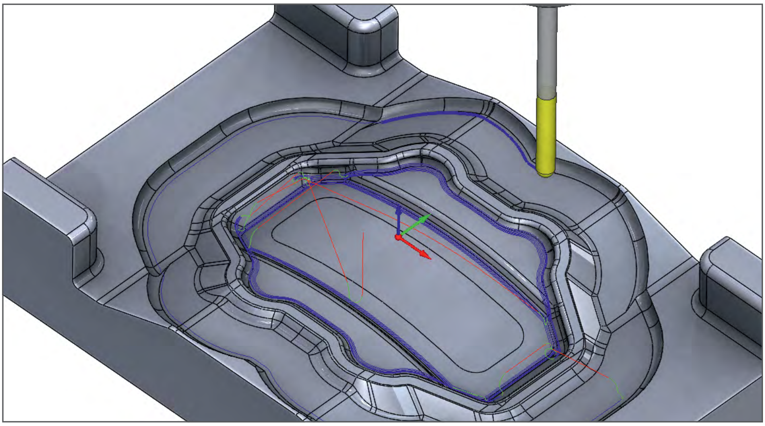
InventorCAM HSM keeps retracts to high Z-levels at a minimum. Angled where possible and smoothed by arcs, retracts do not go any higher than necessary, minimizing air cutting and reducing machining time.

The result of HSM is an efficient and smooth tool path that translates to increased surface quality as well as less wear and longer life for your cutting tools and your machine tools.

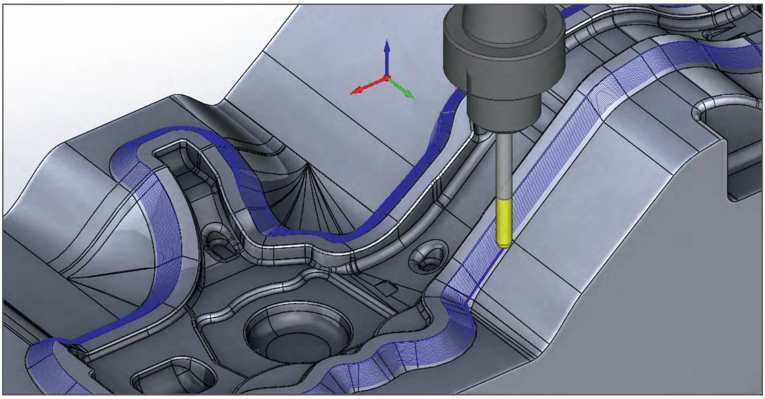
With demands for shorter than ever lead and production times, lower costs and improved quality, InventorCAM HSM is a must in today's machine shops.



The InventorCAM HSR/HSM module features several enhancements to CAM technology that make high speed machining operations possible, including avoiding sharp angles in tool path, ensuring that the tool stays in contact with the material as much as possible, and optimizing non-machining moves to reduce air cutting and generating smooth and tangential lead in/out.



Any HSM strategy can be controlled by specifying the surface slope angle to be machined or by specifying the machining boundary.



A comprehensive set of boundary creation tools is provided, including silhouette boundaries, cutter contact area boundaries, shallow boundaries, rest area boundaries and even user-defined boundaries.

HSM – 3D Machining to the Highest Level

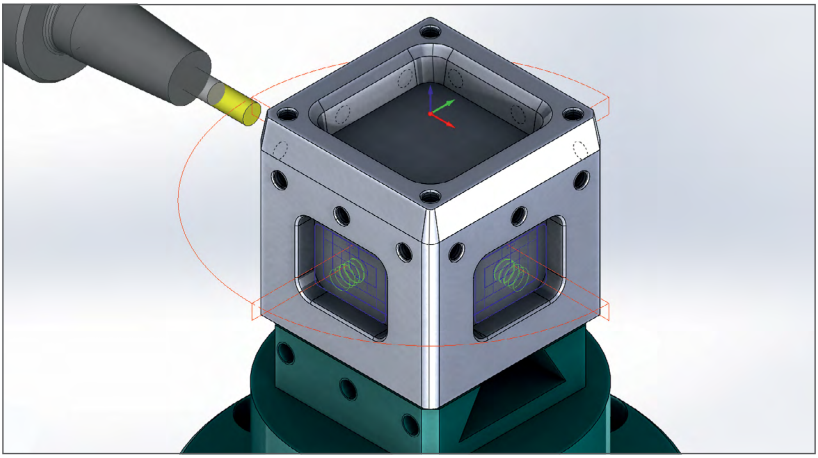
InventorCAM HSM is the powerful solution for all users who demand advanced high speed machining capabilities. It can also be used to improve the productivity of older CNC-Machines with reduced air cutting and smoothing arcs that maintain continuous tool motion.

InventorCAM HSM can take your 3D machining performance to the highest level possible, all with your current machines.

Indexial Multiaxis Machining

Powerful Indexing with Multi-sided Machining – Easiest Coordinate System Definition!

The common scene in many machine shops today is that 4- and 5-Axis CNC-Machines are increasing production, providing faster cycle times.

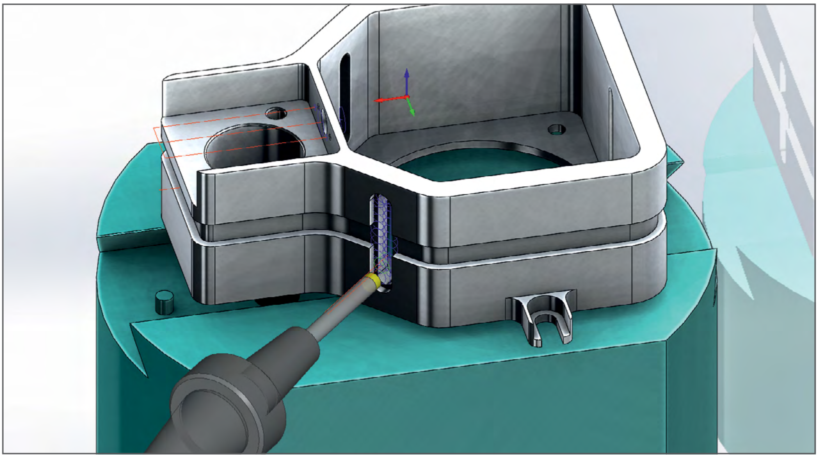


InventorCAM provides an effective and easy way to program on multiple sides of a part. InventorCAM is exceptionally strong in Indexial 4/5-Axis Machining.

Easiest Coordinate System Definition!

Tired of dealing with construction views, copying models, and rotating them in space for new alignments? Do you still copy and transform geometry to separate layers for indexial programming?

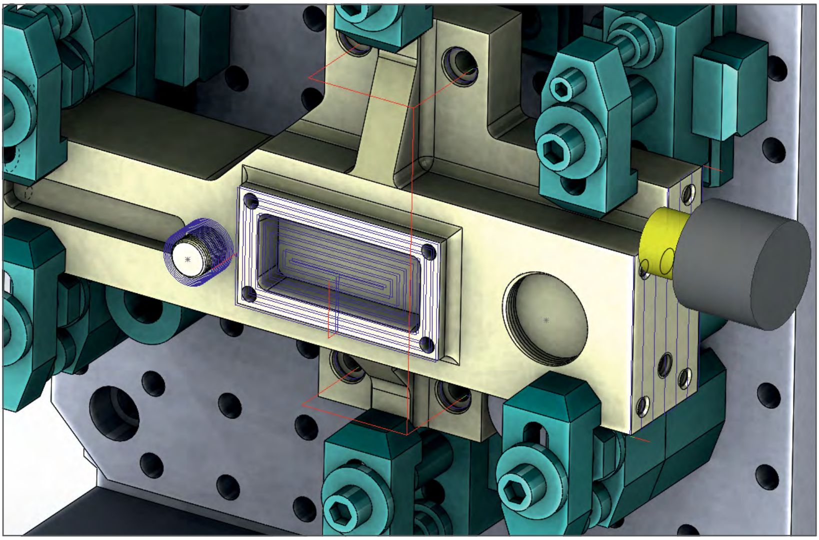
Experience single machine home positions with one-click orientations for indexed setups. InventorCAM speeds up Multi-sided Machining by eliminating multiple coordinate system constructions. Define a InventorCAM Coordinate System and/or positions on the fly, by just picking a face, and then continue programming your part.



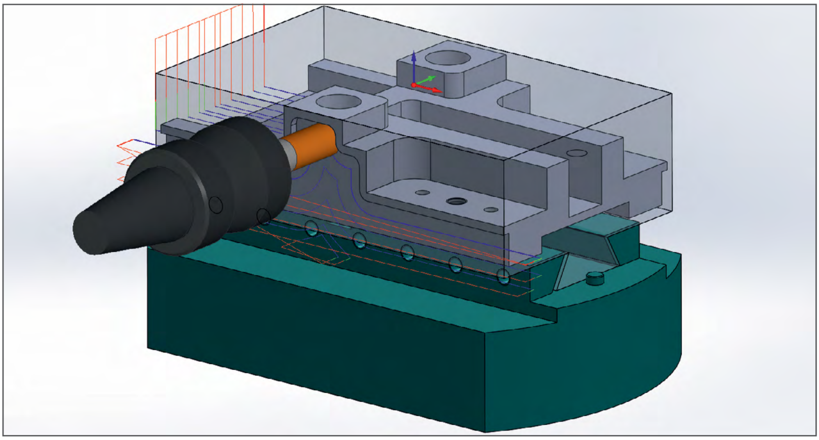
- InventorCAM's "select a face and machine" functionality is the fastest approach to indexial programming
- InventorCAM's CoordSys Manager keeps track of all necessary data for each tool orientation
- InventorCAM's SolidVerify simulation shows tool holders and fixtures, together with material removal for all machining operations

Efficient, Edit-free GCode for Multiaxis CNC-Machines

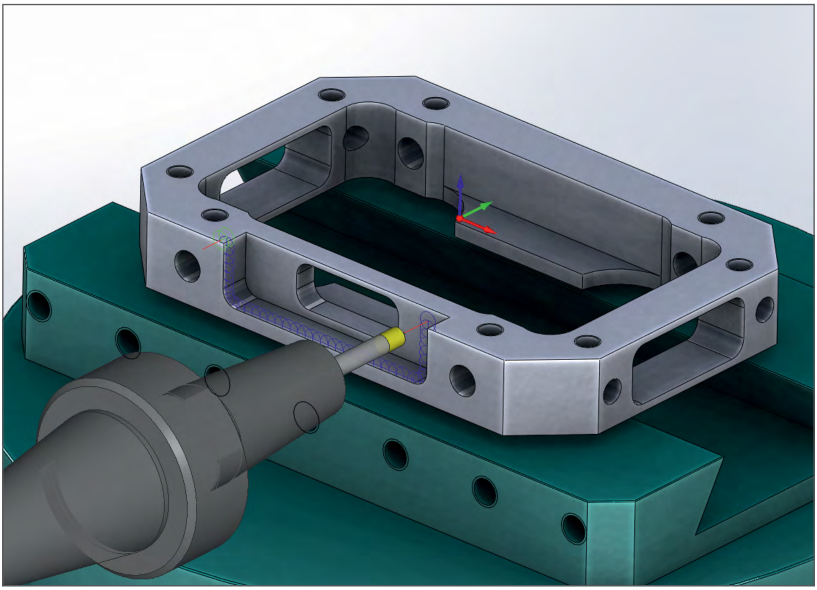
InventorCAM offers multiple options to get efficient, edit-free GCode for Multiaxis CNC-Machines.



InventorCAM's post processors can be configured to handle all rotations and work offset shifting, which eliminates the need for setting up multiple work offsets at the machine. Whether your controller can calculate part rotations internally or it requires the post processor to handle rotations, InventorCAM has it covered.



For controllers with advanced plane rotation or coordinate rotation functions, InventorCAM's post processors are built to use these internal CNC functions. If you have a machine without such functions, users can input the part location inside InventorCAM and the GCode will handle all the GCode transformations for each rotation.

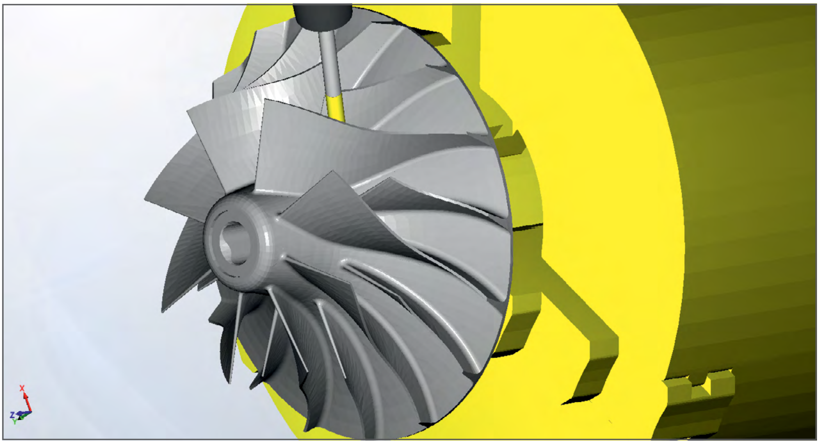


InventorCAM's philosophy towards Indexial 4/5-Axis Machining is simple: from software to GCode, make the process for Multi-sided Machining the same as for Single-sided Machining. There is no need for any special functions or tricks inside the software to machine multi-sided parts – it should just work!

Simultaneous 5-Axis Machining

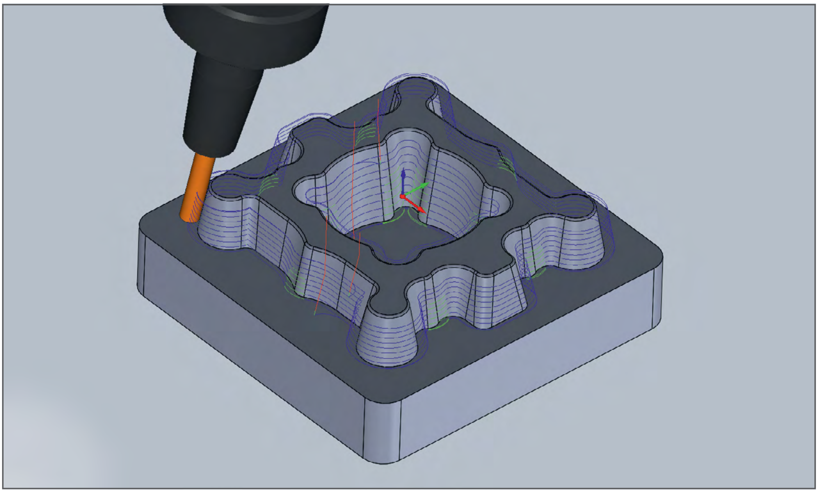
Powerful Simultaneous 5-Axis Tool Paths with a Very Friendly User Interface

Benefit from the most tested and most proven 5-Axis machining tool paths in the industry with a user-friendly interface, collision checking and the most advanced control over all aspects of tool path.



- Wide variety of Simultaneous 5-Axis Machining strategies
- Flowline cutting produces a tool path that follows the natural shape of the component
- Multi-surface finish machining keeps the tool normal to the surface (or with specified lead and lag) to provide a smooth surface finish
- Specific applications solutions for SWARF, Multiblade, Port, Contour 5-Axis, Multiaxis Drilling and conversion of HSM to Sim. 5-Axis
- Advanced tool tilting control and direct control on side tilting and lead/lag angles
- Automatic collision avoidance strategies that check each part of both the tool and holder
- Realistic full 3D Machine Simulation with comprehensive collision and axes limits checking

InventorCAM's Simultaneous 5-Axis Machining supports all Multiaxis machine tools including Head/Head, Table/Table and Head/Table Gantry Machines as well as the latest Mill-Turn Machining Centers.



Flexibility and Control

Each Simultaneous 5-Axis Machining strategy provides sophisticated options for approach/link control and tool axis control.

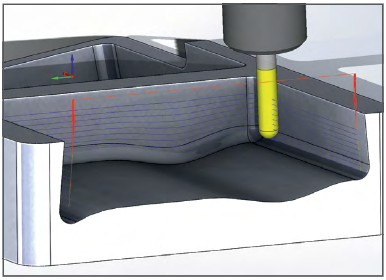
Link and approach movements are fully protected from gouging and different strategies may be used depending on the distance of the link move. InventorCAM also provides options for control over lead/lag and side tilt angles to give complete control over the final tool path.

Collision Avoidance for Tool and Holder

Collision avoidance is supported for both the tool and holder, and a range of strategies is offered for avoiding collisions. The Machine Simulation provides complete visualization of the gouge checking.

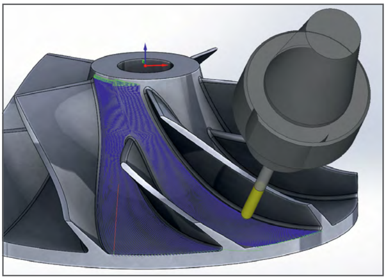
SWARF Machining

SWARF Machining allows the side of the tool to be tilted over to machine the side wall at the correct angle. The SWARF operation uses the whole cutting length of the tool, resulting in better surface quality and shorter machining time.



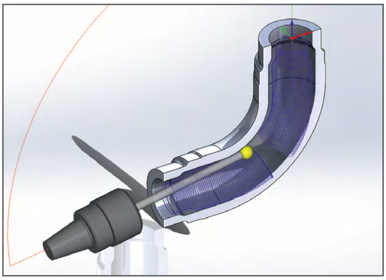
Multiblade Machining

Multiblade Machining easily handles impellers and bladed disks with several strategies to efficiently rough and finish all parts of these complex shapes. Multiple bladed parts are used in many industries and this operation is specifically designed to generate the necessary tool path for the different configurations.



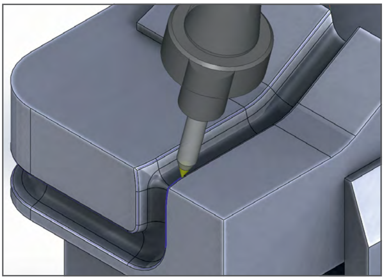
Port Machining

Port Machining is an easy-to-use method for machining ports with tapered lollipop tools, which can reach the full area while offering collision checks for the entire tool. Several strategies are provided to generate roughing and finishing tool paths for making ports from castings or billets.



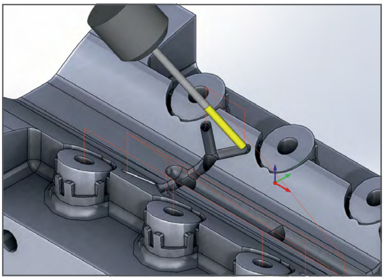
Contour 5-Axis Machining

Contour 5-Axis Machining tilts the tool along a chained 3D profile drive curve, while aligning the tool axis according to defined tilt lines, making it ideal for generating 5-Axis tool path for deburring and trimming.



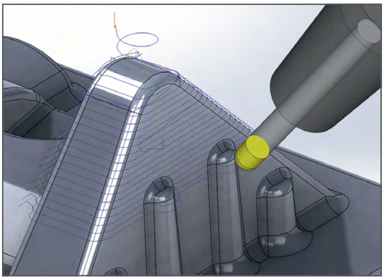
Multiaxis Drilling

Multiaxis Drilling uses InventorCAM's Drill Recognition to automatically recognize all the part hole features and then perform drilling, tapping or boring cycles at any hole direction easily and quickly. In this operation, all the advanced linking, tilting and collision checking are available.



HSM to Sim. 5-Axis Milling

3 to 5 axis Conversion converts HSM 3D tool paths to full 5-Axis collision-protected tool paths. This will maintain optimum contact point between the tool and the part and enable the use of shorter tools for more stability and rigidity.



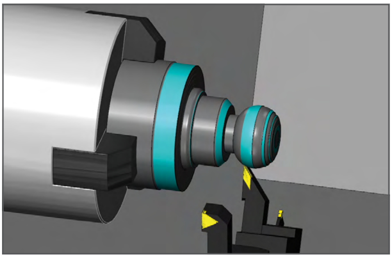
Turning

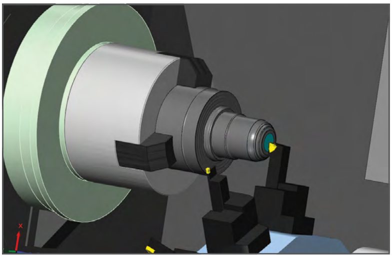
InventorCAM Module for Fast and Efficient Turning

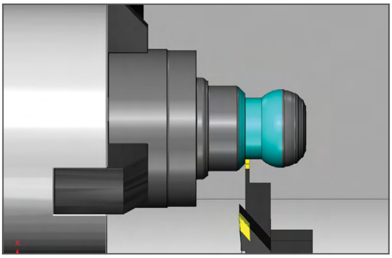
InventorCAM provides the most comprehensive Turning package with powerful tool paths and techniques for fast and efficient turning.

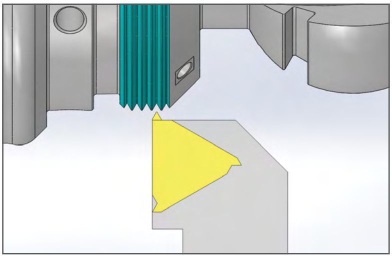
InventorCAM Turning provides functionality for a wide range of machine tools, including 2-Axis Lathes, Multi-turret Configurations, Sub-spindle Turning Centers and Mill-Turn CNC-Machines.

InventorCAM produces advanced rough and finish profile turning, together with support for facing, grooving, threading and drilling in either canned cycles or long GCode.







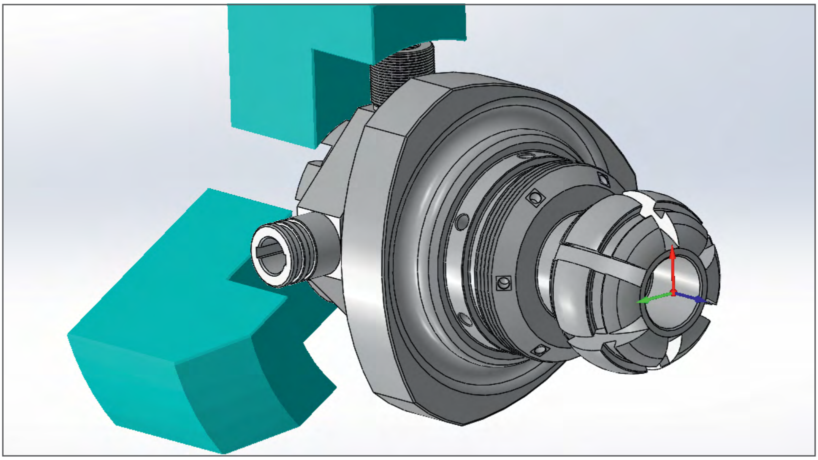


Tool path calculation takes into consideration the tooling insert, tool holder and previously machined stock material, to avoid gouging and to eliminate air cutting.

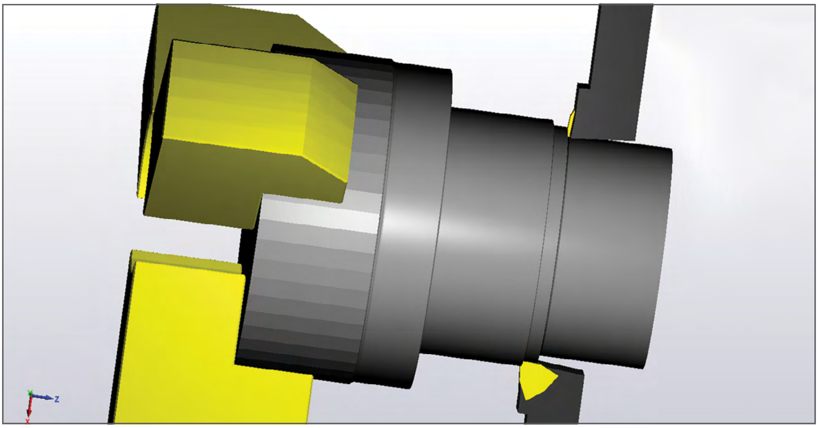
Standard fixture libraries are available and specialized fixtures can be added.

Updated Stock

InventorCAM has the ability to keep the stock updated live within the operations tree. Updated stock is supported from the most basic 2-Axis Turning Center, right through to a CYB Multi-turret, Sub-spindle Mill-Turn CNC-Machine.



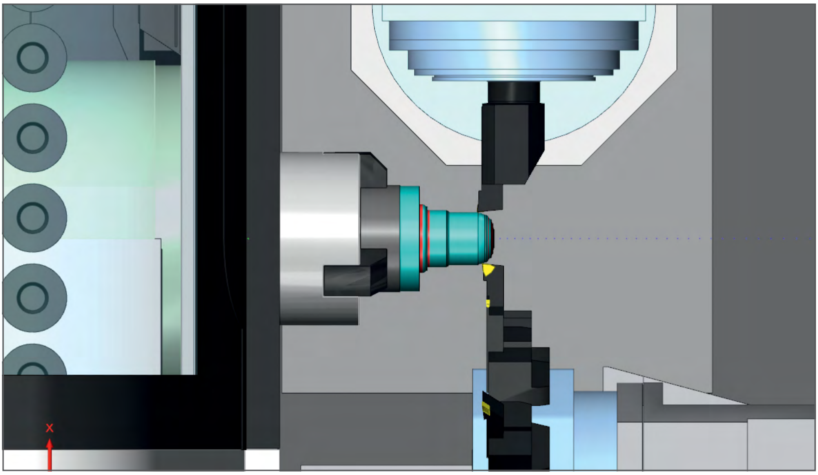
On a Sub-spindle Turning Center, when a component is transferred from the main to the sub-spindle, the Updated stock model is transferred with it. Any subsequent machining on the sub-spindle will detect the stock in the state that it left the main spindle, ultimately providing the most efficient machining sequence possible.



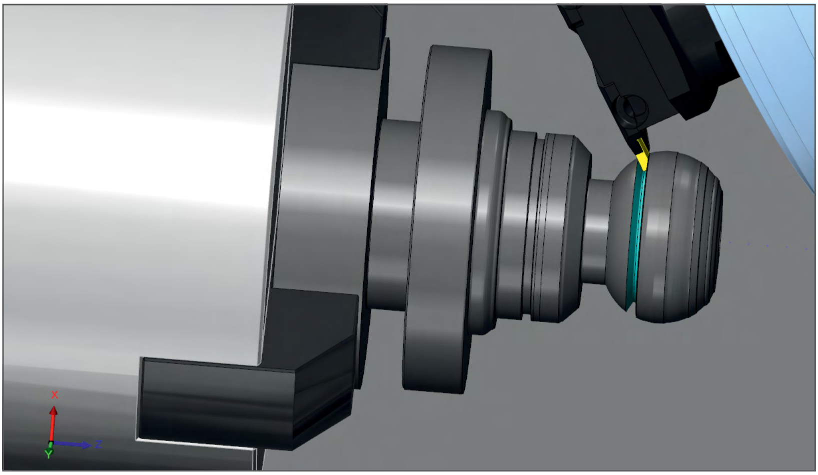
Advanced Turning Operations

InventorCAM's advanced operations bring power to all turning applications.

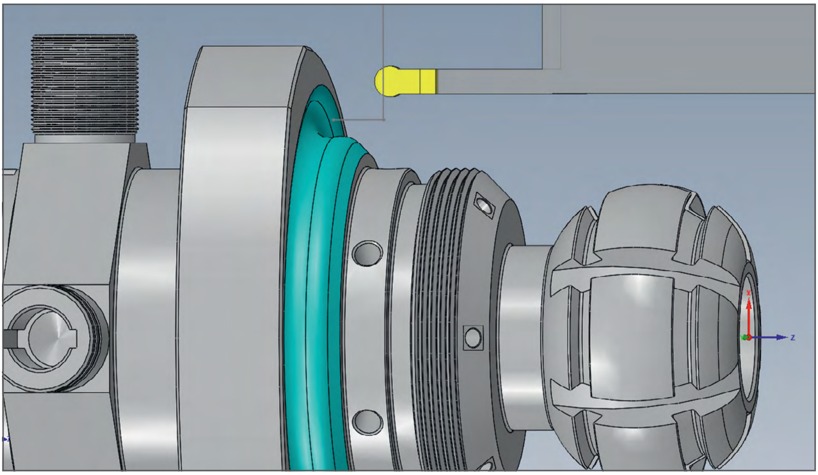
- Balanced Roughing: two tools working simultaneously or in trailing mode to perform the roughing of long and/or large parts



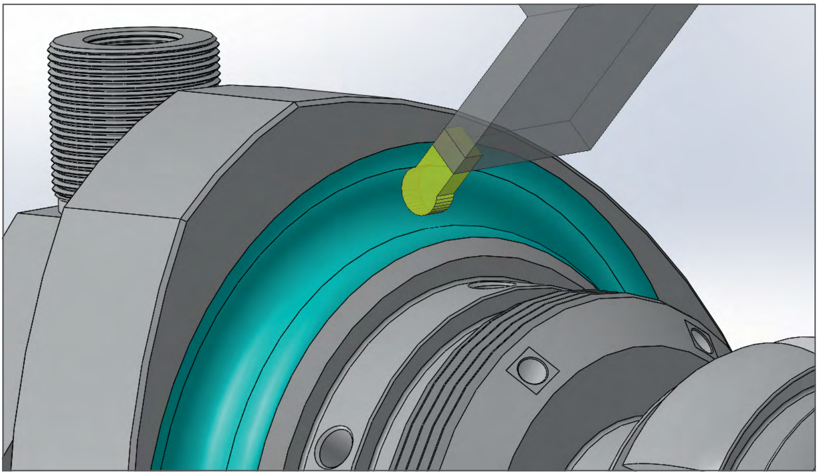
- Angled Grooving: performs internal or external inclined grooves at any defined angle



- Manual Turning: performs turning according to user-defined geometry regardless of Stock and Target models



- 4th Axis Simultaneous Turning: performs machining of curved profiles using the B-Axis tilting capabilities of the tool in order to machine undercut areas in a single machining step



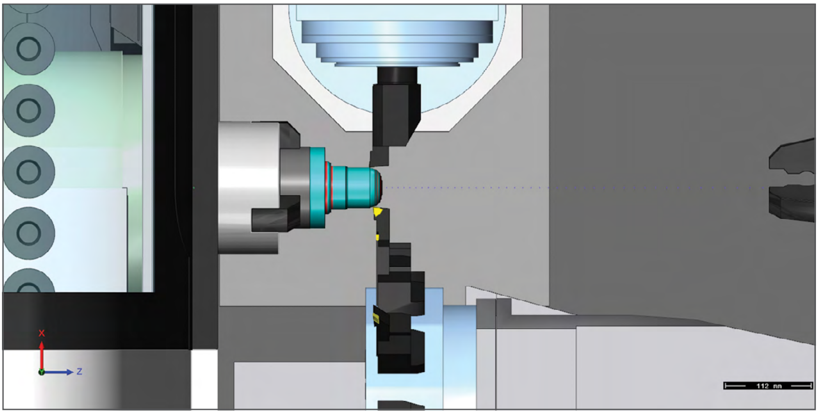
- Multi-turret Synchronization: provides powerful functionality with a machining timeline to perform synchronization of operations on Multi-turret Machines

Mill-Turn

Easy Programming for Complex Multiple Turret and Multiple Spindle CNC-Machines

The fastest growing and most demanding class of CNC-Machines on the market today are multi-task machines that combine several capabilities into one machine – multiple spindles, multiple turrets, material being machined in multiple stages, transferring from spindle to spindle without handling, stock being inserted at one end and finished parts coming out the other.

On a Mill-Turn CNC-Machine, C-, Y- and B-Axis milling and drilling occur within the same program as the turning, providing a fully integrated and associative programming solution.



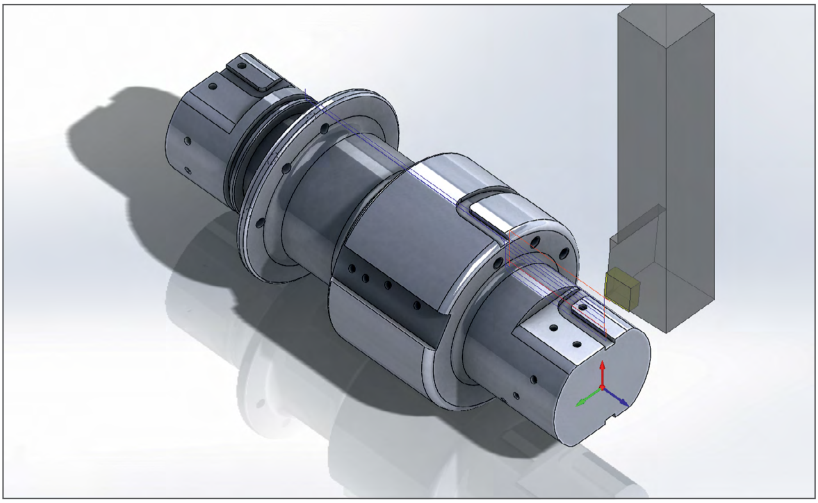
4/5-Axis Simultaneous Mill-Turn Machines have many uses and allow much more flexibility and capabilities not offered from other machine configurations. With this in mind, many of these have multiaxis, upper turrets, lower turrets, CYB and sub-spindles.

InventorCAM has the advanced technology to support the programming of all the latest multi-function CNC-Machines, providing powerful tools that are easy to learn and use, offering ultimate flexibility and configurability.

- Use of tail stocks, steady rest, sub-spindles, rotary and linear turrets along with C-Axis, Y-Axis and B-Axis, are regular features on today's Mill-Turn machine tools, and in this collision risky environment, the programming of these machines is made simple and safe with InventorCAM Turning and Milling in a single environment
- Support for multiple turret and multiple spindle programming with Multi-turret Synchronization and full Machine Simulation, seamlessly integrated into one extremely powerful package
- All InventorCAM Turning and Milling operations, including the unique, revolutionary iMachining, are available for the programming of Mill-Turn Machines, and all ancillary devices can also be defined and considered during simulation and gouge checking

Easy Programming for Complex Mill-Turn CNC-Machines

InventorCAM goes beyond just programming these complex machines with intelligent management of rest material between all operations, both Milling and Turning, for the most efficient tool paths and reduced cycle times to ensure the highest possible productivity ever imagined.

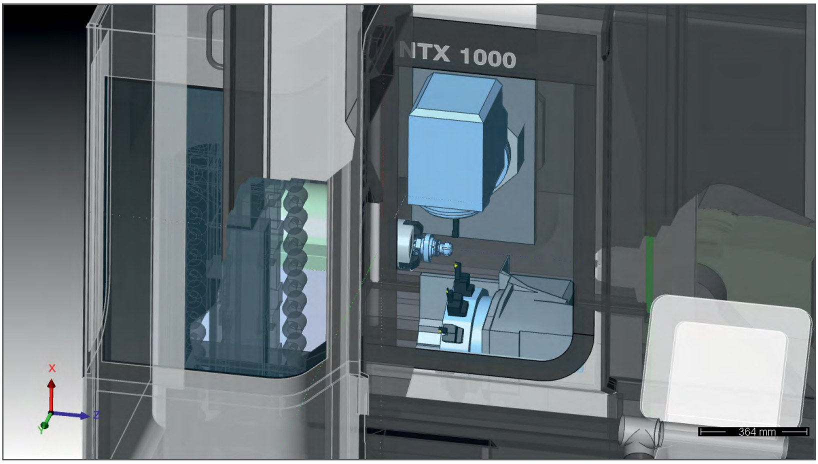


VMID (Virtual Machine ID)

The VMID defines the CNC-Machine components and machine kinematics, enabling users to set up and support the most complicated Mill-Turn Machines easily and effectively.

MCO (Machine Control Operation)

MCO (Machine Control Operation) enables the user to insert various control operations while manufacturing a part. These operations control the CNC-Machine and activate different options and devices such as the opening or closing of fixtures, activating coolants, rotating the part or moving the part from one table to another.

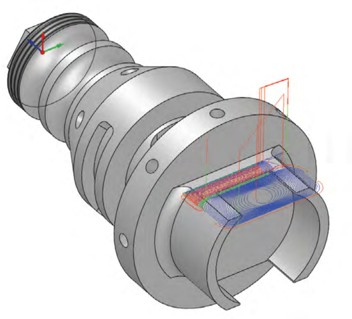


Transfer Between Spindles

Control the transfer of parts between the main and sub-spindle, using Machine Control Operations. Ready-made MCOs provide the best solution for this process.

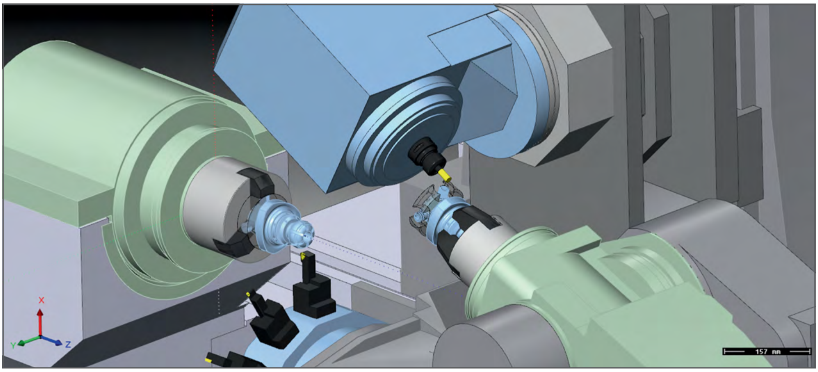
iMachining in Mill-Turn

Using iMachining 2D & 3D saves you both programming and cycle time. Additionally, iMachining has its very important, unmatched advantages of exerting smaller cutting forces, eliminating vibrations and excessive tool wear, even in situations of non-rigid workpiece holding.



Mill-Turn Machine Simulation

Mill-Turn Machine Simulation in InventorCAM offers a full kinematic simulation package, supporting simulation of all Turning and Milling operations and of all CNC-Machine components and devices. The simulator offers full collision detection and avoidance between machine components, workpiece, fixtures, tools and holders.

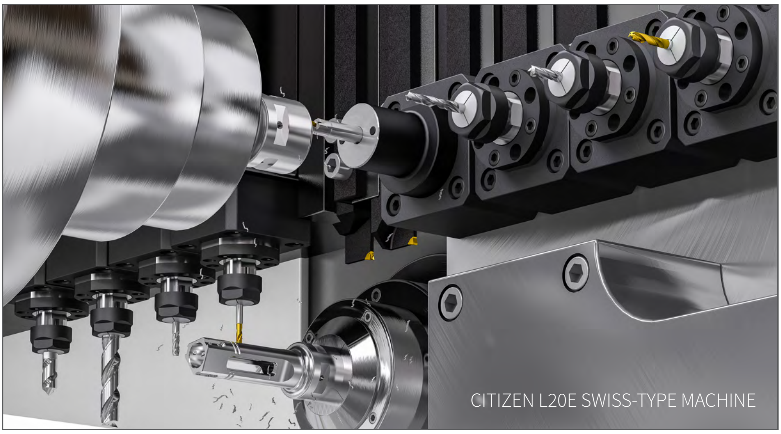


All the cycles and movements are supported along with the full graphics of the machine components and auxiliary devices such as tail stock and steady rest, providing safety as the part is fully tested before reaching the actual machine tool.

Mill-Turn – Swiss-Type

Speed Up Your Complex CNC-Machines and Maximize Productivity

Modern Multi-Axis Machining Centers and Swiss-Type Machines are designed to manufacture workpieces at maximum productivity by combining as many Milling and Turning operations as possible.

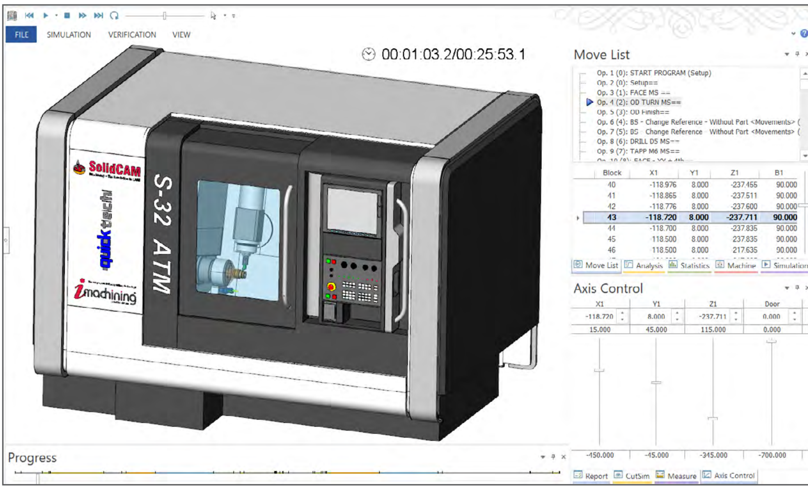

CITIZEN L20E SWISS-TYPE MACHINE

Parts are typically small, complex and often have high precision requirements. Swiss-Type Machines have a sliding headstock and a guide bushing through which the stock is fed, supported and nearby later cut. Such machines can also have a large number of axes, are exceptionally stable and regularly provide remarkable machining cycles.


STAR SR 38 TYPE-B SLIDING HEAD STOCK MACHINE

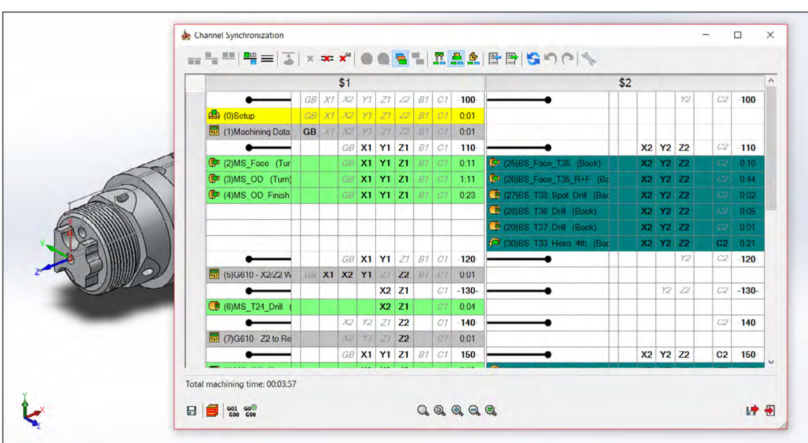
Machine Support is Extensive

While already supporting the most complex CNC-Machines having unlimited number of axes and channels, InventorCAM is constantly adding more Advanced Mill-Turn and Swiss-Type Machines with various configurations to its extensive machine tool database.

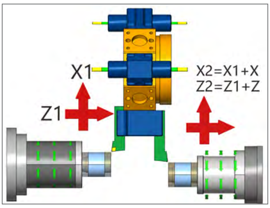


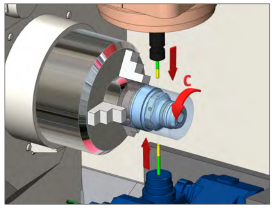
Shortest Possible Cycle Times

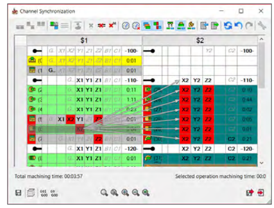
The Channel Synchronization Manager guides you through synchronizing and optimizing all your machining operations for maximum production output.



InventorCAM enables you to control an unlimited number of channels and supports any amount of machine functions and cutting modes.







- InventorCAM handles three different super-imposition modes. A pair of axes can be superimposed one to another, where the slave axis follows the master axis. For applicable machines, this mode is detected automatically.
- Reduce machining time by sharing axes and drive units. Synchronize two turning operations on different turrets at the same time and, under specific conditions, use the same spindle or synchronize two milling operations on different turrets on the same rotary axis.
- The Channel Synchronization's clash engine displays any issue with logical comments. The intelligent system holds the logic and checks the possibilities of the synchronization while taking into account the machine kinematics.

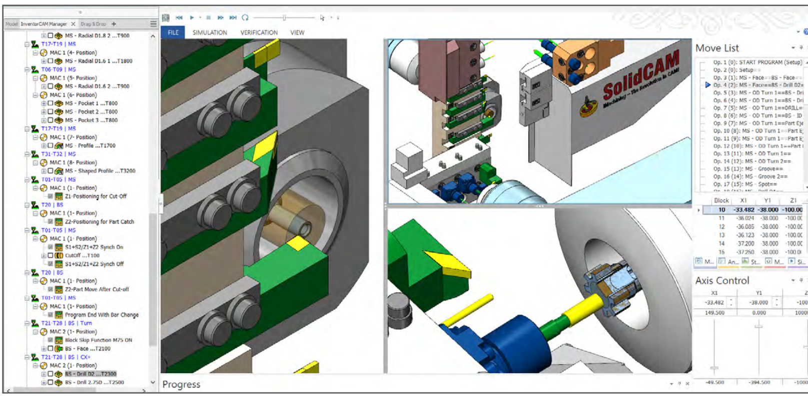
Advanced Machine Control with MCOs

On Swiss-Type Machines, InventorCAM's Machine Control Operations further maximize productivity. Besides your machining operations, you can define any number of machine actions such as:

- Change tool
- Move machine components
- Transfer stock
- Clamp/unclamp fixture
- Program barfeeder
- Control coolants
- Machine mode
- Axes and phase synchronization
- Output any G/M command

Advanced Machine Simulation

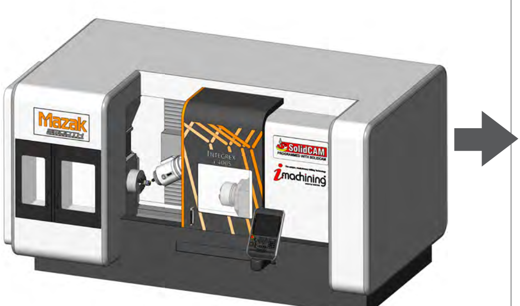
InventorCAM's Full Machine Simulation package can verify all Turning, Milling and MCO operations of the actual machine tool. It provides full collision detection and avoidance between machine components, workpiece, fixtures, tools and holders.



Maximize your productivity by visually proving-out the tool path of your program before physically machining the part.

Advanced Post Processors

InventorCAM's Virtual Machine ID (VMID) is part of the post processor file that defines the kinematic structure and the controller GCode output options for your Swiss-Type Machine. Using the VMID definitions, the customizable General Post Processor Language (GPPL) translates the tool path into efficient, edit-free GCode.




```
O0001 ( MAZAK_I400S )
(INTEGREX-i - 400 S)
(part      : MAZAK_I400S)
(created   :  9-MAY-2019 )

#800=-458.7 (Work-Offset G54 - Z1)
#801=0. (Work-Offset G54 - C1)
(----------------------------)

G21
M901
G92 S2000 R1
G92 S2000 R2
G90 G0 G53 G0 X0. Y0.
G90 G0 G53 G0 Z0.
M108
G90 G53 G0 B0.
M107

G10 L2 P1 X-490. Z#800 C#801
G10 L2 P2 X-490. Z#802 U#803
```

Solid Probe

InventorCAM Essential Module for Tool Setup and On-the-Spot Measurement

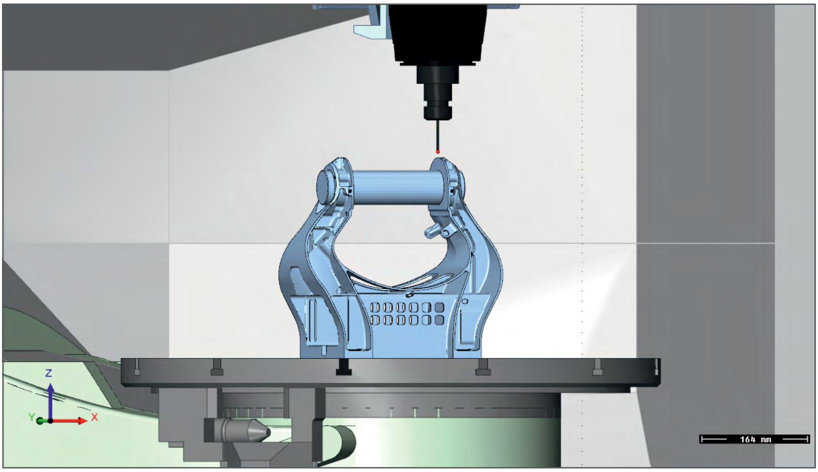
InventorCAM's integrated Solid Probe module provides Home Definition (Setup) and On-Machine Verification (Measurement) capabilities, using probes on the CNC-Machine.

Driving your probe with InventorCAM Solid Probe enables you to perform Setup as well as quick and easy Measurement for the best quality parts directly on your CNC-Machine, without breaking the Setup. Solid Probe also provides Tool Presetter support.

Full visualization of all the probe movements, provided by InventorCAM Machine Simulation, enables you to avoid any potential damage to the probe tool.

InventorCAM Solid Probe is a Must Module for Every Machinist Using Probes

InventorCAM's integrated Solid Probe module does it all.



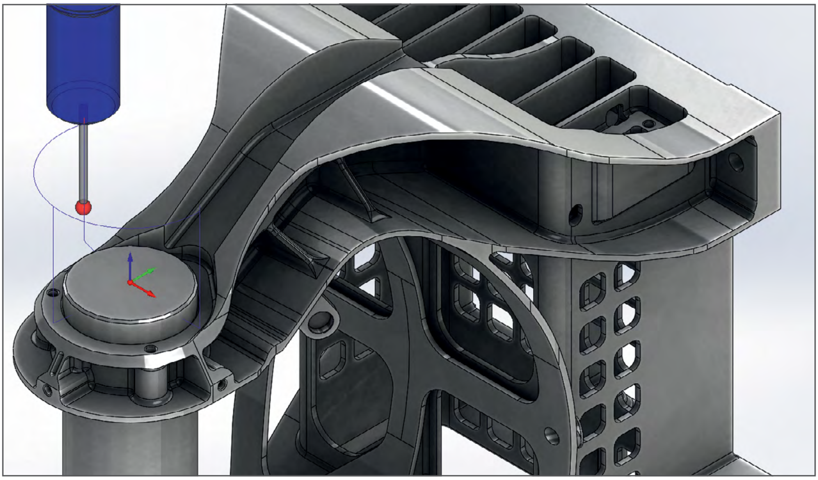
- Easy Home Definition
- Easy On-Machine Verification
- Tool Presetter support
- Easy geometry selection on solid model
- Supports wide range of Solid Probe Cycles
- Visualization of all probe tool movements
- Supports various controllers

Combined Probe and Machining Operations

Machining and Probe operations appear together in the InventorCAM Manager and can share the same geometries defined on the CAD model. When the solid model is changed, both Machining and Probe operations are automatically updated to use the new geometry. Many types of Solid Probe Cycles are provided to simplify measurement challenges and to smooth workflows alongside your Machining operations.

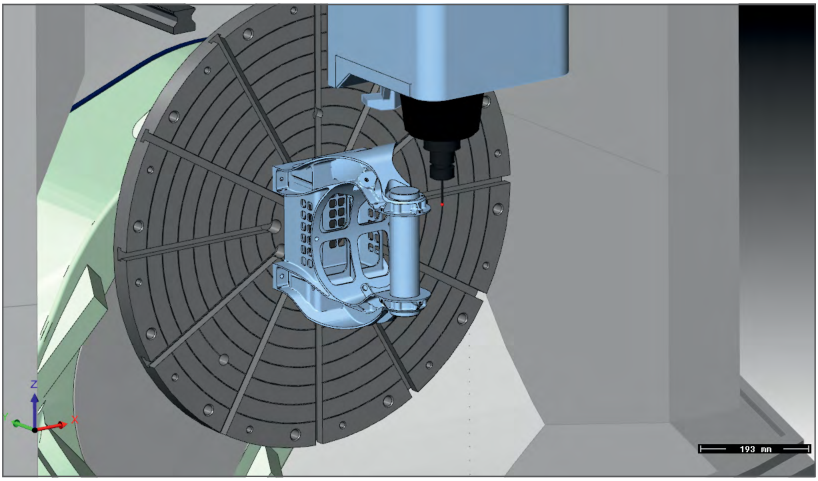
Home Definition

Solid Probe offers 16 different cycles to easily define your home positions, replacing manual Setup procedures that are often time consuming.



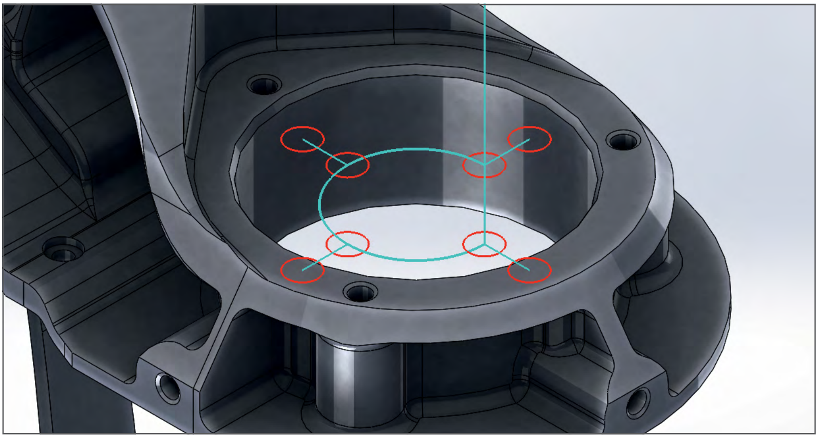
On-Machine Verification

Solid Probe Cycles are used for Measurement of machined surfaces without transferring the part to a Coordinate Measurement Machine (CMM) – the part can be inspected directly on the machine tool itself.



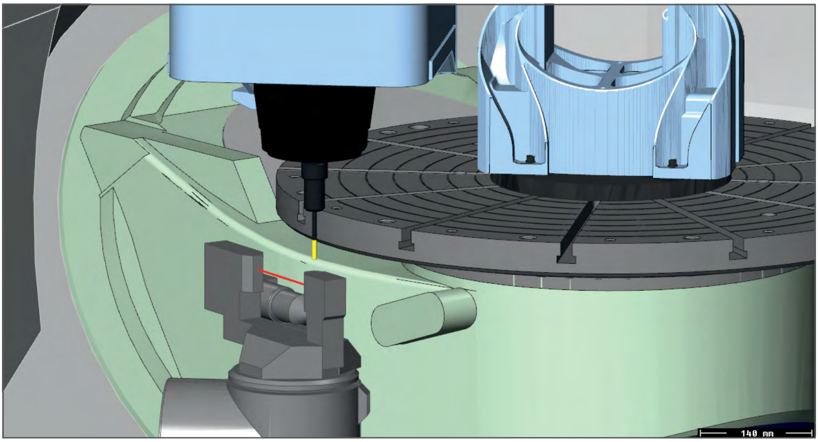
Preview of Cycle Movements

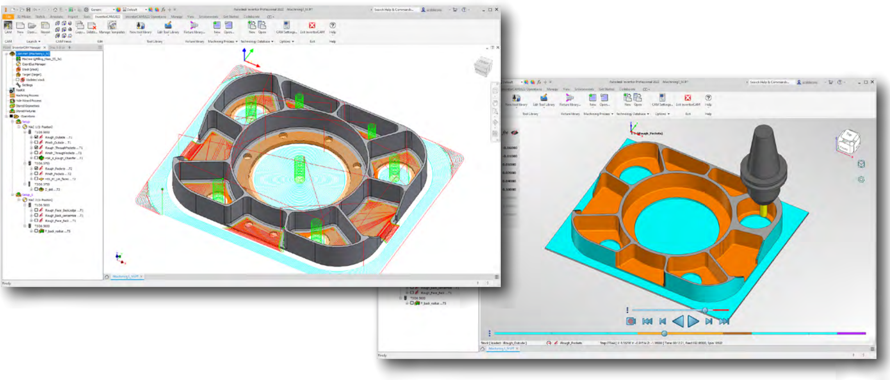
Solid Probe can use the same geometry as your 2.5D Milling operations with full control over tolerances and various sorting options and with a direct preview of the probe tool movements.



Tool Presetter Support

Solid Probe includes Tool Presetter support to check your milling and turning tools between Machining operations and tool change events. It also provides tool breakage detection to ensure continuous and safe machining.





Click  to open the InventorCAM part: **iMachining1_IV.prz**

The operations in this example illustrate the use of InventorCAM's iMachining 2D technology to machine the part shown above. The machining is performed on a 3-Axis CNC-Machine in two setups, using two InventorCAM Coordinate Systems.

- **Outside shape machining (iRough_Outside; iFinish_Outside)**

  These iMachining operations define the machining of the outside shape of the part. Two chains are defined, with the first being the stock boundary and the second being the profile around the part. The stock chain is marked as open, which specifies the tool should start machining from that chain. A Ø12.5 mm (0.5 in) end mill is used. iRough has a 0.25 mm (0.01 in) allowance on the walls, which is then removed by the iFinish operation.

- **Through pockets machining (iRough_ThroughPockets; iFinish_ThroughPockets)**

  These iMachining operations define the machining of the five circular through pockets. Five chains are defined, which represent the five through pockets. A Ø12.5 mm (0.5 in) end mill is used and since the pockets are closed with no Pre-Drilling data or entry chains defined, the tool performs helical ramping to enter into the pockets.

- **Rough machining of angled surfaces (HSR_R_Rough_Chamfer)**

  This HSR operation defines the rough machining of the four large chamfers on the ribs of the part. Two boundaries are picked off the edges the make up the chamfers. A Ø12.5 mm (0.5 in) end mill is used with the Tool Relation set to Centered. A 1.27 mm (0.05 in) Step down is specified with a 0.13 mm (0.005 in) allowance to remain on the surfaces.

- **Pockets machining (iRough_Pockets; iFinish_Pockets)**

  These iMachining operations define the machining of the three semi-open pockets and the seven closed pockets. Since all ten pockets have the same depth, they can all be machined in one operation. A Ø9.5 mm (0.375 in) bull nose mill with a corner radius of 1.6 mm (0.0625 in) is used. iRough has a 0.25 mm (0.01 in) allowance on the walls, which is then removed by the iFinish operation.

- **Finish machining of angled surfaces (HSS_PC_Lin_faces)**

  This HSS operation defines the finish machining of the four large chamfers on the ribs of the part. The Ø9.5 mm (0.375 in) bull nose mill is used and the Linear strategy generates Parallel cuts with a 0.5 mm (0.02 in) Maximum step over. Customized linking is used to allow short repositions and smooth transitions when starting each cut.

- **Holes machining (D_drill)**

  This Drilling operation defines the drilling of the holes through the lower surface of the part.

- **Bottom ledge machining (iRough_Face_BackLedge)**

  This iMachining operation defines the bottom ledge machining on the underside of the part. Two chains are defined, with the first being the stock boundary and the second being the bottom of the floor radius. Using a Ø12.5 mm (0.5 in) end mill, the tool starts machining from the stock chain and collapses in towards the radius.

- **Cutting excess material from through hole (iRough_back_centerHole)**

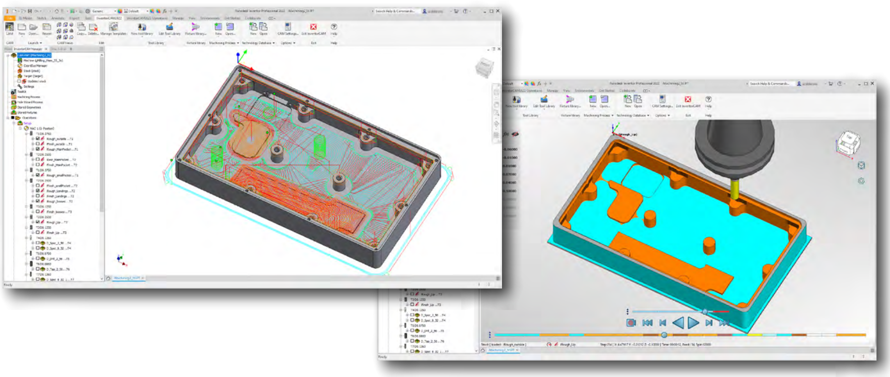
  This iMachining operation is defined to machine away the excess material from the center through hole of the part. This excess material was used for clamping in the first setup.

- **Bottom face machining (iRough_Face_Back)**

  This iMachining operation defines the bottom face machining on the underside of the part. Using a Ø12.5 mm (0.5 in) end mill, a morphing spiral tool path is performed from the outer open chain, collapsing towards the inner closed chain.

- **Floor radius finishing (F_back_radius)**

  This Profile operation defines the 6.35 mm (0.25 in) floor radius finishing on the underside of the part. The chain is defined as the bottom edge of the radius. A Ø12.5 mm (0.5 in) ball nose mill is used with the Tool side set to Center. A 0.13 mm (0.005 in) offset remains on the floor after the first roughing pass and is then removed with a single finishing pass.



Click to open the InventorCAM part: **iMachining2_IV.prz**

The operations in this example illustrate the use of InventorCAM's iMachining 2D technology to machine the part shown above. The machining is performed on a 3-Axis CNC-Machine in three setups, using three InventorCAM Coordinate System.

- **Outside shape machining (iRough_outside; iFinish_outside)**

  These iMachining operations define the machining of the outside shape of the part. A Ø9.5 mm (0.375 in) end mill is used. iRough has a 0.25 mm (0.01 in) allowance on the walls, which is then removed by the iFinish operation.

- **Main pocket machining (iRough_MainPocket; iRest_MainPocket; iFinish_MainPocket)**

  These iMachining operations define the machining of the main pocket area. iRough uses a Ø9.5 mm (0.375 in) end mill and has a 0.25 mm (0.01 in) allowance on the walls. Using a Ø6 mm (0.25 in) end mill, iRest has a reduced 0.13 mm (0.005 in) allowance on the walls, which is then removed by the iFinish operation.

- **Small pocket machining (iRough_smallPocket; iFinish_smallPocket)**

  These iMachining operations define the machining of the small shallow pocket between the landing and side connector opening. iRough uses a Ø9.5 mm (0.375 in) end mill and has a 0.25 mm (0.01 in) allowance on the walls, which is then removed by the iFinish operation using a Ø6 mm (0.25 in) end mill.

- **Landings machining (iRough_Landings; iFinish_Landings)**

  These iMachining operations define the machining of the two landings, one open and the other semi-open. Since both pockets have the same depth, they can be machined in one operation. iRough uses a Ø9.5 mm (0.375 in) end mill and has a 0.25 mm (0.01 in) allowance on the walls, which is then removed by the iFinish operation using a Ø6 mm (0.25 in) end mill.

- **Bosses machining (iRough_bosses; iFinish_bosses)**

  These iMachining operations define the machining of the two center and ten perimeter bosses. iRough uses a Ø6 mm (0.25 in) end mill and has a 0.25 mm (0.01 in) allowance on the walls, which is then removed by the iFinish operation using a Ø3 mm (0.125 in) end mill.

- **Upper ledge machining (iRough_Lip; iFinish_Lip)**

  These iMachining operations define the machining of the upper ledge. iRough uses a Ø6 mm (0.25 in) end mill and has a 0.25 mm (0.01 in) allowance on the walls, which is then removed by the iFinish operation using a Ø3 mm (0.125 in) end mill.

- **Holes machining (D_Spot_2_56; D_Spot_8_32; D_Drill_2_56; D_Tap_2_56; D_Spot_8_32_1; D_Spot_8_32_2)**

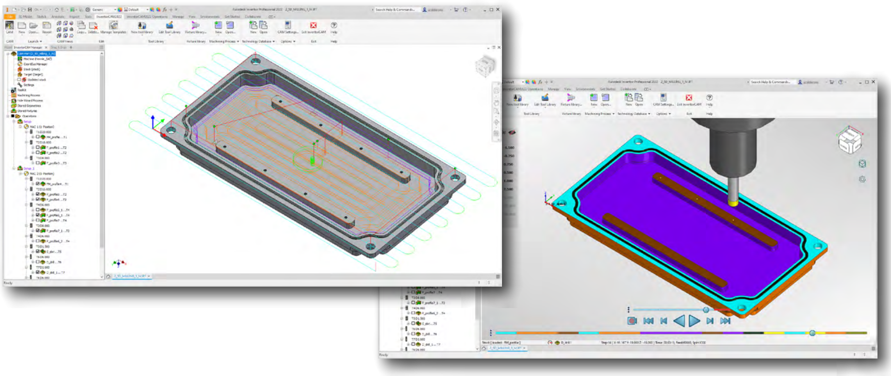
  These Drilling operations define the center drilling, the drilling and tapping of the holes located on the faces of the ledge and bosses.

- **Bottom face machining (iRough_faceBack)**

  This iMachining operation defines the bottom face machining on the underside of the part using a Ø9.5 mm (0.375 in) end mill.

- **Side connector machining and holes drilling (iRough_plugSlot; iRest_plugSlot; iFinish_plugSlot; D_Spot_094; D_Drill_094)**

  These operations define the connector machining and the holes drilling through the side of the part.
  The iMachining operations perform the rough, rest and finish machining of the connector opening.
  The Drilling operations perform the center drilling and drilling of the holes.



Click to open the InventorCAM part: **2_5D_Milling_1_IV.prz**

The operations in this example illustrate the use of several InventorCAM 2.5D Milling technologies to machine the part shown above. The machining is performed on a 3-Axis CNC-Machine in two setups, using two InventorCAM Coordinate Systems.

- **Upper face machining (FM_profile)**

  This Face Milling operation defines the machining of the part upper face. A Ø20 mm (0.75 in) end mill is used with the Hatch technology to remove 2 mm (0.08 in) of material from the stock top face.

- **Outside shape machining (F_profile1; F_profile2)**

  These Profile operations define the machining of the two external contours, one above the ledge and the other below the corner bolt seats. Both operations use a Ø16 mm (0.625 in) end mill to remove the material, and an additional finish pass is executed to remove the remaining 0.5 mm (0.02 in) from the walls.

- **Corner bolt seats machining (F_profile3)**

  This Profile operation defines the machining of the corner bolt seats. The material is removed in one step down cut using an Ø8 mm (0.3125 in) end mill, and an additional finish pass is executed to remove the remaining 0.5 mm (0.02 in) from the walls.

- **Bottom face machining (FM_profile4)**

  This Face Milling operation defines the machining of the part bottom face. A Ø20 mm (0.75 in) end mill is used with the Hatch technology to remove 2 mm (0.08 in) of material from the stock bottom face.

- **Pocket roughing (P_profile5; P_profile6; P_profile6_1)**

  These Pocket operations define the rough machining of the pocket. Using a Ø16 mm (0.625 in) end mill, the first operation machines down to the top face of the islands and the second operation machines down to the lower face of the pocket in between the islands. In the third operation, a Ø6 mm (0.25 in) end mill is used to machine only the rest material in the narrow areas between the islands and pocket walls. A machining allowance of 0.2 mm (0.008 in) remains on the wall and floor surfaces.

- **Pocket and islands finishing (F_profile5_1; F_profile7; F_profile7_1; P_profile6_2)**

  These operations define the finish machining of the pocket and islands to remove the allowances remaining after the previous operations. Using a Ø6 mm (0.25 in) end mill, the first two Profile operations perform the wall finishing. To perform the floor finishing, the third Profile operation uses an Ø8 mm (0.3125 in) end mill for the islands and the Pocket operation uses the Ø6 mm (0.25 in) end mill for the pocket.

- **Slot machining (S_slot)**

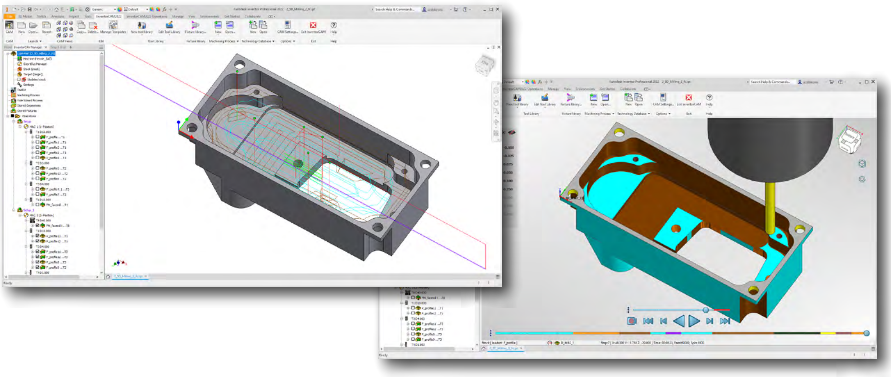
  This Slot operation defines the machining of the groove in the part bottom face. A Ø1.5 mm (0.0625 in) end mill is used to remove the material in three step down cuts.

- **Holes machining (D_drill; D_drill_1)**

  These Drilling operations define the center drilling and drilling of the four through holes located at the part corners.

- **Threaded holes machining (D_drill1; D_drill1_1; D_drill1_2)**

  These Drilling operations define the center drilling, the drilling and tapping of the holes located on the faces of the islands.



Click to open the InventorCAM part: **2_5D_Milling_2_IV.prz**

The operations in this example illustrate the use of several InventorCAM 2.5D Milling technologies to machine the part shown above. The machining is performed on a 3-Axis CNC-Machine in two setups, using two InventorCAM Coordinate Systems.

- **Upper faces machining (F_profile; F_profile1 F_profile2; FM_facemill)**

  These operations define the machining of the part upper faces. Using a Ø10 mm (0.4063 in) end mill, the first two Profile operations remove the majority of material next to and around the protuberance; the Clear offset option is used at the roughing stage and an allowance of 0.5 mm (0.02 in) is removed from the walls in the second operation. The third Profile operation performs roughing and finishing of the two steps. The Face Milling operation machines the protuberance top face.

- **Outside shape machining (F_profile3)**

  This Profile operation defines the machining of the external contour. The material is removed in three step down cuts using a Ø10 mm (0.4063 in) end mill, and additional finish passes are executed to remove the remaining 0.5 mm (0.02 in) from the walls.

- **Connector opening and shallow pocket machining (P_profile4; P_profile5; F_profile13; F_profile6; P_profile4_1)**

  These Pocket and Profile operations define the machining of the connector opening and the shallow pocket on the upper face of the part. Three tools (Ø10 mm (0.4063 in), Ø3 mm (0.125 in) and Ø4 mm (0.1563 in) end mills) are used for the roughing and finishing.

- **Corner bolt seats machining (F_profile7)**

  This Profile operation defines the rough machining of the corner bolt seats. A Ø4 mm (0.1563 in) end mill is used to perform the roughing in several step down cuts.

- **Bottom face machining (FM_facemill1)**

  This Face Milling operation defines the machining of the part bottom face. A Ø40 mm (1.6 in) face mill is used with the Hatch technology to remove material from the stock bottom face.

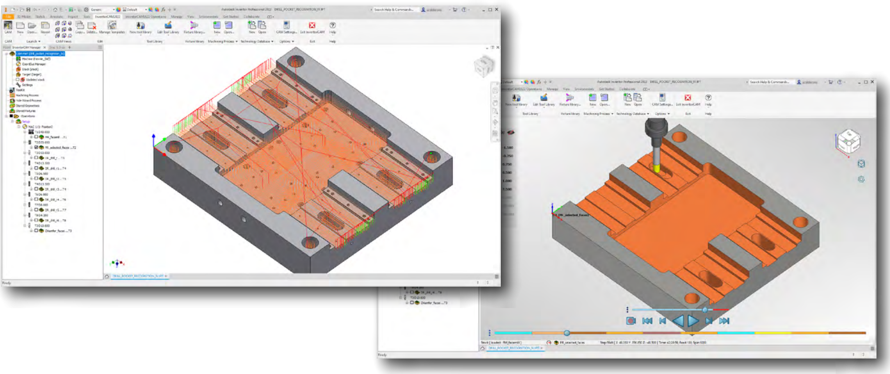
- **Pockets machining (P_profile11; P_profile12; F_profile11; F_profile12; P_profile8; F_profile9)**

  These operations define the machining of the pockets.
  The first pair of Pocket and Profile operations machine the upper level pocket. A Ø10 mm (0.4063 in) end mill is used for the roughing and a Ø4 mm (0.1563 in) end mill is used to remove the 0.2 mm (0.008 in) allowance remaining on the pocket walls.
  The second pair of Pocket and Profile operations machine the middle level pocket. The roughing and finishing is performed using the same tools, options and allowances as the previous pair of operations.
  The third pair of Pocket and Profile operations machine the lower level pocket using the Ø4 mm (0.1563 in) end mill.

- **Holes machining (D_drill – D_drill2_1)**

  These Drilling operations define the center drilling and drilling of the holes located on all the part faces in the second setup.

2.5D Milling – Feature Recognition



Click to open the InventorCAM part: **drill_pocket_recognition_IV.prz**

The operations in this example illustrate the use of InventorCAM's Automatic Feature Recognition and Machining technology to machine the part shown above. The machining is performed on a 5-Axis CNC-Machine in one setup, using one InventorCAM Coordinate System.

- **Upper face machining (FM_facemill)**

  This Face Milling operation defines the machining of the part upper face. A Ø40 mm (1.6 in) face mill is used with the Hatch technology to remove material from the stock top face.

- **Pockets machining (PR_selected_faces)**

  This Pocket Recognition operation automatically recognizes and defines the machining of all the part pocket features. Using a Ø20 mm (0.75 in) end mill, the Open pocket technology performs the approach movement from an automatically calculated point outside the material. The tool descends to the necessary depth and then moves horizontally into the material. A special strategy is applied to the through pockets, which require deepening in order for them to be machined completely.

- **Center drilling (DR_drill_r)**

  This Drill Recognition operation automatically recognizes and defines the center drilling of all the part hole features. A Ø10 mm (0.4083 in) spot drill is used and the chamfer depth is applied automatically for each group of holes.

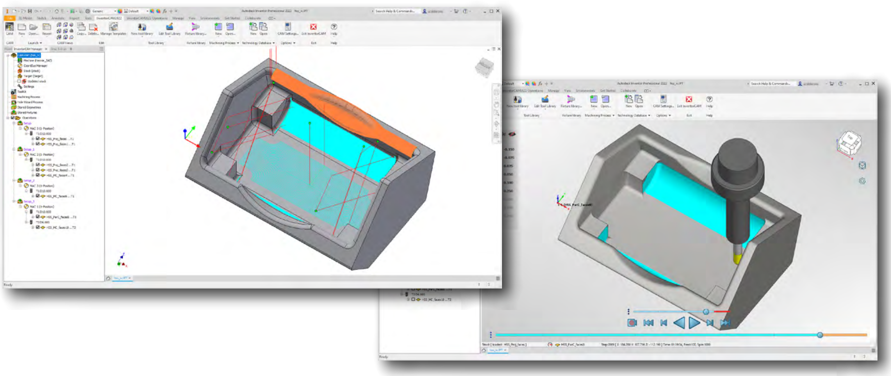
- **Drilling (DR_drill_r1 – DR_drill_r6)**

  These Drill Recognition operations define the drilling of all the part hole features. For each operation, InventorCAM automatically recognizes the hole features according to specific filters. The drilling depth is determined automatically and the through holes are extended in order for them to be machined completely.

- **Chamfer machining (CHamfer_faces)**

  This Edge Deburring Recognition operation automatically recognizes and defines the machining of all the part chamfer features. A Ø10 mm (0.4083 in) spot drill is used to machine a 0.3 mm (0.012 in) Chamfer depth. The tool descends and creates chamfers on the part edges with a Cutting diameter of 1 mm (0.04 in), while maintaining a 4.5 mm (0.18 in) Safety offset.

High Speed Surface Machining (HSS)



Click to open the InventorCAM part: **hss_iv.prz**

The operations in this example illustrate the use of several InventorCAM High Speed Surface Machining (HSS) strategies to machine the part shown above.

- **Projection machining (HSS_Proj_faces; HSS_Proj_faces1; HSS_Proj_faces2; HSS_Proj_faces3)**

  These HSS operations use the Projection strategy to define the machining of four small internal fillet areas. A Ø10 mm (0.4063 in) ball nose mill is used to fit the corresponding surface radius. For Roughing and More, the Depth cuts option ensures the whole depth is machined in several cutting passes.

- **Morph between curves machining (HSS_MC_faces4; HSS_MC_faces6)**

  These HSS operations use the Morph between two boundary curves strategy to define the machining of two large internal fillet areas. This strategy generates a constant tool path that is distributed evenly between the fillet boundaries. The Ø10 mm (0.4063 in) ball nose mill is used. Gouge check is enabled to automatically detect and avoid possible collisions between the tool and the faces of the machining area.

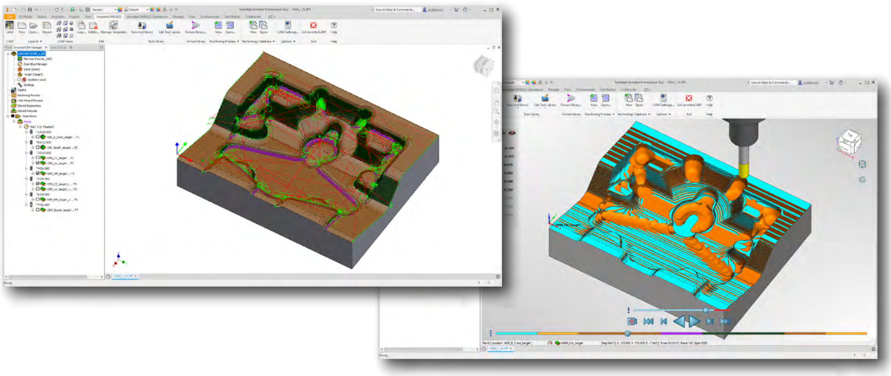
- **Parallel to curves machining (HSS_ParC_faces8)**

  This HSS operation uses the Parallel to curves strategy to define the machining of the part lower face. With this strategy, faces can be machined with cutting passes parallel to the selected curve. In this case, a pocket-style tool path is generated within the boundaries of the selected face using the Ø10 mm (0.4063 in) ball nose mill.

- **Morph between curves machining (HSS_MC_faces10)**

  This HSS operation uses the Morph between two boundary curves strategy to define the machining of the external fillet and an inclined face adjacent to the fillet. Using a Ø6 mm (0.2344 in) ball nose mill, the tool path is generated according to a 0.00375 mm (0.0002 in) Scallop value in order to obtain excellent surface quality. Gouge check is enabled to automatically detect and avoid possible collisions between the tool and the faces of the machining area.

3D Mill High Speed Machining (HSR/HSM)



Click to open the InventorCAM part: **hsm_1_IV.prz**

The operations in this example illustrate the use of several InventorCAM 3D Mill High Speed Machining (HSR/HSM) strategies to machine the part shown above.

- **Rough machining (HSR_R_Cont_target)**

  This HSR Contour roughing operation defines the roughing of the mold cavity. A Ø20 mm (0.75 in) end mill is used along with a Step down of 1 mm (0.04 in). Step down is made adaptive by automatically inserting extra passes. 0.1 mm (0.004 in) is specified for the Precision and the Profile step-in is set equal to the tool diameter. A machining allowance of 0.5 mm (0.02 in) remains on the surfaces for further HSM semi-finishing and finishing operations.

- **Rest roughing (HSR_RestR_target)**

  This HSR Rest roughing operation defines the removal of unmachined material remaining after the previous operation and uses a Ø12 mm (0.4844 in) ball nose mill. A 1.5 mm (0.06 in) Step down is specified in addition to the same machining allowance as in the previous roughing operation.

- **Steep faces semi-finishing (HSM_CZ_target)**

  This HSM Constant Z machining operation defines the semi-finishing of the steep faces (from 40° to 90°). A Ø10 mm (0.4063 in) ball nose mill is used. The Apply fillets option adds virtual fillets to smooth the tool path at the corners and a machining allowance of 0.25 mm (0.01 in) remains on the surfaces for further HSM finishing operations.

- **Shallow faces semi-finishing (HSM_Lin_target)**

  This HSM Linear machining operation defines the semi-finishing of the shallow faces (from 0° to 42°). The Ø10 mm (0.4063 in) ball nose mill is used. Again, the Apply fillets option adds virtual fillets to smooth the tool path at the corners and a machining allowance of 0.25 mm (0.01 in) remains on the surfaces for further HSM finishing operations.

- **Corners semi-finishing (HSM_RM_target)**

  This HSM Rest machining operation defines the semi-finishing of the inside corners. Adding such an operation helps avoid tool overload in the corner areas before finishing. A Ø6 mm (0.2344 in) ball nose mill is used with a Reference tool of Ø12 mm (0.4844 in) to determine the rest material that requires machining. The machining allowance of 0.25 mm (0.01 in) remains on the surfaces for further HSM finishing operations.

- **Steep faces finishing (HSM_CZ_target_1)**

  This HSM Constant Z machining operation defines the finishing of the steep faces (from 40° to 90°). An Ø8 mm (0.3125 in) ball nose mill is used with the Apply fillets option.

- **Shallow faces finishing (HSM_Lin_target_1)**

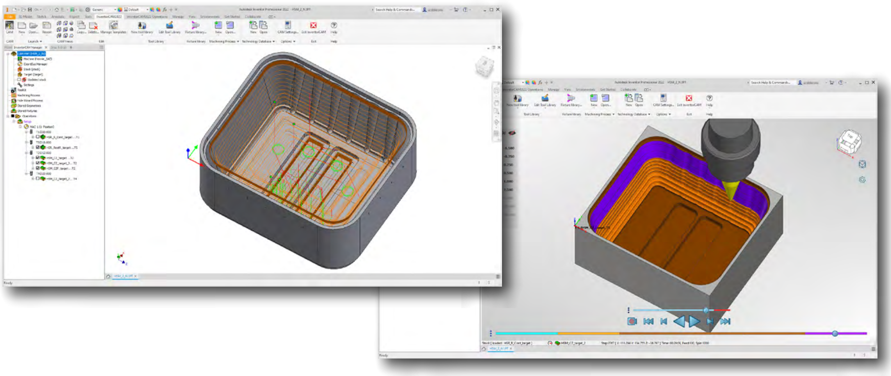
  This HSM Linear machining operation defines the finishing of the shallow faces (from 0° to 42°). The Ø8 mm (0.3125 in) ball nose mill is used with the Apply fillets option.

- **Corners finishing (HSM_RM_target_1)**

  This HSM Rest machining operation defines the finishing of the inside corner areas. A Ø4 mm (0.1563 in) ball nose mill is used with a Reference tool of Ø10 mm (0.4063 in) to determine the rest material that requires machining.

- **Chamfer machining (HSM_Bound_target)**

  This HSM Boundary machining operation defines the chamfering of the upper edges using a Ø6 mm (0.2344 in) chamfer drill.



Click to open the InventorCAM part: **hsm_2_IV.prz**

The operations in this example illustrate the use of several InventorCAM 3D Mill High Speed Machining (HSR/HSM) strategies to machine the part shown above.

- **Rough machining (HSR_R_Cont_target)**

  This HSR Contour roughing operation defines the roughing of the electronic box. A Ø30 mm (1.1563 in) end mill is used along with a Step down of 10 mm (0.4 in). Step down is made adaptive by automatically inserting extra passes. 0.1 mm (0.004 in) is specified for the Precision and the Profile step-in is set equal to the tool diameter. A machining allowance of 0.5 mm (0.02 in) remains on the surfaces for further HSM semi-finishing and finishing operations.

- **Rest roughing (HSR_RestR_target)**

  This HSR Rest roughing operation defines the removal of unmachined material remaining after the previous operation. A Ø16 mm (0.625 in) bull nose mill with a corner radius of 1 mm (0.04 in) is used. A 5 mm (0.2 in) Step down is specified in addition to the same machining allowance as in the previous roughing operation.

- **Upper faces machining (HSM_CZ_target)**

  This HSM Constant Z machining operation defines the vertical faces finishing on the upper ledge down to a certain depth. A Ø12 mm (0.4844 in) bull nose mill with a corner radius of 0.5 mm (0.02 in) is used.

- **Lower faces machining (HSM_CZ_target_3)**

  This HSM Constant Z machining operation defines the vertical faces finishing in the two lower pockets down to a certain depth. The Ø12 mm (0.4844 in) bull nose mill is used.

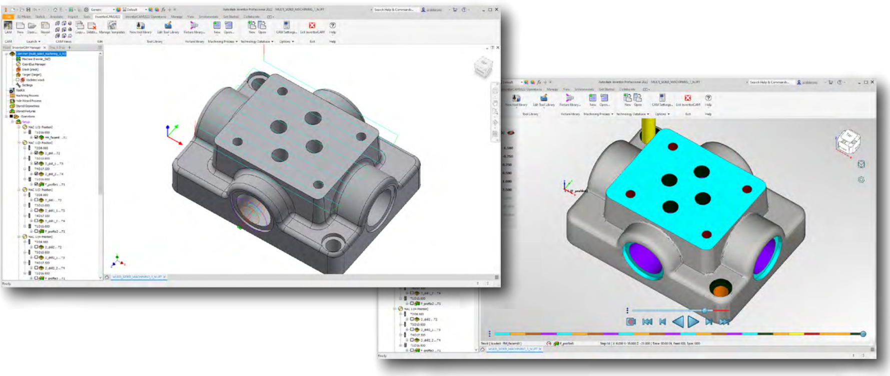
- **Horizontal faces machining (HSM_CZF_target)**

  This HSM Horizontal machining operation defines the finishing of the part horizontal faces. The Ø12 mm (0.4844 in) bull nose mill is used.

- **Inclined faces machining (HSM_CZ_target_2)**

  This HSM Constant Z machining operation defines the finishing of the part inclined faces. A taper mill is used along with a Step down of 10 mm (0.4 in). Such a tool can be used for the machining of internal or external walls with a constant draft angle.

Indexial Multiaxis Machining



Click to open the InventorCAM part: **multi_sided_machining_1_IV.prz**

The operations in this example illustrate the use of InventorCAM's Indexial Multiaxis Machining capabilities to machine the part shown above. The machining is performed on a 5-Axis CNC-Machine in one setup, using one InventorCAM Coordinate System with multiple positions.

- **Top face machining (FM_facemill)**

  This Face Milling operation defines the machining of the part top face using Machine Coordinate System #1, Position #1 (MAC 1 (1- Position)). A Ø16 mm (0.625 in) end mill is used with the Hatch technology to remove 2 mm (0.08 in) of material from the stock top face.

- **Front hole drilling and ledge machining (D_drill; D_drill_1; D_drill_2; F_profile1)**

  These operations define the hole drilling and the ledge machining in the part front face using MAC 1 (2- Position).
  The Drilling operations perform the center drilling and drilling of the holes in three steps using consecutively larger tools (Ø8 mm (0.3125 in), Ø10 mm (0.4083 in) and Ø17.5 mm (0.6875 in) drills).
  The Profile operation performs the roughing and finishing of the ledge around the hole in one step down cut using a Ø16 mm (0.625 in) end mill.

- **Left hole drilling and ledge machining (D_drill1; D_drill1_1; D_drill1_2; F_profile2)**

  These operations define the drilling of the hole and machining of the ledge in the part left face using MAC 1 (3- Position). The Drilling and Profile operations use the same tools and options as the previous group of operations.

- **Back hole drilling and ledge machining (D_drill2; D_drill2_1; D_drill2_2; F_profile3)**

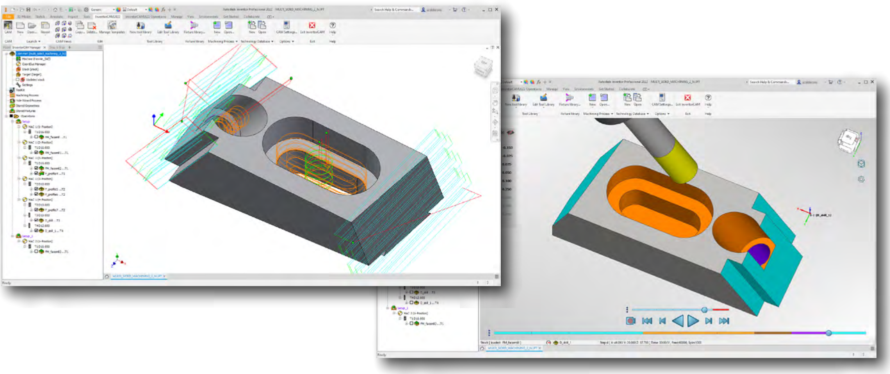
  These operations define the drilling of the hole and machining of the ledge in the part back face using MAC 1 (4- Position). The Drilling and Profile operations use the same tools and options as the previous groups of operations.

- **Right hole drilling and ledge machining (D_drill3; D_drill3_1; D_drill3_2; F_profile4)**

  These operations define the drilling of the hole and machining of the ledge in the part right face using MAC 1 (5- Position). The Drilling and Profile operations use the same tools and options as the previous groups of operations.

- **Top holes and counterbore machining (P_profile5; D_drill4 – D_drill4_2; D_drill5; D_drill5_1; F_profile6)**

  These operations define the machining of the holes on the part top face and the two through holes and counterbore features on the lower opposing corners using MAC 1 (1- Position).



Click to open the InventorCAM part: **multi_sided_machining_2_IV.prz**

The operations in this example illustrate the use of InventorCAM's Indexial Multiaxis Machining capabilities to machine the part shown above. The machining is performed on a 5-Axis CNC-Machine in two setups, using two InventorCAM Coordinate Systems with the first one having multiple positions.

- **Top face machining (FM_facemill)**

  This Face Milling operation defines the machining of the part top face using Machine Coordinate System #1, Position #1 (MAC 1 (1- Position)). A Ø16 mm (0.625 in) end mill is used with the Hatch technology to remove material from the stock top face.

- **Front inclined face machining (FM_facemill1)**

  This Face Milling operation defines the machining of the front inclined face using MAC 1 (2- Position). A Ø16 mm (0.625 in) end mill is used with the Hatch technology to remove material from the front of the stock.

- **Back inclined faces machining (FM_facemill2; F_profile4)**

  These operations define the machining of the back inclined faces using MAC 1 (3- Position). The Face Milling operation uses a Ø16 mm (0.625 in) end mill with the Hatch technology to remove material from the back of the stock. The Profile operation uses the same tool to perform roughing and finishing of the two steps.

- **Slots machining (P_profile5; P_profile6)**

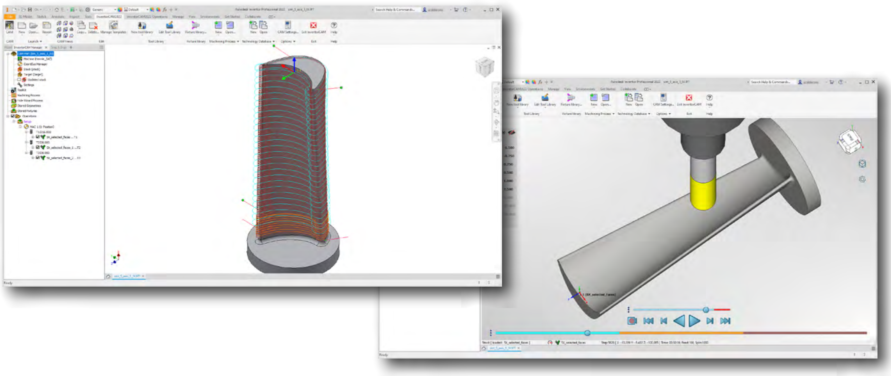
  These Profile operations define the machining of the slots in and through the part top face using MAC 1 (1- Position). Both operations use a Ø10 mm (0.4063 in) end mill for the roughing and finishing.
  The first Pocket operation machines the upper level larger slot. Additional finish passes are executed to remove the remaining 0.2 mm (0.008 in) of material from the wall and floor.
  The second Pocket operation machines the lower level smaller slot. Additional finish passes are executed to remove the remaining 0.2 mm (0.008 in) of material from the wall.

- **Hole and counterbore machining (P_profile7; D_drill; D_drill_1)**

  These operations define the machining of the through hole and counterbore feature along the back inclined faces using MAC 1 (4- Position).

- **Bottom inclined face machining (FM_facemill3)**

  This Face Milling operation defines the machining of the bottom inclined face using MAC 2 (1- Position).



Click to open the InventorCAM part: **sim_5_axis_1_IV.prz**

The operations in this example illustrate the use of InventorCAM's Simultaneous 5-Axis Machining to semi-finish and finish the turbine blade part shown above.

- **Blade Semi-finishing (5X_selected_faces; 5X_selected_faces_1)**

  These Sim. 5-Axis Milling operations use the Parallel cuts strategy to define the semi-finishing of the part.

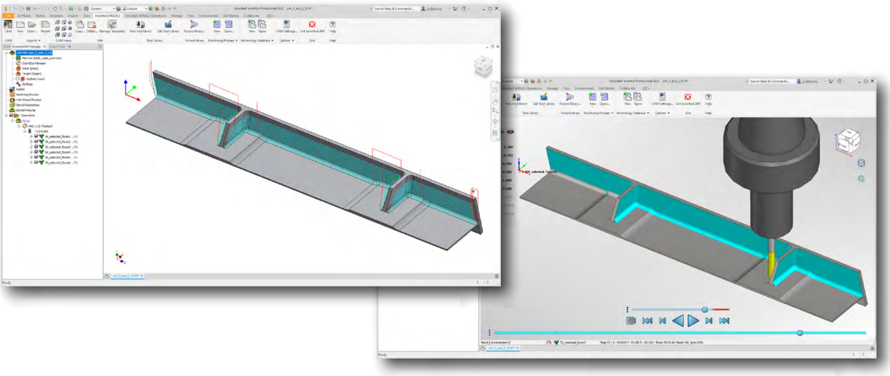
  The first operation provides the semi-finish for the turbine blade surfaces near the end using a Ø16 mm (0.625 in) bull nose mill with a corner radius of 4 mm (0.16 in). Tilted relative to cutting direction defines the tool tilting with a lag angle of 20° and the front face of the tool is specified for the tool contact point. This combination of parameters performs the machining trochoidally by the end surface of the tool. Gouge checking is enabled to detect and avoid possible collisions between the tool, the geometry drive surfaces and the check surfaces of the turbine blade base.

  The second operation provides the semi-finish for the turbine blade surfaces near the base. This area was not machined in the previous operation because of the gouge protection. An Ø8 mm (0.3125 in) bull nose mill with a corner radius of 2 mm (0.08 in) is used. Tilted relative to cutting direction defines the tool tilting with a lag angle of 20° and a side tilting angle of 10°. The front face of the tool is specified for the tool contact point. The same options are enabled for gouge checking as in the previous operation.

  The remaining material from both operations will be removed during the finish machining.

- **Blade finishing (5X_selected_faces_2)**

  This Sim. 5-Axis Milling operation uses the Parallel cuts strategy to define the finishing of the turbine blade surfaces. An Ø8 mm (0.3125 in) bull nose mill with a corner radius of 2.5 mm (0.1 in) is used. Tilted relative to cutting direction defines the tool tilting with a lag angle of 20° and a side tilting angle of 10°. The front face of the tool is specified for the tool contact point. Gouge checking is enabled to detect and avoid possible collisions between the tool and the check surfaces of the turbine blade base. The tool path is generated by a number of cuts that are parallel between the end and base of the turbine blade.



Click to open the InventorCAM part: **sim_5_axis_2_IV.prz**

The operations in this example illustrate the use of InventorCAM's Simultaneous 5-Axis Machining to finish the aerospace part shown above.

- **Inclined walls finishing**
  (5X_selected_faces1; 5X_selected_faces2; 5X_selected_faces3)

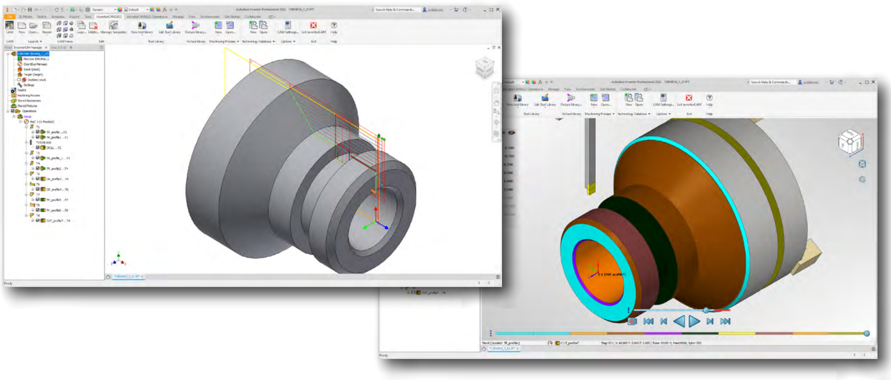
  These Sim. 5-Axis Milling operations use the Parallel cuts strategy to define the finishing of the inclined walls, which are forming undercut areas.

  The three operations provide the finish for the inclined walls that partially surround each area using a Ø4 mm (0.1563 in) ball nose mill. Aside from the geometry selection differences, the Operation definitions are the same. Tilted relative to cutting direction defines the tool tilting with a side tilting angle of 90°. Auto is specified for the tool contact point. This combination of parameters performs the machining axially by the side surface of the tool. For each operation, gouge checking is enabled to detect and avoid possible collisions between the tool, the geometry drive surfaces and the check surfaces of the adjacent fillets. The tool path is generated by a number of cuts parallel to each other along the inclined walls.

- **Fillet finishing**
  (5X_selected_faces4; 5X_selected_faces5; 5X_selected_faces6)

  These Sim. 5-Axis Milling operations use the Projection (User-defined) strategy to define the finishing of the fillets adjacent to the inclined walls.

  The three operations provide the finish for the adjacent fillets in each area using the Ø4 mm (0.1563 in) ball nose mill. Aside from the geometry selection differences, the Operation definitions are the same. Tilted through curve defines the tool tilting with a variable tilting angle. Auto is specified for the tool contact point. This combination of parameters performs a smooth transition between different tool axis orientations. For each operation, gouge checking is enabled to detect and avoid possible collisions between the tool and the geometry drive surfaces. The tool path is generated by a single pass that follows the lower curves of the fillets projected on the drive surfaces.



Click  to open the InventorCAM part: **turning_1_IV.prz**

The operations in this example illustrate the use of several InventorCAM Turning technologies to machine the part shown above.

- **Contour roughing (TR_profile)**

  This Turning operation defines the rough machining of the external surfaces using an external roughing tool. The Rough option is chosen for the work type to perform the machining in a number of equidistant passes. Long external mode is used to create the passes above the geometry and along the rotational axis of the part. A 0.1 mm (0.004 in) Z-Offset remains on the surface for further finishing operations.

- **Front face turning (TR_profile1)**

  This Turning operation defines the machining of the front face using an external roughing tool. Face front mode is used to create the roughing and finishing passes along the front face of the part.

- **Drilling (DRILL)**

  This Drilling operation defines the drilling of the hole through the center of the part using a Ø28 mm (1.1015 in) flat drill.

- **Contour finishing (TR_profile_1)**

  This Turning operation defines the finish machining of the external surfaces using the Finish only option to perform a finishing cut on the rest material.

- **Internal finishing (TR_profile2)**

  This Turning operation defines the finish machining of the internal surfaces including the chamfer. The Finish only option is chosen for the work type and Long internal mode is used to perform the finishing cut under the geometry and along the rotational axis of the part.

- **Grooving (GR_profile3; GR_profile4)**

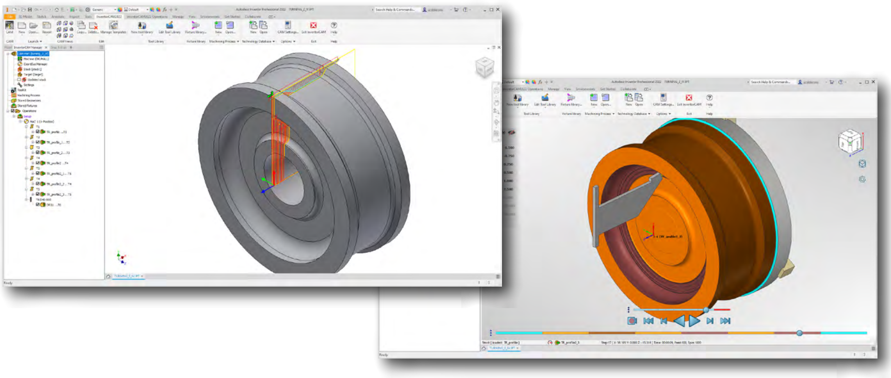
  These Grooving operations define the machining of the external and internal grooves using the appropriate grooving tools.

- **Threading (TH_profile5; TH_profile6)**

  These Threading operations define the machining of the external and internal threads using the appropriate threading tools. A user-defined Thread size and Pitch unit are specified with thread finishing.

- **Part cutoff (CUT_profile7)**

  This CutOff operation defines the parting (cutoff) of the machined part from the stock bar using an external grooving tool.



Click to open the InventorCAM part: **turning_2_IV.prz**

The operations in this example illustrate the use of several InventorCAM Turning technologies to machine the part shown above.

- **Contour roughing (TR_profile)**

  This Turning operation defines the rough machining of the external surfaces using an external roughing tool. The Rough option is chosen for the work type to perform the machining in a number of equidistant passes. Long external mode is used to create the passes above the geometry and along the rotational axis of the part.

- **Contour rest machining and finishing (TR_profile_1; TR_profile_2)**

  These Turning operations define the rest and finish machining of the external surfaces. In the first operation, an external roughing tool with different setup angle is used to remove unmachined material from the external contour. In the second operation, the Finish only option is chosen for the work type to perform the finishing cut.

- **Front face and internal roughing (TR_profile2; TR_profile2_1)**

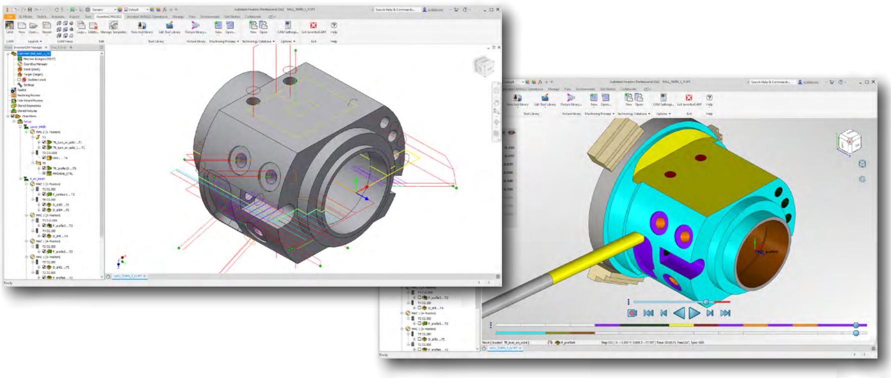
  These Turning operations define the rough machining of the front face and internal surfaces using the Rough option with Front face mode. The first operation performs roughing of the external and internal front faces. The second operation uses a different tool mounting orientation to perform roughing cuts on the large internal fillet.

- **Front face and internal finishing (TR_profile2_2; TR_profile2_3)**

  These Turning operations define the finish machining of the front face and internal surfaces using the Finish only option. As in the previous operations, different tool mounting orientations are used to perform finishing cuts on the rest material.

- **Drilling (DRILL)**

  This Drilling operation defines the drilling of the hole through the center of the part using a Ø40 mm (1.575 in) flat drill.



Click to open the InventorCAM part: **mill_turn_1_IV.prz**

The operations in this example illustrate the use of InventorCAM's Mill-Turn module to machine the part shown above. The machining is performed on a 5-Axis Mill-Turn CNC-Machine using several InventorCAM Milling and Turning technologies and Indexial Multiaxis Machining capabilities.

- **Contour and internal turning (TR_turn_on_solid; TR_turn_on_solid_1; DRILL; TR_profile10)**

  These Turning operations define the machining of the external and internal cylindrical surfaces using the appropriate tools, machining modes, work types, roughing types and finishing methods.

- **Facial milling (F_contour1; D_drill3; D_drill4)**

  These Milling operations define the machining of the screw slot and the drilling of the through hole and three blind holes using Machine Coordinate System #1, Position #1 (MAC 1 (1- Position)).

- **Side faces machining (P_profile3)**

  This Pocket operation defines the indexial machining of the side faces using MAC 1 (3- Position). The Contour strategy is combined with a negative Wall offset value to generate an overlapping tool path that completely machines the faces. The Transform option creates a circular pattern of operations around the revolution axis in order to machine both faces.

- **Side face drilling (D_drill)**

  This Drilling operation defines the indexial drilling of the two holes located on the side face using MAC 1 (3- Position).

- **Slot machining (F_profile5)**

  This Profile operation defines the indexial machining of the slot using MAC 1 (4- Position). The material is removed at a constant depth followed by an additional finish pass.

- **Radial holes machining (D_drill1; P_profile6; D_drill2; P_profile7)**

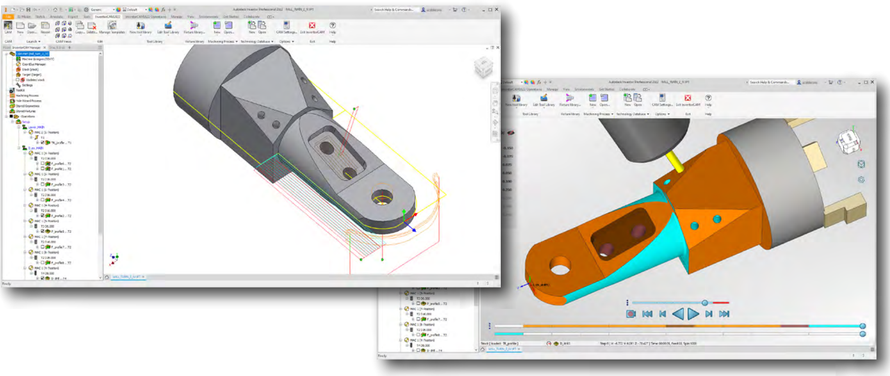
  These operations define the indexial machining of the three counterbore holes on the external cylindrical surface.

  The first Drilling and Profile operations use MAC 1 (5- Position) to perform the hole drilling and counterbore machining on one side of the screw slot.

  The following Drilling and Profile operations use MAC 1 (6- Position) to perform the holes drilling and counterbores machining on the other side of the screw slot.

- **Wrapped pocket machining (P_profile9)**

  This Pocket operation defines the simultaneous machining of the pocket on the external cylindrical surface using MAC 1 (2- Position). Wrapped is selected for the geometry type and the Contour strategy is used to perform the pocket machining.



Click to open the InventorCAM part: **mill_turn_2_IV.prz**

The operations in this example illustrate the use of InventorCAM's Mill-Turn module to machine the part shown above. The machining is performed on a 5-Axis Mill-Turn CNC-Machine using several InventorCAM Milling and Turning technologies and Indexial Multiaxis Machining capabilities.

- **Contour turning (TR_profile)**

  This Turning operation defines the machining of the cylindrical surface.

- **Cube machining (F_profile6)**

  This Profile operation defines the indexial machining of the cube faces at the base of the part using Machine Coordinate System #1, Position #4 (MAC 1 (4- Position)). The Transform option creates a circular pattern of operations around the revolution axis in order to machine all the faces.

- **Horizontal faces machining (F_profile1)**

  This Profile operation defines the indexial machining of the horizontal faces at the front of the part using MAC 1 (4- Position). The Transform option creates a circular pattern of operations around the revolution axis in order to machine both faces.

- **Inclined faces machining (F_profile3; F_profile4)**

  These Profile operations define the machining of the inclined faces that are adjacent to the previously machined horizontal faces using MAC 1 (5 and 6- Positions) and the B-Axis tilting capabilities of the tool.

- **Radial face machining (F_profile2)**

  This Profile operation defines the machining of the cylindrical face at the front of the part using MAC 1 (4- Position).

- **Inclined pocket machining (P_profile5)**

  This Pocket operation defines the machining of the pocket on the previously machined inclined face using MAC 1 (5- Position) and the B-Axis tilting capabilities of the tool.

- **Cube inclined faces machining (F_profile7; F_profile8)**

  These Profile operations define the machining of the inclined faces on the cube corners using MAC 1 (7 and 8- Positions) and the B-Axis tilting capabilities of the tool.

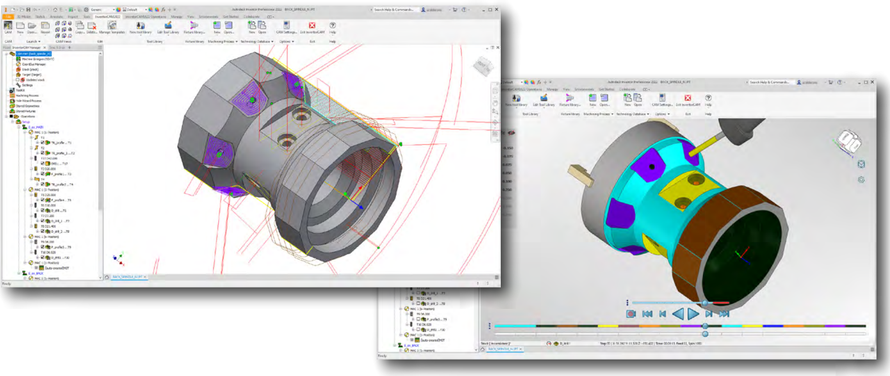
- **Through holes drilling (D_drill; D_drill1)**

  These Drilling operations define the drilling of the hole through the horizontal faces at the front of the part using MAC 1 (4- Position) and the drilling of the holes through the adjacent inclined faces using MAC 1 (6- Position) and the B-Axis tilting capabilities of the tool.

- **Cube inclined faces drilling (D_drill2; D_drill3)**

  These Drilling operations define the drilling of the holes on the cube inclined faces using MAC 1 (7 and 8- Positions) and the B-Axis tilting capabilities of the tool.

Mill-Turn – 2 Spindles



Click to open the InventorCAM part: **back_spindle_IV.prz**

The operations in this example illustrate the use of InventorCAM's Mill-Turn module with front and back spindles functionality to machine the part shown above. The machining is performed on a 5-Axis Mill-Turn CNC-Machine using several InventorCAM Milling and Turning technologies and Indexial Multiaxis Machining capabilities.

- **Main spindle contour and internal turning (TR_profile; TR_profile_1; DRILL; TR_profile2)**

  These Turning operations define the machining of the external and internal cylindrical surfaces on the front side using the appropriate tools, machining modes, work types, roughing types and finishing methods.

- **Main spindle facial milling (F_profile1)**

  This Profile operation defines the machining of the facets around the external surface on the front side using Machine Coordinate System #1, Position #1 (MAC 1 (1- Position)). The material is removed at a constant depth followed by an additional finish pass.

- **Main spindle middle part pads machining (F_profile4)**

  This Profile operation defines the indexial machining of the pads around the cylindrical surface on the middle of the part using MAC 1 (5- Position). The Transform option creates a circular pattern of operations around the revolution axis in order to machine all the pads.

- **Main spindle middle part holes machining (D_drill; D_drill_1; D_drill_2)**

  These operations define the indexial machining of the counterbore holes on the previously machined pads using MAC 1 (5- Position). The Transform option creates a circular pattern of operations around the revolution axis in order to machine all the holes and counterbores.

  The first two Drilling operations perform the center drilling and drilling of the holes.

  The last Drilling operation performs the counterbores machining.

- **Main spindle inclined pads machining/drilling (P_profile5; D_drill1)**

  These Profile and Drilling operations define the indexial machining/drilling of the pads and holes around the conical surface near the back side using MAC 1 (6- Position) and the B-Axis tilting capabilities of the tool. The Transform option creates a circular pattern of operations around the revolution axis in order to machine all the pads and holes.

- **Back spindle contour turning and facial milling (TR_profile6; F_profile7)**

  These Turning and Profile operations define the initial machining of the external cylindrical surface on the back side and the final machining of the facets around the external surface using MAC 3 (1 and 2- Positions).

- **Back spindle internal turning (DRILL_1; TR_profile8)**

  These Turning operations define the machining of the internal cylindrical surfaces on the back side using the appropriate tools, machining mode, work type, roughing type and finishing method.

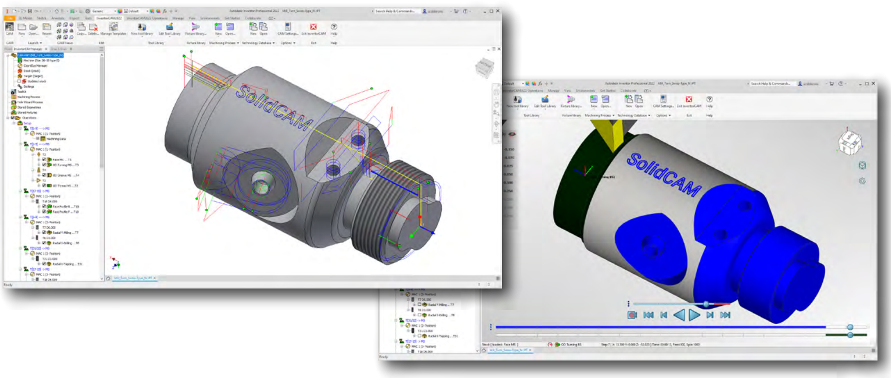
- **Back spindle face machining (F_profile9; D_drill2; D_drill2_1)**

  These operations define the machining of the features on the back face using MAC 3 (2- Position).

  The Profile operation performs the ledge machining.

  The Drilling operations perform the center drilling and drilling of the holes on the ledge corners.

Mill-Turn – Swiss-Type



Click to open the InventorCAM part: **Mill_Turn_Swiss-Type_IV.prz**

The operations in this example illustrate the use of InventorCAM's Mill-Turn module with MCO cycles to machine the part shown above. The machining is performed on a Swiss-Type CNC-Machine using InventorCAM Milling and Turning technologies. Typical parts are small and complex and often require high precision. Swiss-Type CNCs have a sliding headstock and a guide bushing through which the stock is fed, supported and nearby later cut. Such machines can also have a large number of axes, are exceptionally stable and regularly provide remarkable machining cycles.

- **Machining Data Definition (Machining Data)**

  This Machine Control Operation (MCO) defines the main Machining Data parameters of the machine that are required for machining processes.

- **Front face and contour turning (Face MS; OD Turning MS)**

  These Turning operations define the machining of the face and cylindrical surfaces on the front side using the appropriate tools, machining modes, roughing/work types and finishing methods.

- **Grooving and threading (OD Groove MS; OD Thread MS)**

  These operations define the machining of the groove and thread features respectively using an external grooving and threading tool.

- **Front face machining (Face Profile R; Face Profile F)**

  These Profile operations define the rough and finish machining of the rounded feature on the front face of the part.

- **Horizontal pad machining (Radial Y-Milling)**

  This Pocket operation defines the open pocket machining of the pad feature near the front side in preparation for the holes machining.

- **Holes machining (Radial X-Drilling; Radial X-Tapping)**

  These Drilling operations define the drilling and tapping of the holes located on the previously machined horizontal pad feature.

- **Inclined pad/faces machining (20deg Middle Pad/Boss Face; 40deg Adj Face)**

  These operations define the machining of the three features that intersect the part at an angle.

  The first Pocket operation performs the machining of the circular pad into the cylindrical surface on the middle of the part and the Profile operation machines the top face of the tapered boss feature.

  The second Pocket operation performs the machining of the face adjacent to the horizontal pad near the groove feature.

- **Tapered boss machining (5X Sim Boss; Drill B1 - D3)**

  These operations define the machining of the tapered boss located on the previously machined circular pad feature.

  The Sim. 5-Axis Milling operation machines the external surface of the boss using the Parallel to curves strategy.

  The Drilling operation performs the drilling of the hole into the boss center.

- **Wrapped logo machining (SolidCAM logo)**

  This Profile operation defines the engraving of the SolidCAM logo on the part cylindrical surface using the Wrapped functionality.

- **Part pick up (Main Spindle - B1 Safety; Main Spindle/Back Spindle - PickUps)**

  These MCOs define the process required to catch the part after cutoff from the main spindle and to prepare the machine for part transfer.

- **Part cutoff (CutOff - MS)**

  This Cutoff operation defines the cutting of the part off the main spindle.

- **Part transfer (Main Spindle/Back Spindle - Part Transfers)**

  These MCOs define the process required for transferring the part from the main spindle to the back spindle.

- **Back face and contour turning (Face BS; OD Turning BS)**

  These Turning operations define the machining of the face and cylindrical surfaces on the back side using the appropriate tools, machining modes, roughing/work types and finishing methods.

- **Part release (Back Spindle - X2 Z2 to Reference; Back Spindle - Part Release)**

  These MCOs define the process required to prepare the machine for and to release the part from the back spindle.

System Requirements

The system requirements for running InventorCAM are similar to those required for Autodesk Inventor:

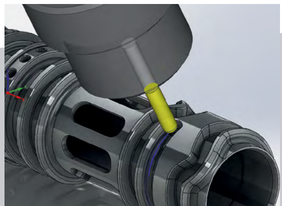
- Microsoft® Windows 10 Pro, 64-bit operating system
- Intel® or AMD® 3.0 GHz CPU or greater, 4 or more cores
- 16 GB RAM or more (32 GB minimum is recommended for large CAM-Parts)
- 2 GB GPU with 80 GB/s Bandwidth, or better (visit knowledge.autodesk.com/certified-graphics-hardware for Inventor certified graphics hardware)
- SSD drives recommended for optimal performance
- 1280 x 1024 or higher display resolution
- Mouse or other pointing device
- Internet connection for web downloads, online activation of license, and use of eSupport system
- Adobe Acrobat version 9 or higher is recommended for viewing InventorCAM user documentation

InventorCAM 2022

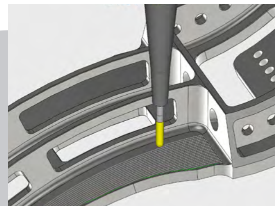
InventorCAM 2022

Modules Overview

iMachining 2D

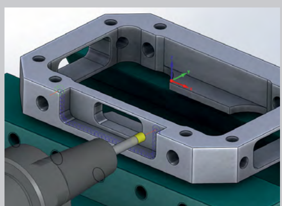


2.5D Milling

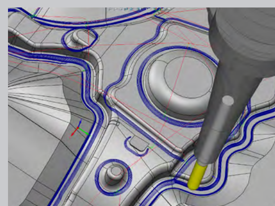


HSS (High-Speed Surface Machining)

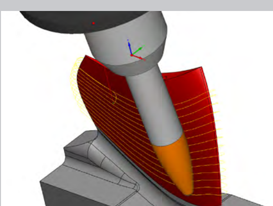
iMachining 3D



Indexial Multi-Sided Machining

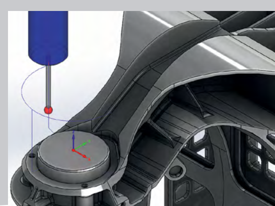


HSM (High-Speed Machining)



Sim. 5-Axis Machining

Turning & Advanced Mill-Turn



Solid Probe

facebook.com/SolidCAM
facebook.com/iMachining

youtube.com/SolidCAMProfessor
youtube.com/SolidCAMiMachining

iMachining Successes
solidcam.com/successes/imachining-successes

www.solidcam.com

InventorCAM
The Solid Platform for Manufacturing