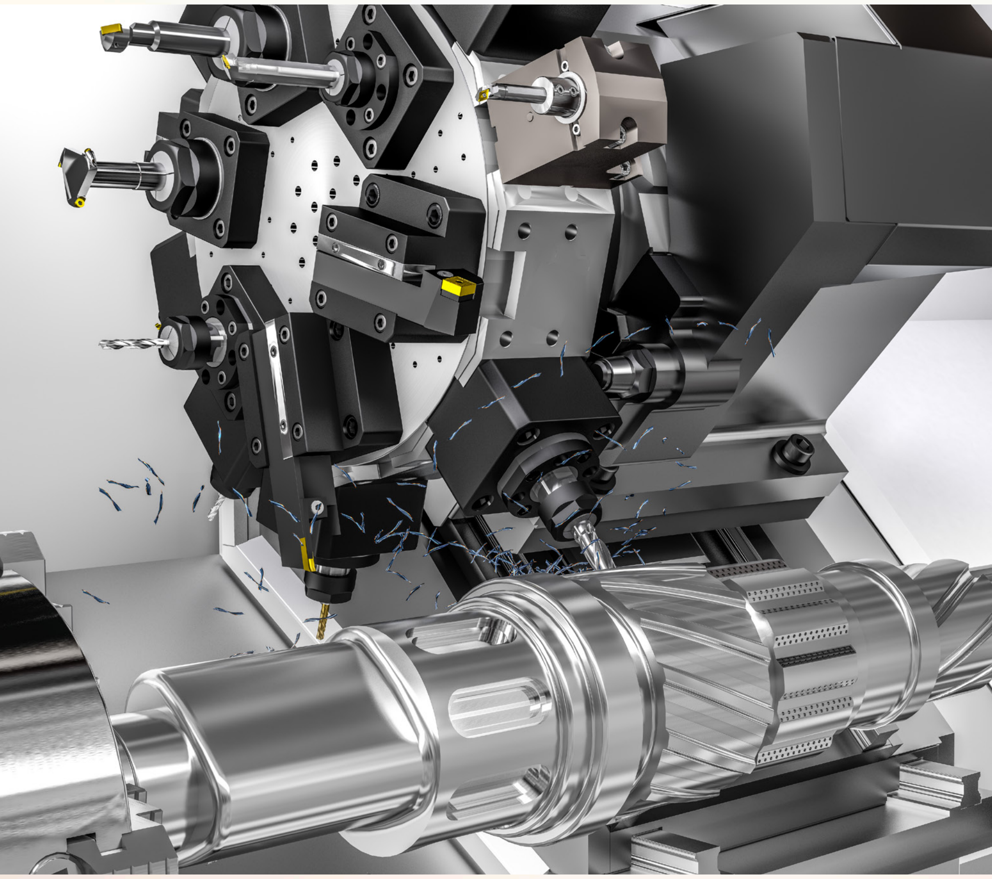
SolidCAM
The Solid Platform for Manufacturing

SolidCAM 2023

Modules Overview



- iMachining 2D
- iMachining 3D
- 2.5D Milling
- Indexial Multi-Sided
- HSS Machining
- 3D High Speed Milling
- Simultaneous 5-Axis
- Turning
- Advanced Mill-Turn
- Swiss-Type
- Solid Probe

SolidCAM MillTurn & Swiss | The unique, revolutionary Milling technology iMachining® patent by SolidCAM | SolidCAM Additive Manufacturing

Table of Contents

Document number: SCMOENG1_2023

About this Booklet

This Modules Overview Booklet provides information about the available CAM Modules that are offered by SolidCAM.

The uses and features of the SolidCAM Modules are demonstrated in the programming of various CAM-Parts. To download the CAM-Parts for use with this booklet or for "Getting Started" purposes, visit:

www.solidcam.com/subscription/getting-started-cam-parts

This Booklet and the provided CAM-Parts are supplemented by SolidCAM Professor videos. To view the videos, visit:

www.solidcam.com/videos/solidcam-professor-video-tutorials/modules-overview

To explore more Professor videos that cover additional features and uses of SolidCAM, visit:

www.solidcam.com/videos/solidcam-professor-video-tutorials

For the most up-to-date PDF version of this Booklet, visit:

www.solidcam.com/subscription/documentation

ABOUT SOLIDCAM

With 40 years of experience in the development and support of SolidCAM, the most powerful CAM solution has been created that takes your CNC-Machines to maximum productivity. Founded in 1984, SolidCAM's strategy of integrating with the most popular CAD systems has created tremendous growth and established SolidCAM as the ultimate solution for integrated CAM systems. SolidCAM's Add-ins for SOLIDWORKS, Solid Edge and Autodesk Inventor provide seamless, single-window integration and full associativity to their design models.

SOLIDCAM MISSION

In today's world of manufacturing, productivity counts, and every CNC-Machine must be utilized to its maximum capabilities. SolidCAM provides the ability to drive CNC-Machines in the most efficient and productive way.

In 2011, the revolutionary iMachining was launched, taking the machining industry to a new level. Still today, the patented iMachining technology offers tens of thousands of users unique, amazing benefits, including more than 70% faster machining times and dramatically increased tool life. In addition, iMachining's patented Technology Wizard guides machinists all over the world to perfect machining results on every single part.

SolidCAM is the perfect solution for multi-tasking machining needs, with the ultimate in programming flexibility and configurability. With full channel synchronization, you can optimally program multi-turret and multi-spindle operations, then watch SolidCAM's simulation of your part being machined in multiple stages. SolidCAM best supports Mill-Turn and Swiss-Type Machines, including multi-channel synchronization.

SolidCAM's large customer base spans across all industries including aerospace, automotive, electronics, medical, optics, energy, mold making, prototyping, and more. Customers include job shops, medium size engineering and manufacturing companies, large aerospace and automotive companies as well as technical colleges and vocational institutions.

The perfect CAM Solution involves both the best software and the best support. SolidCAM's technical and post-processor support, appreciated by tens of thousands of customers, is a core company philosophy that is routinely enhanced to provide the best possible service.

THE SOLIDCAM ADVANTAGE

- Easiest-to-use CAM System with short learning curve
- Seamlessly integrated in SolidWorks, Solid Edge and Autodesk Inventor, with extensive import of all common CAD data formats
- Patented iMachining – amazing and unique technology
- The leading integrated CAD/CAM Solution that can control the most complex Mill-Turn & Swiss-Type CNC-Machines
- SolidCAM supports Hybrid Manufacturing – combining 3D Metal Printing with CNC Manufacturing

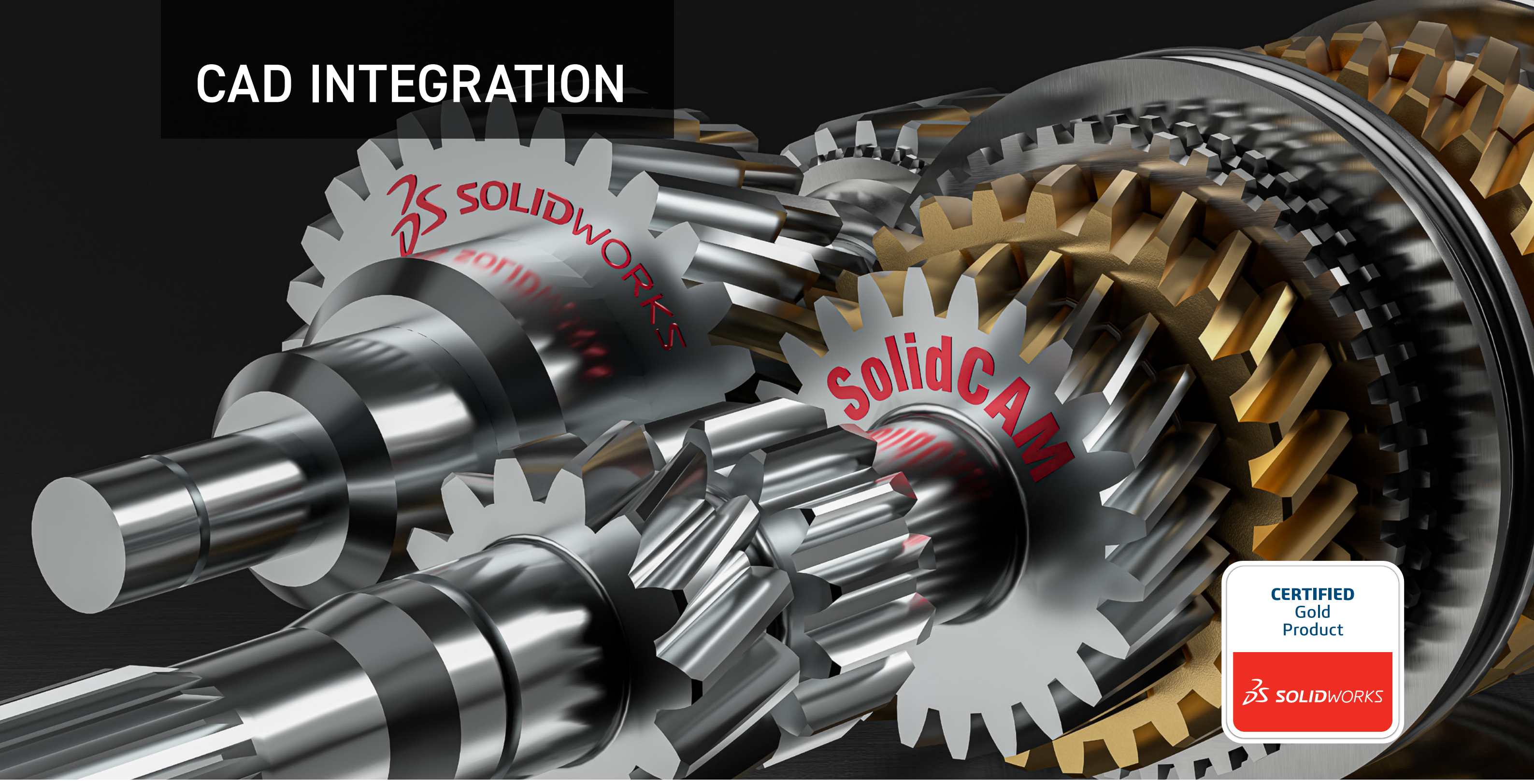
SOLIDCAM TECHNOLOGY PARTNERS

SolidCAM's worldwide cooperation with a large number of leading suppliers of CNC-Machine tools, CNC controllers, cutting tools, tool holders, fixtures and clamping, as well as tool path verification and tool data integrators, has great benefits for its large customer base.

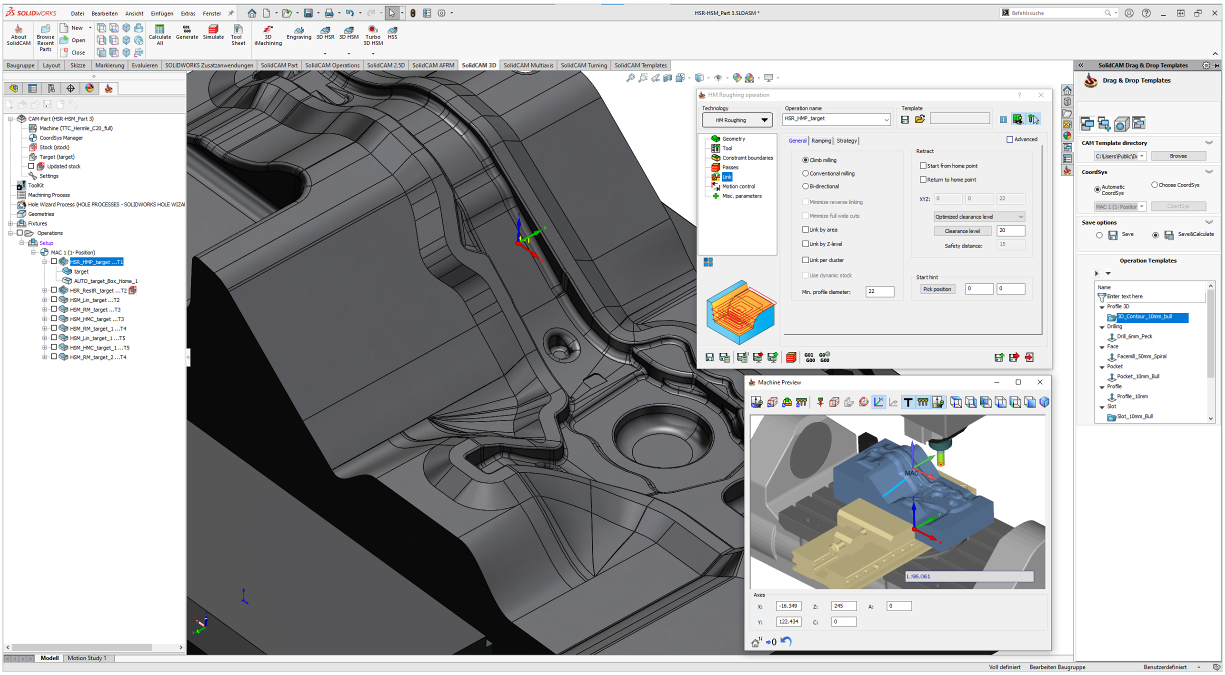
GROB
brother
star
CITIZEN
TSUGAMI
NOMURA SWISS
KENNAMETAL
SIEMENS
TORNOS
DMG MORI
CGTech VERICUT
FANUC
NEXTURN SWISS TURN LEADER
HEDELIUS
Hoffmann Group
HEIDENHAIN
OKUMA OPEN POSSIBILITIES
HERMLE besser fräsen
ALBRECHT Präzisions Spannfutter
ALZMETALL we drive productivity
röders TEC
Member IMC Group ISCAR
FAHRION PRÄZISION
ZOLLER expect great measures
EMUGE FRANKEN
SECO
MHT INTELLIGENZ FÜR WERKZEUGMASCHINEN
CERATIZIT GROUP
WIDIA
MAPAL
SPREITZER
SCHUNK
BLUM focus on productivity
GÜHRING

For a more complete listing, please visit www.solidcam.com

CAD INTEGRATION



SolidCAM is the Complete, 'Best-in-Class' CAM Suite for Profitable CNC Programming in SOLIDWORKS



Major Benefits of SolidCAM seamlessly integrated in SOLIDWORKS:

- SOLIDWORKS look and feel through seamless single window integration – with full support for modern 4K displays
- Full associativity: tool paths automatically update when the SOLIDWORKS model changes
- SolidCAM works in the SOLIDWORKS assembly mode to define fixtures, tooling and vices

With the single window integration, all machining operations can be defined, calculated and verified without leaving the SOLIDWORKS assembly environment.

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

SolidCAM + SOLIDWORKS is scalable for all CNC-Machine types and applications. The integrated CAD/CAM Solution, is available from SolidCAM, as a bundle-package.

Customers rave about SolidCAM's integration in SOLIDWORKS

This approach shortens the learning curve for programmers, offers greater geometry editing and manipulation power to manufacturing and provides a common tool for supporting interaction between designers and machinists."

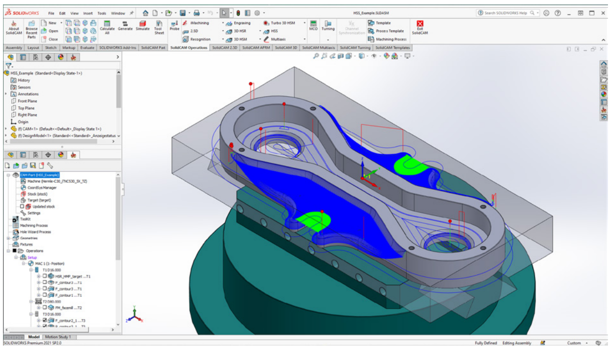
If changes are made on the manufacturing side, we capture them on both the design side and the manufacturing side because SOLIDWORKS and SolidCAM are fully associative."

The integrated approach facilitates discussion and resolution of manufacturing issues because everyone is working with the same model and modeler. We communicate issues and features a lot better working with an integrated system."

The integrated approach has a lot of advantages, including saving time, accessing a single geometry file, and using the intelligence of our design data in a more efficient, systematic way."

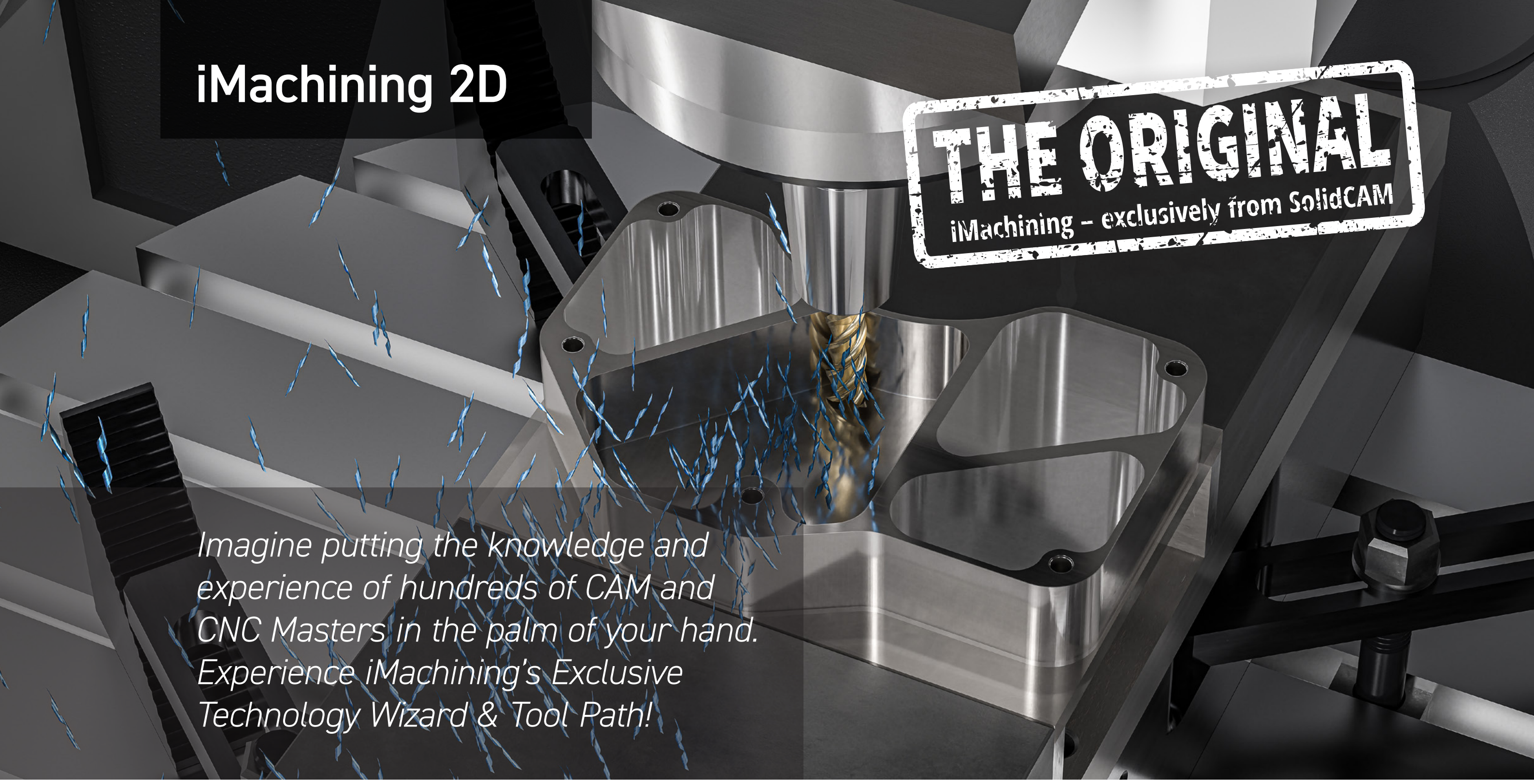
SolidCAM is the Swiss pocket knife for machining. With the modules for the 2.5D, 3D, Simultaneous 5-Axis machining and rotary milling, all daily machining tasks can be done quickly – from the complex drilling pattern to the most demanding 5-Axis impeller. Program the part, simulate and off you go on the machine. The software delivers, what it has promised!"

You Never Have to Leave the SOLIDWORKS Window!



Since loading the SolidCAM trial version integrated in SOLIDWORKS, I've been able to program complex parts and run them without concern. The machine seems to run smoother than before, cutters last longer and confidence levels are high. I am able to train others here to use SolidCAM with ease. The software is pretty self-explanatory and the tutorials are easy to follow."

The tight integration with SOLIDWORKS makes my design-to-production life cycle easy and fast. The SolidCAM support team is rock solid. I do some pretty complex 4-Axis production projects and SolidCAM handles them very nicely."

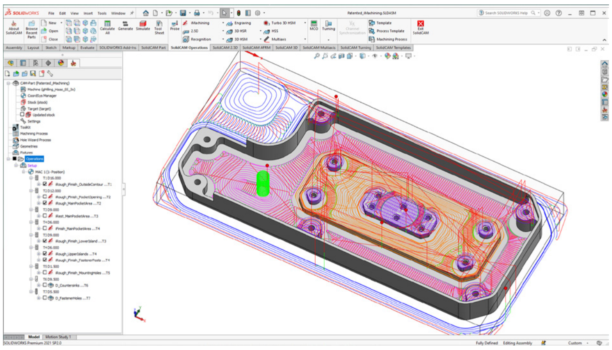


iMachining 2D

Imagine putting the knowledge and experience of hundreds of CAM and CNC Masters in the palm of your hand. Experience iMachining's Exclusive Technology Wizard & Tool Path!

Patented iMachining: "Truly Amazing"

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



The Revolution in CNC Machining

- Increased productivity due to shorter cycles times - 70% savings and more!
- Dramatically increased tool life – 5 times and more
- Unmatched hard material machining
- Outstanding small tool performance
- 4-Axis and Mill-Turn iMachining
- Automatic, optimal feeds and speeds
- High programming productivity
- Shortest learning curve in the CAM industry

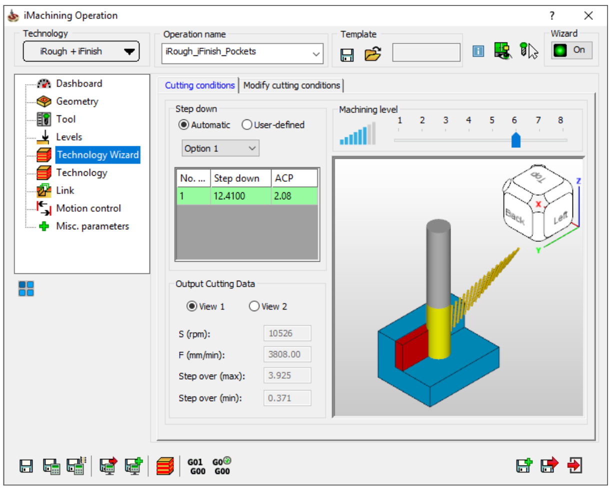
Unique Technology Wizard

SolidCAM's iMachining has the exclusive patented Technology Wizard, the industry's first and only Wizard that automatically calculates optimal cutting conditions for every segment of the iMachining tool path.

The Wizard provides synchronized values of feed rate, spindle speed, axial depth of cut, cutting angles and chip thickness based on the mechanical properties of the workpiece and tool, while also taking into account the technical limits of the CNC-Machine.

The "iMachining level slider" lets the user choose from 8 selectable levels, to automatically adjust for "real-world" fixture, tool holding and machine conditions. The slider makes it easy to overcome standard problems with spindle rigidity, fixture rigidity and cutting tool stability.

All SolidCAM customers worldwide, who use iMachining, are enjoying immense savings and have gained a real competitive advantage.



iRough, iRest, iFinish and Multi-tool Technologies

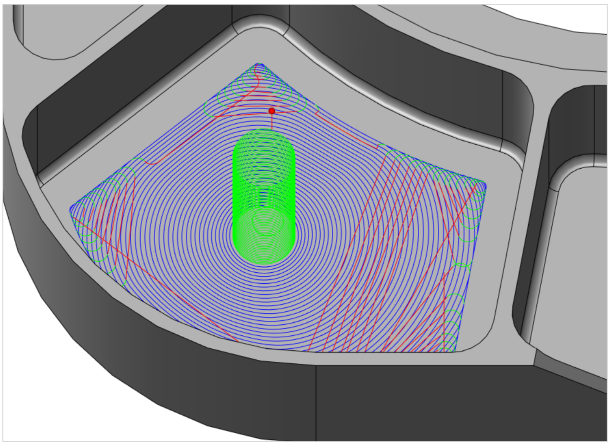
- Combined roughing, finishing and rest material functionality in one single job.
- Multi-tool: Easily define and edit related jobs that use multiple tools, all from within a single interface. Geometry and Levels are synchronized and rest material is tracked automatically.
- iRough + iFinish: Optimized roughing and finishing in the same job with the same tool. Ideal for prototyping and the machining of soft materials.
- iFinish: Suitable for hard materials and precise machining with separate tool for finishing floors and walls.
- Optimized rest roughing and bottom finishing of 2.5D features with various strategies.
- Automatic recognition and removal of rest material remaining through the drill tip.

Distinctive & Proprietary iMachining Tool paths!

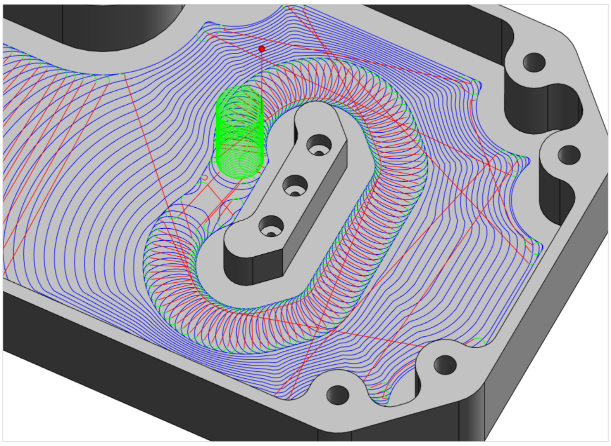
Morphing Spirals – iMachining uses an advanced, patented morphing spiral that gradually conforms to the geometry of the feature being machined rather than a conventional offset tool path. This maximizes tool to stock contact or "tool in the cut" time.

Channels and Moats: Divide & Conquer – In order to most efficiently attack large areas of material removal as well as stand-alone islands, they are separated or subdivided into smaller sections using iMachining's patented Moating technology. This maximizes the continuous morphed spiral cutting.

Eliminate Wasted Time & Motion – iMachining tool paths only cut the stock that needs to be removed, eliminating "air cuts". From the initial approach, right to the last cut, rest material tracking ensures every tool path is always efficiently cutting material.

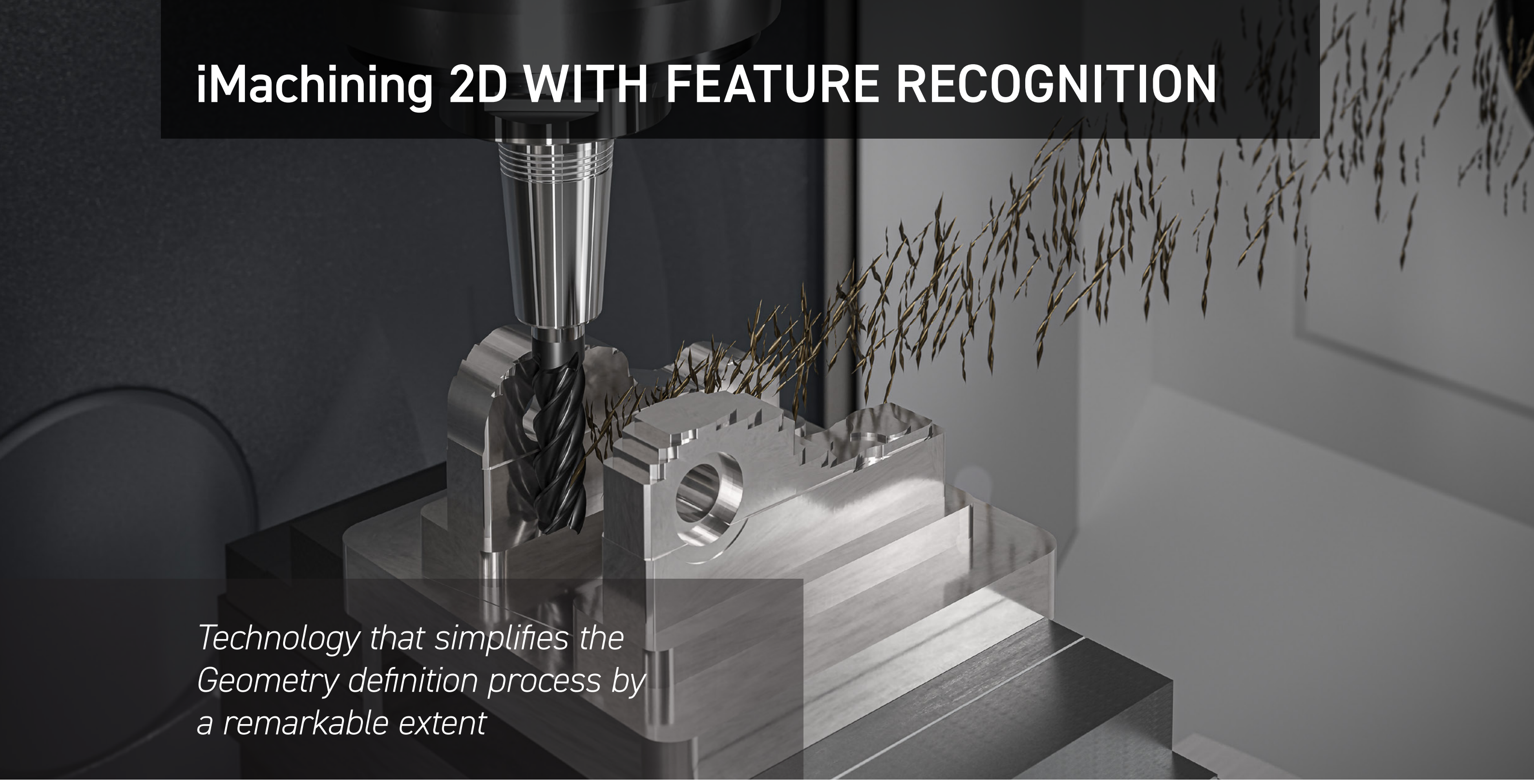


Most efficient iMachining Morphing Spirals tool path



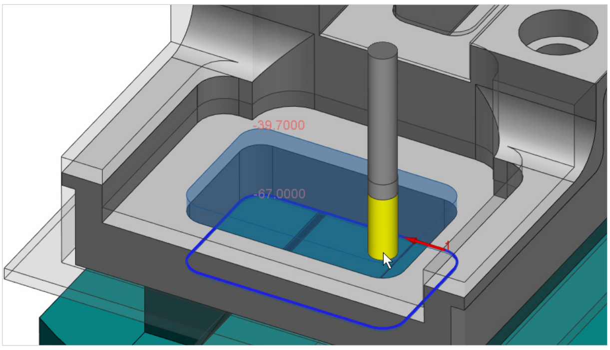
Moating: Intelligent Division of Areas to maximize Morphed Spirals

iMachining 2D WITH FEATURE RECOGNITION



Technology that simplifies the Geometry definition process by a remarkable extent

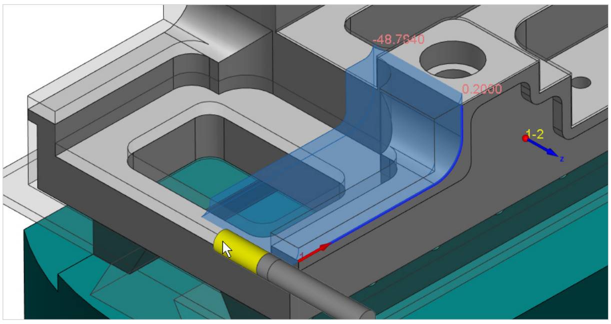
iMachining 2D's Feature Recognition technology detects and defines the part machinable features by utilizing the solid model data, with minimal input from the user.



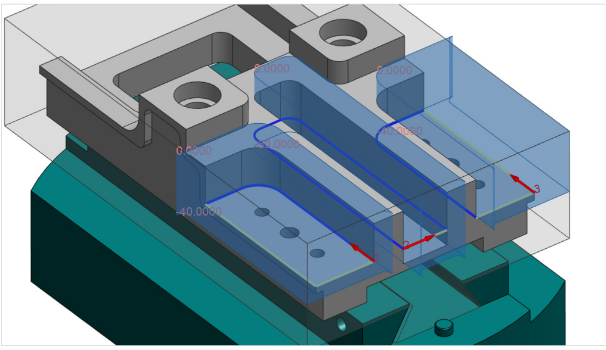
Chains Recognition for Through Pockets

Feature Recognition Modes

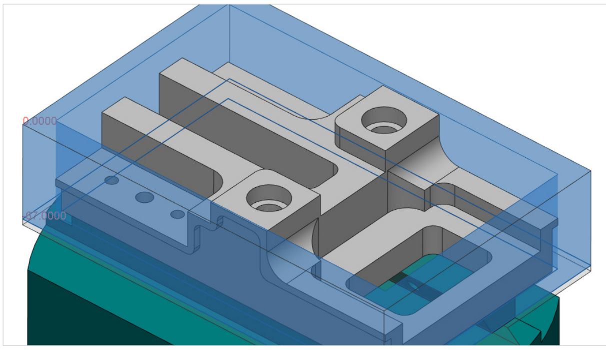
- **Faces:** Smart Face technology builds chains by just the simple selection of faces. Entire pocket features and their levels, which can consist of varying depths, are recognized automatically.
- **Chains:** Machinable areas are recognized by chains in combination with the solid model data. Perfect for features not having a floor face to select, such as when milling through pockets and side profiles.
- **Outside Feature Recognition:** Machinable stock surrounding the target is recognized and its levels are detected automatically.
- **Chains without Feature Recognition:** Option to use SolidCAM's standard chaining method, without iMachining's Recognition and Protection functionality.



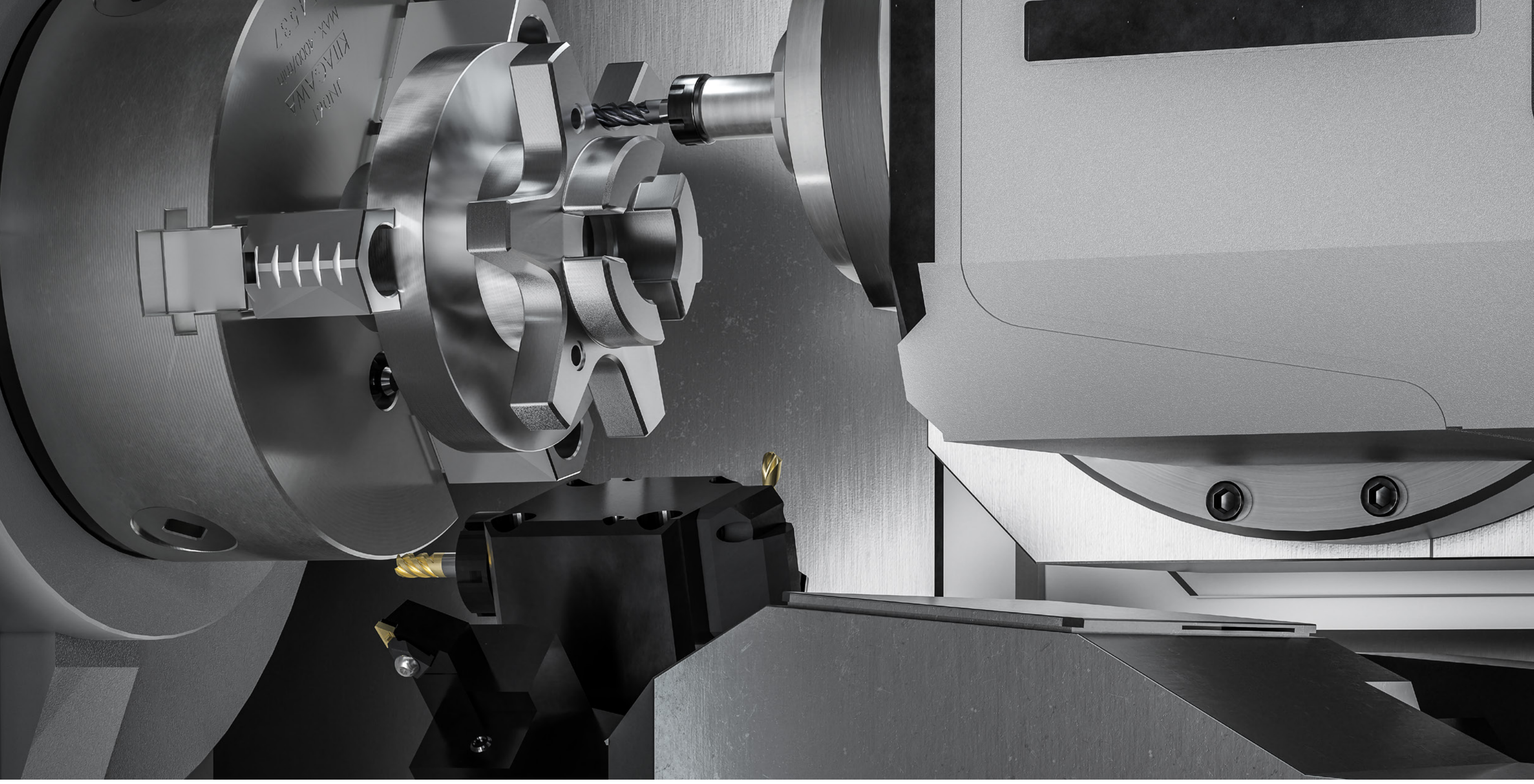
Chains Recognition for Side Profiles



Faces Recognition



Outside Feature Recognition



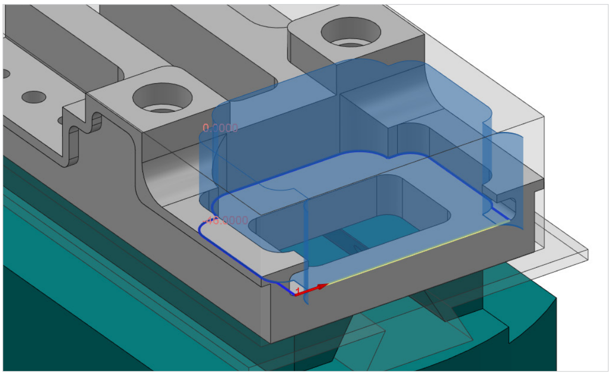
Recognition + Protection

Taking into account the Stock/Updated Stock and Target models, iMachining 2D automatically:

- Detects and avoids part features that create undercut areas
- Detects and extends stock material in open pocket areas
- Detects rest material at every stage of the machining process
- Protects the solid geometries against cutting tool collisions

Dynamic Display of Depths and iMachining Region

iMachining generates and displays a preview of the machinable regions and their levels. The machining geometry can have varying depths and its preview is dynamically updated on job editing, all of which can be visualized in the CAD graphics area.



iMachining's Faces Recognition: Features that make undercut areas are handled with ease

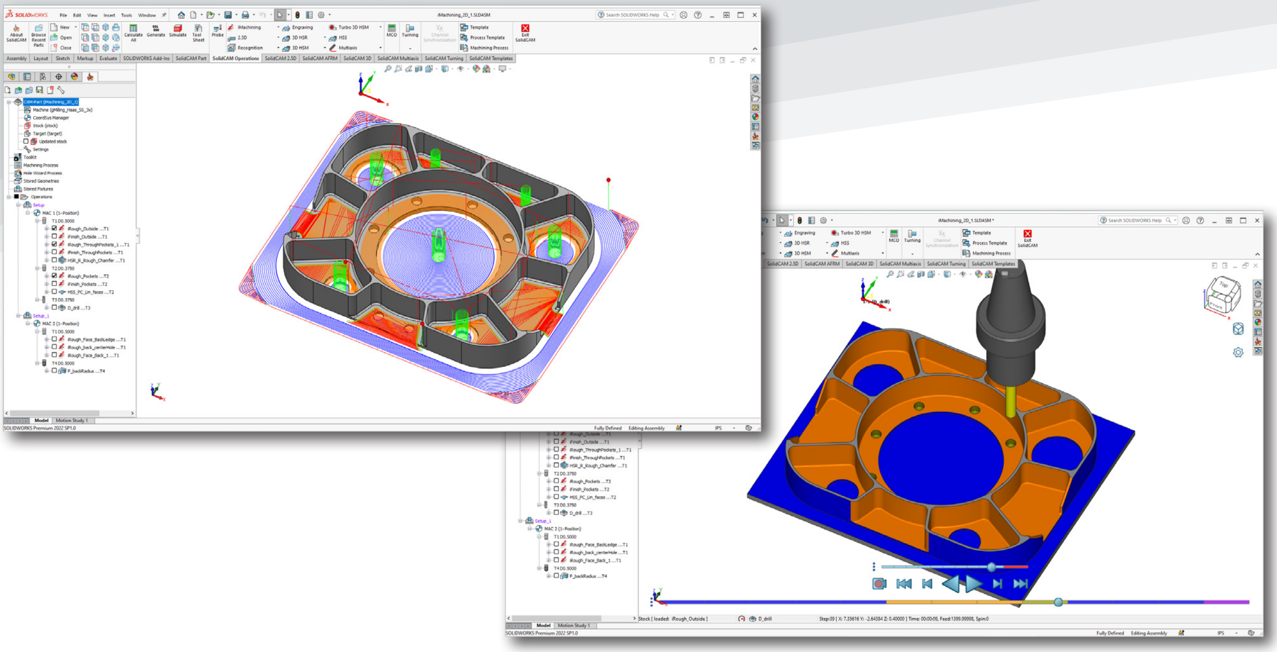
We have found all the claims for iMachining to hold true for our applications in Dixons Surgical – incredible tool life, faster cycles, lighter cutting loads reducing vibration in poor work holding situations (Mill-Turn), and protection of small diameter cutters. The user interface is very clear and simple, and programming iMachining is faster than traditional strategies."

Jay Dixon, Dixons Surgical Instruments

We discovered that SolidCAM reduced our NC programming time by half. On our previous CAD/CAM system, we had to substantially edit GCodes to make the program operate. Now, with SolidCAM, the post-processor produces perfect NC-code, making it far simpler and quicker to produce a new CAM program."

Bob Luck, Alcon Components Ltd

iMachining 2D CAM-PART EXAMPLES



SolidCAM Part: iMachining_2D_1.prz

The operations in this example illustrate the use of SolidCAM'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 SolidCAM 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_1; 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.

- **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.

- **Bottom ledge machining (iRough_Face_BackLedge)**

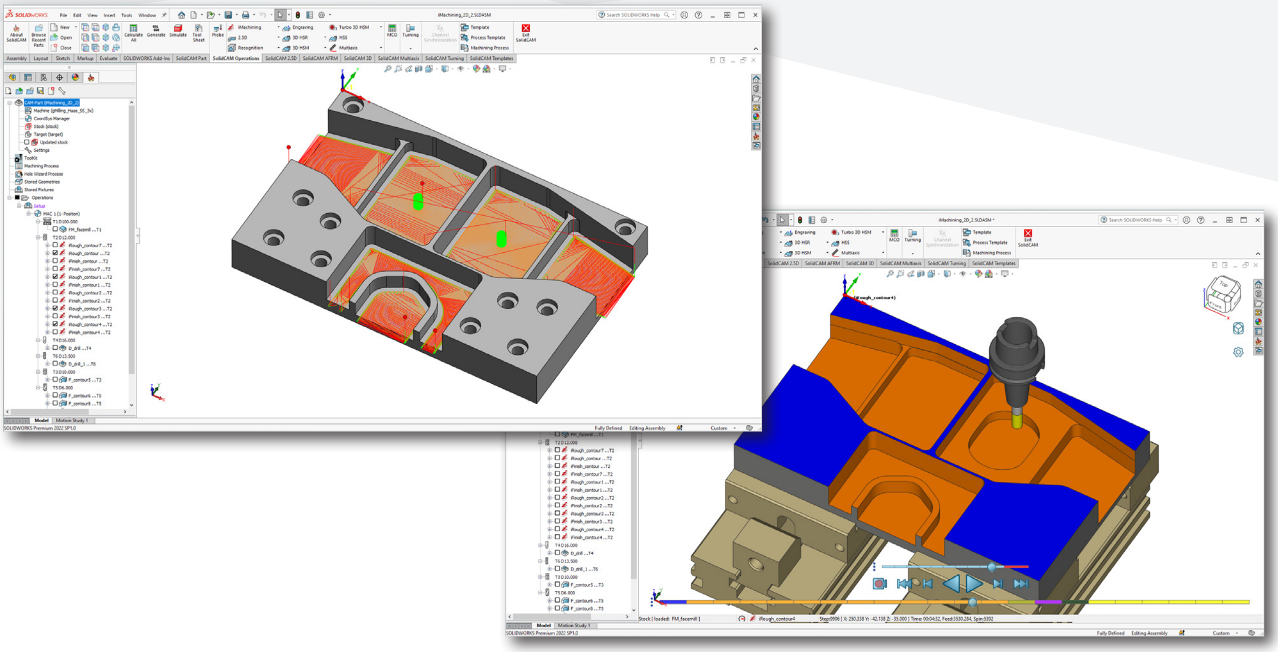
  This iMachining operation defines the bottom ledge machining on the underside of the part. Two chains are defined, with the first being the stock boundary and the second being the bottom of the floor radius. Using a Ø12.5 mm (0.5 in) end mill, the tool starts machining from the stock chain and collapses 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_1)**

  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 inward towards the inner closed chain.



SolidCAM Part: iMachining_2D_2.prz

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

- **Upper face machining (FM_facemill)**

  This Face Milling operation defines the machining of the part upper face. A Ø100 mm (4 in) face mill is used with the Hatch technology to remove 1 mm (0.04 in) of material from the stock top face.

- **Pocket machining (iRough_contour7; iRough_contour; iFinish_contour; iFinish_contour7; iRough_contour1; iFinish_contour1)**

  These iMachining operations define the machining of the semi-open pocket. A Ø12 mm (0.4844 in) end mill is used. The iRough operations have a 0.25 mm (0.01 in) allowance on the walls, which is then removed by the iFinish operations.

- **Pockets machining (iRough_contour2; iFinish_contour2; iRough_contour3; iFinish_contour3; iRough_contour4; iFinish_contour4)**

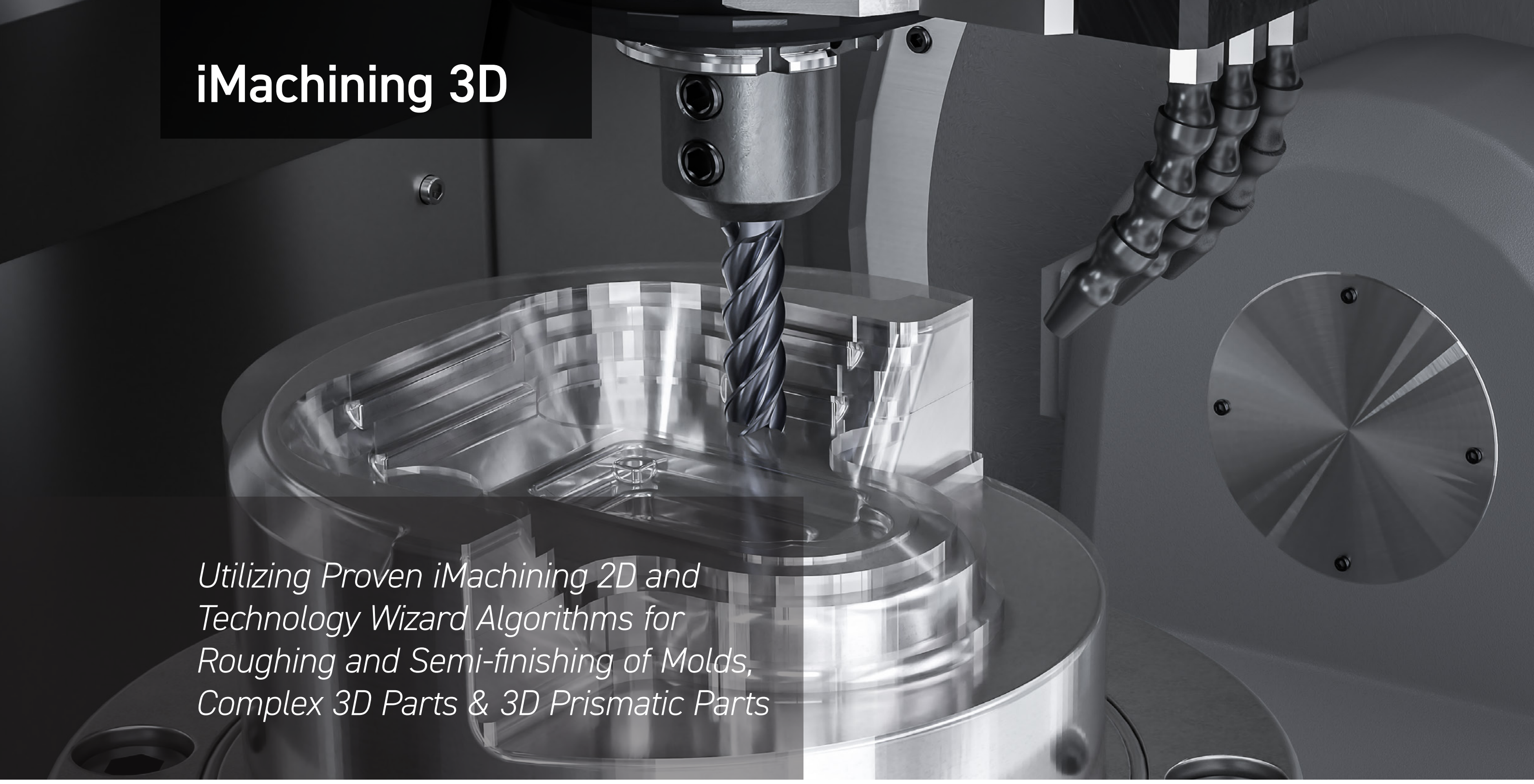
  These iMachining operations define the machining of the semi-open pockets and the two closed pockets. The first iRough and iFinish operation machines the two larger semi-open pockets. The second and third pair of operations machine the smaller semi-open and closed pockets. A Ø12 mm (0.4844 in) end mill is used. The iRough operations have a 0.25 mm (0.01 in) allowance on the walls, which is then removed by the iFinish operations.

- **Holes machining (D_drill; D_drill_1; F_contour5)**

  These Drilling operations define the center drilling and drilling of the holes located on the upper surface of the part. The Profile operation is defined to finish the walls of the holes.

- **Chamfer machining (F_contour6 – F_contour12)**

  These Profile operations define the machining of the chamfers on the part edges using a Ø6 mm (0.2344 in) chamfer drill and the Chamfer option.



iMachining 3D

Utilizing Proven iMachining 2D and Technology Wizard Algorithms for Roughing and Semi-finishing of Molds, Complex 3D Parts & 3D Prismatic Parts

iMachining 3D provides amazing 3D machining results, regularly saving 70% in machining time, reaching up to 90% in many cases.

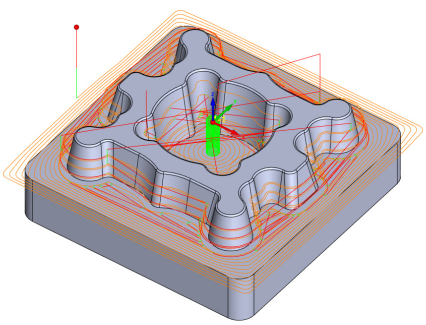
iMachining 3D automatically produces a complete, ready to run CNC program, with optimal cutting conditions, achieved by the expert knowledge-based Technology Wizard, to rough and rest rough an entire 3D part in a single operation.

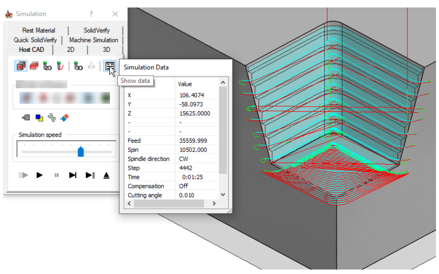
iMachining 3D uses sophisticated analysis algorithms to determine the optimal order of its rough and rest rough tool paths. Combined with its unique local machining feature, full-depth Step down, intelligent Step-up and smart positioning, iMachining 3D achieves the shortest possible cycle time for roughing and semi-finishing of molds, complex 3D parts and 3D prismatic parts.

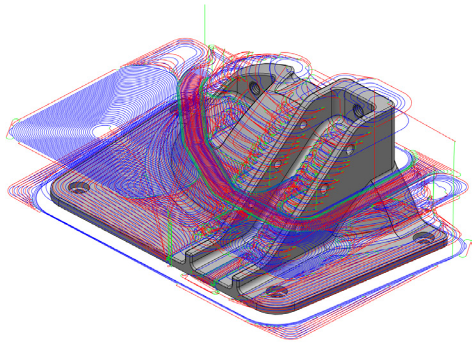
iMachining 3D provides a complete machining solution when combined with other SolidCAM technologies, such as 3D HSM for finishing molds and complex 3D parts or iMachining 2D for finishing 3D prismatic parts.

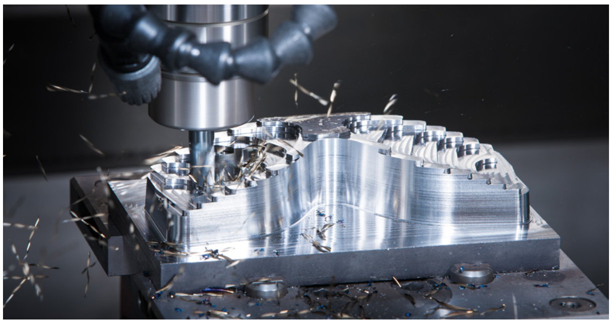
iMachining 3D is a Must-Have!

- Quick solid geometry selection and automatic target model protection
- Optimized machining of each Z-Step, using proven iMachining 2D technology
- Deep roughing with the whole flute length, resulting in shorter cycle times and increased 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 retracts and long position moves, producing the shortest times in the industry
- A dynamically Updated Stock model and "Cut only the Rest material" mode eliminates all air cutting
- Tool path automatically adjusts to avoid collisions between the tool holder and Updated Stock model, at every stage of the machining process



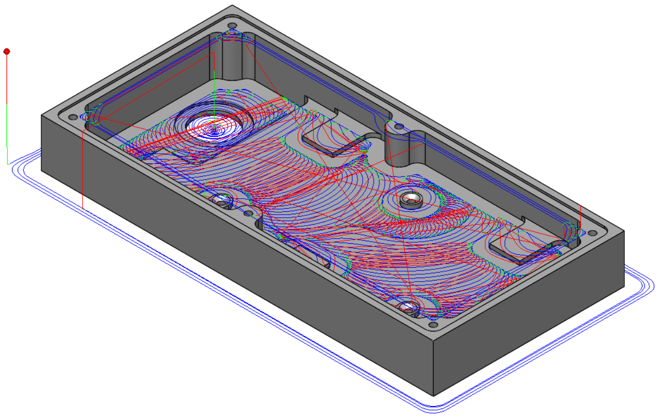






iMachining 3D for Prismatic Parts

Programming times for prismatic parts are drastically reduced with iMachining 3D. In a single operation, rough and rest rough an entire 3D prismatic part that includes any number of pockets and islands, without chaining or sketching a single contour. With just the solid geometry and cutting tool as input, iMachining 3D calculates the rest for you - automatically and optimally.



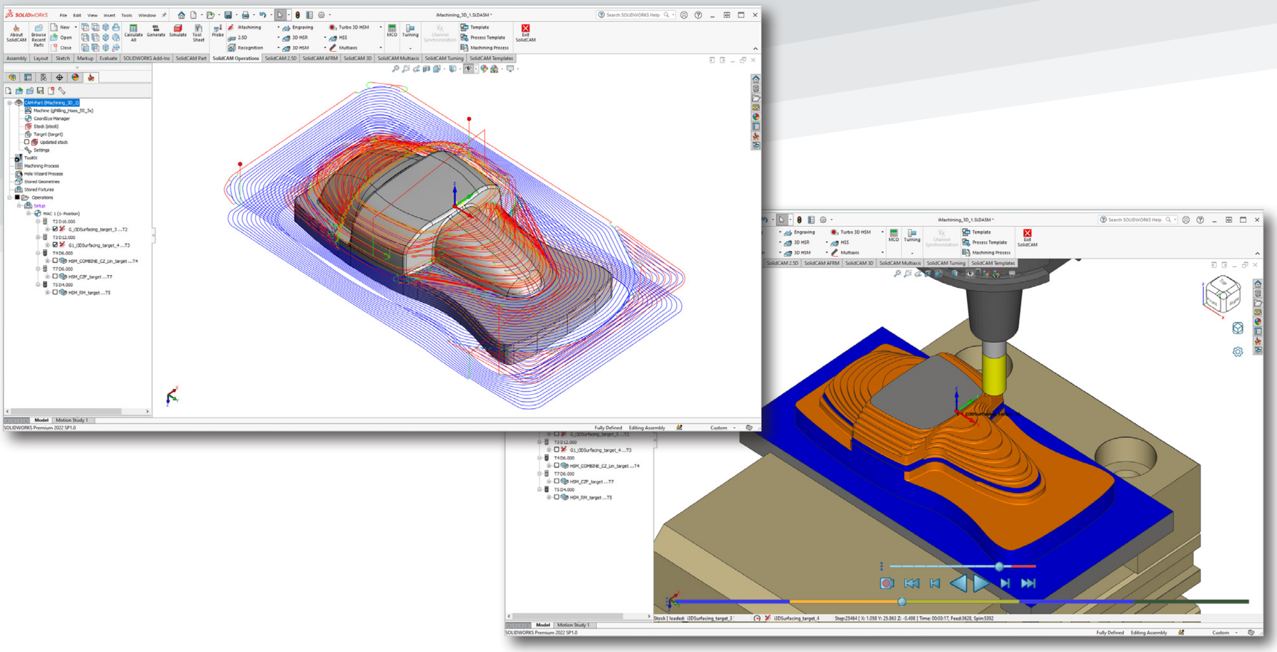
" Growth at the company is continuing at a very healthy rate with an emphasis on lean manufacturing. The introduction of SolidCAM has contributed to this expansion, enabling the company to manufacture more complex parts in a shorter lead time, raising throughput, and maximizing the productivity of our machines."

Dan Patrick, Big Bear Plastics

" SolidCAM has enabled us to get the time down on downtime! It's allowed much more synergy going from one complex product to the next, more so if you have a complete new set-up on tooling. We need to make sure that from one set-up to the next that the downtime is as minimal as possible."

Shaun Palmer, Director, Oracle Precision Ltd

iMachining 3D CAM-PART EXAMPLES



SolidCAM Part: iMachining_3D_1.prz

The operations in this example illustrate the use of SolidCAM's iMachining 3D technology to perform the machining of the part shown above.

- **Roughing (G_i3DSurfacing_target_3)**

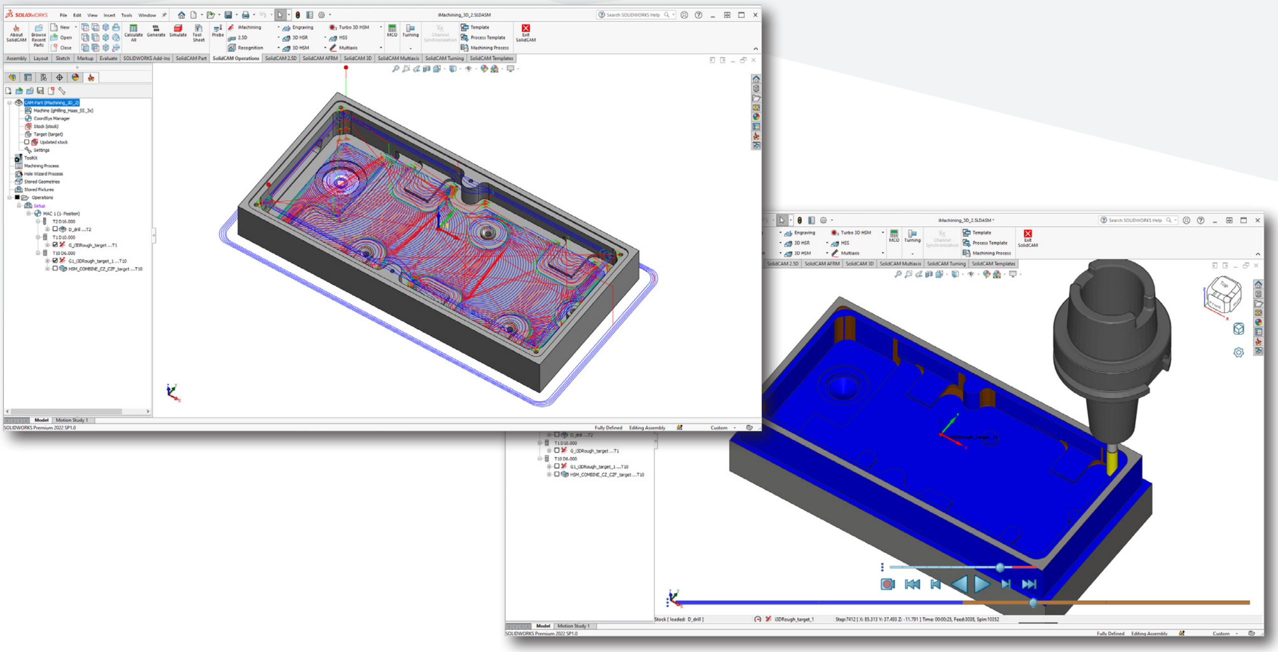
  This iMachining 3D operation is defined to remove the majority of material using the 3D General strategy and a Ø16 mm (0.625 in) end mill. For this operation, rest roughing is disabled. 2 Step downs are performed and a machining allowance of 0.2 mm (0.008 in) remains on the surfaces.

- **Rest machining (G1_i3DSurfacing_target_4)**

  This iMachining 3D operation defines the removal of unmachined material remaining after the previous operation and uses a Ø12 mm (0.4844 in) bull nose mill with a corner radius of 0.6 mm (0.024 in). After 2 Step downs, rest roughing is performed in Step-up mode with the current tool to remove additional rest material on sloped surfaces according to the automatically defined value of Scallop. A machining allowance of 0.2 mm (0.008 in) remains on the surfaces.

- **Finish machining (HSM_COMBINE_CZ_Lin_target; HSM_CZF_target; HSM_RM_target)**

  These HSM operations use the Combine, Horizontal and Rest machining strategies to define the part finishing.



SolidCAM Part: iMachining_3D_2.prz

The operations in this example illustrate the use of SolidCAM's iMachining 3D technology to perform the machining of the part shown above.

- **Through hole machining (D_drill)**

  This Drilling operation defines the drilling of the through hole. The hole will be used for safe tool entry in the subsequent iMachining operation. A Ø16 mm (0.625 in) drill is used.

- **Roughing (G_i3DRough_target)**

  This iMachining 3D operation is defined to remove the majority of material using the 3D General strategy and a Ø10 mm (0.3906 in) end mill. The 3D Prismatic strategy can also be used in this case. After just 1 Step down, rest roughing is performed in Step-up mode with the current tool to remove rest material on higher horizontal surfaces. A machining allowance of 0.38 mm (0.015 in) remains on the wall and floor surfaces.

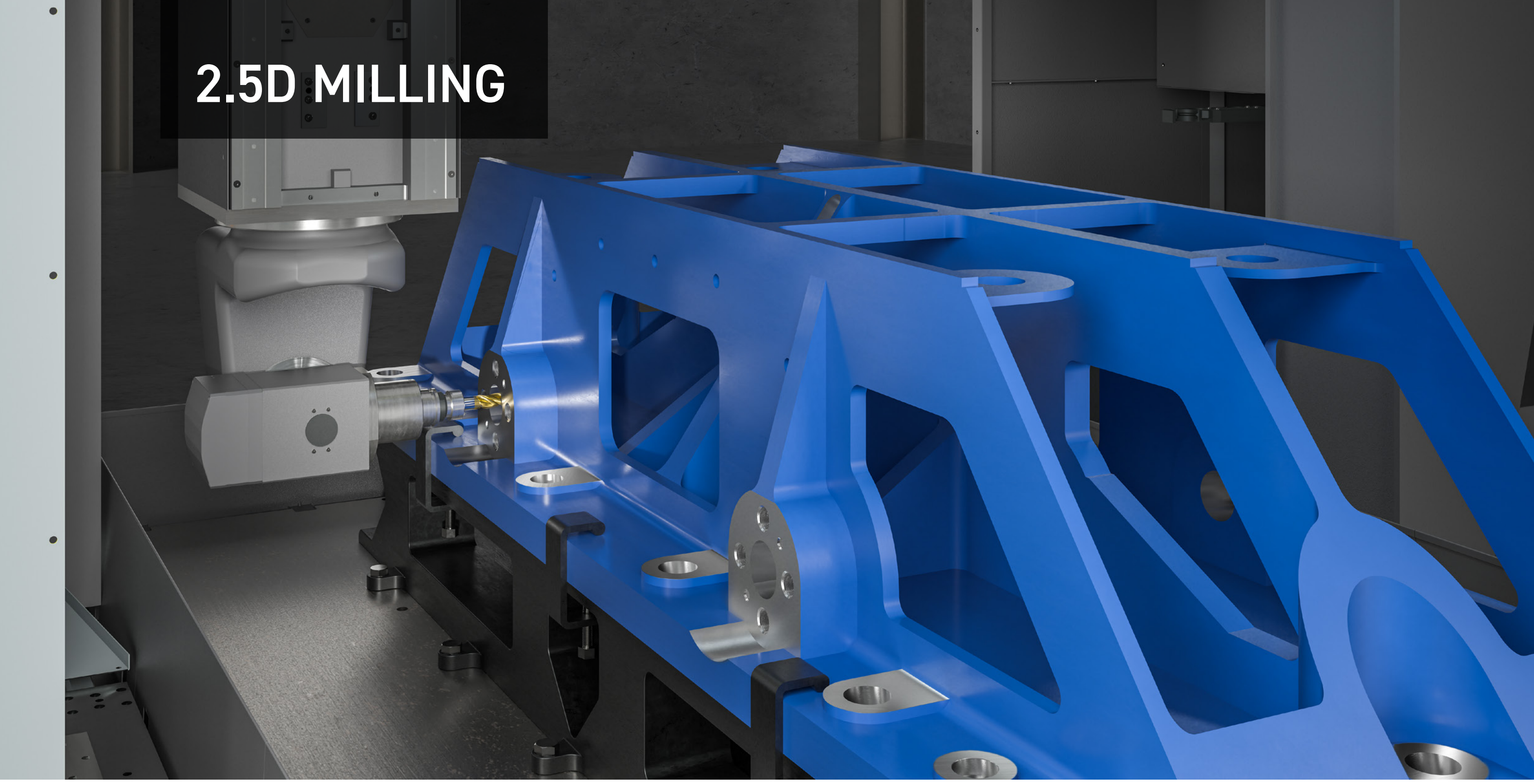
- **Rest machining (G1_i3DRough_target_1)**

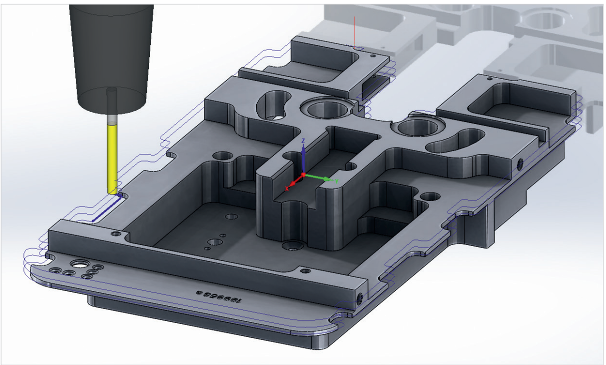
  This iMachining 3D operation defines the removal of unmachined material remaining in the corners after the previous operation and uses a Ø6 mm (0.2344 in) end mill. A machining allowance of 0.38 mm (0.015 in) remains on the wall and floor surfaces.

- **Finish machining (HSM_COMBINE_CZ_CZF_target)**

  This HSM operation combines the Constant Z and Horizontal strategies to define the part finishing.

2.5D MILLING





The most straightforward, easy-to-use interface that is seamlessly integrated in SOLIDWORKS, 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 CNC machining operations. Quickly place fixtures and components for full visualization.

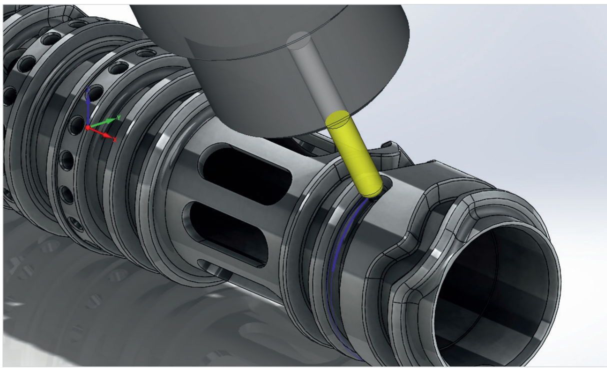
Best of Both Worlds: Complete Interactive Control + Feature Recognition

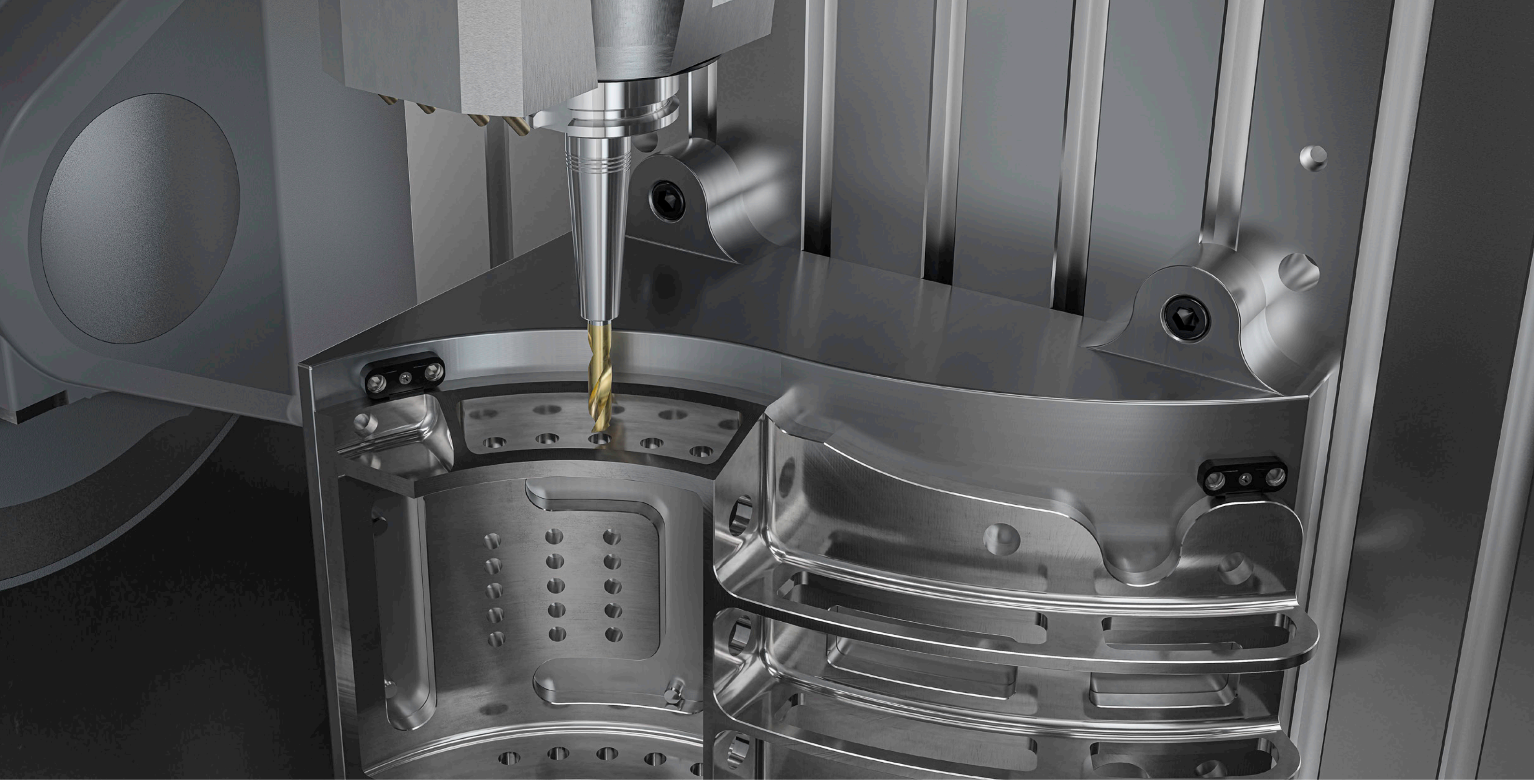
SolidCAM provides both interactive and automated 2.5D Milling operations on SOLIDWORKS models. Designed for both the novice and advanced user, SolidCAM offers the best of both worlds, with your choice of fully controlled selection of geometry, parameters and CNC programming strategies or Automated Pocket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

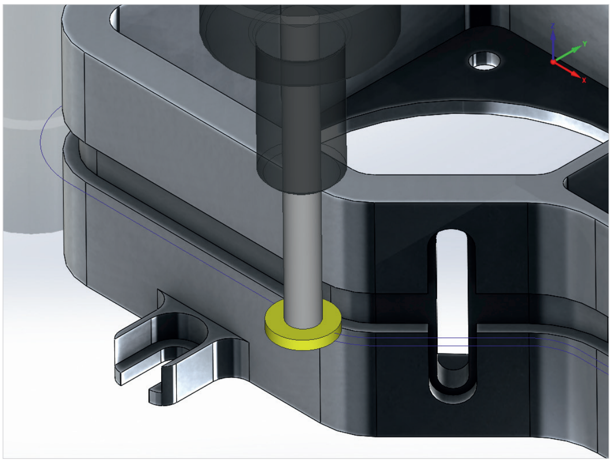
Interactive 2.5D Mill Operations

Besides the standard 2.5D Milling profiling, pocketing and drilling operations, SolidCAM 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 remaining 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
- Variable levels of pockets and profiles in a single job
- Engraving of text on flat and wrapped faces and medial line engraving of a 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 movements from linear to rotary
- Special operation for the machining of side slots with an undercut using a T-slot tool

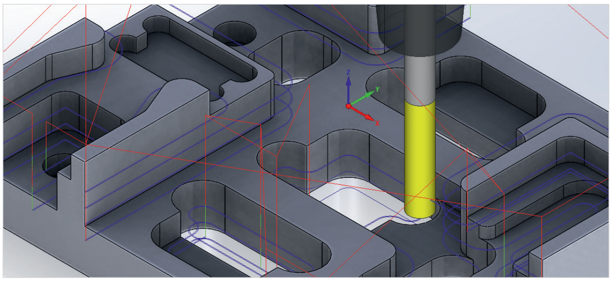






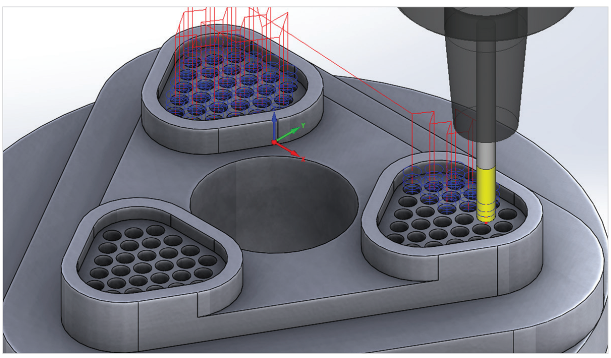
Pocket Recognition

SolidCAM's powerful pocketing strategy is taken to the next level, by automatically identifying all pockets on the CAD model. All strategies and options of the standard Pocket operation are available, combined with variable levels and depths recognized from the model faces.



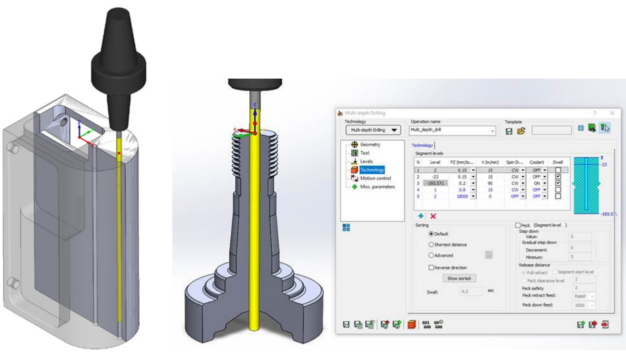
Drill Recognition

Automatic recognition and grouping of holes from the solid model with option to modify resulting geometry. A single Drill Recognition operation can machine groups of holes on varying levels and depths.

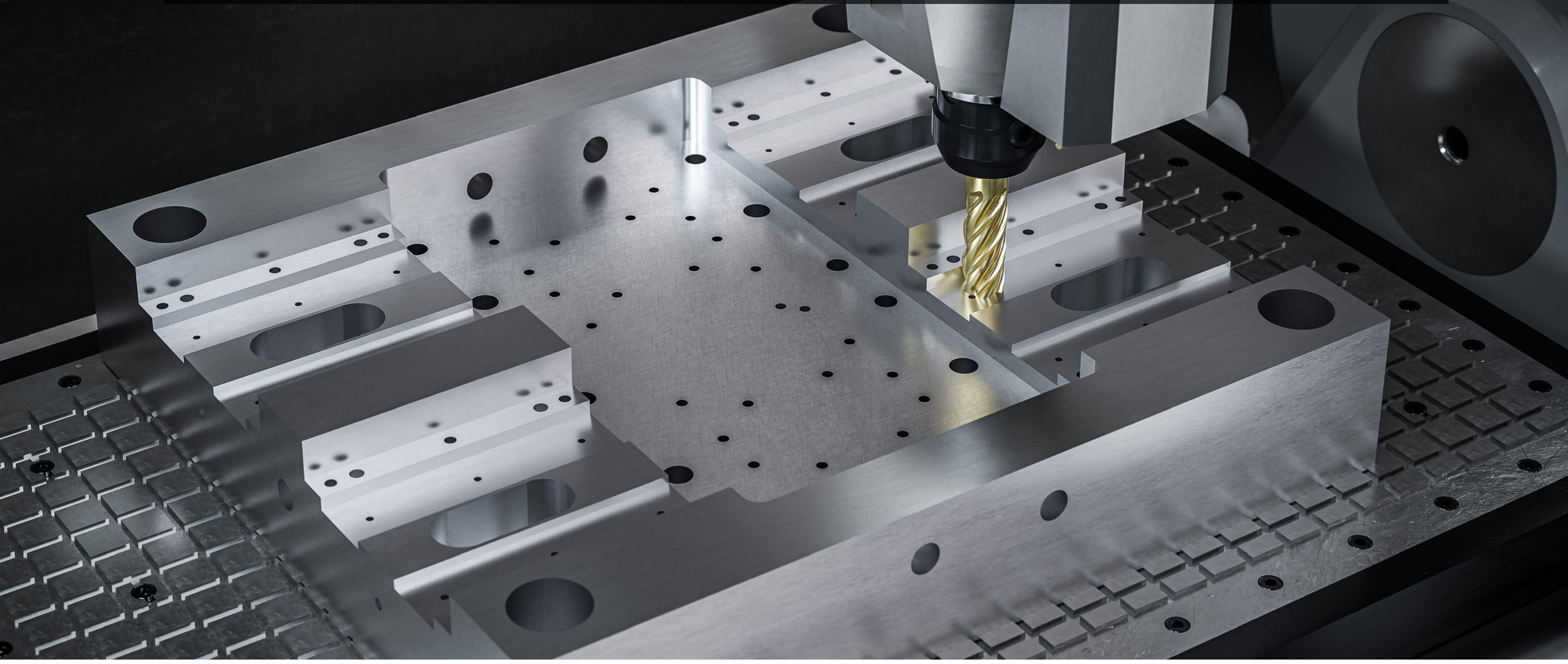


Multi-Depth-Drilling

This powerful Drilling operation gives you full control, allowing you to customize your drilling operation at every step and every depth. This is the perfect Drilling operation for Deep drilling and Cross hole dilling.



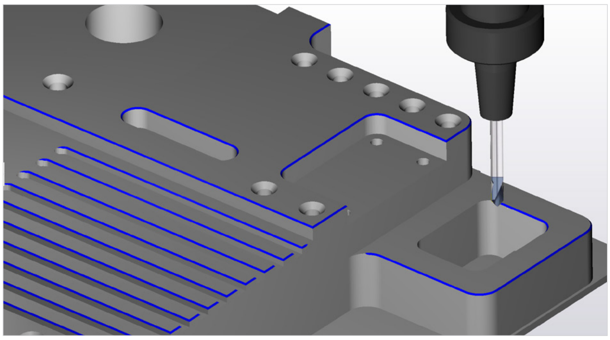
AUTOMATIC FEATURE RECOGNITION & MACHINING



Advanced Pocket Recognition

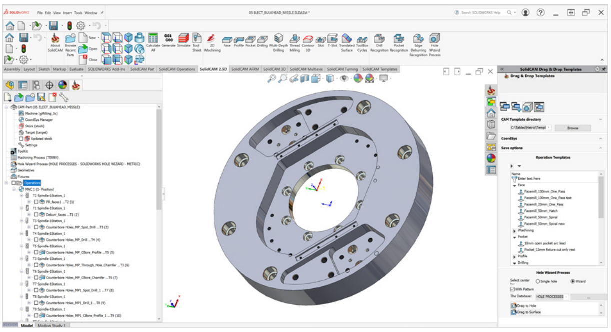
SolidCAM AFRM sets the new standard. Instead of machining each individual pocket in a separate operation, all pockets, no matter whether they are open, closed, blind or through pockets, are being identified with their corresponding depth and Z-level and machined in one operation. Full fixture protection in pocket, pocket recognition and 2D drilling allows you to machine your parts while protecting your fixtures.

- All strategies and options of the standard pocket operation are available, combined with variable upper levels and depths recognized from the model faces. User controls the choice of the Tool, Technology and Cutting Strategy.
- Automatic recognition and machining of fillets on the pocket floor
- Automatic rest material recognition on each pocket
- The perfect tool for multi pocketed parts

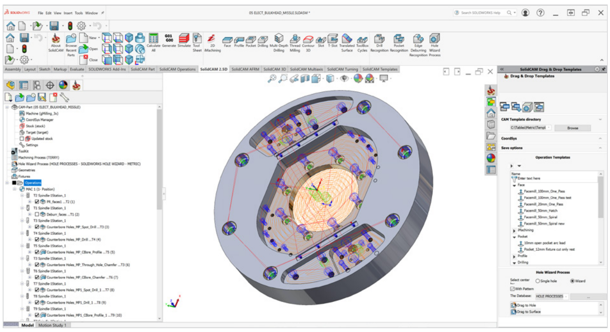


Chamfering and Deburring

SolidCAM automatically recognizes all sharp edges where a chamfer can be applied. The user only sets the depth of the chamfer, the cutting diameter of the tool and a safety offset. SolidCAM's chamfer recognition automatically avoids vertical walls and machines as much as possible, while protecting the part from collisions with the shank. The option Dynamic Tool Diameter results in less wear and tear of the tool, which is particularly helpful when machining hard materials.



Drag & Drop Hole Wizard Process applied to a single hole feature

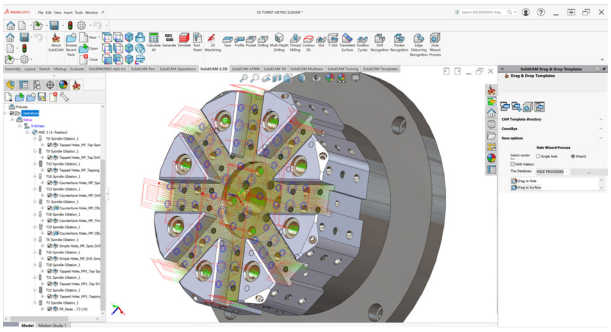


Drag & Drop Hole Wizard Processes applied to the entire part

Advanced Drill Recognition

SolidCAM automatically identifies all drills on the solid model and generates the necessary CNC operations.

- To select the drills to be machined, powerful filtering tools such as diameter, Z-level or drill depth are available.
- Spot drills can be generated on all drill positions, where the depth relates to the diameter of the drill tool being used.

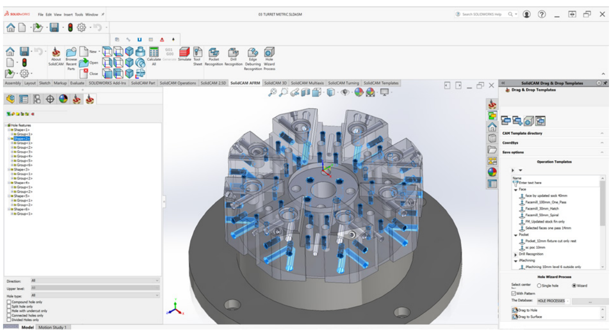


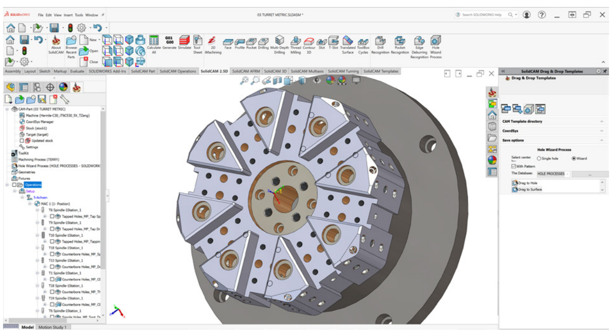
Drill features detection & Automatic Tool path generation

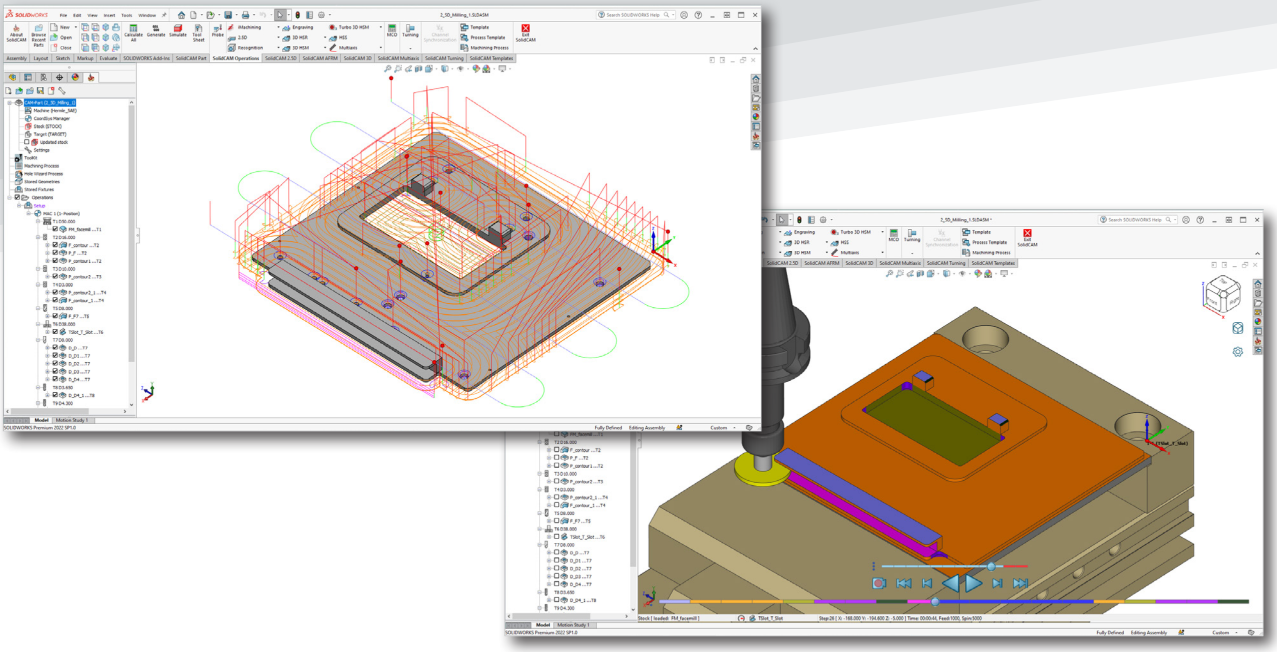
SolidCAM's Hole Wizard, with Drag & Drop Machine Processes

SolidCAM's Hole Wizard, with Drag & Drop Machine Processes, optimizes the task of programming multiple operation for complex holes.

- All holes in the CAD feature, including subordinate patterns are recognized.
- All geometry and dimensional parameters of the CAD feature are available for use in the machining process, including Hole Wizard tolerances.
- Complex logic, including conditional equations, provides greater flexibility.
- Simple, Counter Sunk, Counter Bored & Tapped hole sets are programmed with a single mouse click.







SolidCAM Part: 2_5D_Milling_1.prz

The operations in this example illustrate the use of SolidCAM's 2.5D Milling technologies to machine the part shown above. The machining is performed on a 5-Axis CNC-Machine in one setup, using one SolidCAM Coordinate System.

- **Upper face machining (FM_facemill)**

  This Face Milling operation defines the machining of the part upper face. A Ø50 mm (2 in) face mill is used with the Hatch technology to remove 0.5 mm (0.02 in) of material from the stock top face.

- **Outside shape machining (F_contour)**

  This Profile operation defines the machining of the outside shape. The material is removed in three step down cuts using a Ø16 mm (0.625 in) end mill, and an additional finish pass is executed to remove the remaining 0.3 mm (0.012 in) from the wall.

- **Island and pockets machining (P_F; P_contour1; P_contour2)**

  These Pocket operations define the machining of the island and pockets. The first operation creates the large island using a Ø16 mm (0.625 in) end mill. The Open pocket technology uses a profile strategy and approaches from the outside, leaving a 0.2 mm (0.008 in) allowance on the walls of the island for a finishing pass. In the second operation, the same tool is used to create nubs on the island upper face. In the third operation, the through pocket is created inside the island using a Ø10 mm (0.4063 in) end mill.

- **Corners rest and finish machining (P_contour2_1; F_contour_1)**

  These operations define the inside corners machining of the through pocket and outside shape. Both the Pocket operation and the Profile operation use a Ø3 mm (0.125 in) end mill with the Rest material option to only remove the material that remains after the larger tool from the previous operations.

- **Chamfer machining (F_F7)**

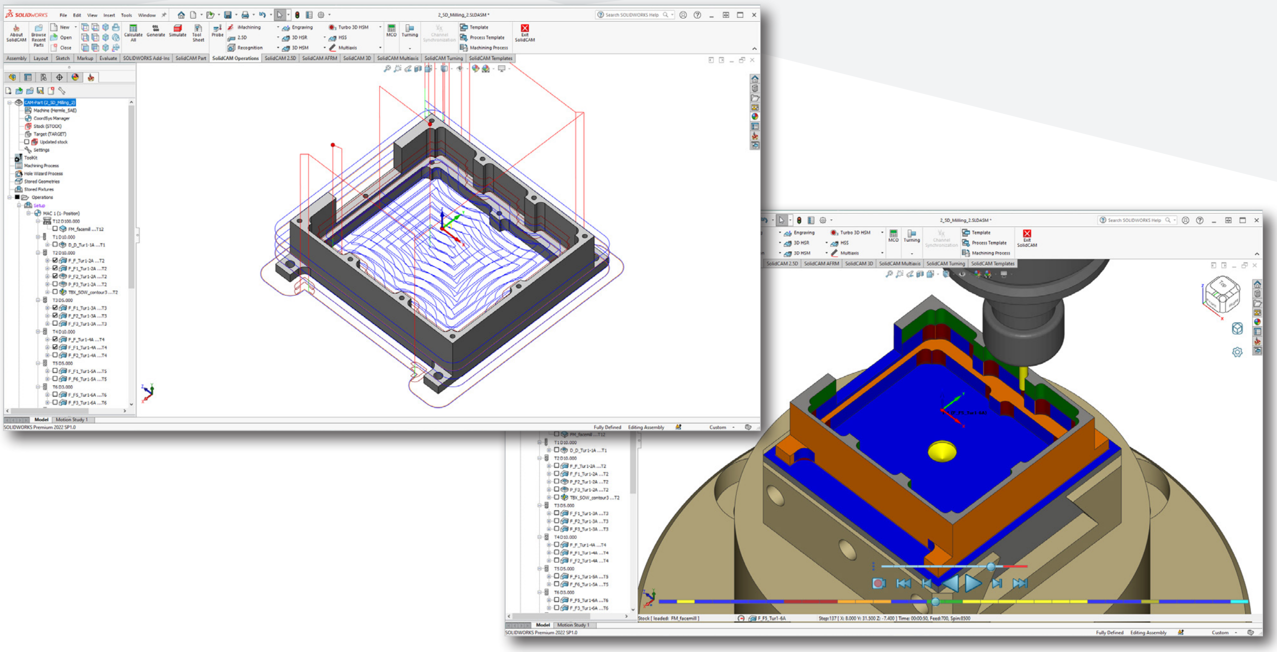
  This Profile operation defines the machining of the chamfers on the part edges using an Ø8 mm (0.3125 in) chamfer drill and the Chamfer option.

- **Slot machining (TSlot_T_Slot)**

  This T-Slot operation is defined to machine the side groove. A Ø38 mm (1.5 in) slot mill is used and finish passes are executed on the groove floor, ceiling and walls.

- **Holes machining (D_D – D_D2_1)**

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



SolidCAM Part: 2_5D_Milling_2.prz

The operations in this example illustrate the use of SolidCAM's 2.5D Milling technologies to machine the part shown above. The machining is performed on a 5-Axis CNC-Machine in two setups, using two SolidCAM Coordinate Systems.

- **Upper face machining (FM_facemill)**

  This Face Milling operation defines the part upper face machining in one pass using a Ø100 mm (4 in) face mill.

- **Pockets machining (D_D_Tur1-1A; P_F2_Tur1-2A; P_F3_Tur1-2A)**

  These operations define the rough machining of the center pocket areas. In the Drilling operation, a hole is drilled using a Ø10 mm (0.4083 in) drill that will be used for safe tool entry in the subsequent Pocket operations. A Ø10 mm (0.4063 in) rough mill is then used with the Contour strategy to perform the pocket machining.

- **Outside shape machining (F_F_Tur1-2A; F_F1_Tur1-2A)**

  These Profile operations define the rough machining of the outside shape. A Ø10 mm (0.4063 in) rough mill is used, leaving a 0.2 mm (0.008 in) allowance on the walls.

- **Slot machining (TBX_SOW_contour3)**

  This ToolBox cycles operation is defined to machine a wide open slot in the part side wall. A spiral tool path effectively removes the material using the whole cutting length of the Ø10 mm (0.4063 in) rough mill.

- **Corners rest and finish machining (F_F1_Tur1-3A; F_F1_Tur1-5A)**

  These Profile operations define the inside corners machining of the external tabs. Both operations use the Rest material option to only remove the material that remains after the larger tool from the previous operations. In the second operation, the tool finishes the corners at the full depth.

- **Semi-finish and finish machining (F_F_Tur1-4A; F_F1_Tur1-4A; F_F2_Tur1-3A; F_F3_Tur1-3A; F_F2_Tur1-4A; F_F6_Tur1-5A; F_F5_Tur1-6A; F_F3_Tur1-6A)**

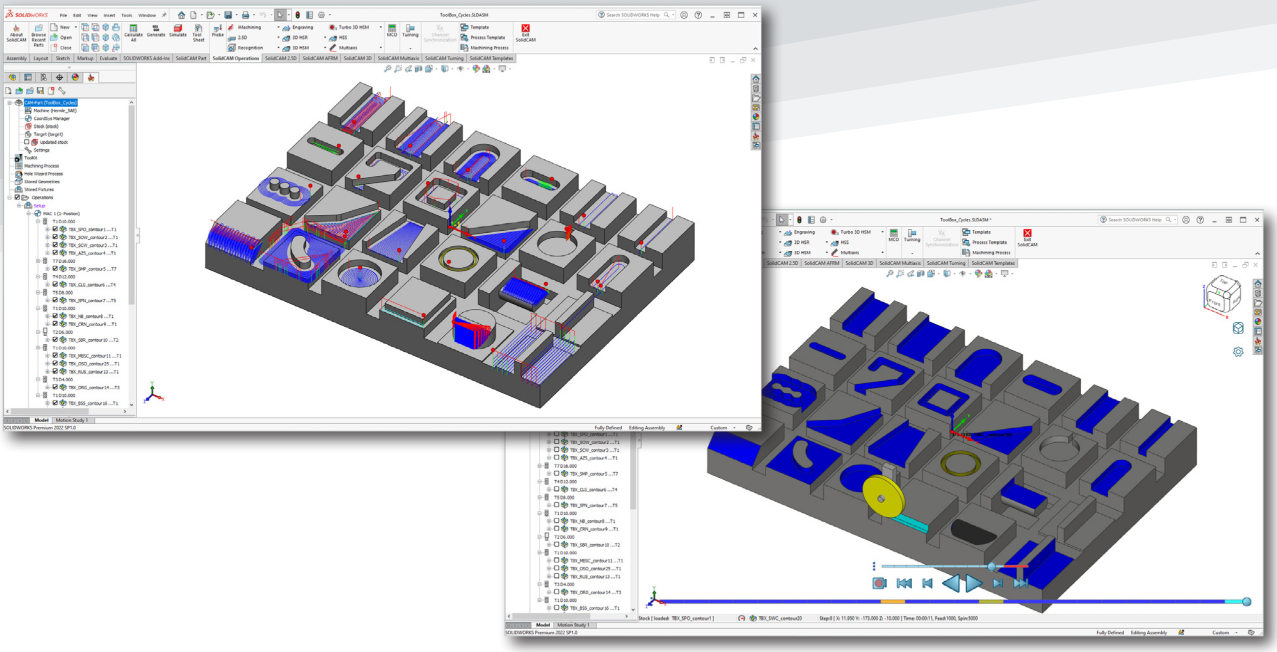
  These Profile operations define the semi-finish and finish machining of the part external and internal surfaces using different end mills. A smaller end mill is used for the finishing.

- **Chamfer machining (F_F7_Tur1-7A; F_F3_Tur1-7A; F_F8_Tur1-7A)**

  These Profile operations define the chamfering of the part edges using a Ø2 mm (0.0781 in) chamfer drill.

- **Holes machining (D_D1_Tur1-7A – D_D4_Tur1-11A)**

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



SolidCAM Part: ToolBox_Cycles.prz

The operations in this example illustrate the use of SolidCAM's ToolBox Cycles to machine the part shown above. The machining is performed on a 5-Axis CNC-Machine in one setup, using one SolidCAM Coordinate System with multiple positions. The following operations are included but not limited to:

- **Closed slot machining and spine slot machining (TBX_CLS_contour6; TBX_SPN_contour7)**
  The CLS cycle enables you to machine a closed slot with the width equal to the tool diameter. The machining is performed in the zigzag manner; the last cut is horizontal to clean the slot floor. In the SPN cycle, SolidCAM automatically determines the spine of the slot and performs the machining according to the spine shape.

- **Four nubs machining (TBX_NB_contour8)**
  This cycle enables you to machine the profile of a pocket while preserving the rest of the material fixed to the wall using small thin bridges.

- **Simple corner machining (TBX_CRN_contour9)**
  This cycle enables you to machine an open corner area with a number of cuts equidistant to the selected geometry.

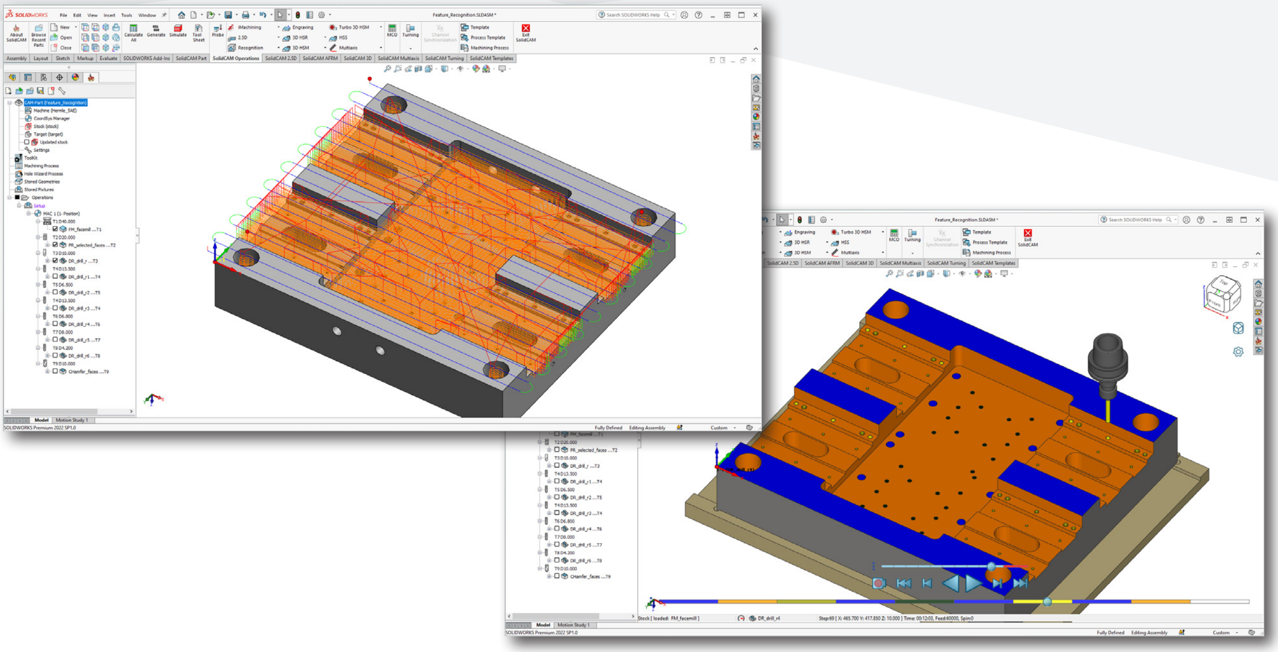
- **Simple multi-bosses machining (TBX_MBSC_contour11)**
  This cycle enables you to machine several bosses starting from one face. The machining is performed in a number of equidistant Clear offset passes parallel to the selected bosses geometry.

- **One side open slot machining (TBX_OSO_contour25)**
  This cycle enables you to machine a one-sided slot using two chains: a main chain and a drive chain. The machining is performed with a spiral tool path.

- **O-Ring machining (TBX_ORG_contour14)**
  This cycle enables you to machine slots that have an O-ring shape. The geometry is defined as a pair of closed chains, with the first being an external chain and the second being an internal chain.

- **Spiral pocket machining (TBX_SPK_contour17)**
  This cycle enables you to create a spiral-shaped tool path inside the pocket. The Spiral cuts are controlled by Cutting angle parameters, ranging from minimal to maximal values.

- **Roll into open slot and roll into closed slot (TBX_ROS_contour21; TBX_RCS_contour22)**
  The ROS cycle enables you to machine an open slot that, while rolling into the material with an arc movement, starts from the slot center and diverges to the walls. The RCS cycle enables you to machine a closed slot that, while rolling into the material with an arc movement, starts from the slot center and diverges to the walls.



SolidCAM Part: Feature_Recognition.prz

The operations in this example illustrate the use of SolidCAM'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 SolidCAM Coordinate System.

- **Upper face machining (FM_facemill)**

  This Face Milling operation defines the machining of the part upper face. A Ø40 mm (1.575 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.

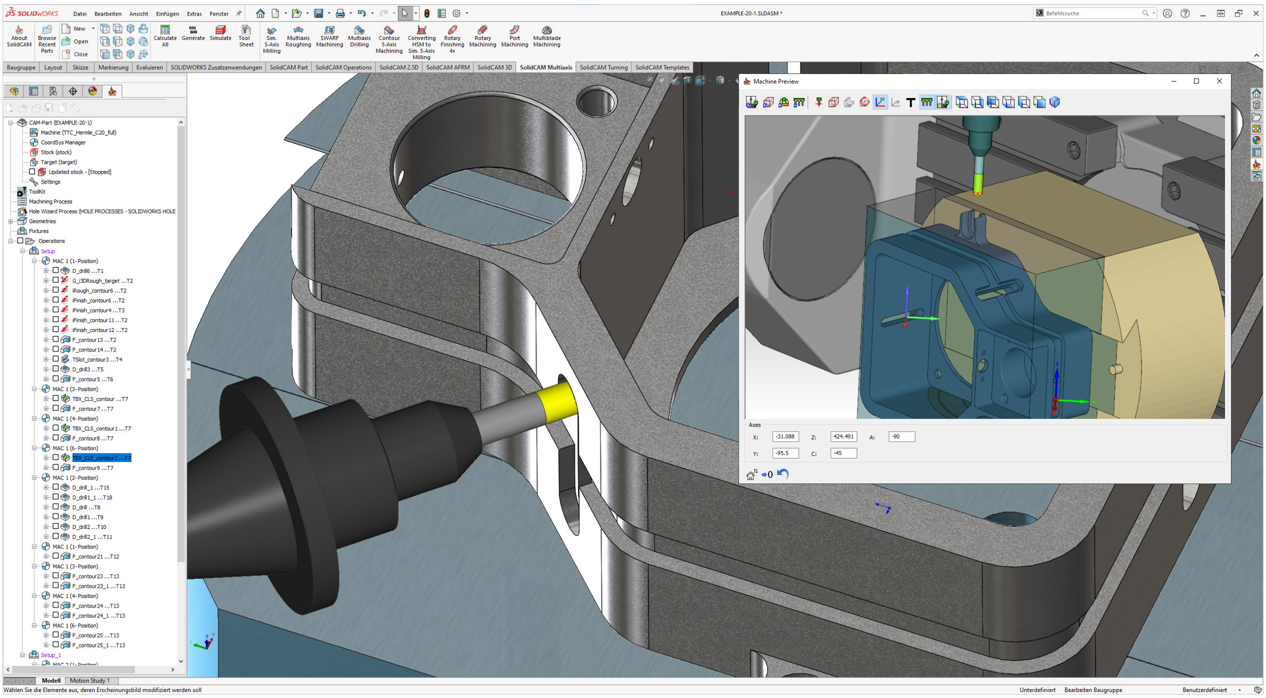
- **Drilling (DR_drill_r1 – DR_drill_r6)**

  These Drill Recognition operations define the drilling of all the part hole features. For each operation, SolidCAM 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 (0.4063 in) chamfer mill is used to machine a 0.5 mm (0.02 in) Chamfer depth. The tool descends and creates chamfers on the part edges with a Cutting diameter of 2 mm (0.08 in), while maintaining a 4 mm (0.16 in) Safety offset.

INDEXIAL 4/5-AXIS MILLING



A common scene in any machine shop today is that 4- and 5-Axis CNC-Machines are increasing production, providing faster cycle times.

SolidCAM excels in Indexial 4/5-Axis Milling and offers the most efficient and effective ways of programming on multiple sides of a part.

Easiest Coordinate Systems Definition for Indexial 5-Axes!

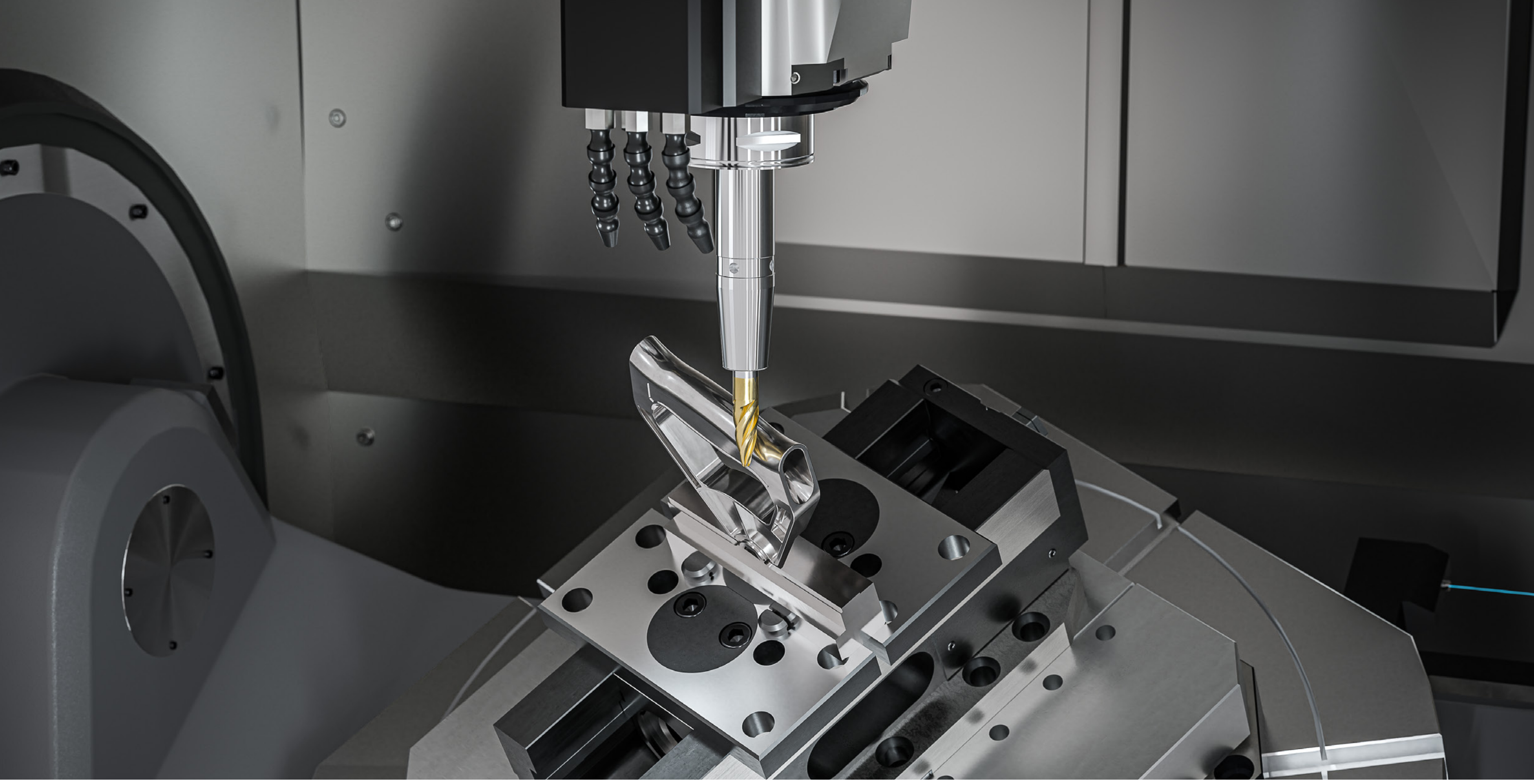
Tired of dealing with construction views, copying models, and rotating them in space for new alignments? Do you still copy and transform geometry to separate layers for indexial programming?

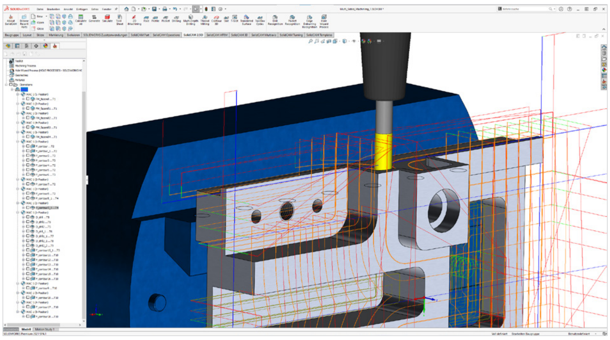
Experience single machine home position, with one-click orientations for indexed setups – SolidCAM speeds up multi-sided machining by eliminating multiple Coordinate System constructions. Define a Coordinate System on the fly by simply picking a face and then continuing to program your part.

SolidCAM's "select a face and machine" feature offers the fastest approach to indexial programming.

SolidCAM's Coordinate System Manager keeps track of all necessary data for each tool orientation.

SolidVerify Simulation displays tool holders and fixtures, alongside material removal for all machining operations.





Efficient, Edit-Free GCode for Multi-Axis Machines

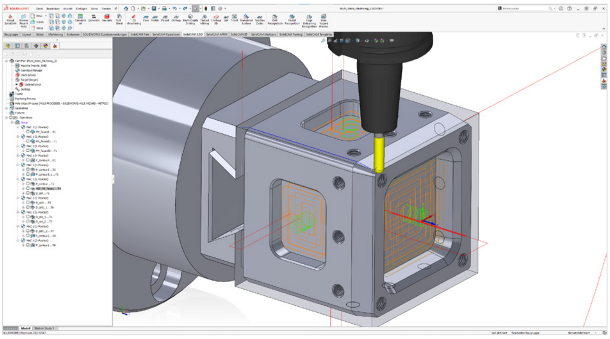
SolidCAM offers multiple options to get efficient GCode for Multi-Axis machines.

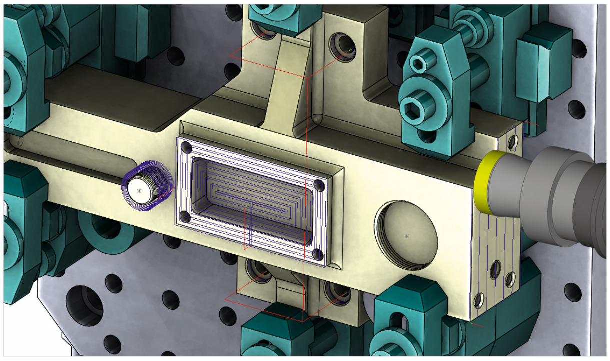
SolidCAM's post-processor can be configured to handle all rotations and work offset shifts, eliminating the need for setting up multiple work offsets at the machine. Whether your controller can calculate part rotations internally or it needs the post-processor to handle rotations, SolidCAM has you covered.

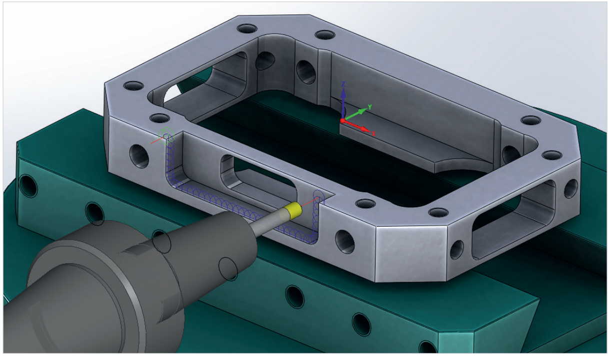
For controllers with advanced plane rotation or coordinate rotation functions, SolidCAM's post-processors are built to utilize these internal CNC functions. If your machine lacks such functions, you can input the part location within SolidCAM and the GCode will automatically handle all of the transformations for each rotation.

SolidCAM's approach to Indexial Milling is straightforward: From Software to GCode, we make the process for Indexial Milling as seamless as single-sided machining.

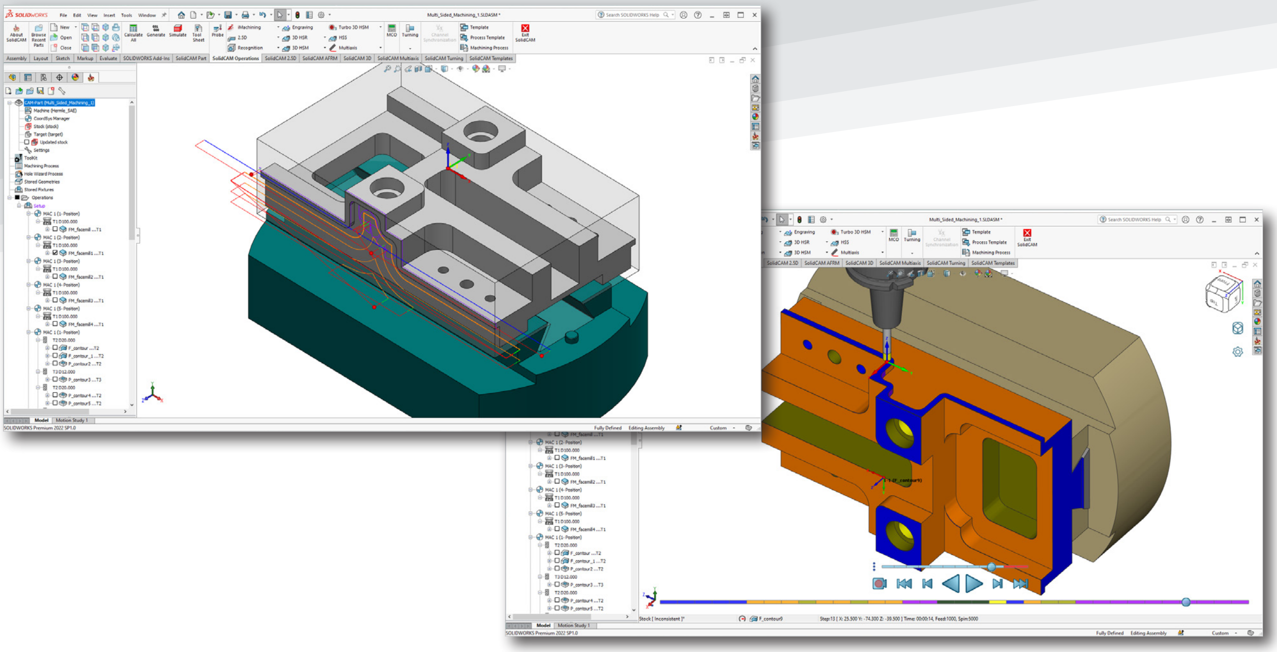
No specialized functions or software tricks are required to machine multi-sided parts – it just works!







INDEXIAL 4/5-AXIS MILLING CAM-PART EXAMPLES



SolidCAM Part: Multi_Sided_Machining_1.prz

The operations in this example illustrate the use of SolidCAM'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 SolidCAM Coordinate System with multiple positions.

- **Top and side faces machining (FM_facemill; FM_facemill1; FM_facemill2; FM_facemill3; FM_facemill4)**

  The first Face Milling operation defines the machining of the part top face using Machine Coordinate System #1, Position #1 (MAC 1 (1- Position)). The remaining Face Milling operations define the machining of the part side faces using MAC 1 (2, 3, 4 and 5- Positions).

- **Step machining (F_contour; F_contour_1)**

  These Profile operations define the rough and finish machining of the step feature using MAC 1 (1- Position).

- **Through pocket machining (P_contour3)**

  This Pocket operation defines the machining of the through pocket located below the step feature using MAC 1 (1- Position).

- **Open and side pockets machining (P_contour2; P_contour4; P_contour5; P_contour6; P_contour7; P_contour8)**

  The first Pocket operation defines the machining of the open pocket located below the step feature and the next three Pocket operations define the machining of the remaining open pockets using MAC 1 (1- Position). The remaining two Pocket operations define the machining of the open pockets located on the opposing side faces using MAC 1 (2 and 3- Positions).

- **Corners rest and finish machining (P_contour8_1; P_contour7_1)**

  These Pocket operations define the inside corners machining of the open pockets created in the previous operations using the same MAC 1 (2 and 3- Positions).

- **Holes machining (D_drill – D_drill2_2)**

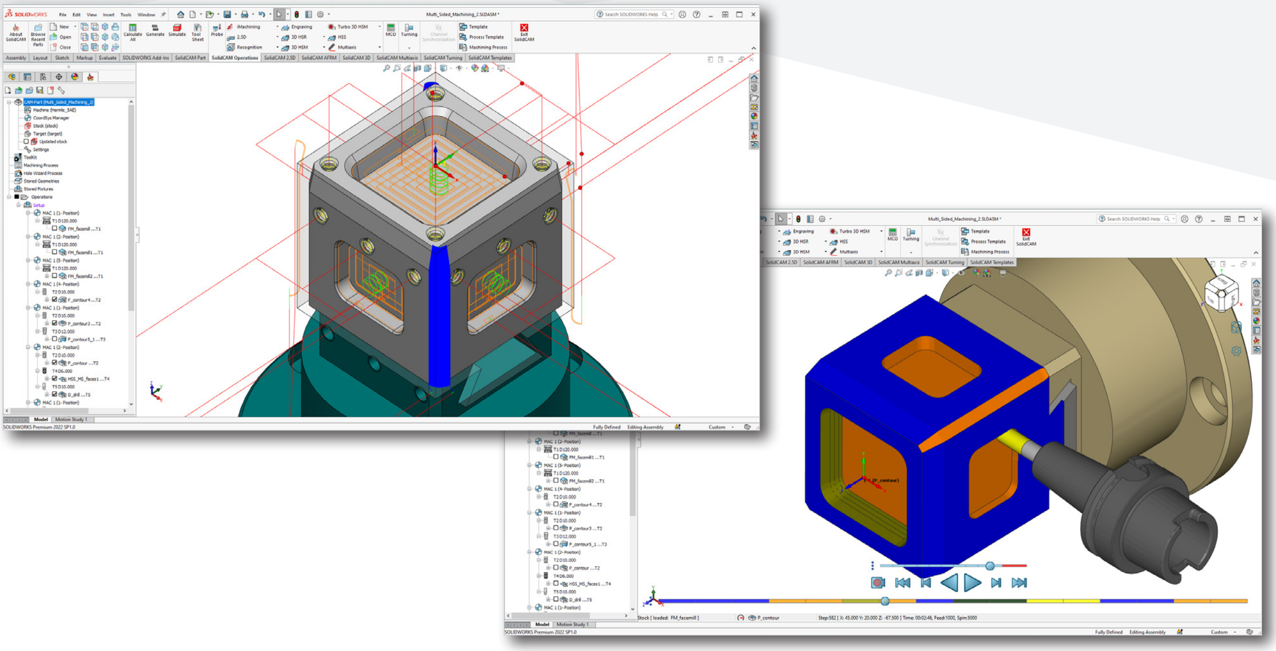
  These Drilling operations define the center drilling, the drilling and tapping of the holes located on all the part faces using MAC 1 (1- Position).

- **Counterbore machining (F_contour15_1)**

  This Profile operation defines the machining of the two counterbore features using MAC 1 (1- Position).

- **Chamfer machining (F_contour9 – F_contour18)**

  These Profile operations define the machining of the chamfers on the part edges using MAC 1 (1, 2, 3, 4 and 5- Positions).



SolidCAM Part: Multi_Sided_Machining_2.prz

The operations in this example illustrate the use of SolidCAM'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 SolidCAM Coordinate System with multiple positions.

- **Top/Side and inclined faces machining (FM_facemill; FM_facemill1; FM_facemill2)**

  The first Face Milling operation defines the machining of the part top face using Machine Coordinate System #1, Position #1 (MAC 1 (1- Position)). The second Face Milling operation defines the machining of the part side faces using MAC 1 (2- Position). The Transform 4th Axis option is used to create a circular pattern of operations around the revolution axis. The last Face Milling operation defines the machining of the inclined faces using MAC 1 (5- Position). The Transform 4th Axis option is used again to create a circular pattern of operations around the revolution axis.

- **Corner faces and top pocket machining (F_contour4; P_contour3; F_contour5_1)**

  The first Profile operation is defined to machine the vertical and inclined faces of two opposing corners using MAC 1 (4- Position). The Transform 4th Axis option is used again to create a circular pattern of operations around the revolution axis. The remaining Pocket and Profile operations define the machining of the top pocket and its inclined faces using MAC 1 (1- Position).

- **Side features machining (P_contour; HSS_MS_faces1; D_drill)**

  These operations define the machining of several side features using MAC 1 (2- Position). For each operation, the Transform 4th Axis option is used to create a circular pattern of relevant operations around the revolution axis.

  The Pocket operation machines the pockets. The HSS Morph between two adjacent surfaces operation machines the external fillets on the two opposing corners not yet machined. The Drilling operation defines the center drilling of the three holes located on each of the side faces.

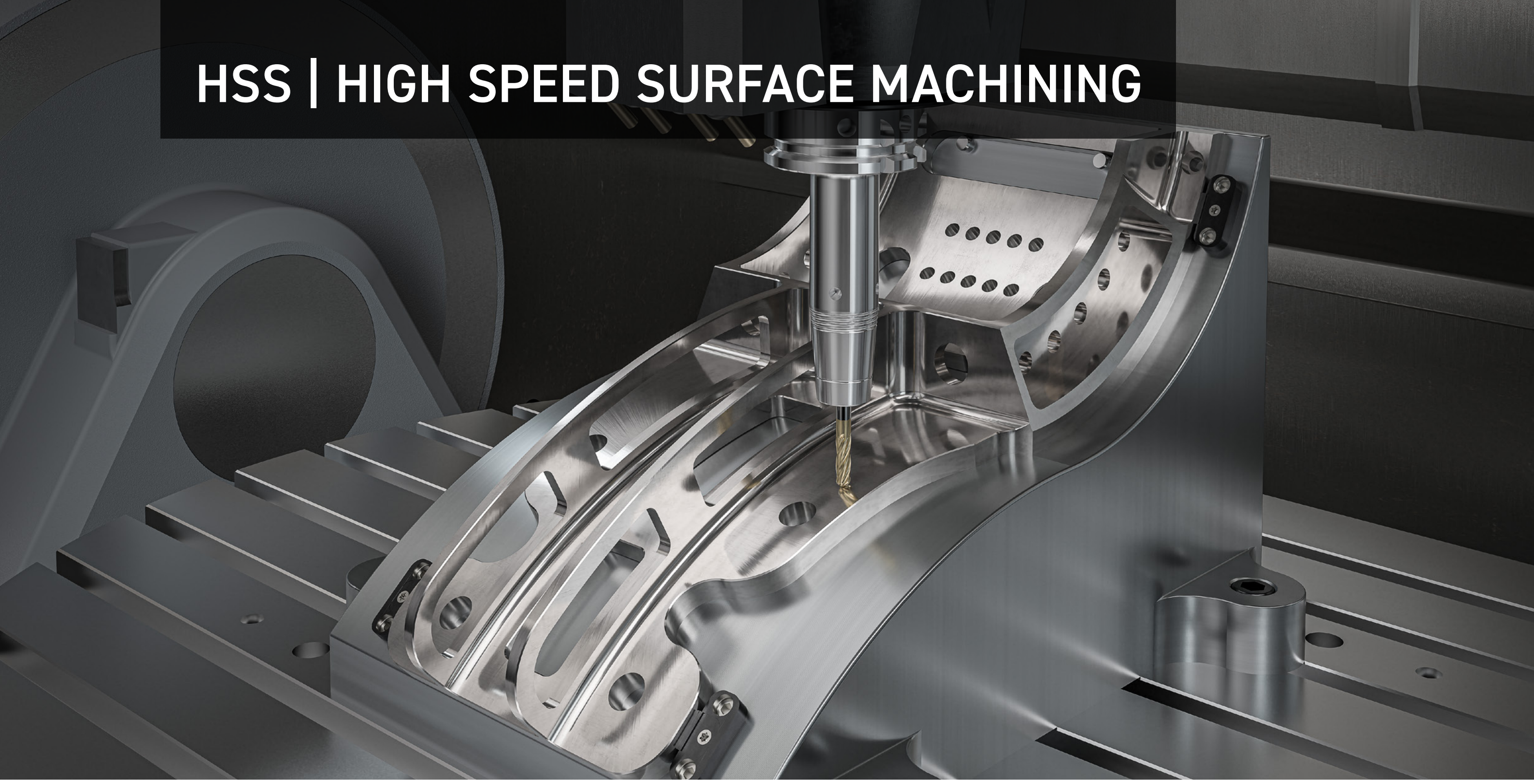
- **Top and side holes machining (D_drill1; D_drill1_1; D_drill1_2; D_drill_1; D_drill_2)**

  The first three Drilling operations define the center drilling, the drilling and tapping of the holes located on the part top face using MAC 1 (1- Position). The remaining Drilling operations define the drilling and tapping of the holes located on the part side faces using MAC 1 (2- Position). The Transform 4th Axis option is used again to create a circular pattern of operations around the revolution axis.

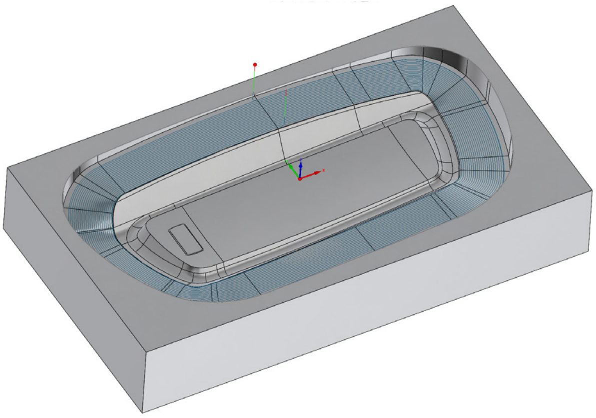
- **Chamfer machining (F_contour5; F_contour1)**

  These Profile operations define the machining of the chamfers on the top and side pocket edges using MAC 1 (1 and 2- Positions). For the last operation, the Transform 4th Axis option is used again to create a circular pattern of operations around the revolution axis.

HSS | HIGH SPEED SURFACE MACHINING



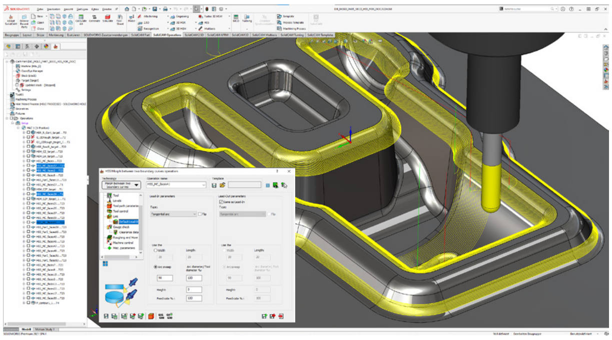
SolidCAM's HSS is a high speed surface machining module for smooth and powerful machining of localized surface areas of the part, including undercuts. It provides easy selection of the surfaces to be machined, eliminating the need to define boundaries. It supports both standard and shaped tools.



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

SolidCAM's HSS Module provides numerous surface machining strategies that produce efficient, smooth, gouge-free and optimal tool paths to finish the selected surfaces.

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

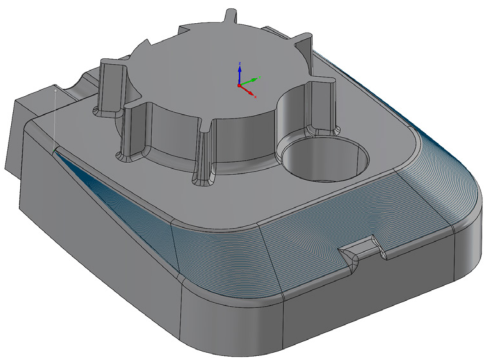


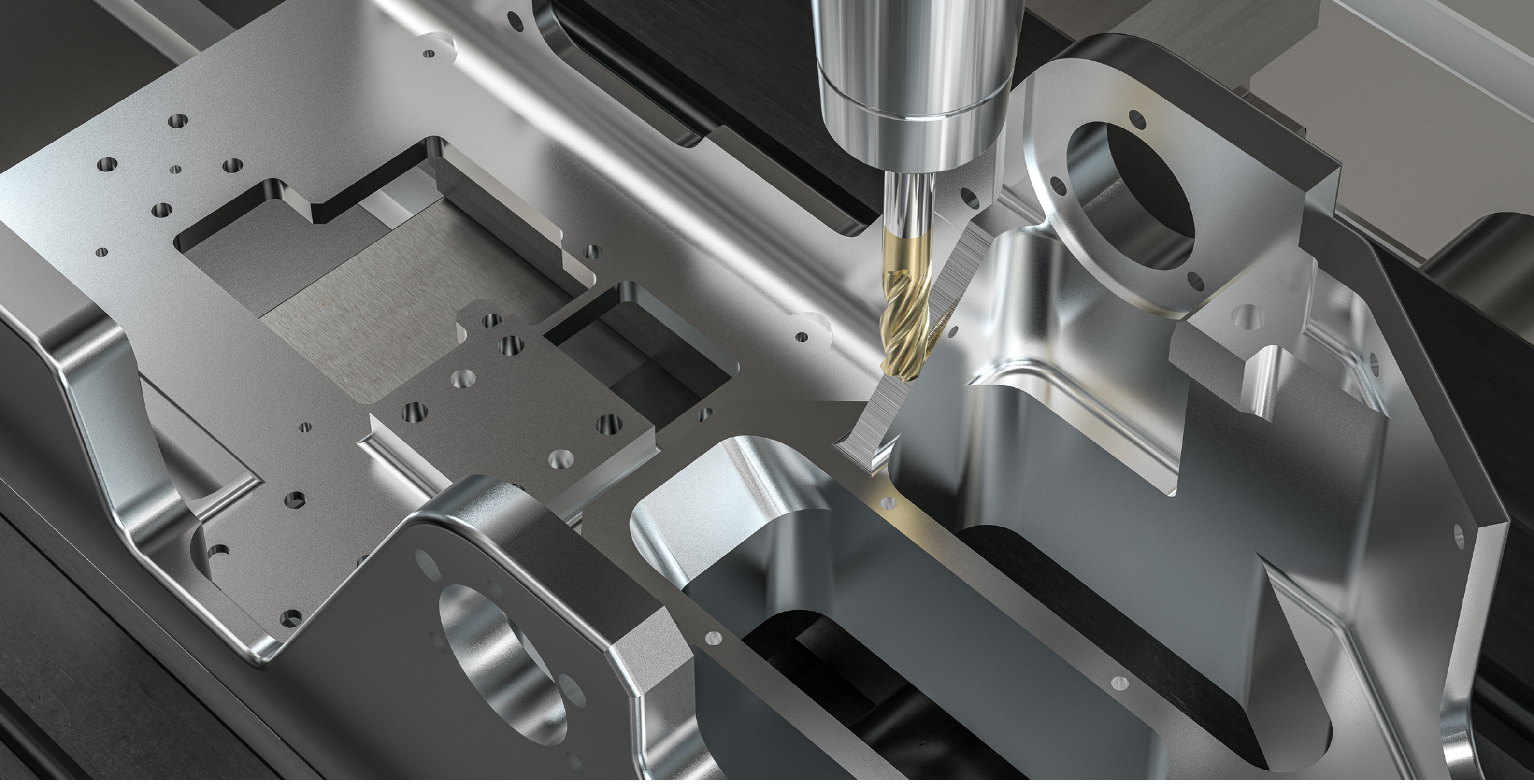
Total Tool Control to Machine Only the Areas You Choose

HSS is the Module that takes your 2.5D Milling far beyond profiles, pockets and faces, providing a 3D machining capability by driving along specific surfaces on prismatic and 3D 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 form a complex 3D shape (e.g., fillets).

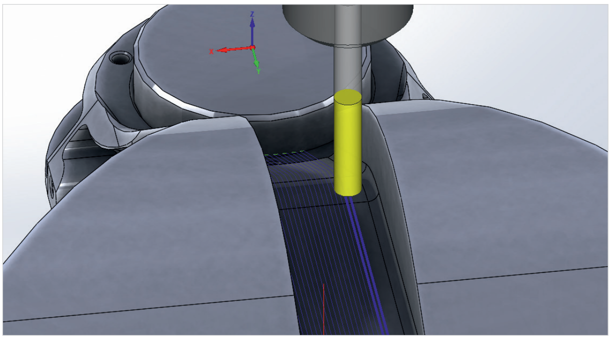
Experience total tool control to machine only areas you choose, without the need for constraint boundaries or construction geometry.





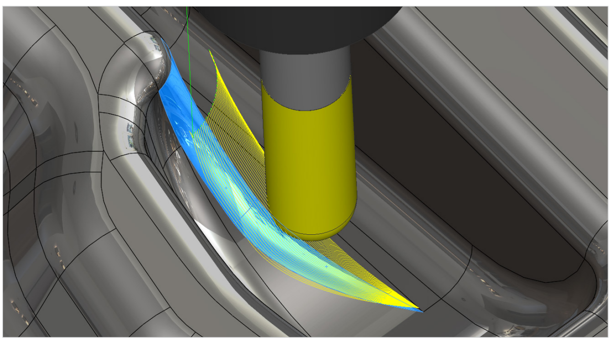
Advanced Gouge Control for Holder, Arbor and Tool

Complete collision control is available for Holder, Arbor and Tool. Adjoining check surfaces that are to be avoided can be selected. Several retract strategies are available under full user control.



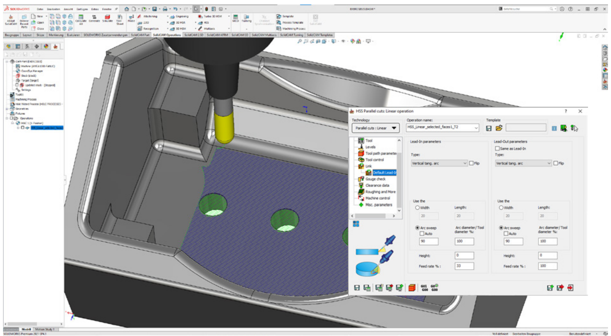
Advanced Linking

Total freedom to control tool entry and tool exit motion, no surface modifications needed. Tool paths can be extended or trimmed, gaps and holes can be jumped and you can choose from multiple lead-in/lead-out options.



Important Module for Every Machine Shop

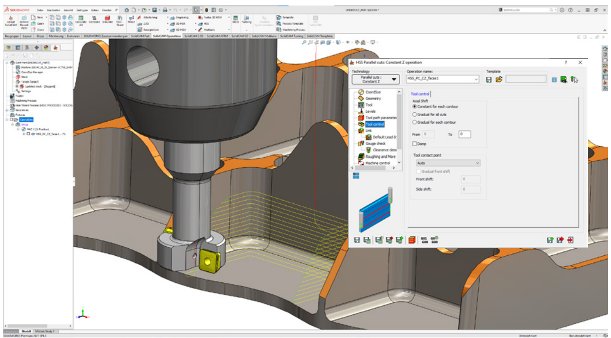
The advantages of SolidCAM's HSS Module translate into significantly increased surface quality. The HSS Module is an important and valuable add-on for every machine shop for the machining of all types of parts.



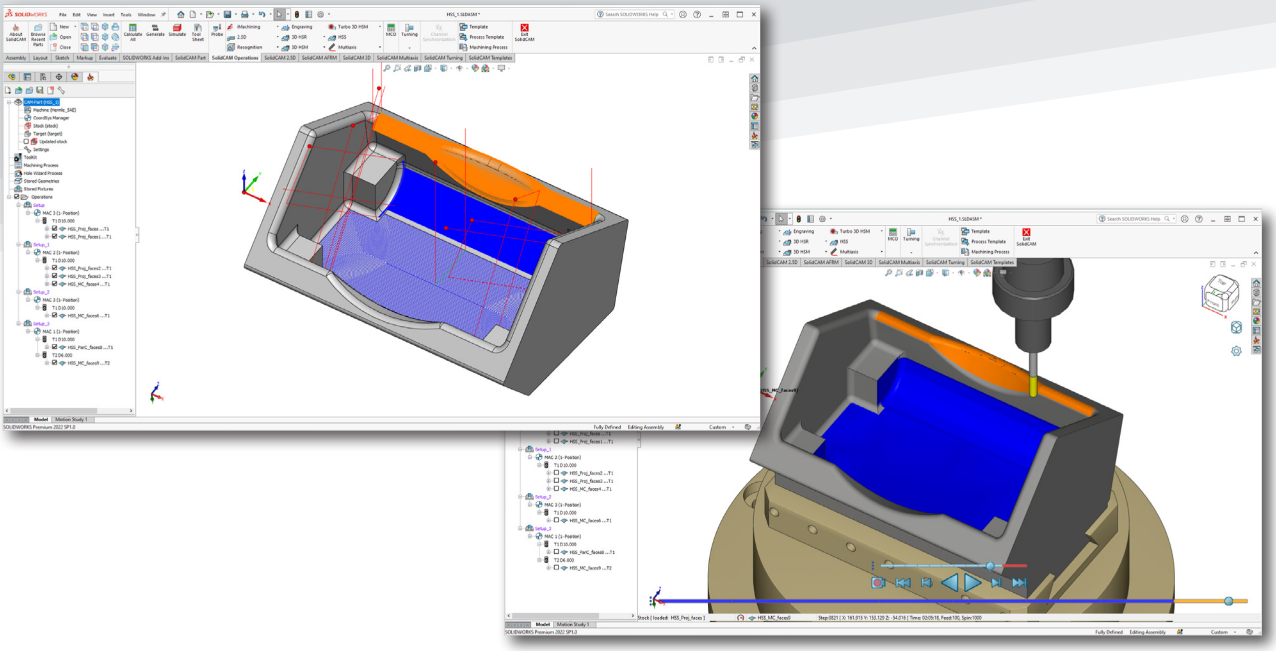
Linear High Speed Finishing tool path on selected surface

Handling Undercut in HSS

Use Tapered, Lollipop or T-Slot Tools for undercuts or geometry that is difficult to machine.



HSS CAM-PART EXAMPLES



SolidCAM Part: HSS_1.prz

The operations in this example illustrate the use of SolidCAM's 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)**

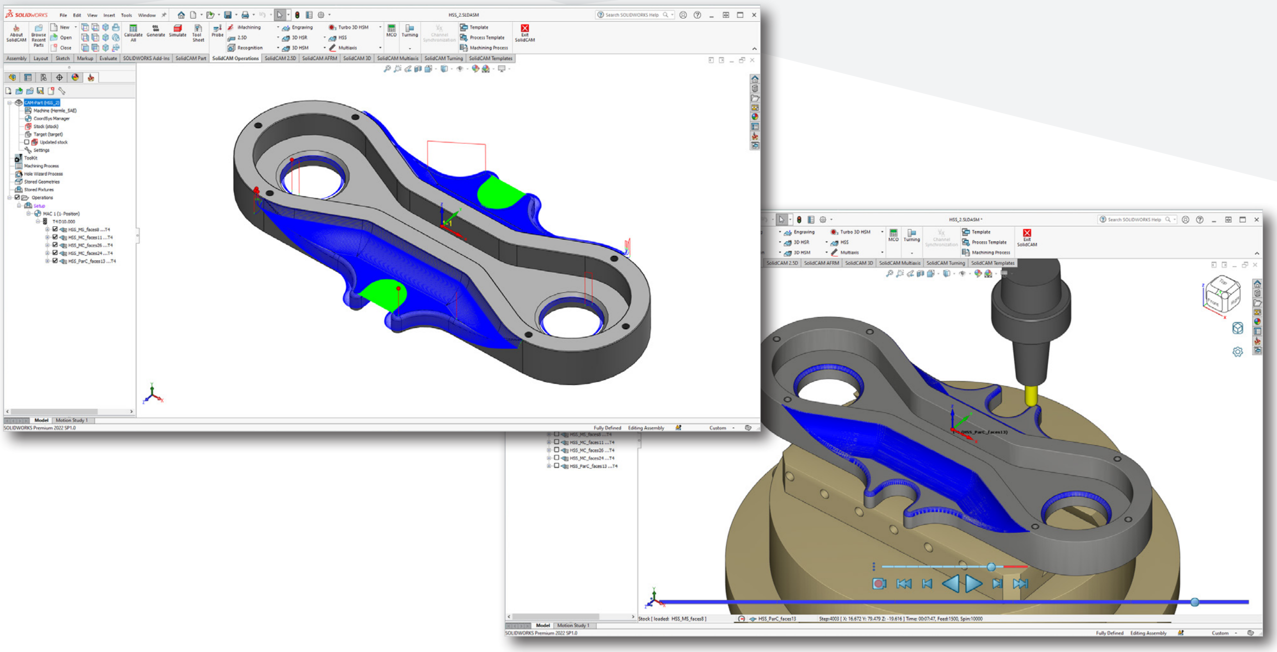
  These HSS operations use the Morph between two boundary curves strategy to define the machining of two large internal fillet areas. This strategy generates a constant tool path that is distributed evenly between the fillet boundaries. The Ø10 mm (0.4063 in) ball nose mill is used. Gouge check is enabled 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_faces9)**

  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 0.004 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.



SolidCAM Part: HSS_2.prz

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

- **Morph between surfaces machining (HSS_MS_faces8)**

  This HSS operation uses the Morph between two adjacent surfaces strategy to define the machining of the two concave side areas on the outside of the part. This strategy generates a constant tool path that is distributed evenly on a drive surface enclosed by two check surfaces. A Ø10 mm (0.4063 in) ball nose mill is used.

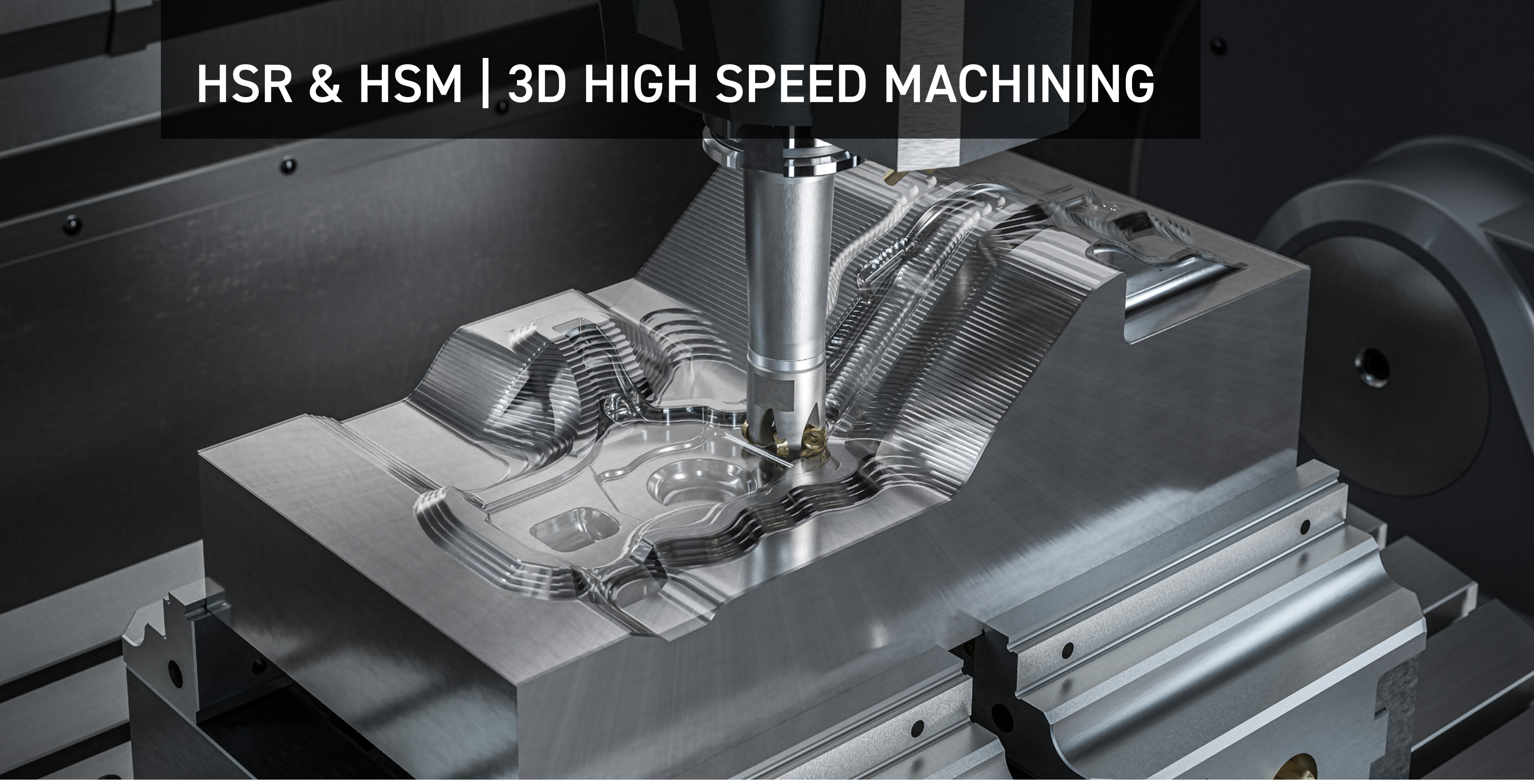
- **Morph between curves machining (HSS_MC_faces11; HSS_MC_faces26; HSS_MC_faces24)**

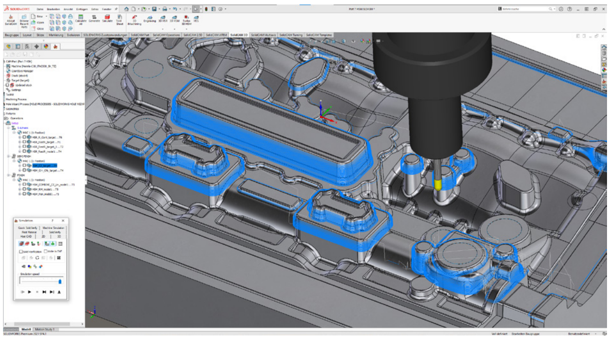
  These HSS operations use the Morph between two boundary curves strategy to define the machining of the upper and lower fillets on the external surfaces and the chamfers on the internal surfaces. This strategy generates a constant tool path that is distributed evenly between the fillet boundaries. The 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_faces13)**

  This HSS operation uses the Parallel to curves strategy to define the machining of the lower faces on the outside of the part. With this strategy, faces can be machined with cutting passes parallel to the selected curve. In this case, a tool path is generated to follow the surfaces and to close the large U-shaped gaps by Direct linking. The Ø10 mm (0.4063 in) ball nose mill is used and the tool leads into each cut with a Vertical tangential arc.

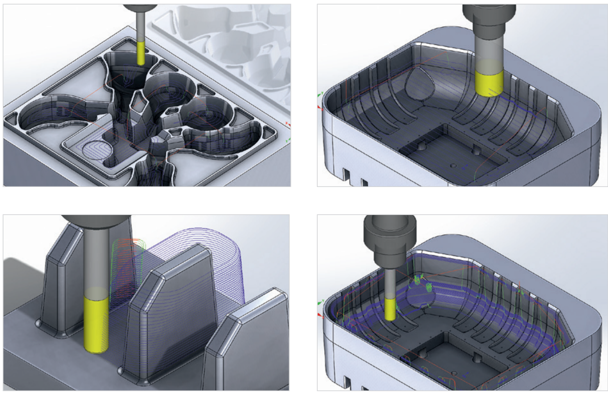
HSR & HSM | 3D HIGH SPEED MACHINING





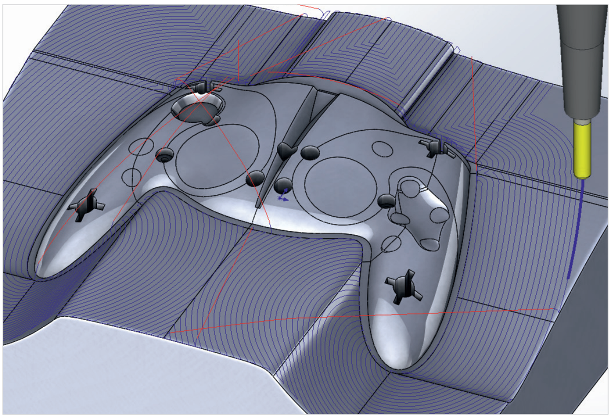
HSR – High Speed Roughing

SolidCAM HSR provides powerful high speed roughing strategies including: HM roughing, contour, hatch, hybrid rib-roughing, and rest roughing.



SolidCAM's High Speed Machining (HSM) offers unique machining and linking strategies for 3D high speed tool paths. It will smooth the paths of both cutting moves and retracts, wherever possible, to maintain a continuous machine tool motion – an essential requirement for maintaining higher feed rates and eliminating dwelling.

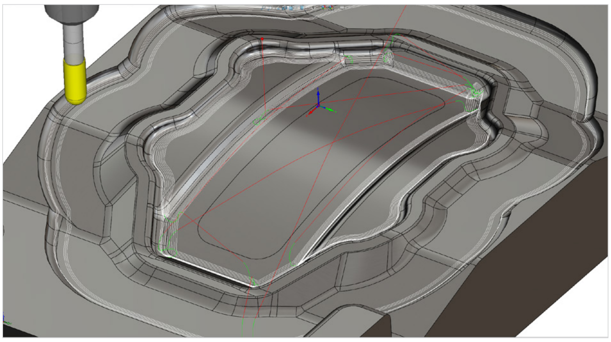
- 3D machining taken to an entirely new level of smoothness, efficiency and smart machining.
- The finest tool paths available anywhere for complex 3D parts, aerospace parts, molds, tools and dies.



High Speed Finishing

With SolidCAM's HSM Module, retracts to high Z-levels are kept to a minimum. Angles are smoothed with arcs wherever possible, ensuring that retractions never exceed what is necessary. This minimizes air cutting and machining time.

- Efficient and smooth tool path that translates to increased surface quality, reduced wear on your tools and a longer life for your machine tools.
- High Speed Machining is a must in today's machine shops to meet demands for ever-shorter lead and production times, lower costs and improved quality.



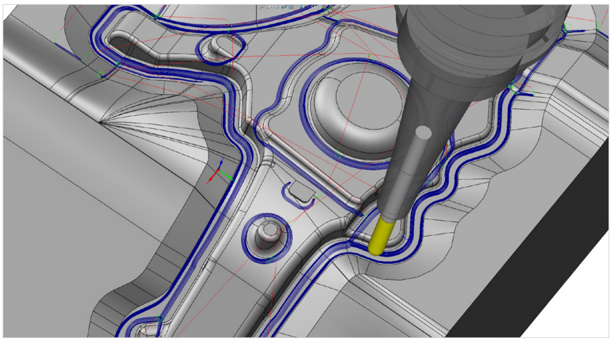
SolidCAM's HSM Module features several enhancements to CAM technology that make High Speed Machining operations possible (e.g., avoiding sharp angles in the tool path and generating smooth and tangential lead-ins and lead-outs).

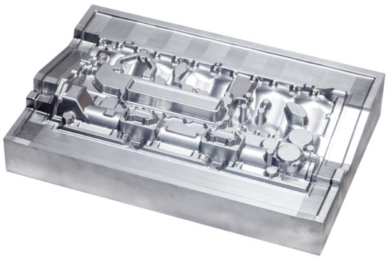
- Tool stays in contact with the material as much as possible, reducing non-machining moves.
- Working area can be precisely controlled using a comprehensive set of options, including silhouette boundaries, cutter contact area boundaries, shallow area boundaries and rest area boundaries.
- HSR/HSM tool paths can be edited after tool path creation using working areas, Z-level limits or a combination of both to control cutting moves or to exclude specific areas from machining.

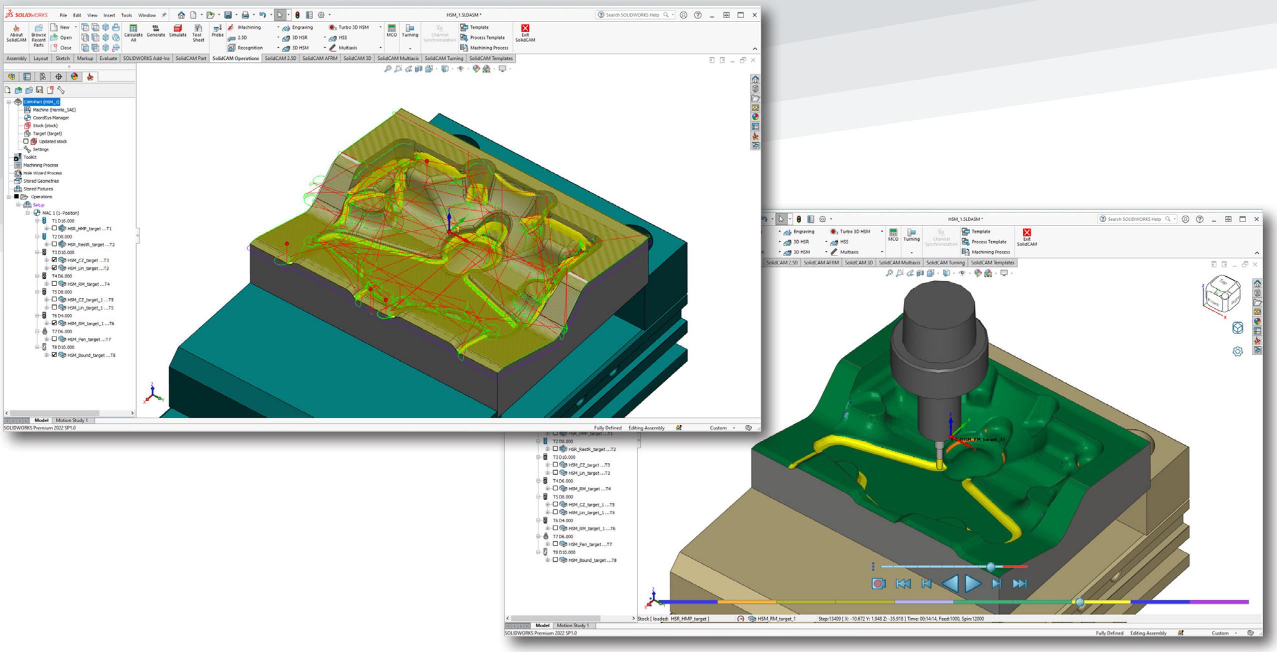
3D Machining to the Highest Level

SolidCAM's HSM Module is a powerful solution for all users who demand advanced High Speed Machining capabilities. It can also be used to improve the productivity of older CNCs with reduced air cutting and smoothing arcs thus maintaining continuous tool motion.

SolidCAM's HSM Module takes 3D machining performance to the highest level – all with your current machines.







SolidCAM Part: HSM_1.prz

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

- **Roughing and rest roughing (HSR_HMP_target; HSR_RestR_target)**

  The HM Roughing operation defines the contour roughing of the mold cavity. To machine the outside/inside areas, Constant is chosen for the Step down type and HM spiral is chosen for the Step over type. A machining allowance of 0.5 mm (0.02 in) remains on the surfaces for further HSM semi-finishing and finishing operations.

  The HSR Rest roughing operation defines the removal of unmachined material remaining after the previous operation. A 1 mm (0.04 in) Step down is specified with the same machining allowance.

- **Steep and shallow faces semi-finishing (HSM_CZ_target; HSM_Lin_target)**

  The HSM Constant Z machining operation defines the semi-finishing of the steep faces (from 40° to 90°). The HSM Linear machining operation defines the semi-finishing of the shallow faces (from 0° to 42°). For both operations, 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 Ø16 mm (0.6299 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 and shallow faces finishing (HSM_CZ_target_1; HSM_Lin_target_1)**

  The HSM Constant Z machining operation defines the finishing of the steep faces (from 40° to 90°). The HSM Linear machining operation defines the finishing of the shallow faces (from 0° to 42°). An Ø8 mm (0.3125 in) ball nose mill is used with the Apply fillets option.

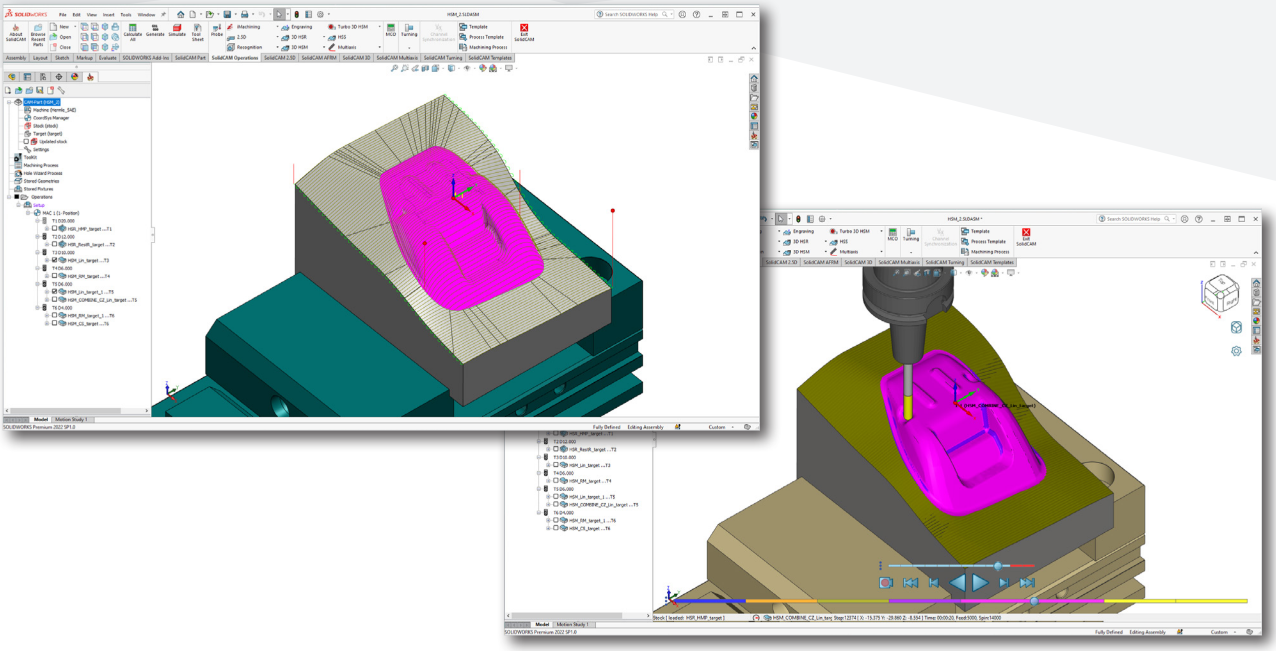
- **Corners finishing and final finishing (HSM_RM_target_1; HSM_Pen_target)**

  The 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 Ø12 mm (0.4724 in) to determine the rest material that requires machining.

  The HSM Pencil milling operation defines the final finishing of the inside corner areas. A Ø6 mm (0.2344 in) lollipop mill is used to remove any material that could not be reached in the previous operations.

- **Chamfer machining (HSM_Bound_target)**

  This HSM Boundary machining operation defines the chamfering of the upper edges using a Ø10 mm (0.4083 in) chamfer drill.



SolidCAM Part: HSM_2.prz

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

- **Roughing and rest roughing (HSR_HMP_target; HSR_RestR_target)**

  The HM Roughing operation defines the contour roughing of the mold core. To machine the outside/inside areas, Constant is chosen for the Step down type and HM spiral is chosen for the Step over type. A machining allowance of 0.3 mm (0.012 in) remains on the surfaces for further HSM semi-finishing and finishing operations.

  The HSR Rest roughing operation defines the removal of unmachined material remaining after the previous operation. A 2 mm (0.08 in) Step down is specified with the same machining allowance.

- **Semi-finishing and corners semi-finishing (HSM_Lin_target; HSM_RM_target)**

  The HSM Linear machining operation defines the part semi-finishing. A 2 mm (0.08 in) Step over is specified and the Apply fillets option adds virtual fillets to smooth the tool path at the corners. The HSM Rest machining operation defines the semi-finishing of the inside corners and it is used to avoid tool overload 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. For both operations, a machining allowance of 0.1 mm (0.004 in) remains on the surfaces for further HSM finishing operations.

- **Linear Finishing (HSM_Lin_target_1)**

  This HSM Linear machining operation defines the part finishing. A Ø6 mm (0.2344 in) ball nose mill is used with the Apply fillets option. The Constraint boundaries are created manually to limit the tool path to the core area only.

- **Combined finishing of steep and shallow areas (HSM_COMBINE_CZ_Lin_target)**

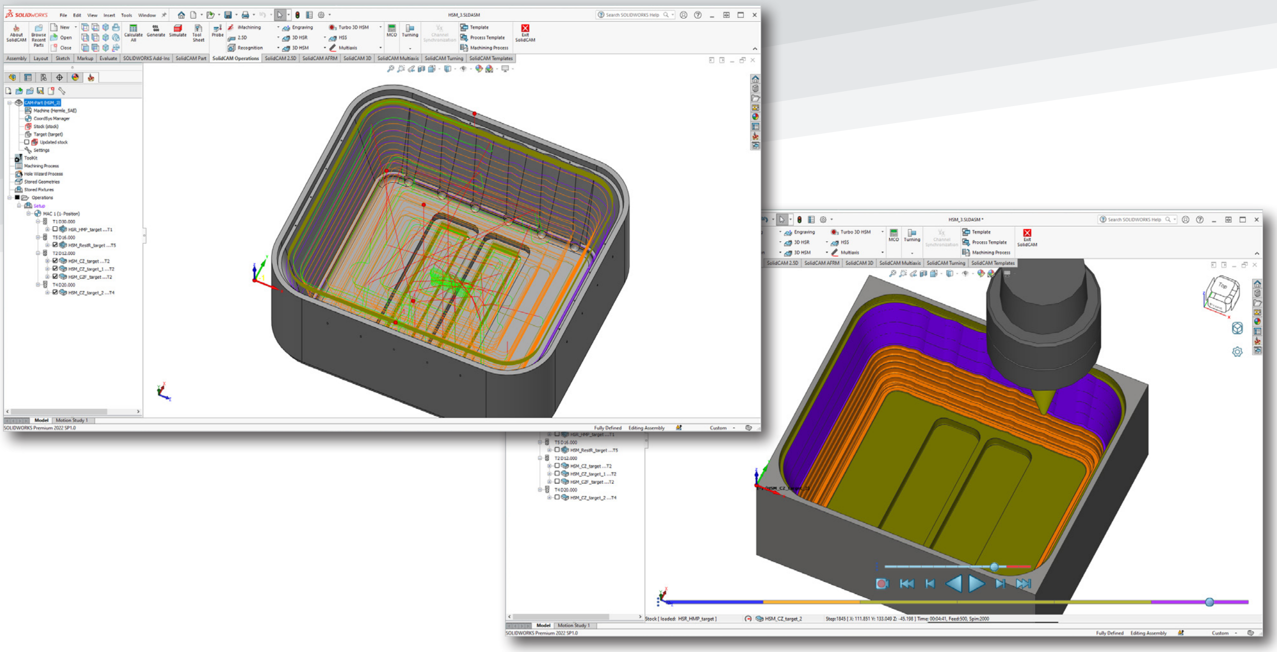
  This HSM Combine machining operation defines the Constant Z finishing of the steep areas (from 35° to 90°) and Linear finishing of the shallow areas (from 0° to 38°) according to the Constraint boundaries manually created in the previous operation. The Ø6 mm (0.2344 in) ball nose mill is used with the Apply fillets option and the Linear passes are defined by a 90° angle.

- **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 Ø8.2 mm (0.3228 in) to determine the rest material that requires machining.

- **Concave areas machining (HSM_CS_target)**

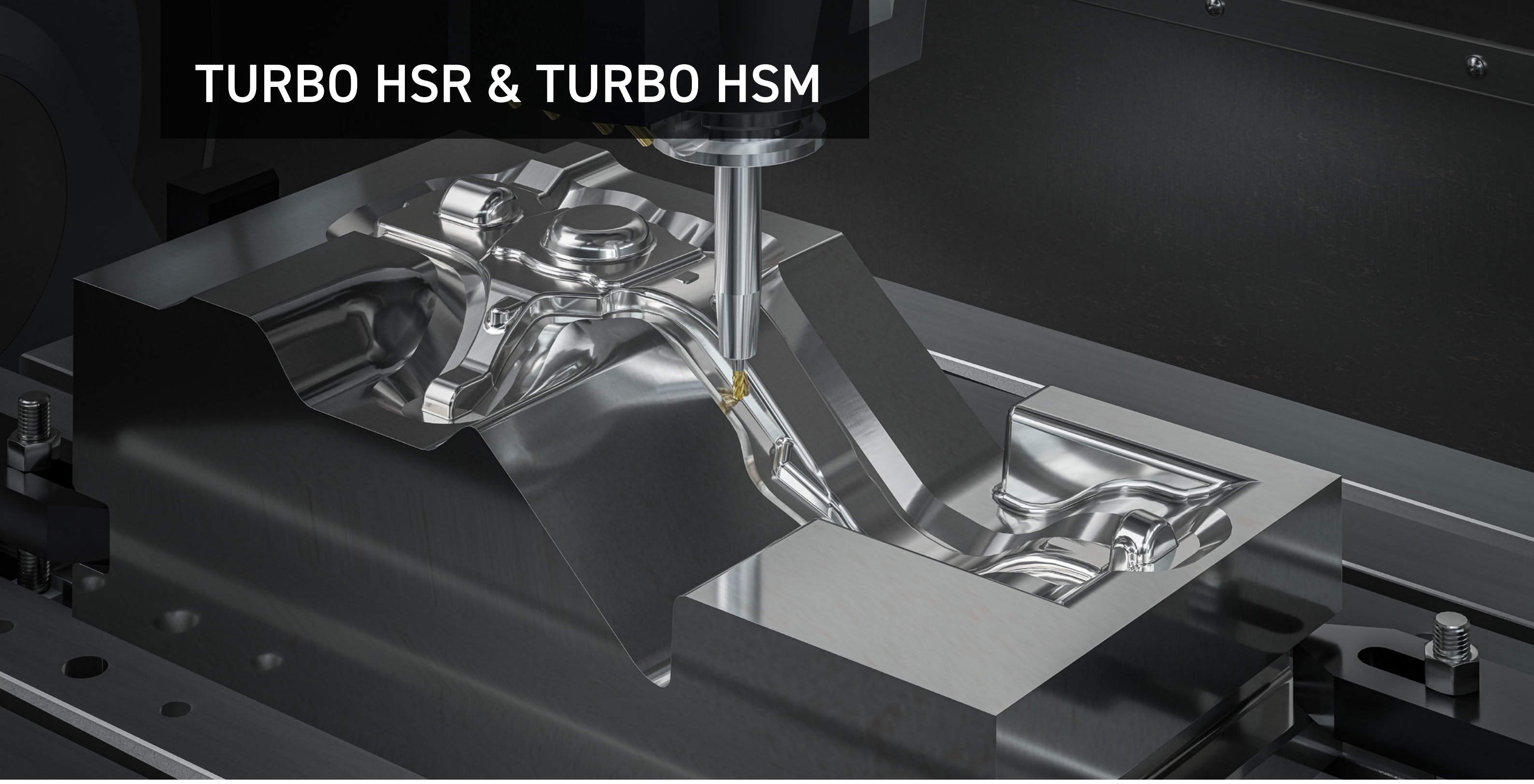
  This HSM 3D Constant step over operation defines the finishing of the two concave areas down to a certain depth. The Ø4 mm (0.1563 in) ball nose mill is used with a step over of 0.05 mm (0.002 in) specified for both the horizontal and vertical directions.



SolidCAM Part: HSM_3.prz

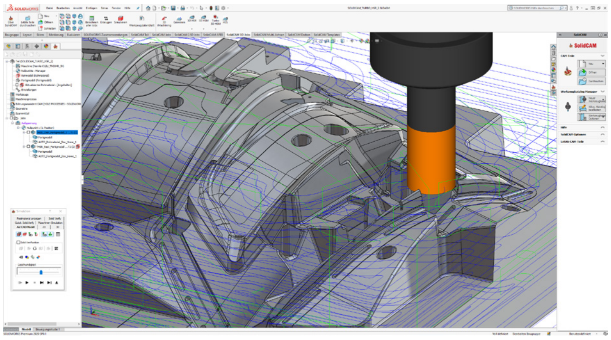
The operations in this example illustrate the use of SolidCAM's 3D Mill High Speed Machining (HSR/HSM) strategies to machine the part shown above.

- **Rough machining (HSR_HMP_target)**
  This HM Roughing operation defines the contour 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). To machine the inside areas, the Constant option is chosen for the Step down type and the Cavity option is chosen for the Step over type. A machining allowance of 0.5 mm (0.02 in) remains on the surfaces for further HSM semi-finishing and finishing operations.
- **Rest roughing (HSM_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_1)**
  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.



SolidCAM's THSR and THSM

SolidCAM's Turbo 3D HSR (THSR) and Turbo 3D HSM (THSM) are powerful High Speed Roughing (HSR) and High Speed Machining (HSM) Modules for much faster calculations than the regular HSR / HSM Modules.



THSR and THSM offer unique machining and linking strategies for generating high speed tool paths. The 3-Axis calculation engine recalculates the tool path at lightning speeds. Its 64-bit architecture completely utilizes all the cores for tool path calculations.

The current THSR strategies (Hatch, Contour, and Rest) remove large volume of excess material rapidly and leave a small amount of stock for semi-finishing and finishing strategies. The biggest advantage of these strategies is that the tool path contours are always collision-free.

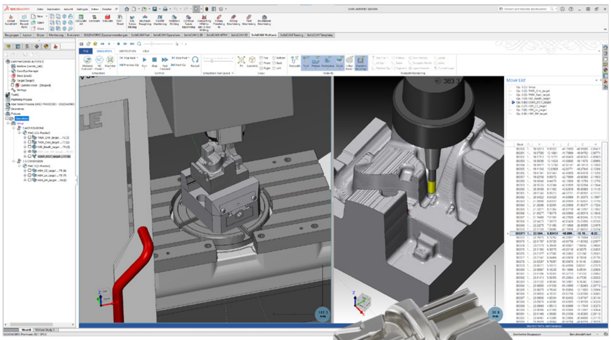
- Extremely fast calculation and generation of tool paths.
- Fewer options – quicker definition of High Speed Machining jobs.
- Advanced gouge checking strategies.
- Most efficient, collision-free tool path.

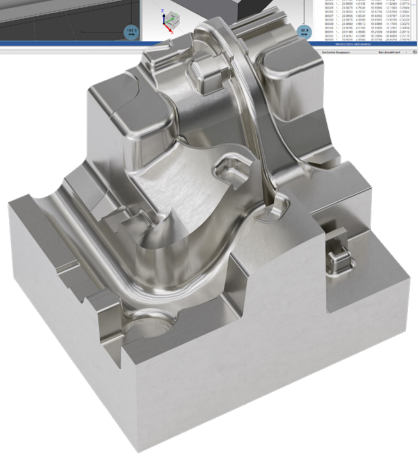
Auto 3+2 High Speed Roughing

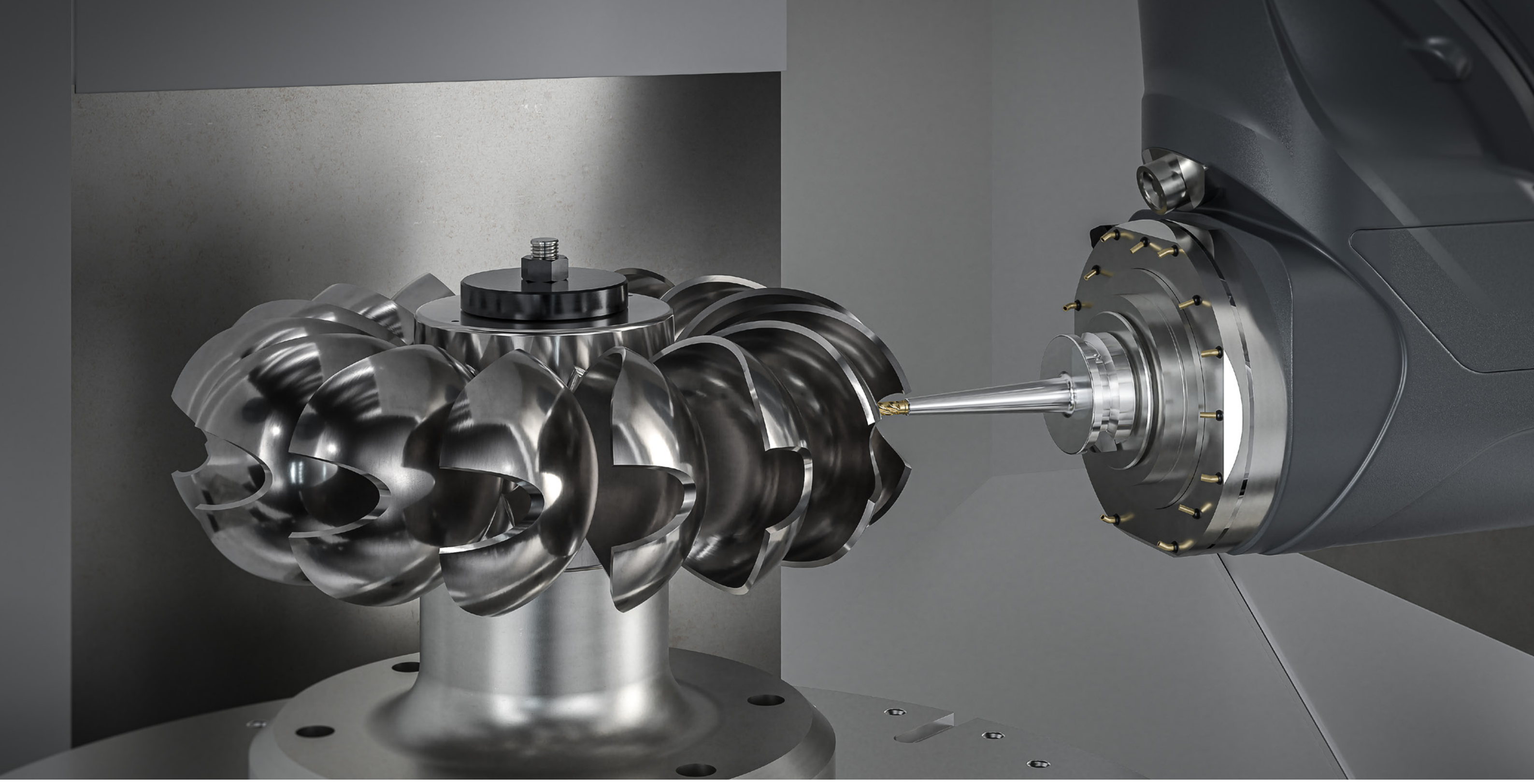
Hybrid High Speed Roughing with 5X transitions between THSR Jobs

Auto 3+2 THSR Hatch and Contour technologies detect and process undercut areas in the specified range of processing angles. The functionalities of these technologies remain mainly the same as Turbo 3D HSR with an added advantage of Auto 3+2 Axis support.

The Auto 3+2 Module is useful to efficiently machine undercut areas and gaining increased access from a single direction. It minimizes the number of operations with different machining directions.



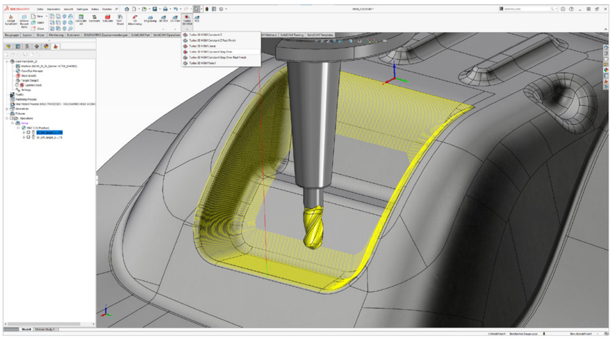


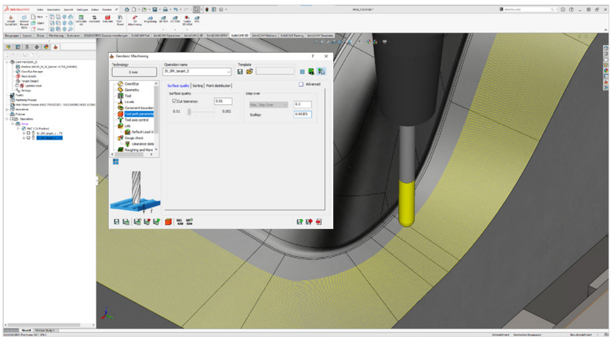


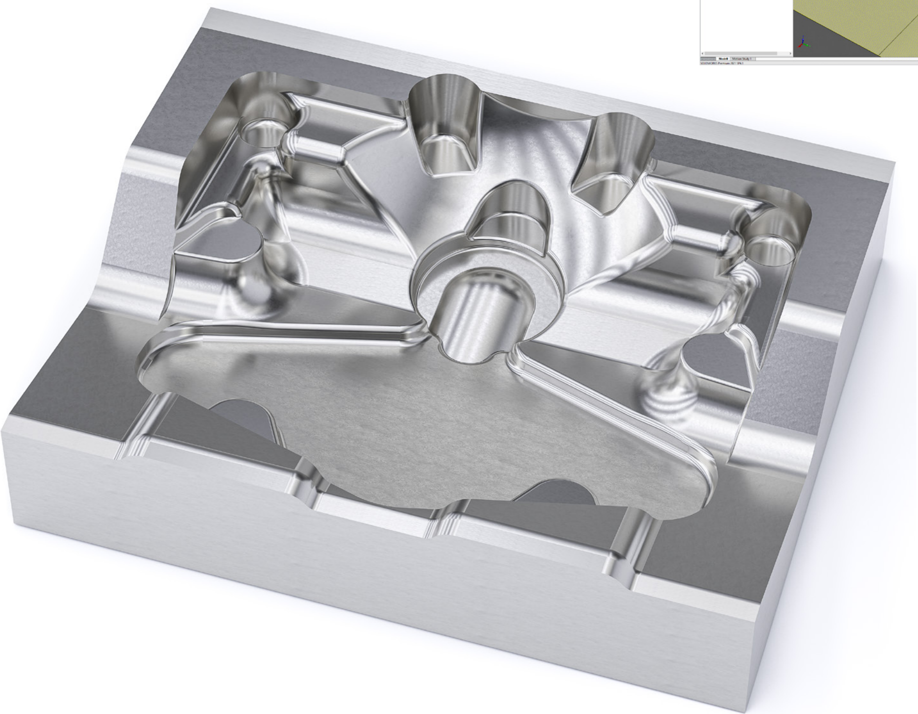
Geodesic Machining

SolidCAM's Geodesic Machining enables machining of complex 3D shapes (Solid Model & Surface Groups) with a tool path that has constant stepover and undercut areas. This module generates a pattern of tool path with measurable constant step over that remains constant, even on steep and shallow walls, as it machines different surfaces or an entire model. SolidCAM uses a global distance field without a fixed direction. Typically, a step over is calculated with reference to a vector direction, but in Geodesic Machining global distance and a step over are used without a fixed direction.

- Various tool path patterns available.
- Constant 3D distance between consecutive cuts.
- Works effectively even in undercut situations.
- Utilizes single entry and exit move.

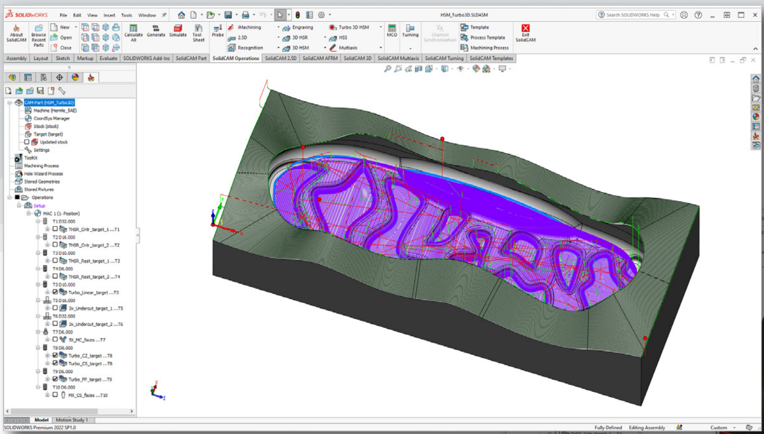


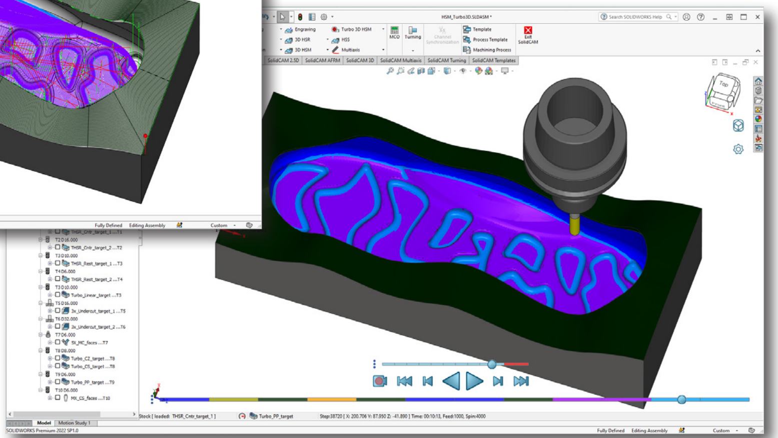




Excellent surface finish of 0.4 µm with SolidCAM HSM strategies

TURBO HSR & TURBO HSM CAM-PART EXAMPLE





SolidCAM Part: HSM_Turbo3D.prz

The operations in this example illustrate the use of SolidCAM's Turbo 3D HSR & Turbo 3D HSM strategies to machine the part shown above.

- **Rough machining (THSR_Cntr_target_1; THSR_Cntr_target_2)**
  These Turbo HSR operations define the contour roughing of the mold cavity. First operation uses a Ø32 mm (1.25 in) bull nose mill with 6 mm (0.24 in) corner radius and specifies a Step down of 4 mm (0.16 in). Second operation uses a Ø16 mm (0.625 in) bull nose mill with 3 mm (0.12 in) corner radius and specifies a Step down of 1 mm (0.04 in). For both, the Output type is set to High surface quality and the Axial Shift is Constant for each contour. The optimal Ramping type is determined automatically. A Global offset of 0.5 mm (0.02 in) is allowed on the surfaces.

- **Rest roughing (THSR_Rest_target_1; THSR_Rest_target_2)**
  These Turbo HSR operations define the rest roughing of the cavity lower depressions. Ø10 mm (0.4063 in) and Ø6 mm (0.2344 in) ball nose mills are used. Respectively, Step downs of 0.5 mm (0.02 in) and 0.25 mm (0.01 in) are specified. Output type is High surface quality, Axial Shift is Constant for each contour, Ramping type is determined automatically and a Global offset of 0.5 mm (0.02 in) is allowed on the surfaces.

- **Parting surface finishing (Turbo_Linear_target)**
  This Turbo 3D HSM Linear operation defines the finish machining of the contoured parting surface. The Ø10 mm (0.4063 in) ball nose mill is used and an Adaptive step over of 0.5 mm (0.02 in) maximum is specified. Zigzag for the Cutting method and an Optimal machining angle in X,Y greatly reduces machining time. Links are performed optimally via operation defaults.

- **Cavity walls machining (3x_Undercut_target_1; 3x_Undercut_target_2)**
  These 3 Axis Undercut Machining operations define the machining of the cavity walls. First operation uses a Ø16 mm (0.625 in) slot mill with 3 mm (0.12 in) corner radius, specifies a Step down of 0.4 mm (0.016 in) and an Offset of 0.5 mm (0.02 in). Second operation specifies the same Step down and uses a Ø32 mm (1.25 in) slot mill with 2 mm (0.08 in) corner radius. Machine only undercuts is enabled for both operations, which are further optimized by the selected Passes parameters.

- **Upper fillet finishing (5X_MC_faces)**
  This Sim. 5-Axis Milling operation defines the finishing of the fillet along the undercut feature using a Ø6 mm (0.2344 in) lollipop mill. The tool is Tilted relative to cutting direction and is driven by the fillet surface and the Morph between two boundary curves strategy. Gouge checking is enabled to detect and avoid possible collisions.

- **Shallow faces finishing (Turbo_CZ_target)**

  This Turbo 3D HSM Constant Z operation defines the finishing of the cavity lower faces and depressions using an Ø8 mm (0.3125 in) ball nose mill. Detect flat areas is enabled with a Step down of 0.2 mm (0.008 in). Also enabled is detection of Shallow areas with a Maximum step over of 0.5 mm (0.02 in). Links are performed optimally via operation defaults.

- **Steep faces finishing (Turbo_CS_target)**

  This Turbo 3D HSM Constant step over operation defines the finishing of the cavity side arches having an Angle range of 0° to 45°. The Ø8 mm (0.3125 in) ball nose mill is used. A Maximum step over of 0.2 mm (0.008 in) is specified and the Links are performed optimally via operation defaults.

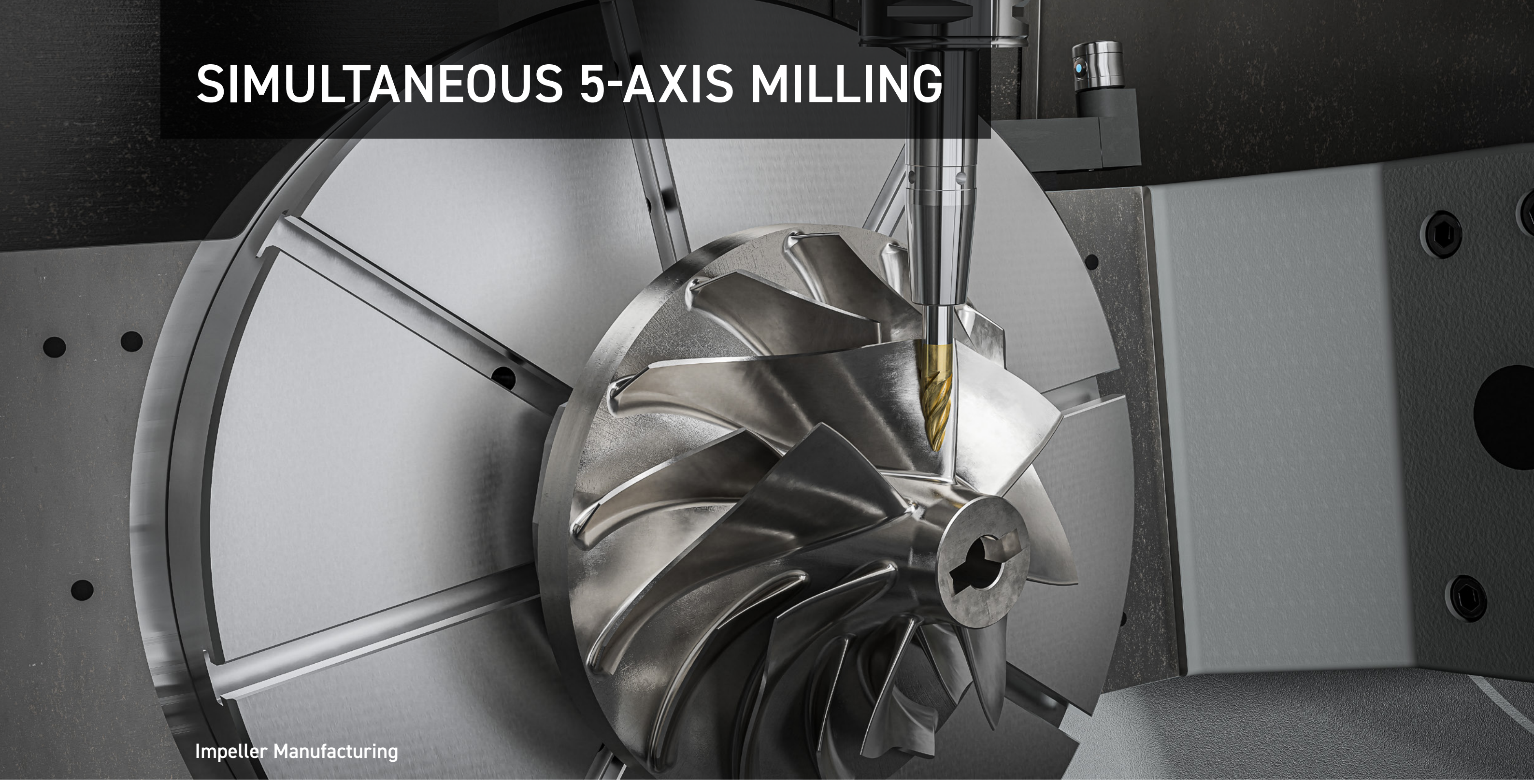
- **Lower fillets finishing (Turbo_PP_target)**

  This Turbo 3D HSM Pencil operation defines the finishing of the fillets in the cavity and lower depressions. A Ø6 mm (0.2344 in) ball nose mill is used. Multi pencil is selected to perform at least 5 multiple passes having a Maximum step over of 0.2 mm (0.008 in). Links are performed optimally.

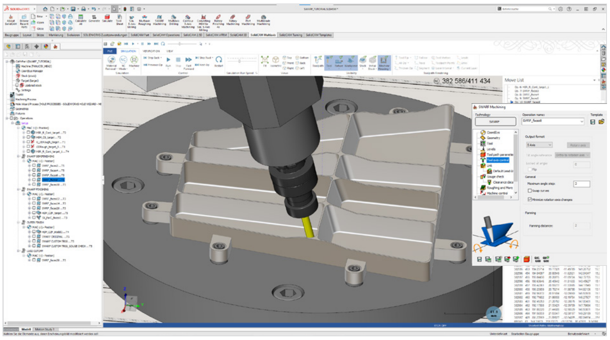
- **Covered depression finishing (MX_CS_faces)**

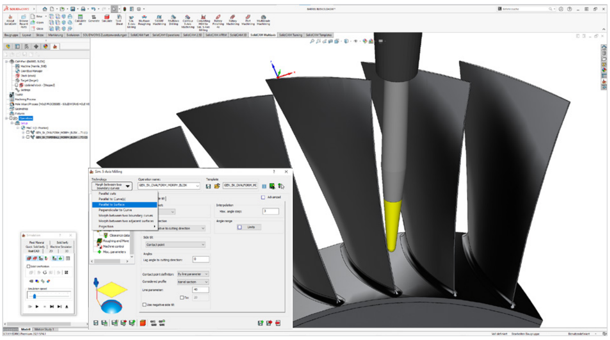
  This Geodesic Machining operation defines the finishing of the lower depression that is partially covered by the undercut feature. Another Ø6 mm (0.2344 in) ball nose mill is used and a Constant Step Over of 0.25 mm (0.01 in) is specified. The Tool axis control and Gouge check parameters generate a consistent and reliable tool path on the shallow and fillet surfaces while avoiding the undercut and parting surfaces.

SIMULTANEOUS 5-AXIS MILLING



Impeller Manufacturing





Support of circular-segment cutters with barrel-, oval- and tapered geometry is constantly being implemented by SolidCAM

Benefit from the most tested and 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 the tool path:

- Wide variety of Simultaneous 5X cutting strategies.
- Flow line cutting that produces a tool path following the natural shape of the component.
- Multi-surface finish machining keeps the tool normal to the surface (or with specified lead and lag) to achieve a smooth surface finish.
- Advanced tool tilting control including direct control over side tilting and lead/lag angles.
- Automatic collision avoidance strategies that assess each part of both the tool and holder.
- Multiaxis rest roughing efficiently removes the remaining material left by the previously used larger diameter cutter.
- Realistic full 3D machine simulation that offers comprehensive collision and axis limits checking.

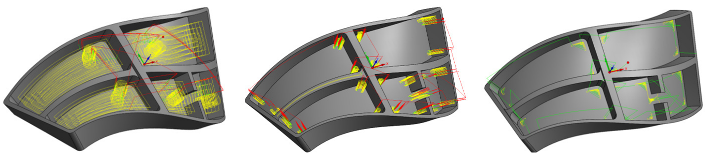
Flexibility and Control

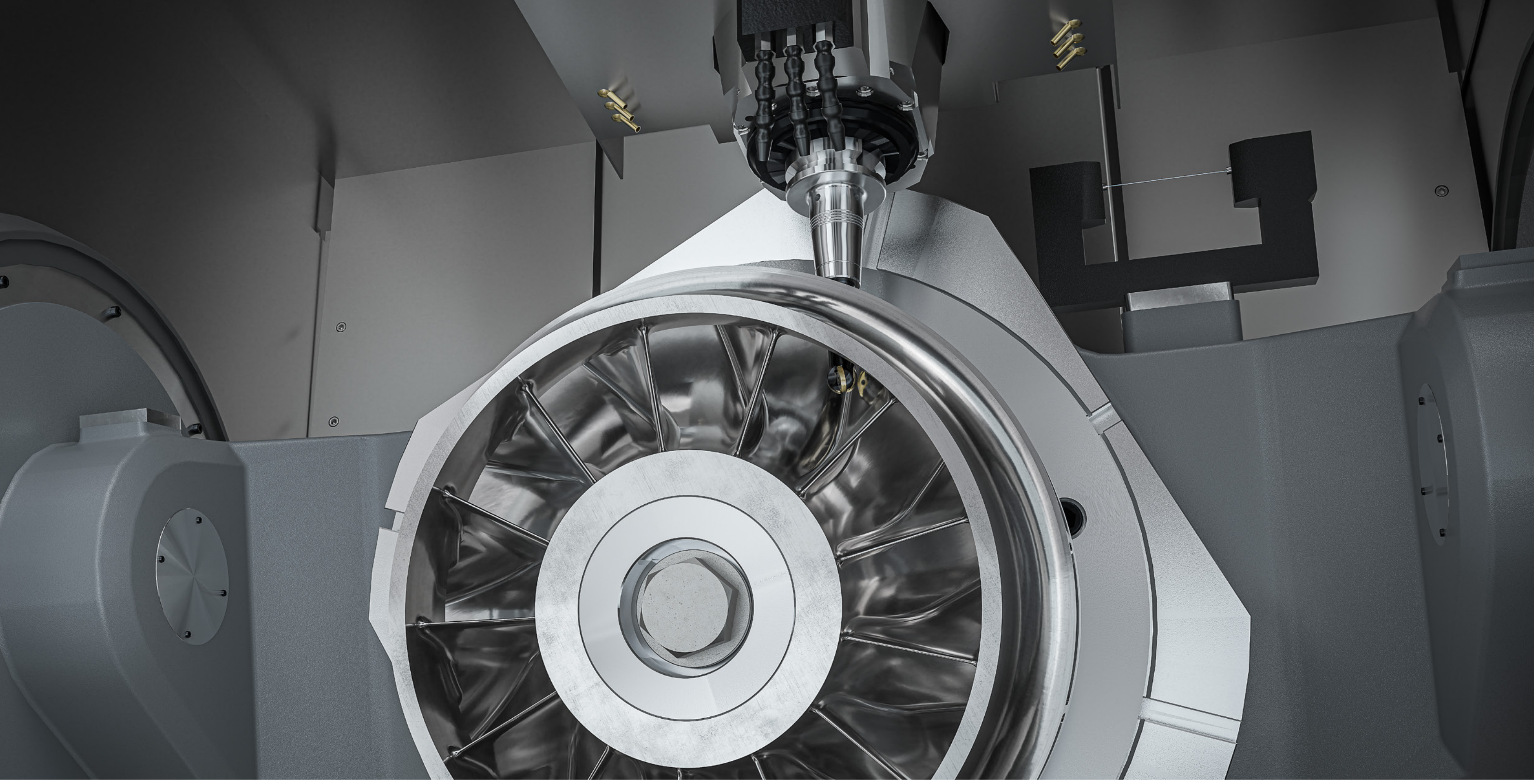
Each 5-Axis machining strategy provides sophisticated options for approach/link control and tool axis control.

Link and approach moves are fully gouge protected and different strategies may be used depending on the distance of the link move. SolidCAM 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

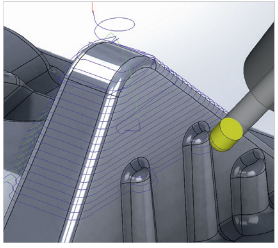
Collision avoidance is supported for both the tool and holder, and a range of strategies is offered for avoiding collisions. The Machine Simulation provides complete visualization of the gouge checking.





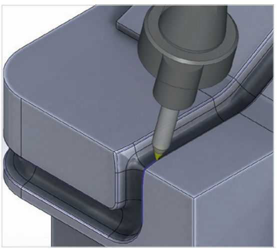
HSM to Sim. 5-Axis Milling

Converts HSM 3D tool paths to full 5-Axis collision-protected tool paths. This maintains optimum contact point between the tool and the part, and it enables the use of shorter tools for more stability and rigidity.



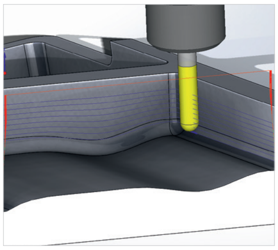
Contour 5X Machining

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.



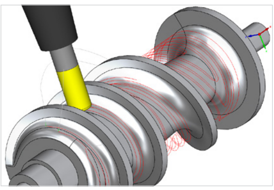
SWARF Machining

Allows tilting of the tool to machine side walls at the correct angle. The SWARF operation utilizes the entire cutting length of the tool, resulting in better surface quality and shorter machining time.



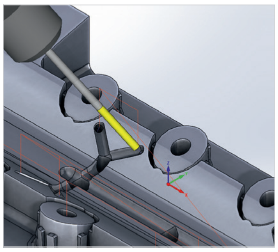
Screw Machining

Generates 4-Axis rotary roughing/finishing tool path for screws using bull nose, ball nose or flat end mills.



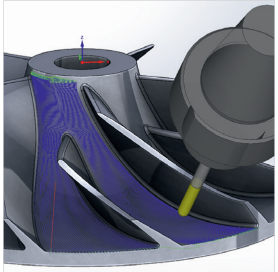
Multiaxis Drilling

Uses SolidCAM's automatic hole recognition technology and then performs drilling, tapping or boring cycles at any hole direction, easily and quickly. All the advanced linking, tilting and collision avoidance strategies are available in this operation.



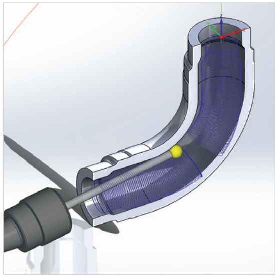
Multiblade Machining

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.

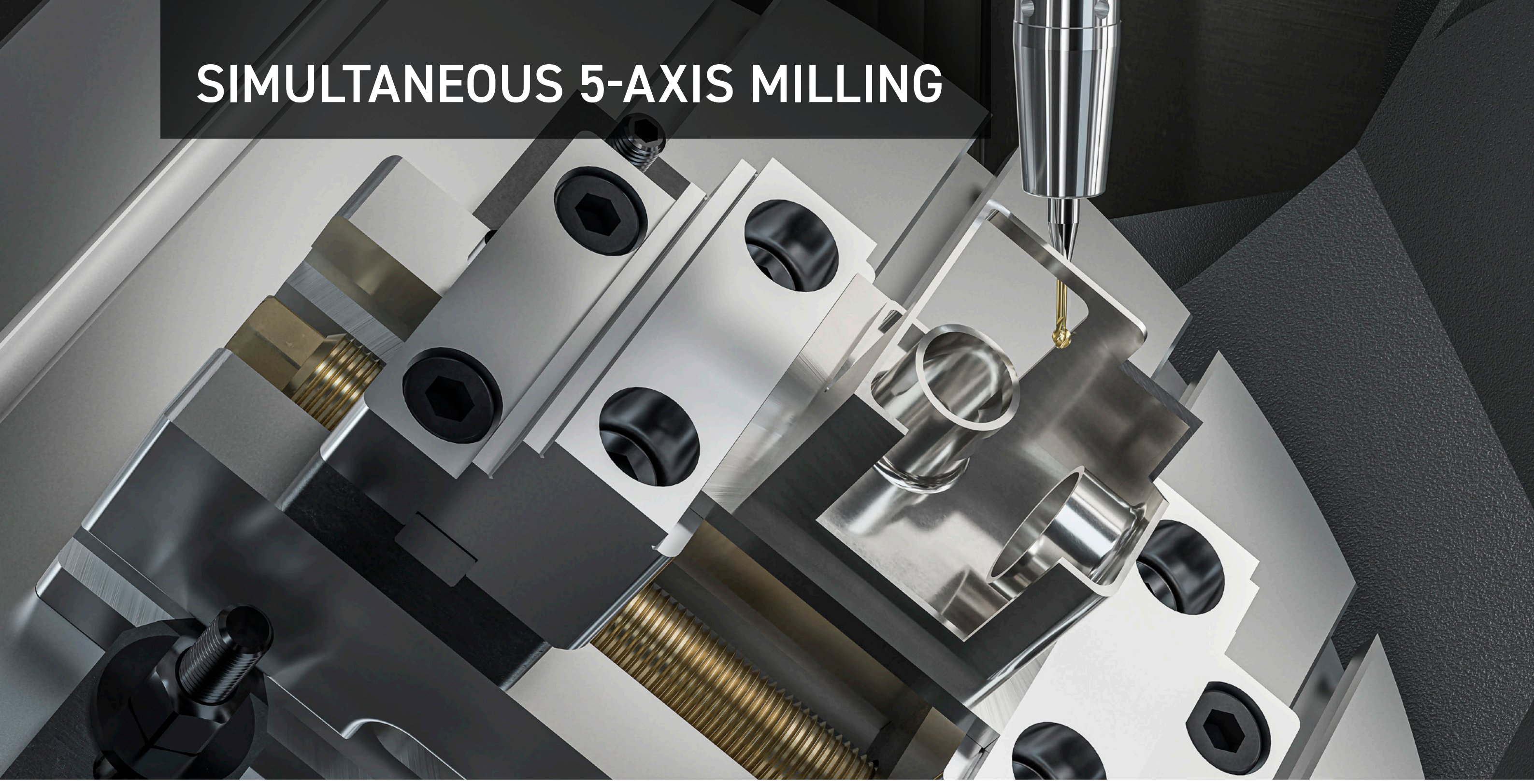


Port Machining

Machines intake and exhaust ducts as well as inlets or outlets of pumps, in castings or steel blocks, with tapered lollipop tools. Roughing and finishing operations can be quickly and easily defined and reliably simulated with complete collision control for the entire tool and holder.

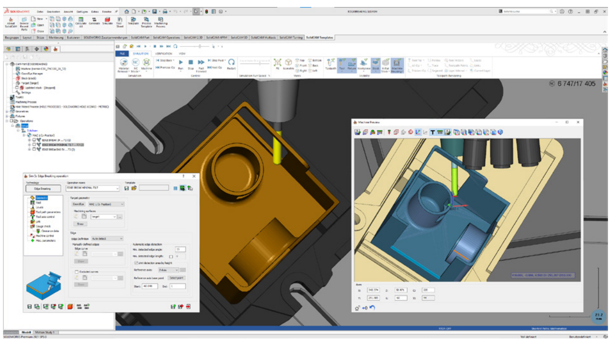


SIMULTANEOUS 5-AXIS MILLING



Simultaneous 5X Edge Breaking

After machining a CAM-Part, a burr can sometimes remain that has straight edges or non-tangent outer surface topologies. This occurs when the tool chips the metal off the edge, potentially comprimising the functionality of the part, endangering the user due to its razor-sharp nature. Removing it is the best option.

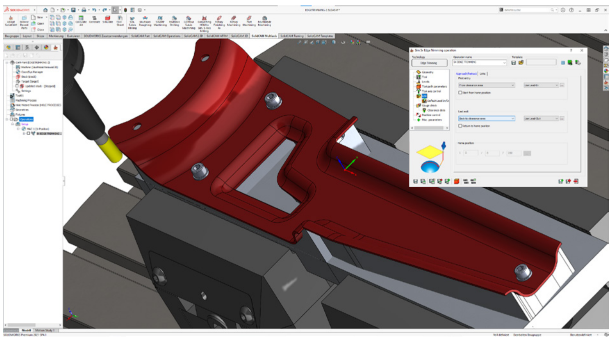


SolidCAM's Edge Breaking operation creates a deburring tool path on the outer edges of a part geometry. The position of the tool relative to the edge is always the bi-vector between the two surfaces of that edge.

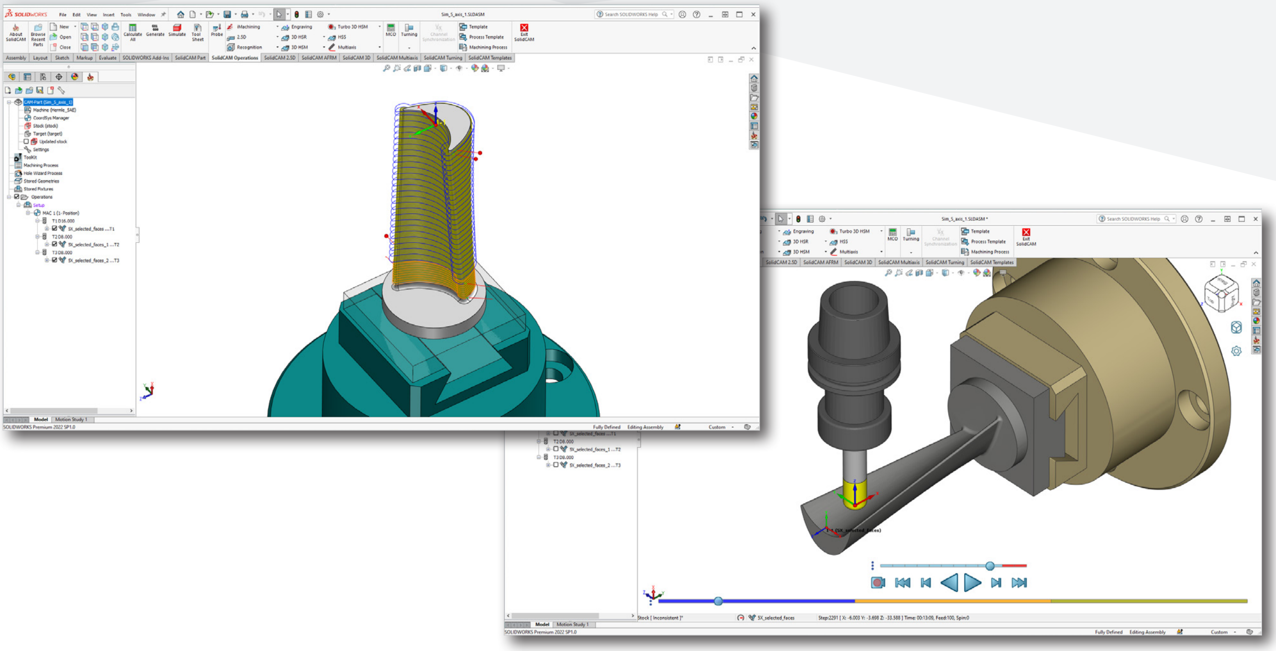
- Enables creation of a fully automatic tool path by just selecting the part geometry.
- Additional features include automatic detection, linking, lead-in and collision avoidance.
- Ball mill cutters and quality geometry input (mesh) are required for the automatic detection feature to work properly.

Simultaneous 5X Edge Trimming

SolidCAM's Edge Trimming operation efficiently machines parts that require edge trimming to get their final shape. This operation uses a highly automated algorithm to create a tool path to trim the edges of thin materials.



- Designed for the edge trimming of thin materials.
- Position of the tool relative to the geometry can be defined by various options from only a 3-Axis output to a more complex 5-Axis output with different tool axis orientation options.
- Axial shift enables tool engagement into the material with a certain value.
- Edge trimming can be automated or user defined, and it offers a variety of corner handling functions to create a seamless tool path.



SolidCAM Part: Sim_5_axis_1.prz

The operations in this example illustrate the use of SolidCAM'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.

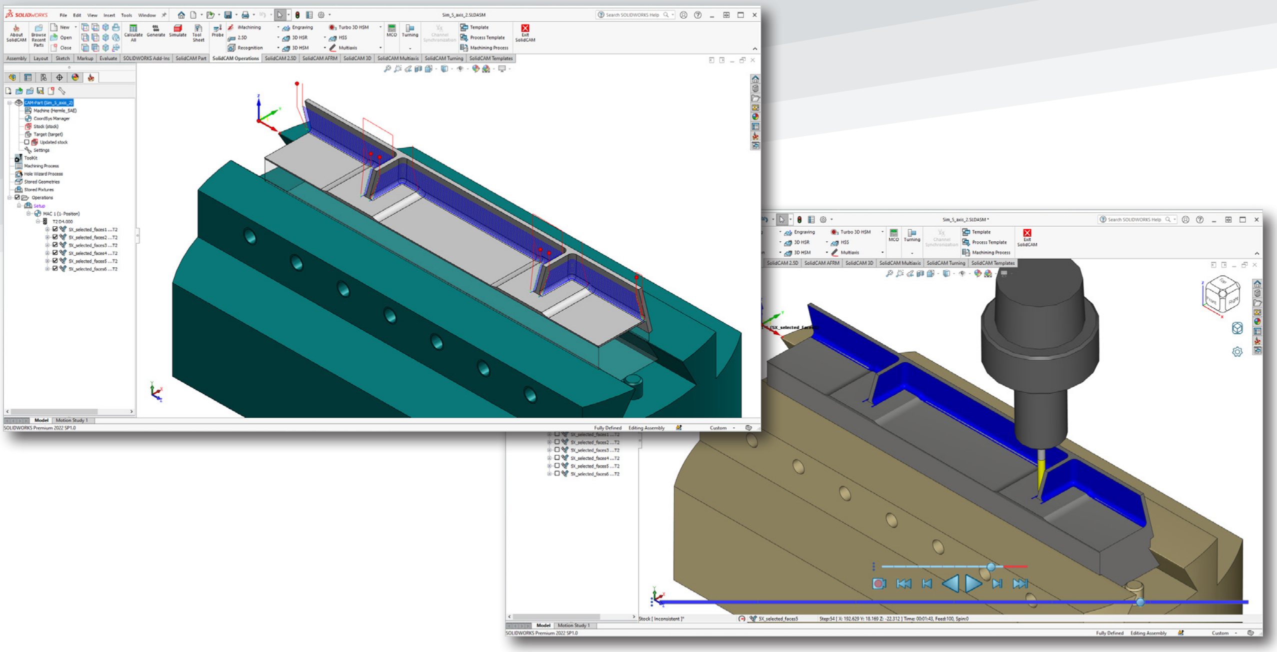
  The first operation provides the semi-finish for the turbine blade surfaces near the end using a Ø16 mm (0.625 in) bull nose mill with a corner radius of 4 mm (0.16 in). Tilted relative to cutting direction defines the tool tilting with a lag angle of 20° and the front face of the tool is specified for 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.



SolidCAM Part: Sim_5_axis_2.prz

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

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

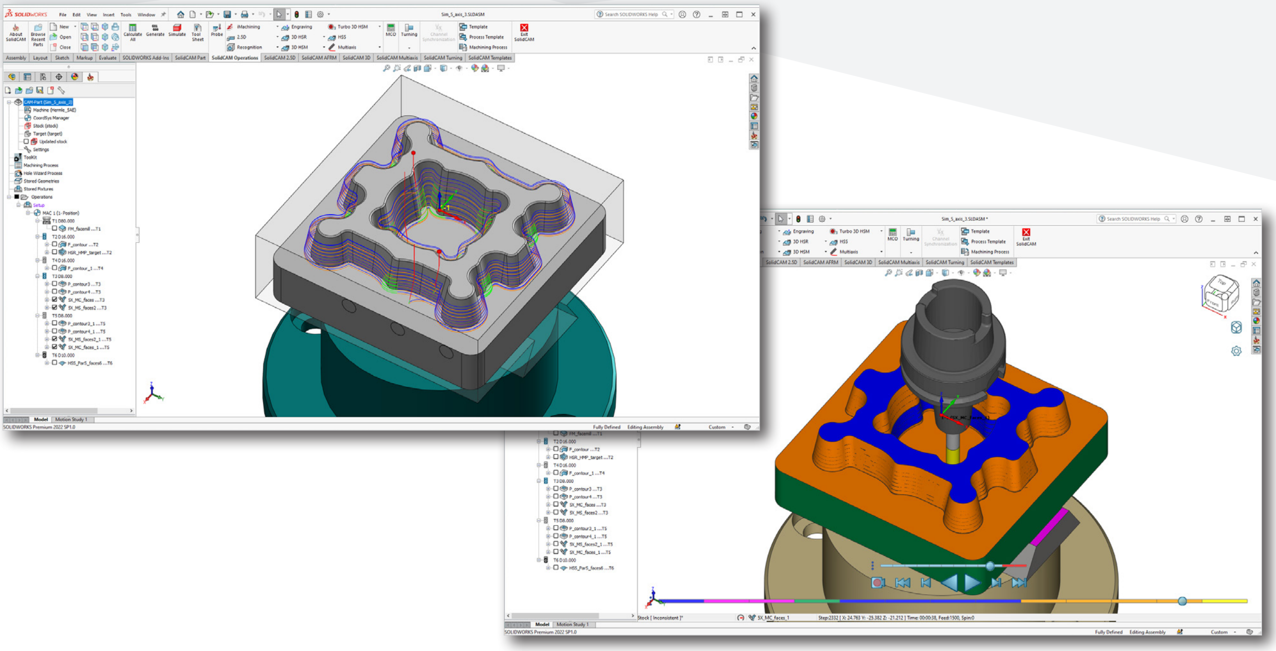
  These Sim. 5-Axis Milling operations use the Parallel cuts strategy to define the finishing of the inclined walls, which are forming undercut areas.

  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 radially via 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.



SolidCAM Part: Sim_5_axis_3.prz

The operations in this example illustrate the use of SolidCAM's Simultaneous 5-Axis Machining to semi-finish and finish both the internal and external inclined walls of the machine part shown above.

- **Rough machining (FM_facemill; F_contour; HSR_HMP_target; P_contour3; P_contour4)**

  These operations define the rough machining of the entire part (i.e., roughing of the upper face and outside shape, internal and external inclined walls as well as the open and closed pockets).

- **Internal inclined walls semi-finishing (5X_MC_faces)**

  This Sim. 5-Axis Milling operation uses the Morph between two boundary curves strategy to define the semi-finishing of the internal inclined walls. An Ø8 mm (0.3125 in) end mill is used and a Drive surface offset of 0.2 mm (0.008 in) is specified. 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 radially via the side 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 pocket floor. A morphed tool path is generated along the geometry drive surfaces and between the selected start edge and end edge curves.

- **External inclined walls semi-finishing (5X_MS_faces2)**

  This Sim. 5-Axis Milling operation uses the Morph between two adjacent surfaces strategy to define the semi-finishing of the external inclined walls. The Ø8 mm (0.3125 in) end mill is used and a Drive surface offset of 0.2 mm (0.008 in) is specified. 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 radially via the side surface of the tool. Gouge checking is enabled to detect and avoid possible collisions between the tool and the geometry drive surfaces. A morphed tool path is generated along the geometry drive surfaces and between the selected start edge and end edge surfaces.

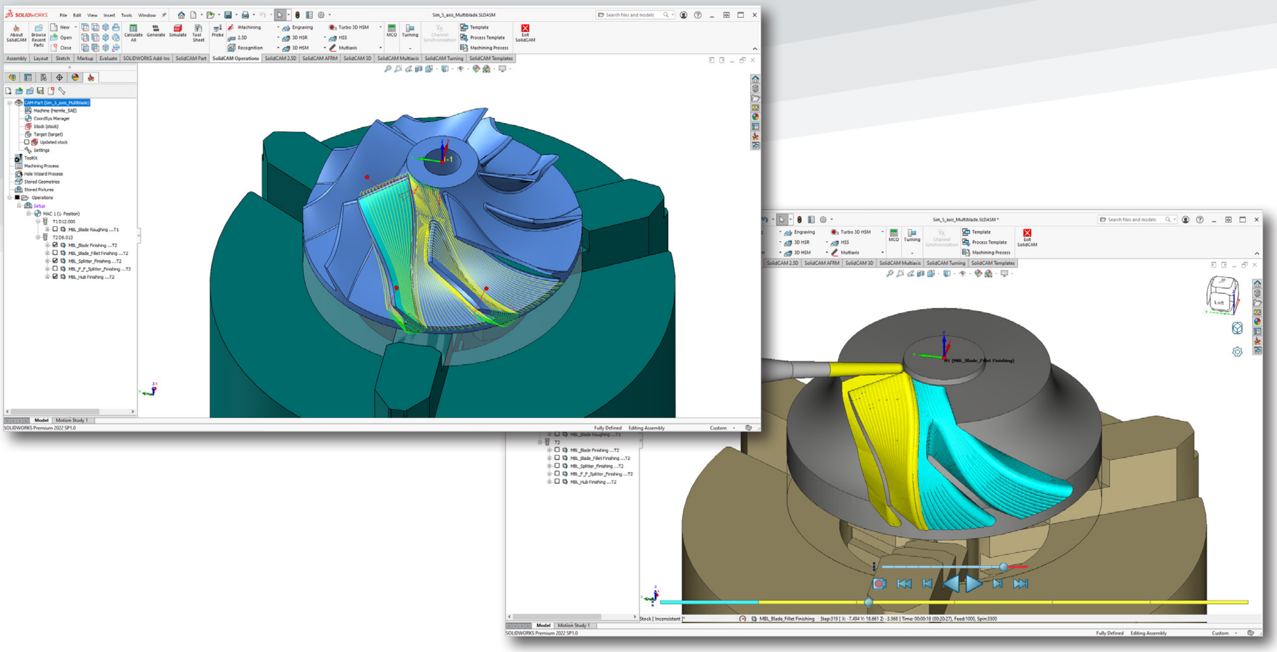
- **Finish machining (F_contour_1; P_contour3_1; P_contour4_1; HSS_ParS_faces6)**

  These operations define the finish machining of the part, except for the internal and external inclined walls (i.e., finishing of the outside shape, pocket floors and the fillets).

- **Internal and external inclined walls finishing (5X_MS_faces2_1; 5X_MC_faces_1)**

  These Sim. 5-Axis Milling operations are defined to finish the internal and external inclined walls. The two previous Sim. 5-Axis Milling operations were simply saved and copied to create these finishing operations with the same parameters. Another Ø8 mm (0.3125 in) end mill is used to perform the finish machining and a value of 0 is specified for the Drive surface offset. Again, the morphed tool path is generated along the geometry drive surfaces.

SIM. 5-AXIS MILLING CAM-PART EXAMPLES



SolidCAM Part: Sim_5_axis_Multiblade.prz

The operations in this example illustrate the use of SolidCAM's Multiblade Machining strategies to machine a section of the multiple bladed part shown above.

- **Roughing (MBL_Blade Roughing)**

  This Roughing operation defines the rough machining of an area between two main blades using a Ø12 mm (0.4844 in) taper mill. For both Layers and Slices, 10 is specified as the maximum number permitted in the tool path. Tool tilting is controlled according to a preferred lead angle of 5° and a maximum lead angle of 45°. A morphed tool path is generated between the hub and shroud, gradually removing the material by layers and slices.

- **Blade finishing (MBL_Blade Finishing; MBL_Splitter Finishing)**

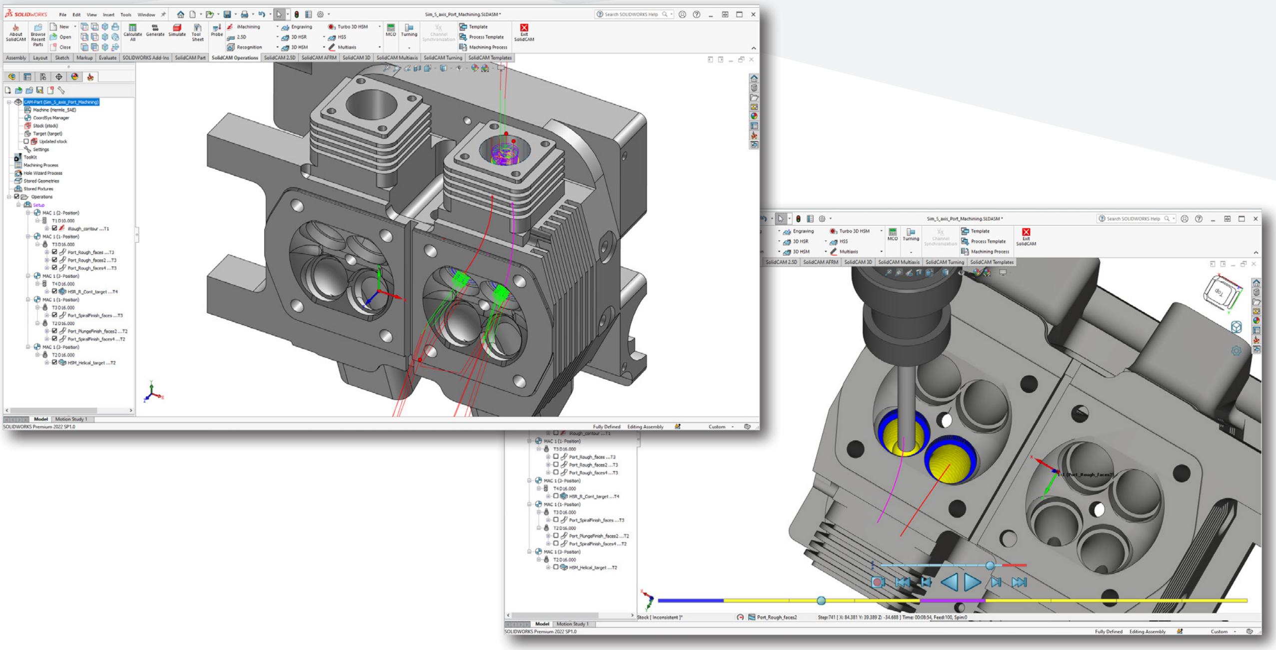
  These Blade finishing operations define the finish machining of two blades (one main and one splitter) using a Ø12 mm (0.4844 in) taper ball nose mill. Aside from the geometry selection differences, the Operation definitions are the same. Full (trim trailing edge) is chosen so the tool follows the contour of the blades but does not roll around the trailing edges. A maximum number of 15 layers is permitted in the tool path. Tool tilting is controlled according to a preferred lead angle of 5° and a maximum lead angle of 45°. A morphed tool path is generated between the hub and shroud, gradually removing the rest material by layers along the contours of each blade.

- **Fillet finishing (MBL_Blade_Fillet Finishing; MBL_F_F_Splitter_Finishing)**

  These Fillet finishing operations define the finish machining of the fillets adjacent to the hub for the previously finished main and splitter blades. The Ø12 mm (0.4844 in) taper ball nose mill is used. Aside from the geometry selection differences, the Operation definitions are the same. Full (trim trailing edge) is chosen so the tool follows the contour of the fillets but does not roll around the trailing edges. 8 cuts are specified along the blade side and 5 cuts are specified along the hub side with a 0.3 mm (0.012 in) side step on both sides. The same angles are defined for the tool tilting as in the previous operations. SolidCAM automatically detects the fillets and the finishing tool path is generated accordingly.

- **Hub finishing (MBL_Hub Finishing)**

  This Hub finishing operation defines the finish machining of the hub area surrounding the previously finished splitter blade. The Ø12 mm (0.4844 in) taper mill is used. A maximum number of 18 slices is permitted in the tool path and the same angles are defined for the tool tilting as in the previous operations. This combination of parameters enables the tool to achieve a smooth tool path around the splitter blade and along the hub surface.



SolidCAM Part: Sim_5_axis_Port_Machining.prz

The operations in this example illustrate the use of SolidCAM's Port Machining strategies to machine internal surfaces of intakes and curved tubes.

- **Pockets roughing (iRough_contour)**

  This iMachining operation defines the rough machining of the top part of two curved tubes using a Ø10 mm (0.4063 in) end mill. The tool performs a helical entry into each of the pockets and then, in 1 Step down, spirals to the outer walls. A machining allowance of 0.05 mm (0.002 in) remains for further finishing operations.

- **Curved tubes roughing (Port_Rough_faces; Port_Rough_faces2)**

  These Roughing operations define the rough machining of the internal surfaces of two curved tubes using a Ø16 mm (0.625 in) lollipop mill. Aside from the geometry selection differences, the Operation definitions are the same. Tool path parameters are specified with a 5 mm (0.2 in) Maximum step over and a 2 mm (0.08 in) Step down. Degouging is enabled to detect and avoid possible collisions between the tool, the adjacent outer surfaces and against any machined surfaces. The tool path is output in 5-Axis format, gradually removing the material within the tubes by layers and slices until the Maximum from top distance is reached.

- **Port roughing (Port_Rough_faces4)**

  This Roughing operation defines the rough machining of the port connecting the two curved tubes using the Ø16 mm (0.625 in) lollipop mill. The same Maximum step over and Step down parameters are specified as in the previous operations. Degouging is enabled to detect and avoid possible collisions between the tool, the adjacent outer surfaces and against any machined surfaces. The tool path is output in 5-Axis format, gradually removing the material within the port by layers and slices until the Maximum from bottom distance is reached.

- **Curved tubes finishing (Port_SpiralFinish_faces; Port_PlungeFinish_faces2)**

  These operations use the Spiral finishing and Plunge finishing strategies to define the finish machining of similar geometries. Both operations use the Ø16 mm (0.625 in) lollipop mill and aside from the geometry selection differences, the Operation definitions are the same. A Maximum step over of 2 mm (0.08 in) is specified and the same Degouging options are used as in the previous roughing operations. The tool path is generated using spiral cuts in one curved tube and plunge cuts in the other to finish the internal surfaces until the Maximum from top distance is reached.

- **Port finishing (Port_SpiralFinish_faces4)**

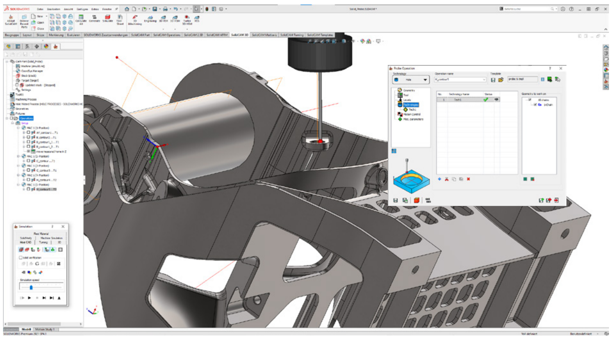
  This Spiral finishing operation defines the finish machining of the port connecting the two curved tubes using the Ø16 mm (0.625 in) lollipop mill. A 1 mm (0.04 in) Maximum step over is specified and the same Degouging options are used as in the previous roughing operation. The tool path is generated using spiral cuts in the port to finish the internal surfaces until the Maximum from bottom distance is reached.

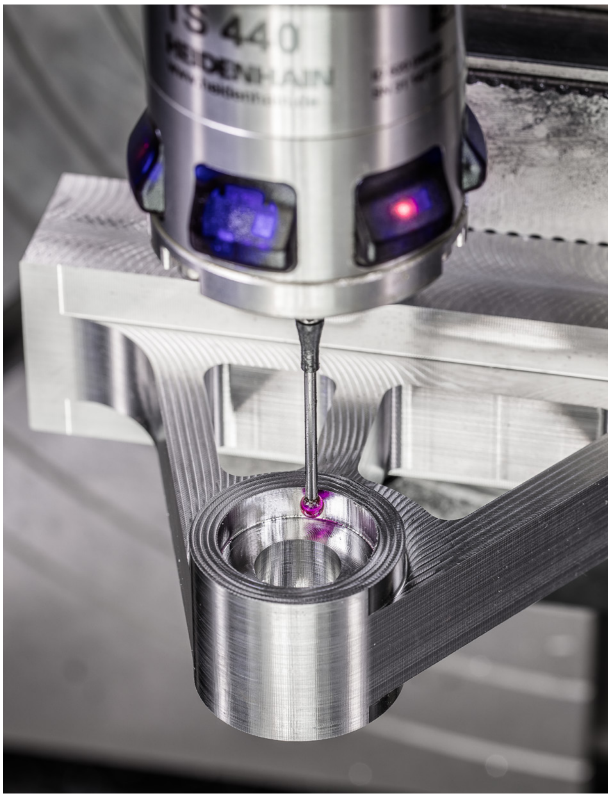
SOLID PROBE

Probing & Measuring Made Easy

SolidCAM's Solid Probe Module provides capabilities for Home Definition and On-Machine Verification, using probes on the CNC-Machine, to perform setup and control the quality of machined parts.

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



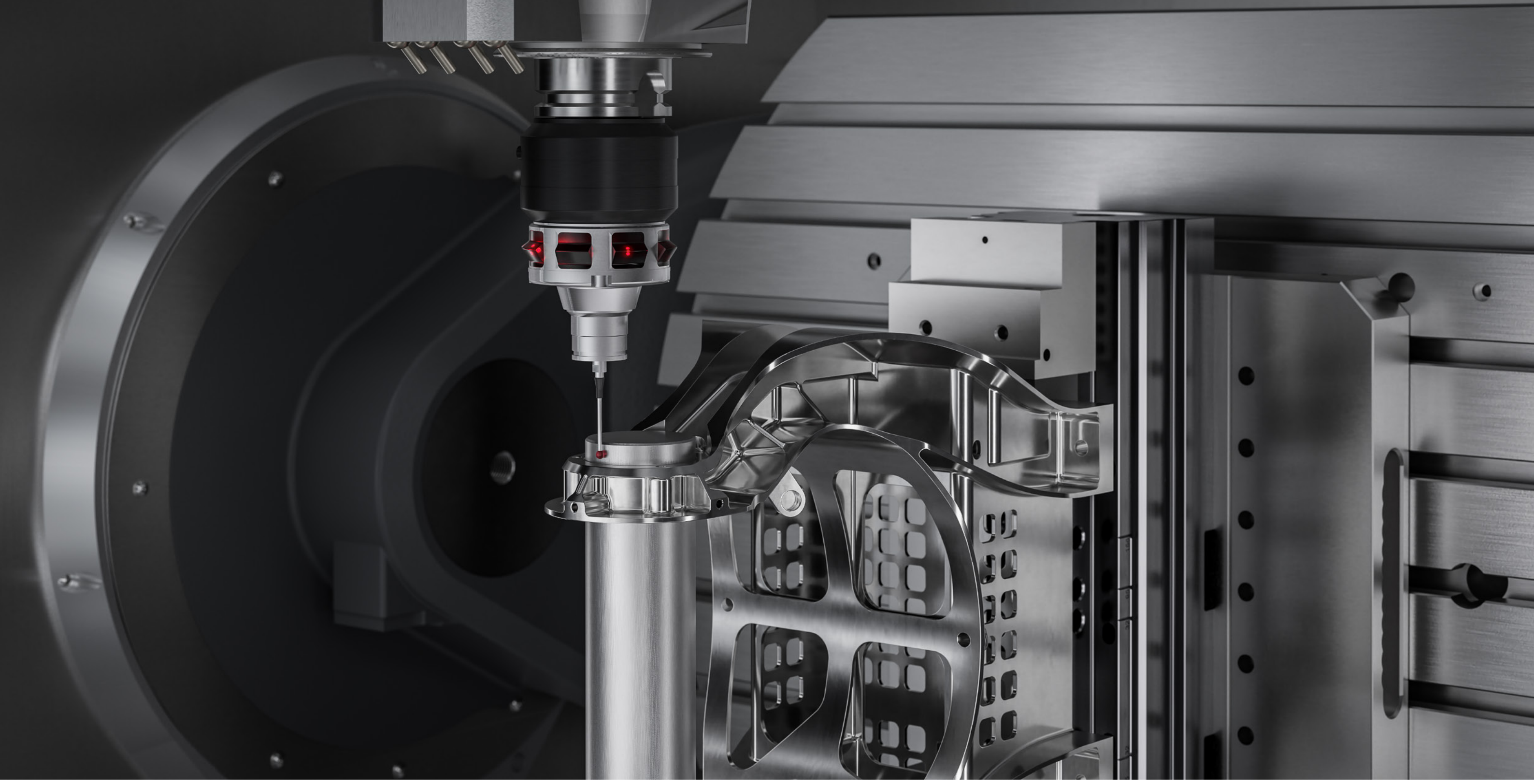


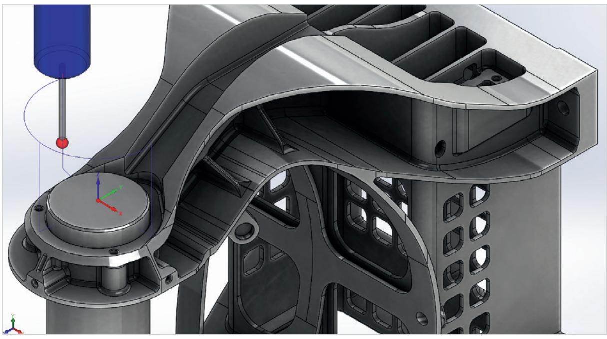
Solid Probe is a Must for Every Machinist using Probes:

- Easy Home Definition
- On-Machine Verification
- Tool Presetter support
- Easy geometry selection on solid model
- Supports a wide range of Probe Cycles
- Visualization of all the Probe tool movements
- Support of different Probe controllers

Combined Probe and Machining Operations

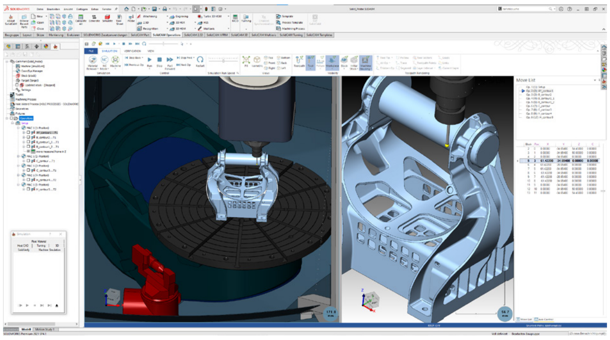
Machining operations and probe operations are intermixed in the CAM Manager and can use the same geometries on the CAD model. When the solid model is changed, both the machining and probe operations can be automatically synchronized to the change.





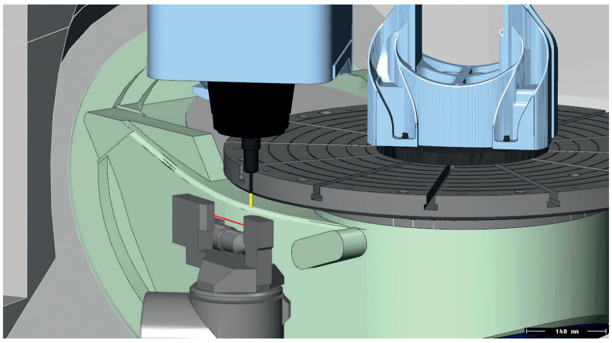
Home Definition

Solid Probe provides an easy solution for home setting, offering 16 different cycles, to define home positions, replacing manual setup procedures.



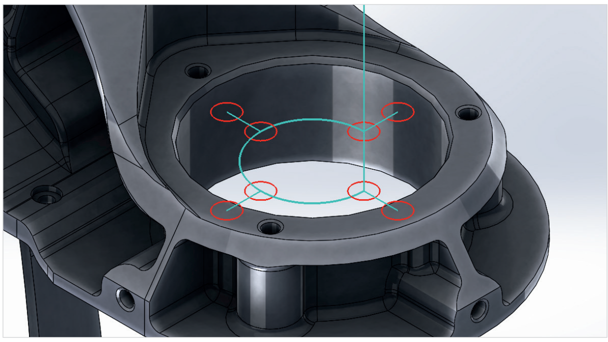
On-Machine Verification

Solid Probe Cycles are used for measuring machined surfaces, without transferring the part to a Coordinate Measurement Machine (CMM). The part can be inspected directly on the CNC-Machine itself.



Tool Presetter Support

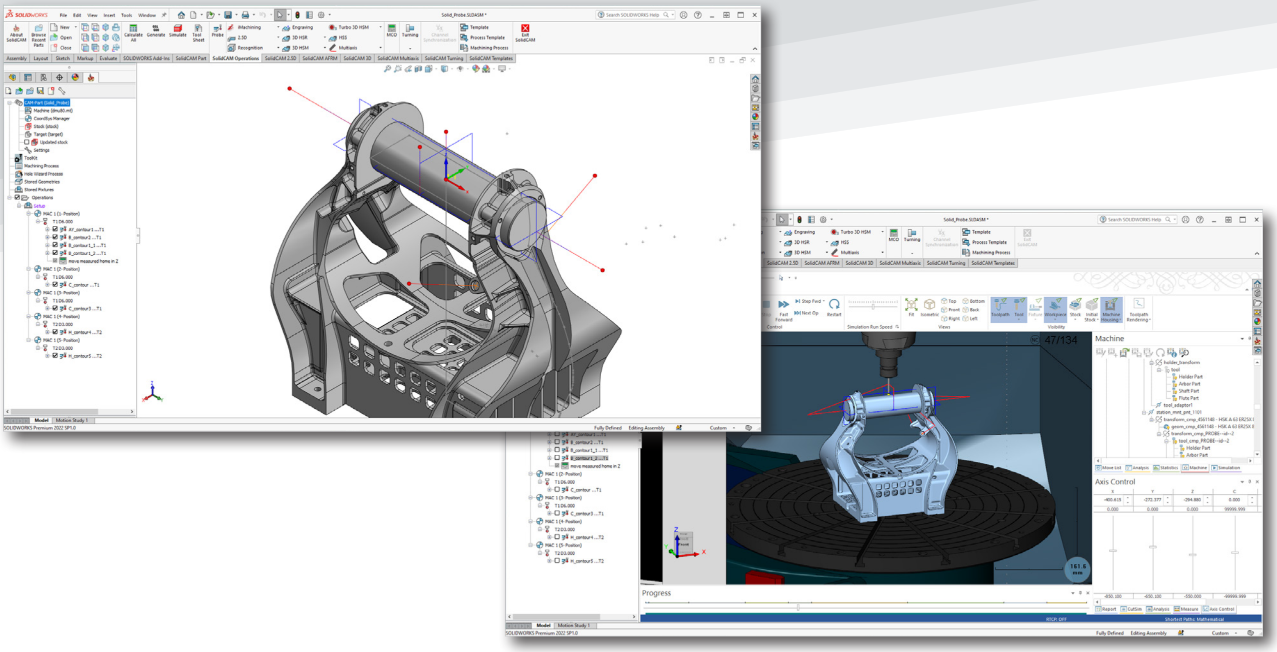
Solid Probe includes Tool Presetter support to check your milling and turning tools, between machining operations and tool change events. It also provides tool breakage detection for continuous and safe machining.



Preview of Cycle Movements

Solid Probe uses the same geometry as the 2.5D Milling operations. Full control over tolerances, various sorting options and preview of cycle movements are provided.

SOLID PROBE CAM-PART EXAMPLE



SolidCAM Part: Solid_Probe.prz

The operations in this example illustrate the use of SolidCAM's Solid Probe Cycles to check the machining precision for different areas of the part shown above. One SolidCAM Coordinate System is defined with multiple positions.

- **Cylinder surface straightness checking (AY_contour1)**

  This Angle Y Cycle is defined to check if the cylinder surface is straight. Machine Coordinate System #1, Position #1 (MAC 1 (1- Position)) is used for the cycle.

- **Cylinder width checking (B_contour2)**

  This Boss Cycle is defined to check the width of the cylinder. MAC 1 (1- Position) is used for the cycle.

- **Cylinder length checking (B_contour1_1)**

  This Pocket Cycle is defined to check the length of the cylinder, measuring the distance between two boundary surfaces. MAC 1 (1- Position) is used for the cycle.

- **Cylinder height checking (B_contour1_2)**

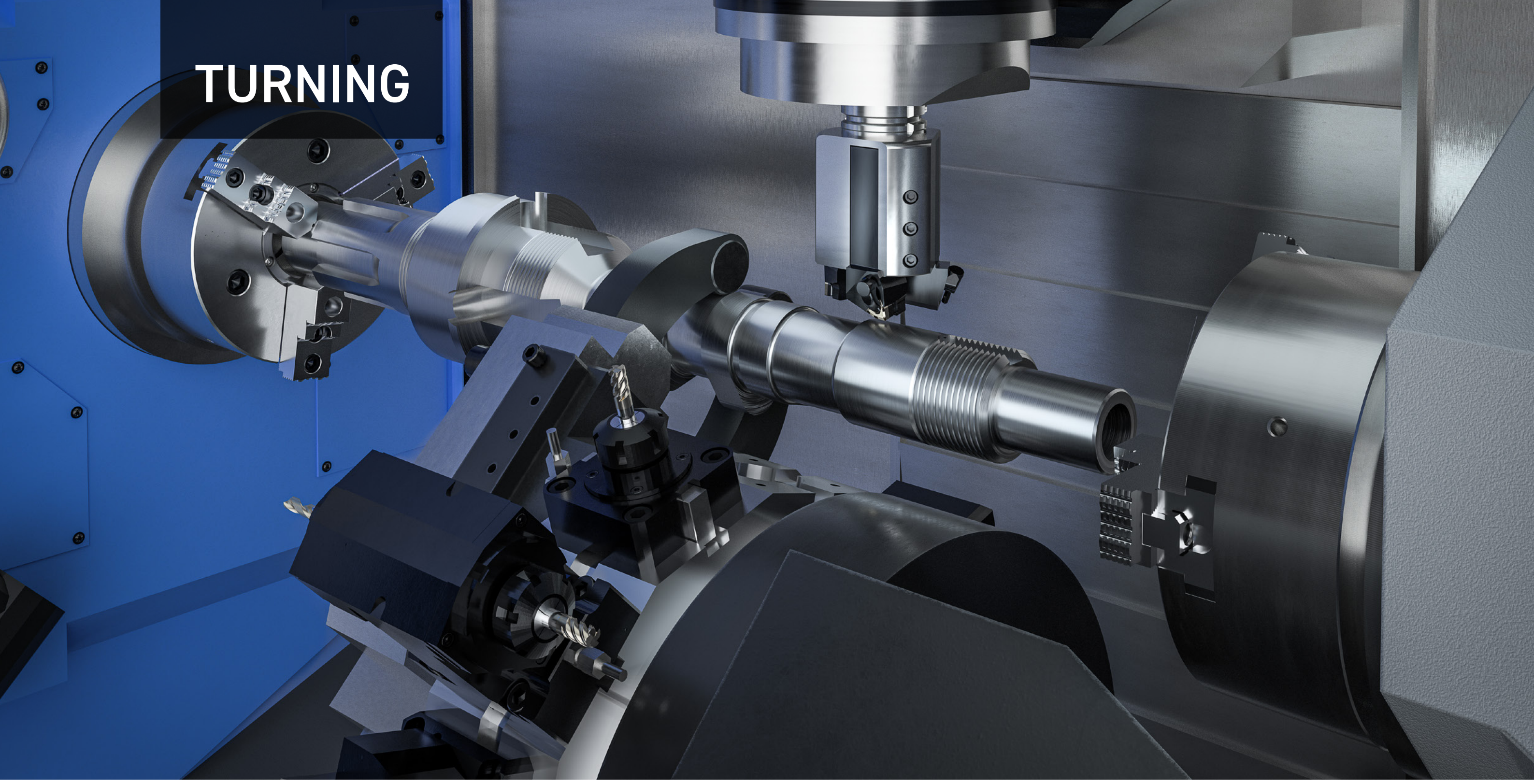
  This Single point Z Cycle is used to check the height of the cylinder, measuring the position of the highest point. MAC 1 (1- Position) is used for the cycle.

- **Cylinder diameter checking (C_contour; C_contour3)**

  These Cylinder Cycles are used to check the diameters on both ends of the cylinder. MAC 1 (2 and 3- Positions) are used for the cycles.

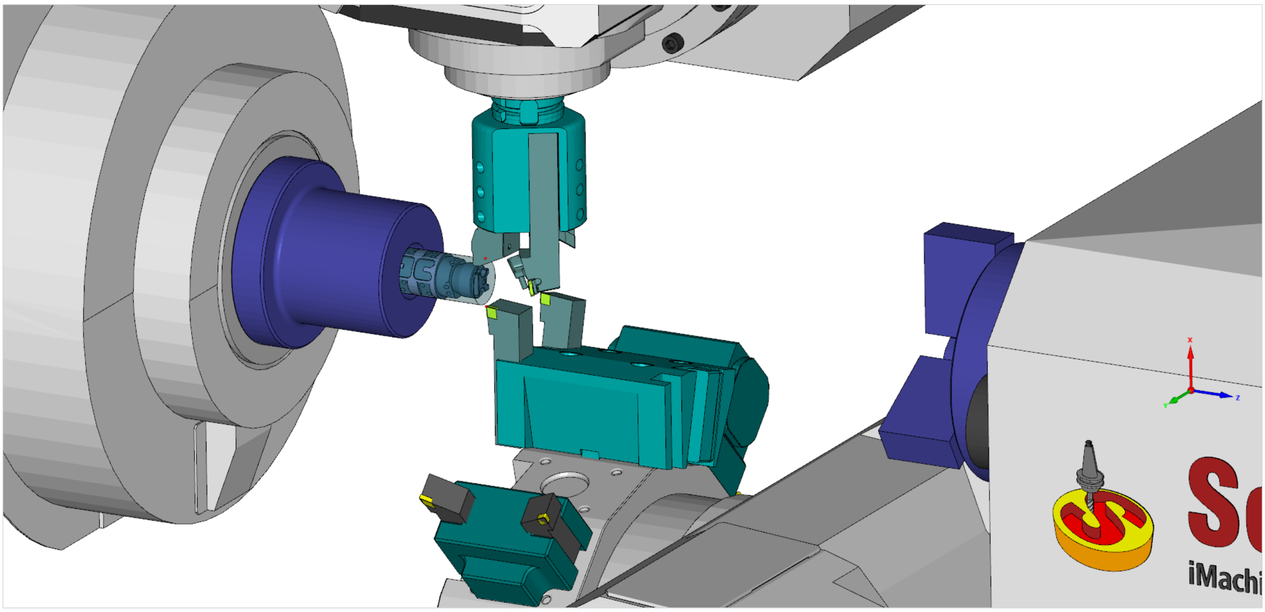
- **Holes diameter checking (H_contour4; H_contour5)**

  These Hole Cycles are used to check the diameters of the holes on the tabs below the cylinder. MAC 1 (4 and 5- Positions) are used for the cycles.

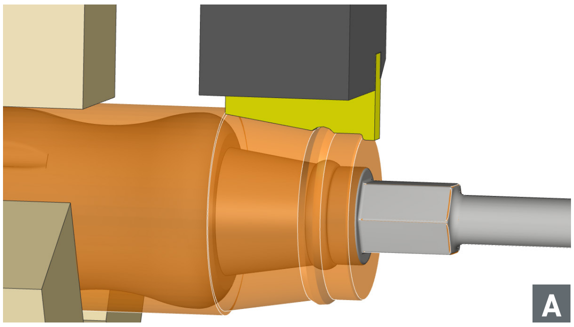


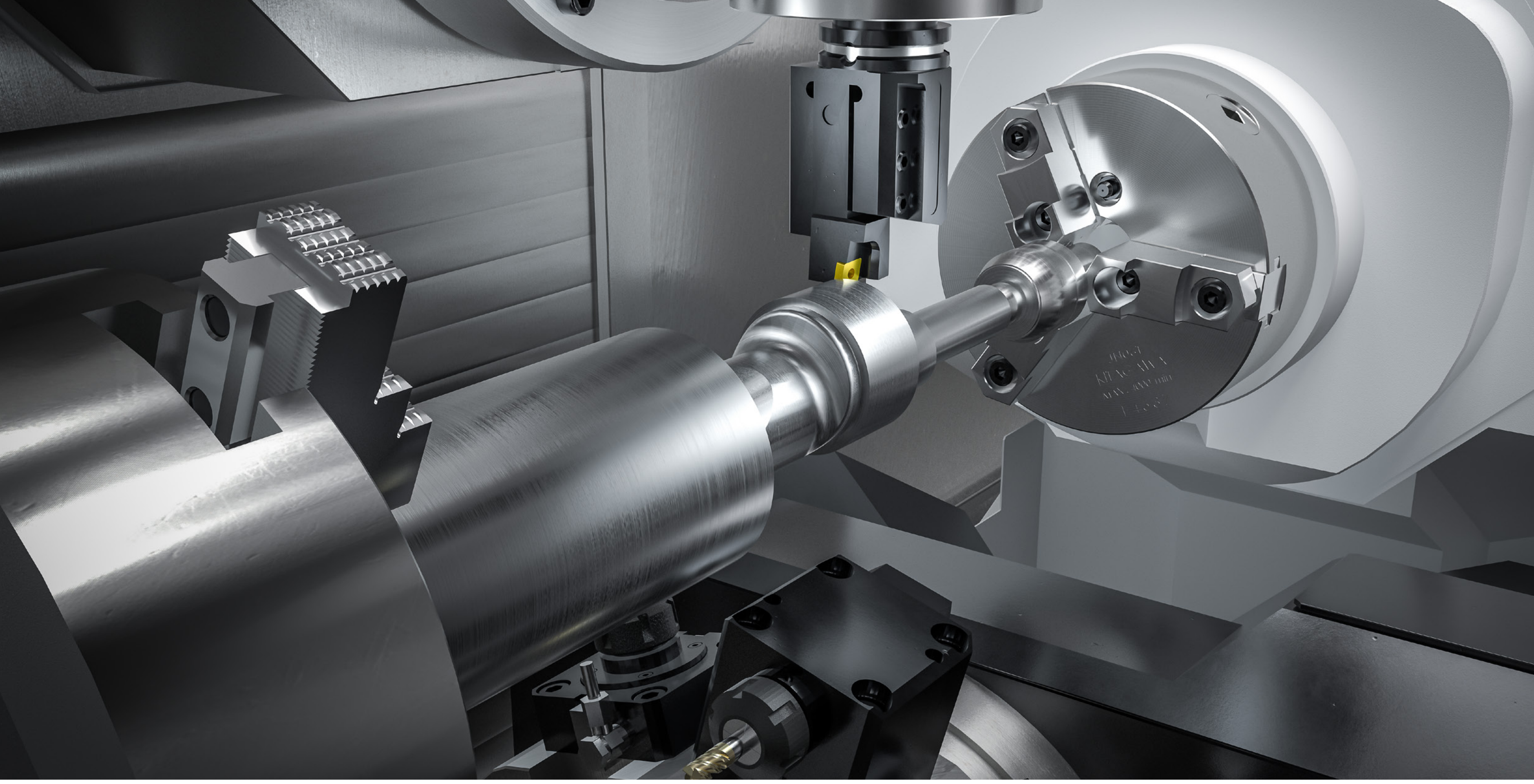
TURNING

SolidCAM Module for Fast and Efficient Turning



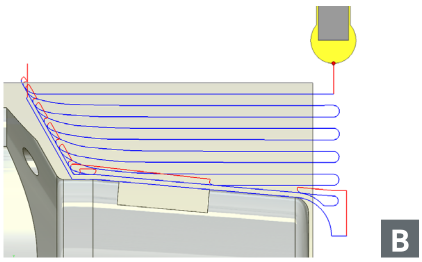
- A comprehensive turning package is provided with powerful tool paths and techniques for fast and efficient turning with fixture and holder protection.
- Advanced rough and finish profile turning is produced, together with support for facing, grooving, threading and drilling cycles.
- Turning geometries and profiles can be generated very quickly and easily modified for production.
- Machine Preview allows you to interactively, in the machine environment, check and verify your setup and machine position, at every stage of the tool path, minimizing programming and setup errors.
- The widest range of machine tools are supported, including 2-Axis lathes and multi-turret configurations with or without sub-spindles.
- Custom turning inserts and shape inserts with multiple cutting edges can be used. [A]

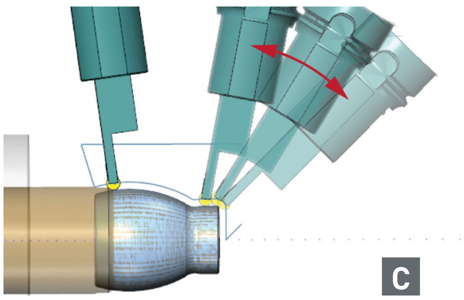


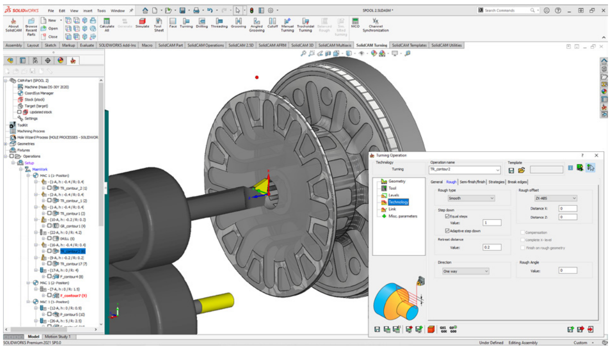


Advanced Turning Capabilities

- Balanced Roughing allows two turning tools working simultaneously, or in trailing mode, to perform the rough turning 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.
- New Trochoidal tool path of round grooving tools for increased efficiency. [B]
- 4th Axis Simultaneous Turning performs machining of curved profile using the B-Axis tilting capabilities of the tool, in order to machine undercut areas in a single machining step. [C]
- Drive Unit Sharing allows two turning tools working simultaneously while a single Drive Unit (spindle) rotates with the same RPM's and direction. [D]



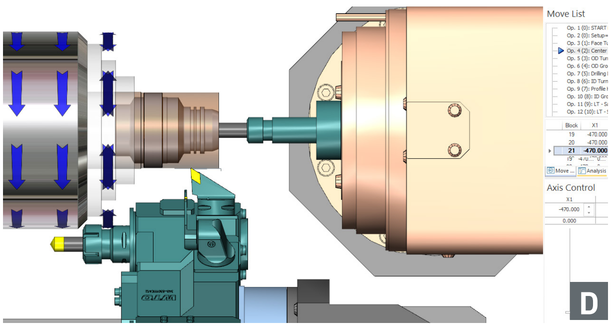


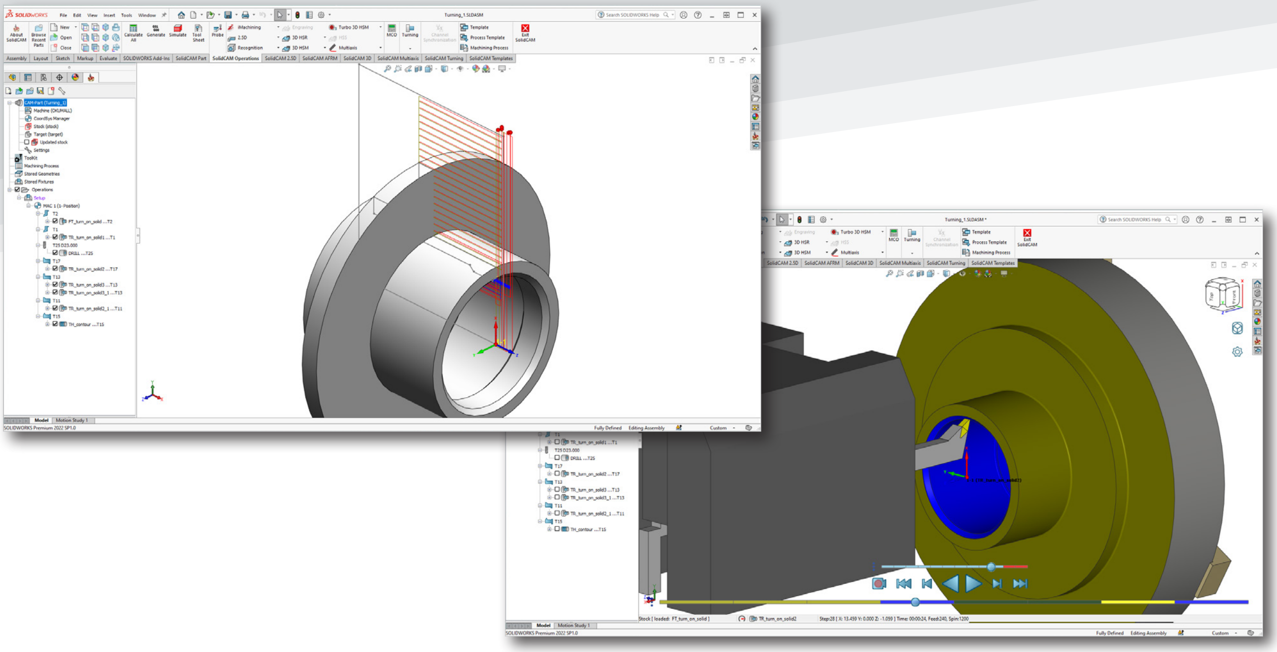


Updated Stock

SolidCAM 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 spindle 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.

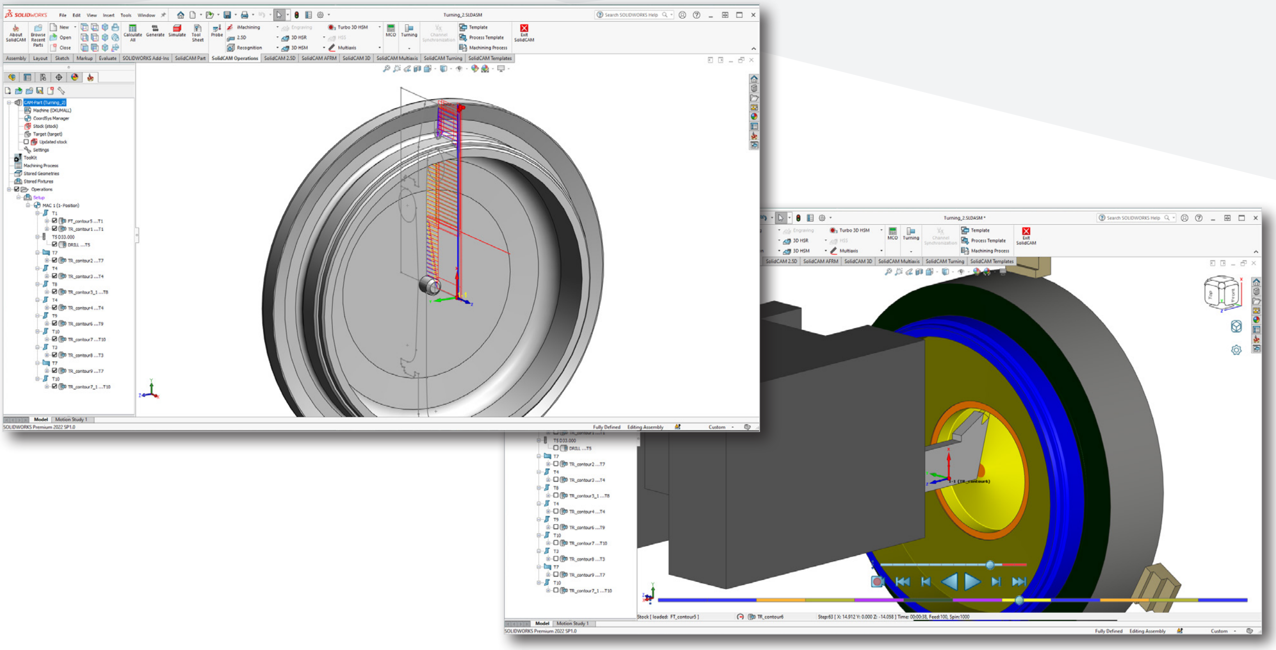




SolidCAM Part: Turning_1.prz

The operations in this example illustrate the use of SolidCAM's Turning technologies to machine the part shown above.

- **Face turning (FT_turn_on_solid)**
  This Face Turning operation defines the rough machining of the front face using an external roughing tool. A 0.1 mm (0.004 in) Z-Offset remains on the surface for further finishing operations.
- **Contour turning (TR_turn_on_solid1)**
  This Turning operation defines the machining of the external surfaces on the front side 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. The specified roughing offsets are removed in the same operation with a finishing cut.
- **Drilling (DRILL)**
  This Drilling operation defines the drilling of the hole through the center of the part using a Ø23 mm (0.9063 in) flat drill.
- **Internal roughing (TR_turn_on_solid2)**
  This Turning operation defines the rough machining of the internal surfaces including the chamfer using an internal roughing tool. The Rough option is chosen for the work type and Long internal mode is used to create passes under the geometry and along the rotational axis of the part. Non-descending motions strategy is used to avoid collisions.
- **Internal grooving (TR_turn_on_solid3)**
  This Turning operation defines the rough machining of the internal groove using an internal grooving tool. The Smooth option is chosen for the roughing type to provide a smoother finish.
- **Internal finishing (TR_turn_on_solid3_1; TR_turn_on_solid2_1)**
  These Turning operations define the finish machining of the internal groove and the internal surfaces using the appropriate tools. The Finish only option is chosen for the work type to perform only finishing cuts.
- **Internal threading (TH_contour)**
  This Threading operation defines the machining of an internal thread using an internal threading tool. A user-defined Thread size and Pitch unit are specified with no thread finishing.



SolidCAM Part: Turning_2.prz

The operations in this example illustrate the use of SolidCAM's Turning technologies to machine the part shown above.

- **Face turning (FT_contour5)**

  This Face Turning operation defines the rough machining of the front face using an external roughing tool. A 0.2 mm (0.008 in) Z-Offset remains on the surface for further finishing operations.

- **Contour roughing (TR_contour1)**

  This Turning operation defines the rough machining of the external surfaces using the 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. Non-descending motions strategy is used to avoid collisions.

- **Drilling (DRILL)**

  This Drilling operation defines the drilling of a hole in the center of the part to make machining the internal contour easier and more efficient. A Ø33 mm (1.375 in) flat drill is used.

- **Internal roughing (TR_contour2)**

  This Turning operation defines the rough machining of the internal surfaces using an internal roughing tool. The Rough option is chosen for the work type and Long internal mode is used to create passes under the geometry and along the rotational axis of the part.

- **Contour rest machining and finishing (TR_contour3; TR_contour3_1; TR_contour4; TR_contour8)**

  These Turning operations define the rest and finish machining of all 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 remaining operations, the Finish only option is chosen for the work type to perform only finishing cuts.

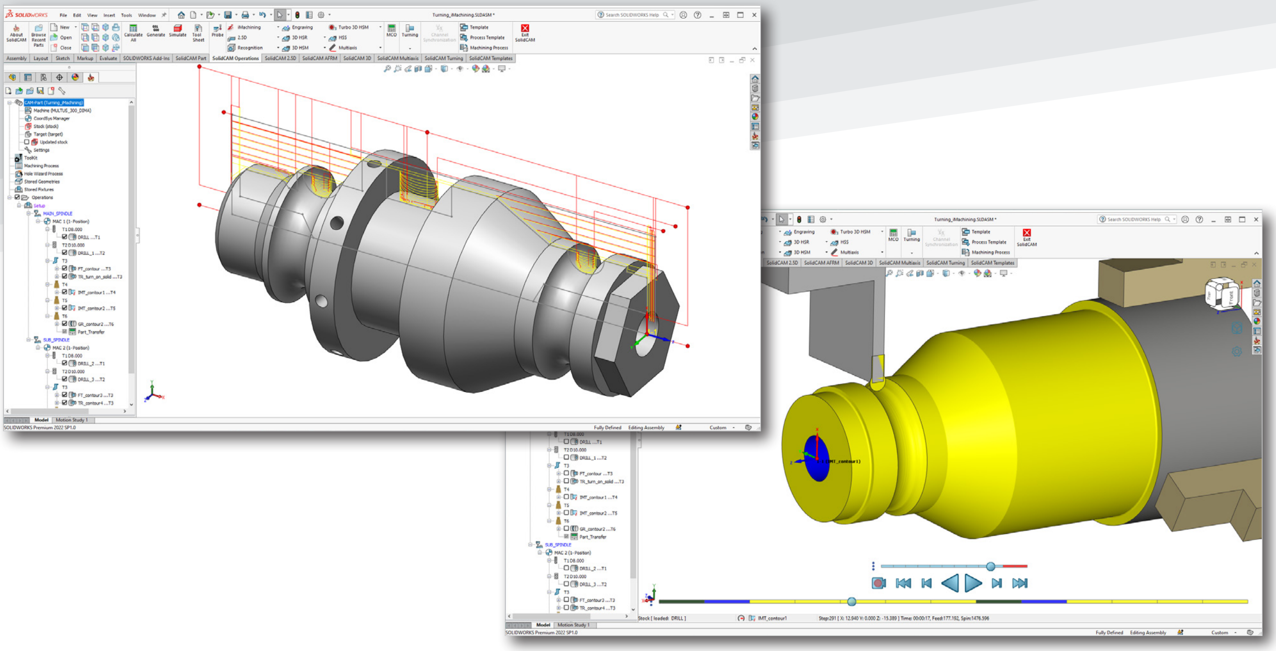
- **Internal rest machining (TR_contour6; TR_contour7)**

  These Turning operations define the rest machining of the internal surfaces.

  The first operation removes unmachined material from the internal contour. The second operation uses a different tool mounting orientation to perform roughing cuts towards the center of the part.

- **Internal finishing (TR_contour9; TR_contour7_1)**

  These Turning operations define the finish machining of the internal surfaces using the Finish only option. As in the previous operations, different tool mounting orientations are used to perform finishing cuts both away from and towards the center of the part.



SolidCAM Part: Turning_iMachining.prz

The operations in this example illustrate the use of SolidCAM's Trochoidal Turning and several other Turning technologies with front and back spindles functionality to machine the part shown above.

- **Main spindle face drilling (DRILL; DRILL_1)**

  These Drilling operations define the drilling of the hole in the center of the front face.

- **Main spindle face turning (FT_contour)**

  This Face Turning operation defines the machining of the front face using an external roughing tool. The specified 0.2 mm (0.008 in) Z-Offset is removed in the same operation with a finishing cut.

- **Main spindle contour turning (TR_turn_on_solid)**

  This Turning operation defines the machining of the external surfaces on the front side using the external roughing tool. Long external mode is used and the Rough option is chosen for the work type. The Smooth option provides a smoother finish during the roughing, which is then followed by a finishing cut. Non-descending motions strategy is used to avoid collisions.

- **Main spindle round grooving (IMT_contour1)**

  This Trochoidal Turning operation defines the machining of the round groove using a grooving tool with round insert. Long external mode is used and a Cutting angles range of 30° to 45° is specified. Zigzag is chosen for the direction to keep the tool in contact with the material and to make machining the groove more efficient.

- **Main spindle flat grooving (IMT_contour2; GR_contour2)**

  These operations define the machining of the flat groove.

  The Trochoidal Turning operation provides roughing of the groove using a grooving tool with round insert. The same technological parameters are used as in the previous operation minus a finishing cut. When the tool reaches the bottom of the groove, iMachining movements are performed in the corners.

  The Grooving operation provides finishing of the groove using a typical grooving tool with square insert. The same geometry is used to remove the material remaining in the corners. The Finish only option is chosen for the work type to perform only finishing cuts on the rest material.

- **Sub-spindle face drilling (DRILL_2; DRILL_3)**

  These Drilling operations define the drilling of the hole in the center of the back face.

- **Sub-spindle face turning (FT_contour3)**

  This Face Turning operation defines the machining of the back face using an external roughing tool. The specified 0.2 mm (0.008 in) Z-Offset is removed in the same operation with a finishing cut.

- **Sub-spindle contour turning (TR_contour4)**

  This Turning operation defines the machining of the external surfaces on the back side using the external roughing tool. Long external mode is used and the Rough option is chosen for the work type. The Smooth option provides a smoother finish during the roughing, which is then followed by a finishing cut. Non-descending motions strategy is used to avoid collisions.

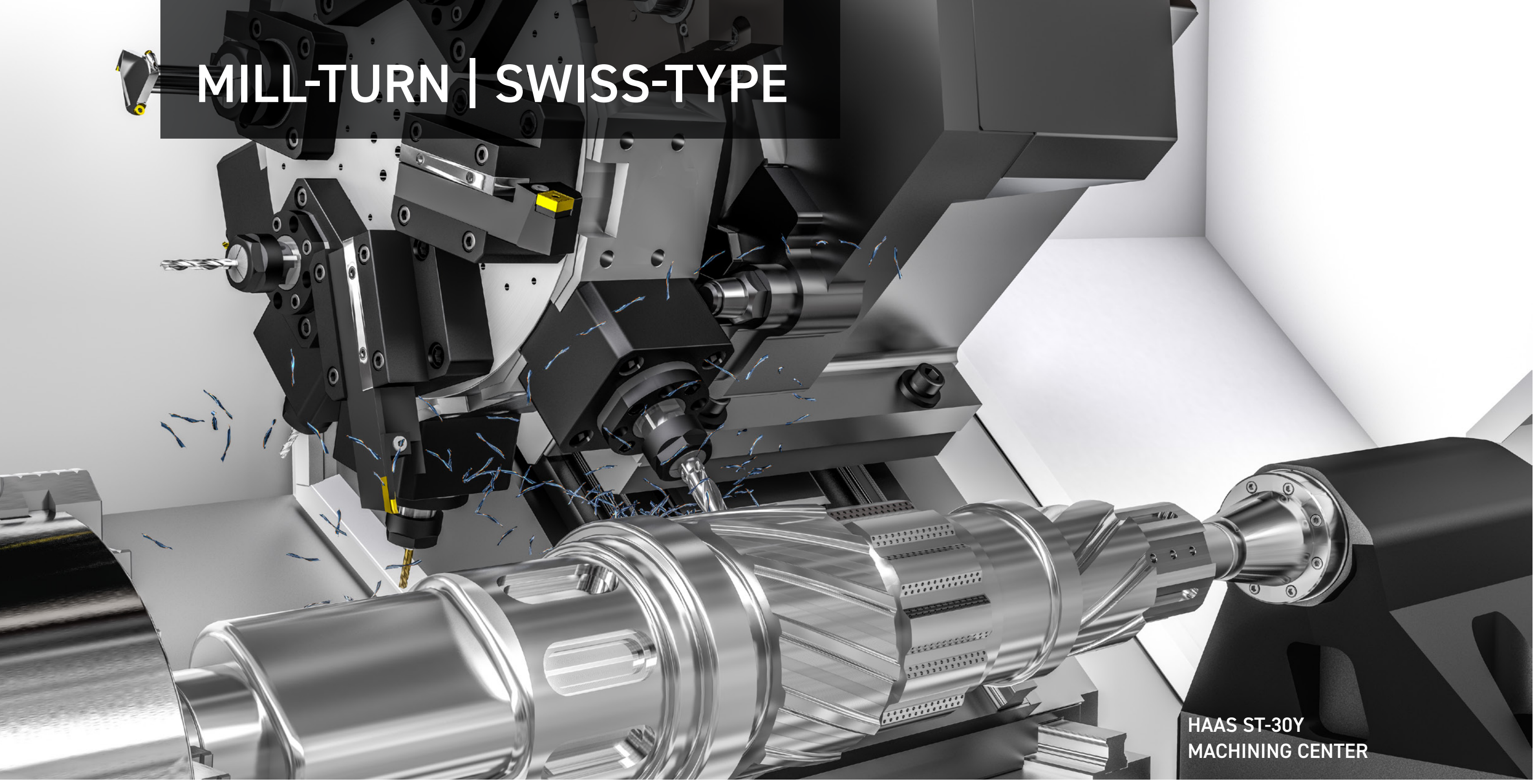
- **Sub-spindle grooving (IMT_contour5; TR_contour6)**

  These operations define the machining of the round and inclined grooves.

  The Trochoidal Turning operation provides the rough and finish machining of the round groove using a grooving tool with round insert. Long external mode is used and a Cutting angles range of 30° to 45° is specified. Zigzag is chosen for the direction to keep the tool in contact with the material and to make machining the groove more efficient.

  The Turning operation provides the rough and finish machining of the inclined groove using an external roughing tool. Long external mode is used and the Rough option is chosen for the work type. The Smooth option provides a smoother finish during the roughing, which is then followed by a finishing cut.

MILL-TURN | SWISS-TYPE

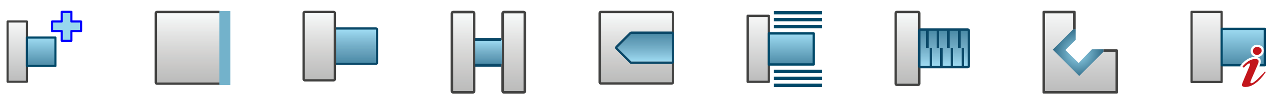


Complete Solution for Advanced Multi-Turret/Spindle Mill-Turn and Swiss-Type Machines

Modern Multiaxis Machining Centers are designed to combine as many Milling and Turning operations as possible to manufacture workpieces at maximum productivity.

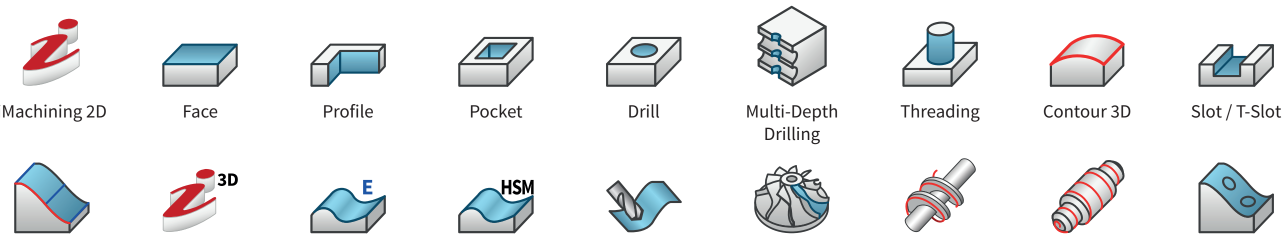
Manual CNC programming of sophisticated parts on complex machines, directly at the machine controller is – if at all humanly possible – unproductive, error-prone and expensive.

TURNING OPERATIONS



Manual | Face | Turning | Grooving | Drilling | Balanced Roughing | Threading | Angled Grooving | Trochoidal Turning

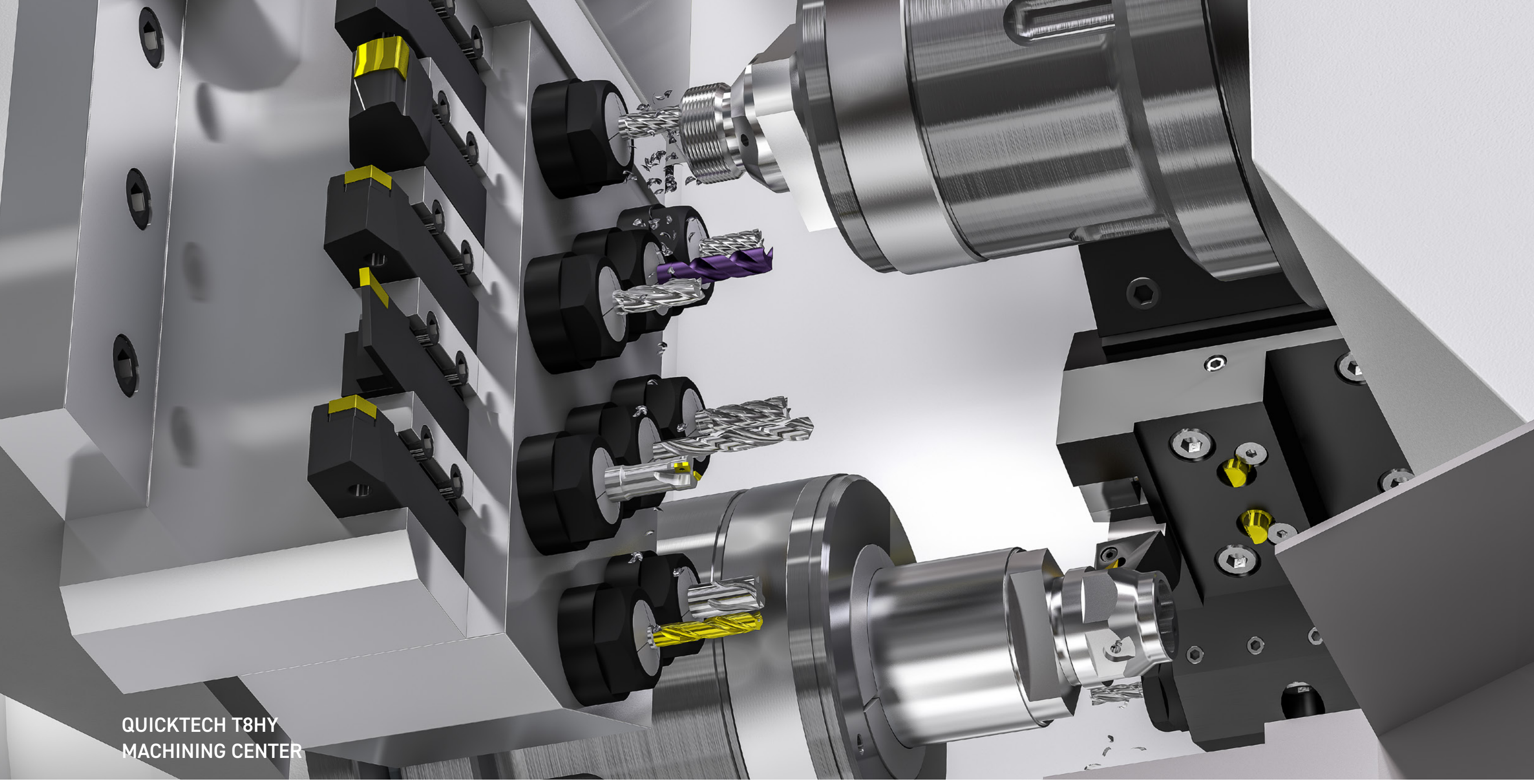
MILLING OPERATIONS



iMachining 2D | Face | Profile | Pocket | Drill | Multi-Depth Drilling | Threading | Contour 3D | Slot / T-Slot

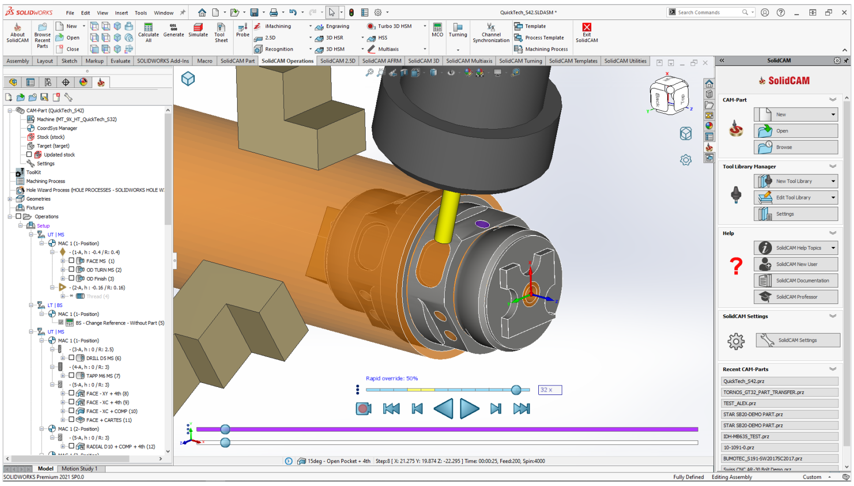
Translated Surface | iMachining 3D | Engraving | HSR / HSM / HSS | Sim. 5X | Blade Machining | Screw Machining | Rotary Machining | Multiaxis Drilling

... and many more!



QUICKTECH T8HY
MACHINING CENTER

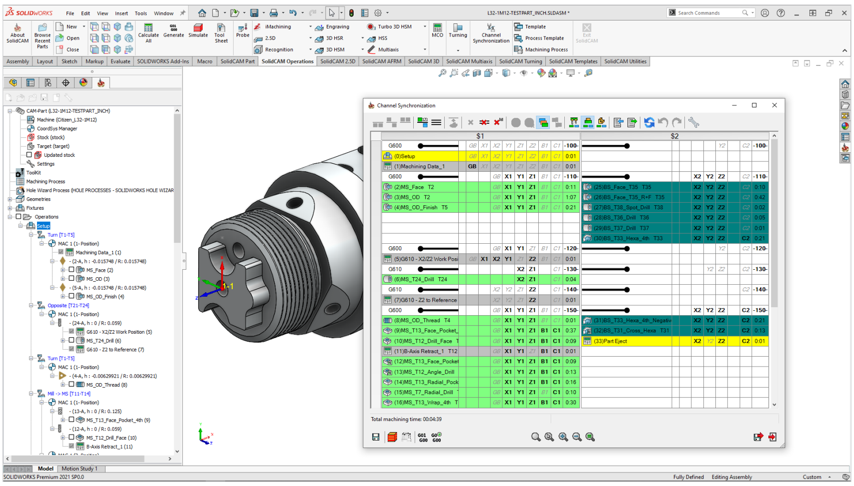
Integrated. Easy-to-Use. Complete.



Within the SolidCAM user interface, seamlessly integrated into your CAD software, you program Milling and Turning operations on the main and back spindles, control turrets, tailstocks, steady rests and linear tool carriers.

Milling operations include the unique and patented iMachining technology, which is available only from SolidCAM.

Short Cycle Times. Maximum Productivity.

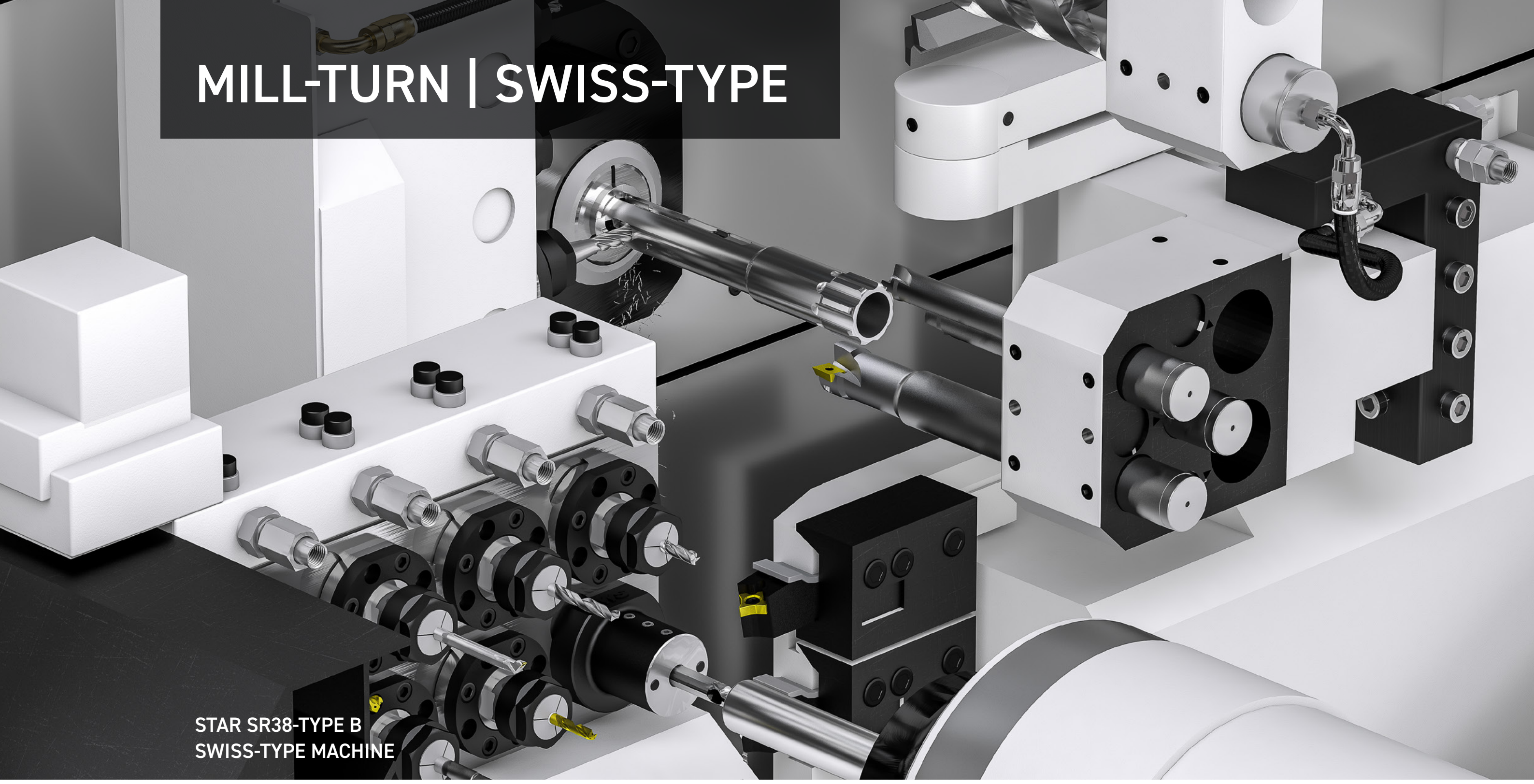


Easy-to-use Operation Sequence Manager guides you through the order of operations, shows clashes and assists you in avoiding them.

It is perfect for synchronizing and optimizing all your machining operations for the maximum production output.

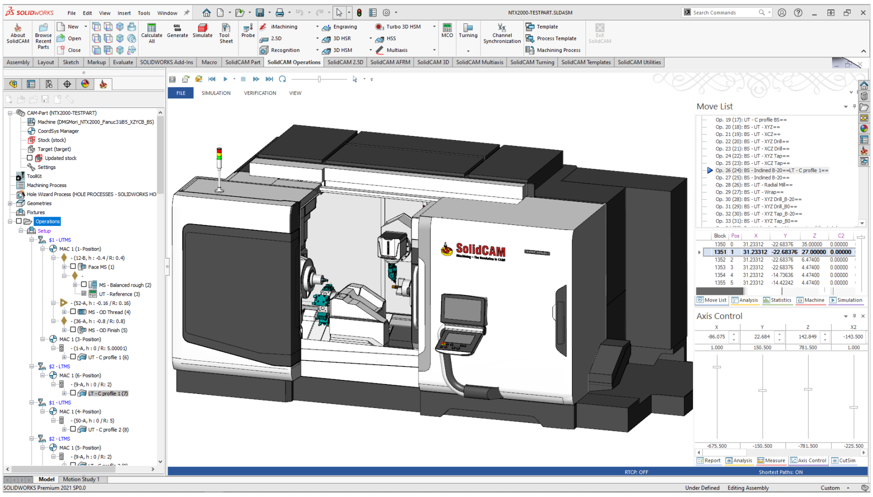
SolidCAM can control an unlimited number of channels and supports any amount of machine functions and cutting modes.

MILL-TURN | SWISS-TYPE



STAR SR38-TYPE B
SWISS-TYPE MACHINE

Speed Up Your Complex CNC-Machines



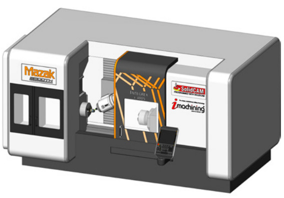
DMG Mori Seiki NTX2000 in Machine Simulation

SolidCAM supports the most complex CNCs with an unlimited number of axes and channels.

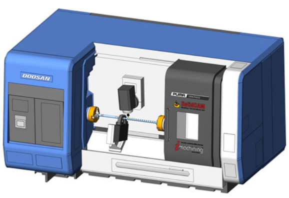
The machine tool database is being updated constantly to include Mill-Turn and Swiss-Type machines with various configurations and capabilities.

Advanced Machine Simulation shows the complete kinematics and all machine elements, providing full tool path simulation and verification for all your machining operations.

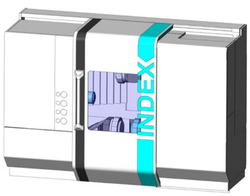
Chiron FZ08MT



Mazak Integrex i-400S

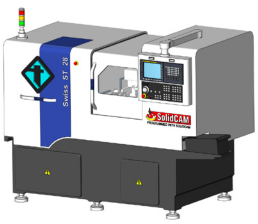


Doosan SMX2600SX

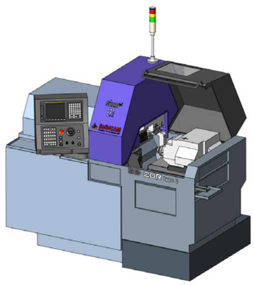


INDEX G200

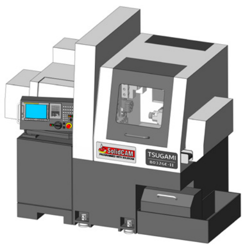
Citizen D25



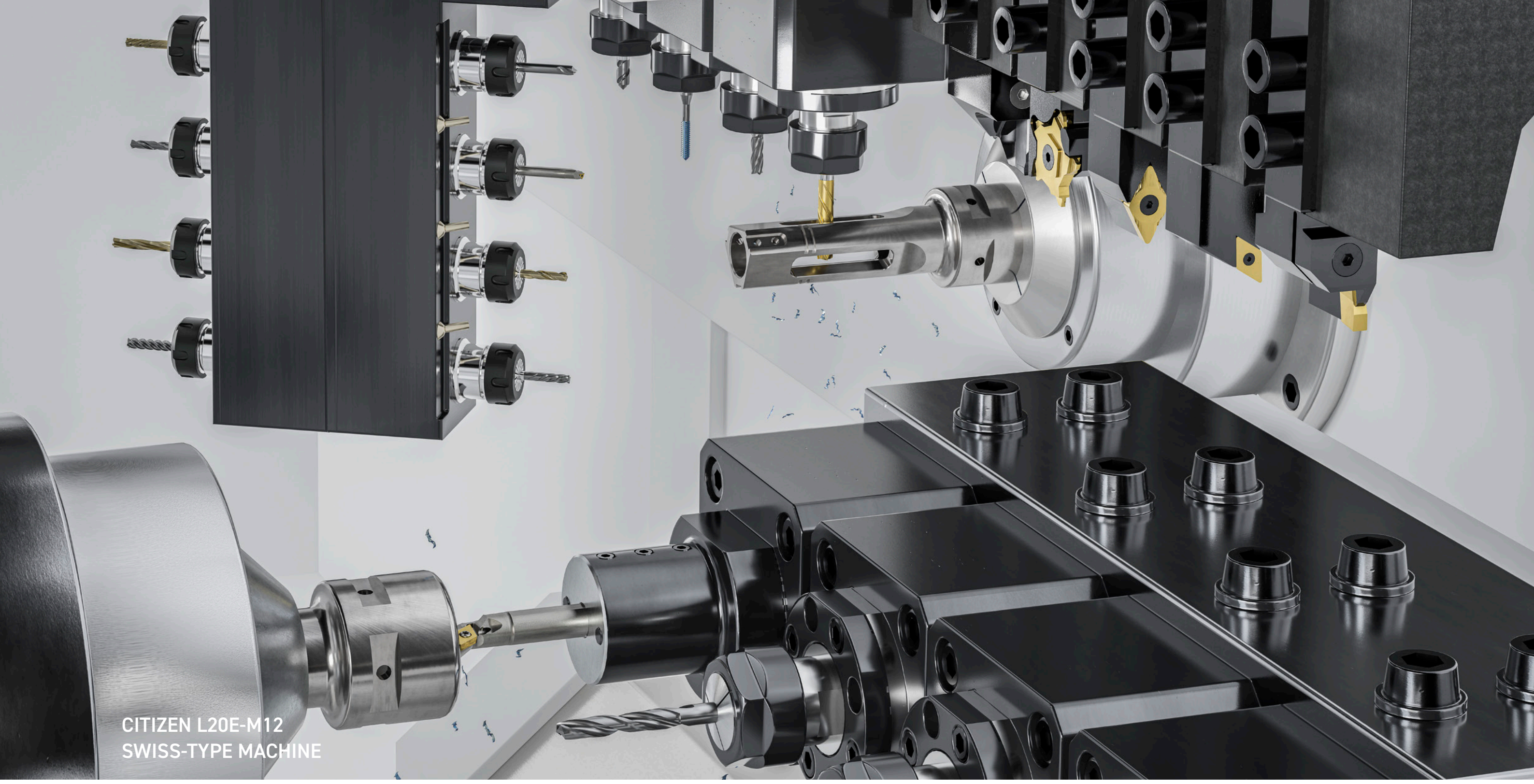
Swiss ST 28



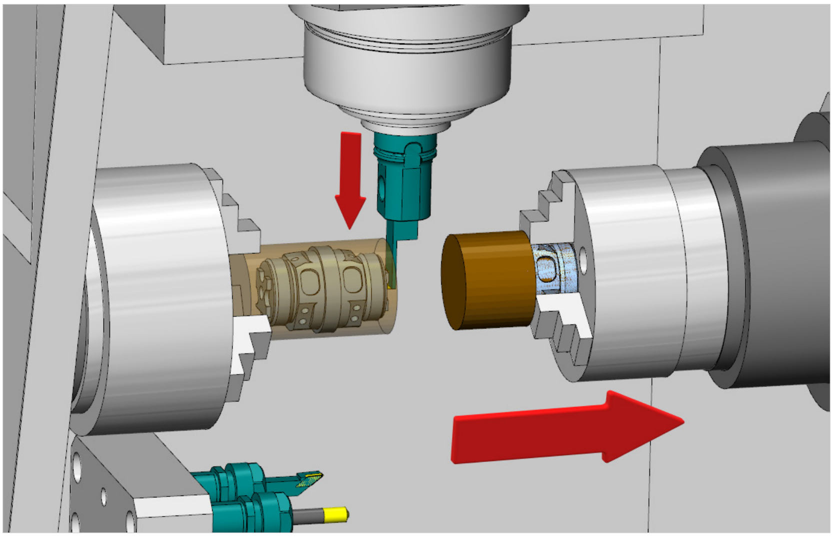
STAR SB20-R type G



Tsugami B0326E-II



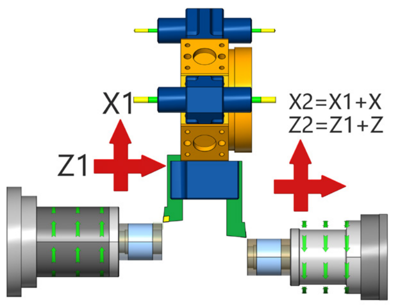
Advanced Rest Material Handling



SolidCAM always keeps the stock updated live, within the Operations tree, to optimize the tool path, avoid air-cutting and to achieve minimal cycle time.

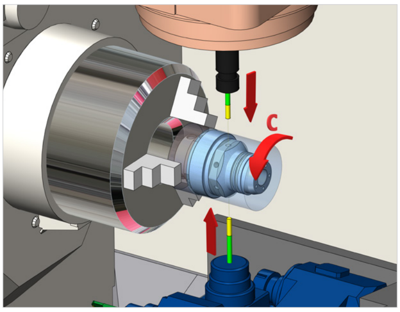
When the workpiece is transferred from the main to the subspindle, the Updated Stock Model (USM) is also transferred to the new position.

Any subsequent machining on the subspindle will detect the stock in the state that it left the main spindle, ultimately providing the most efficient machining possible.



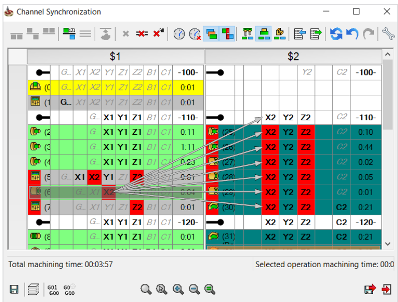
SolidCAM supports three different superimposition modes. A pair of axes can be superimposed one to another, where the slave follows the master.

For applicable Mill-Turn machines, SolidCAM will automatically detect this mode.



Axes and drive units can be shared to reduce machining time.

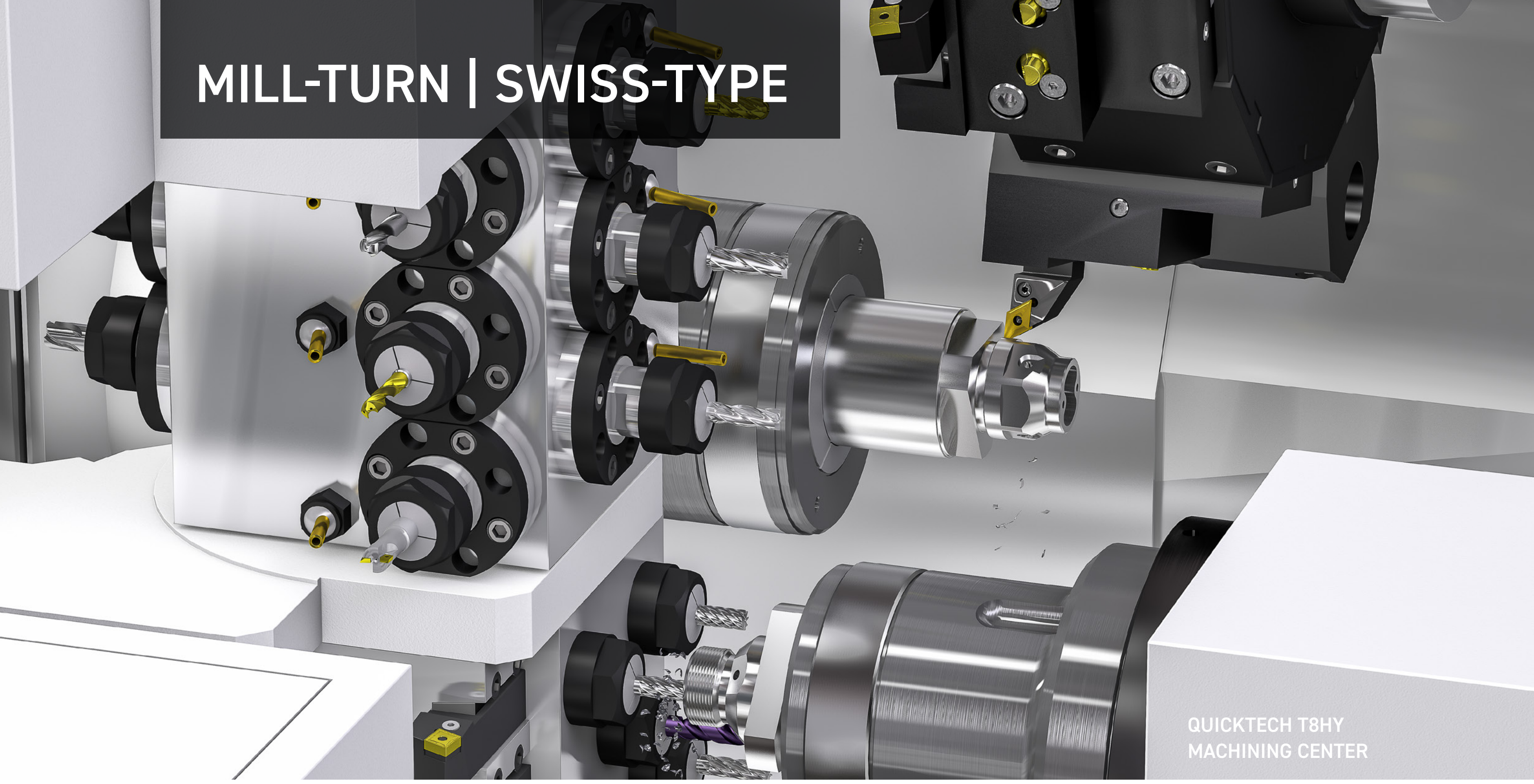
Synchronize your Milling and Turning operations, on different turrets, on the same table device, and under specific conditions.



The Operation Sequence Manager's clash engine displays any issue with logical comments.

The intelligent system holds the logic and checks the possibilities of the synchronization, taking into account the complete machine kinematics.

MILL-TURN | SWISS-TYPE

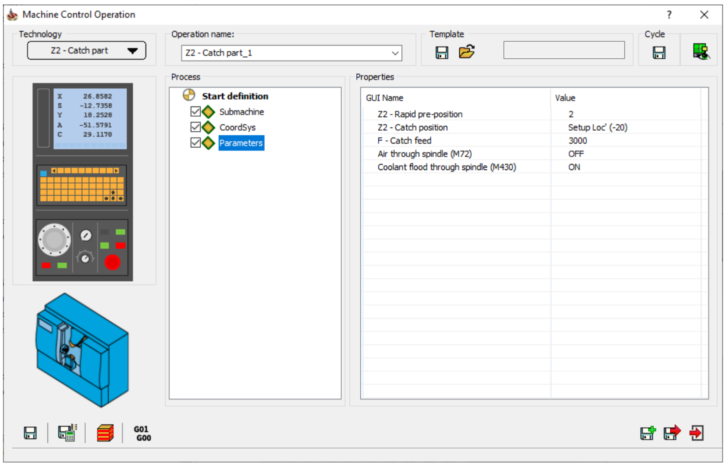


Machine Control Operations: MCO

With MCOs you can define various CNC-Machine actions, in addition to machining operations programmed in SolidCAM.

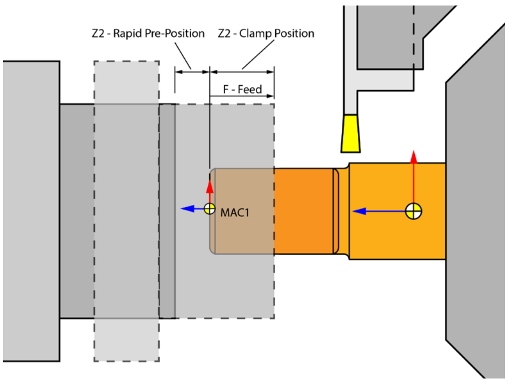
Such actions include:

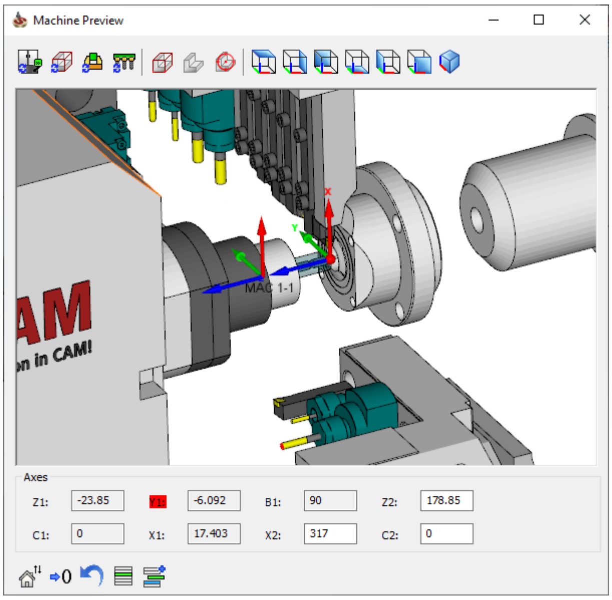
- Change tool
- Move machine components
- Transfer stock
- Clamp/unclamp fixture
- Program bar feeder
- Control coolants
- Machine mode
- Axes and phase synchronization
- Output any G/M command

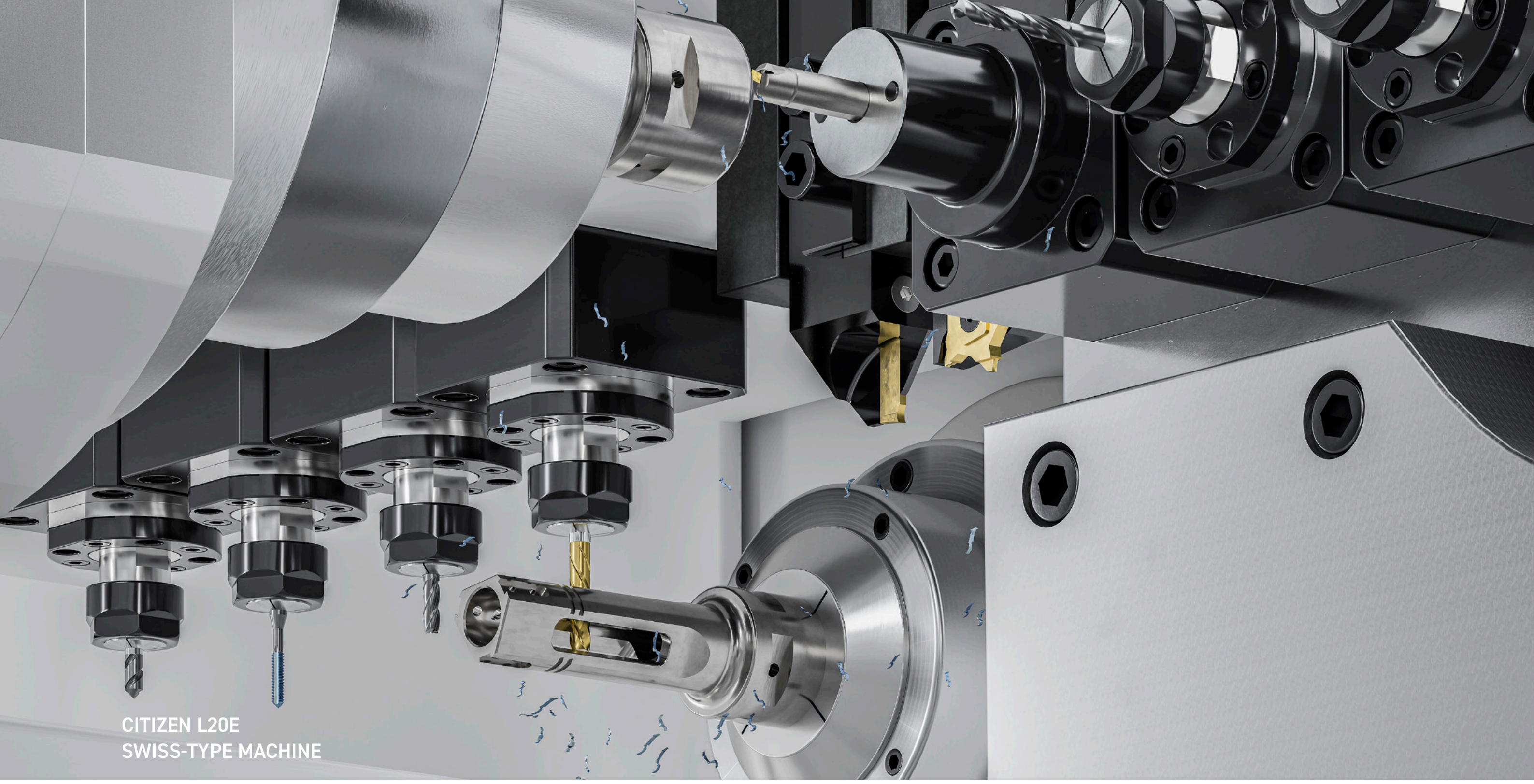


Part transfer between spindles

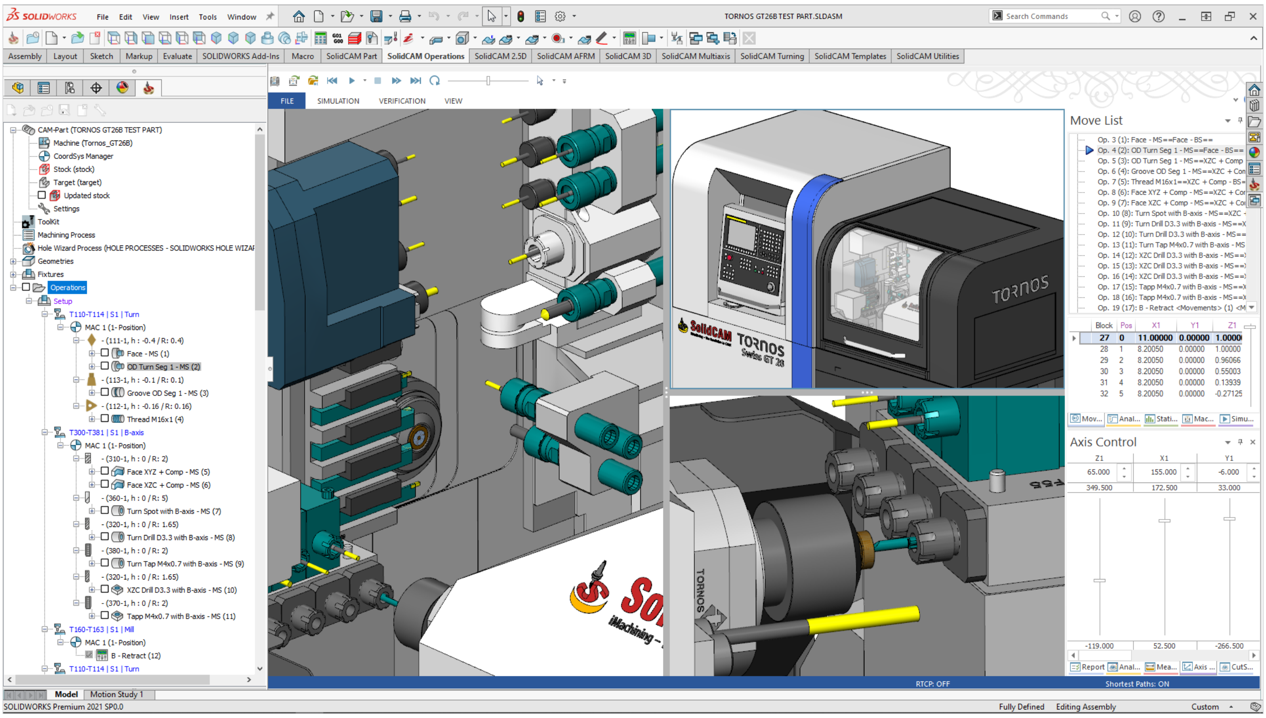
Control the transfer of parts between the main and subspindle, using Machine Control Operations. Ready-made MCOs provide the best solution for this process.



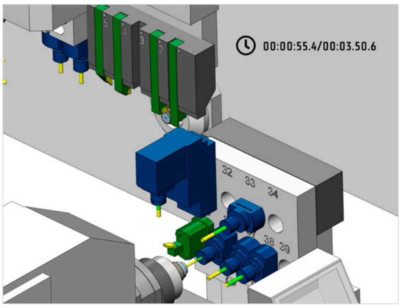




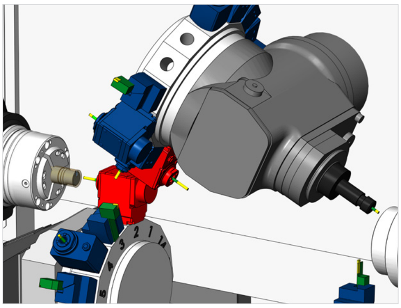
Advanced Machine Simulation



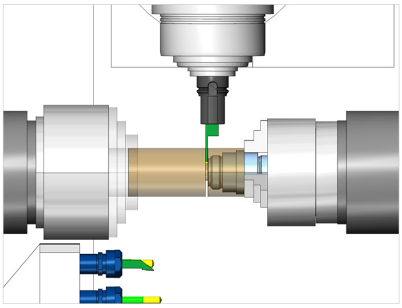
Making visual prove-out and verifying programmed tool path in Machine Simulation on Tornos GT26B.



Calculated cycle time is displayed

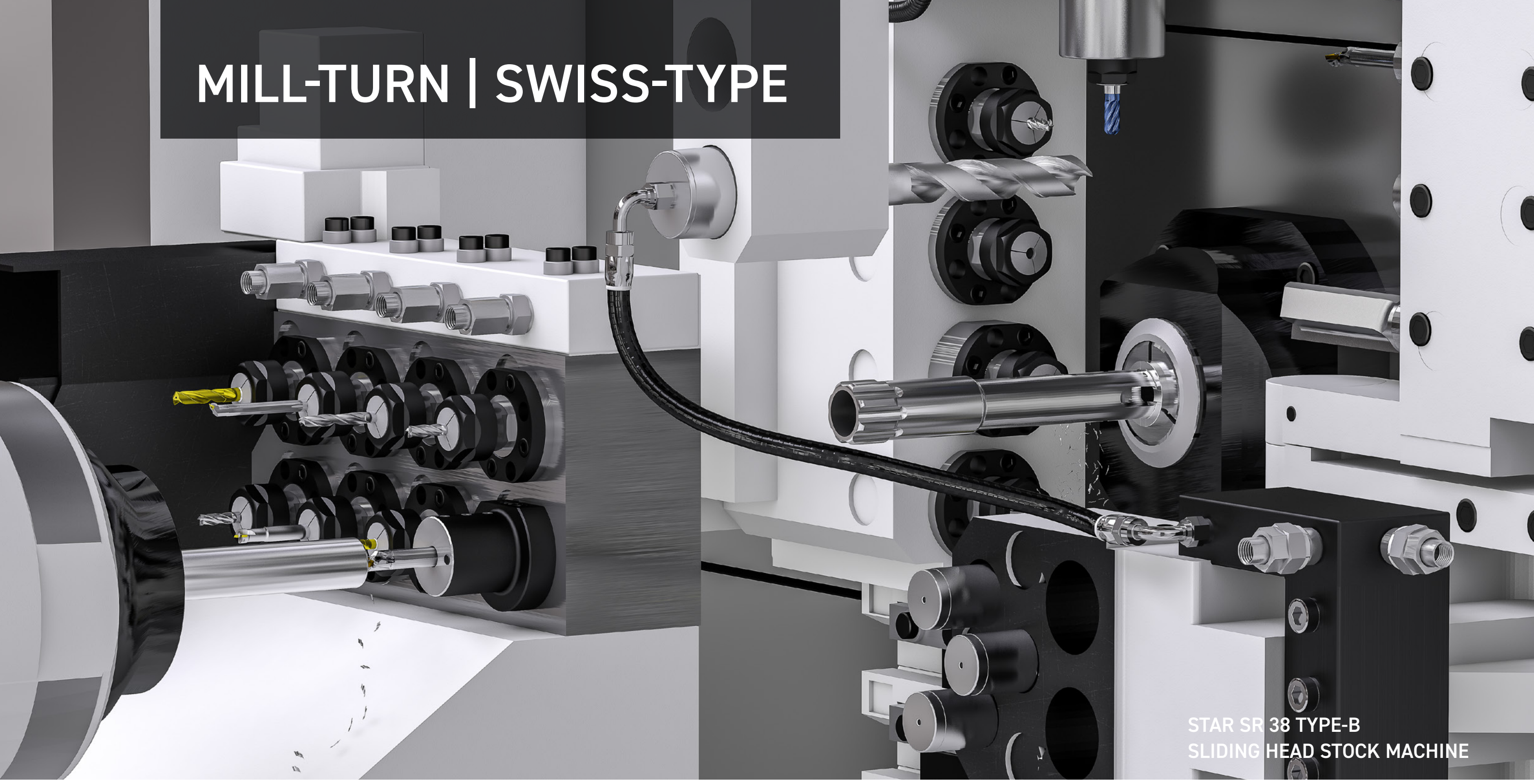


Collisions are detected



Cut-off process is simulated during part transfer

MILL-TURN | SWISS-TYPE



STAR SR 38 TYPE-B
SLIDING HEAD STOCK MACHINE

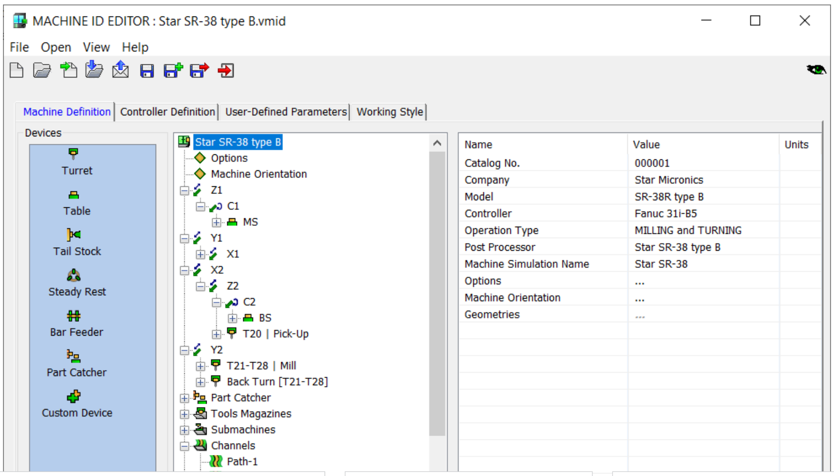
Post-Processors: Well structured. Verified. Trustful.

Open-source post-processors are written in SolidCAM's GPPL (an internal language for writing post-processors) and support defining output for any GCode format or structure for specific NC control unit. With no manual editing needed, generated GCode can be sent straight to the CNC-Machine.

Dedicated Post-Processor Team

Post-processors are defined by a dedicated development team of post writers, all with strong programming and practical machining experience. The Post-Processor Team takes care of customizing the GCode output to the needs and requirements of the specific controller and CNC-Machine.

Worldwide Post-Processor Team




```
%_N_TR_PROFIL3_Kanal1_MPF
;$PATH=/_N_WKS_DIR/_N_SOLIDCAM2018_
RADNABE_NTX1_WPD
N1 CHANDATA(1)
; ---------- KANAL: 1 ---------------
;SOLIDCAM  : 99748 PP:Rev.3.6
;ERSTELLT  :  9-MAY-2019 - 19:56:41
;MASCHINE  : NTX 1000
;WERKSTUECK: SOLIDCAM2018_RADNABE_NTX1000
; ----------------------------------
N2 WAITM(1,1,2)
R10=0 R11=0 R12=298.565 ;G54 X Y Z
R20=0 R21=0 R22=603.919 ;G55 X Y Z
R29=0                   ;G55

$P_UIFR[1]=CTRANS(X,R10,Y,R11,Z,R12,C4,0,C3,0)
;G54
$P_UIFR[2]=CTRANS(X,R20,Y,R21,Z,R22-
R29,C4,0,C3,0) ;G55

N3 WORKPIECE(,,,"CYLINDER",192,2.5,-150,-230,110)
;GOTOF ABDA

GROUP_BEGIN(0,"1: Programmkopf",0,0)
N4 WAITM(2,1,2)
N5 TRANS
N6 ROT
N7 DIAMOF
N8 GETD(Z3)
N9 GETD(B3)
N10 G00 SUPA X330. D0
N11 G00 SUPA Z400. Y0. B1=90.
N12 WAITM(3,1,2)
N13 NP_B3_VAR(0,870)
GROUP_END(0,0)
N14 WAITM(4,1,2)
N1 WAITM(5,1,2)
N1 WAITM(6,1,2)
```

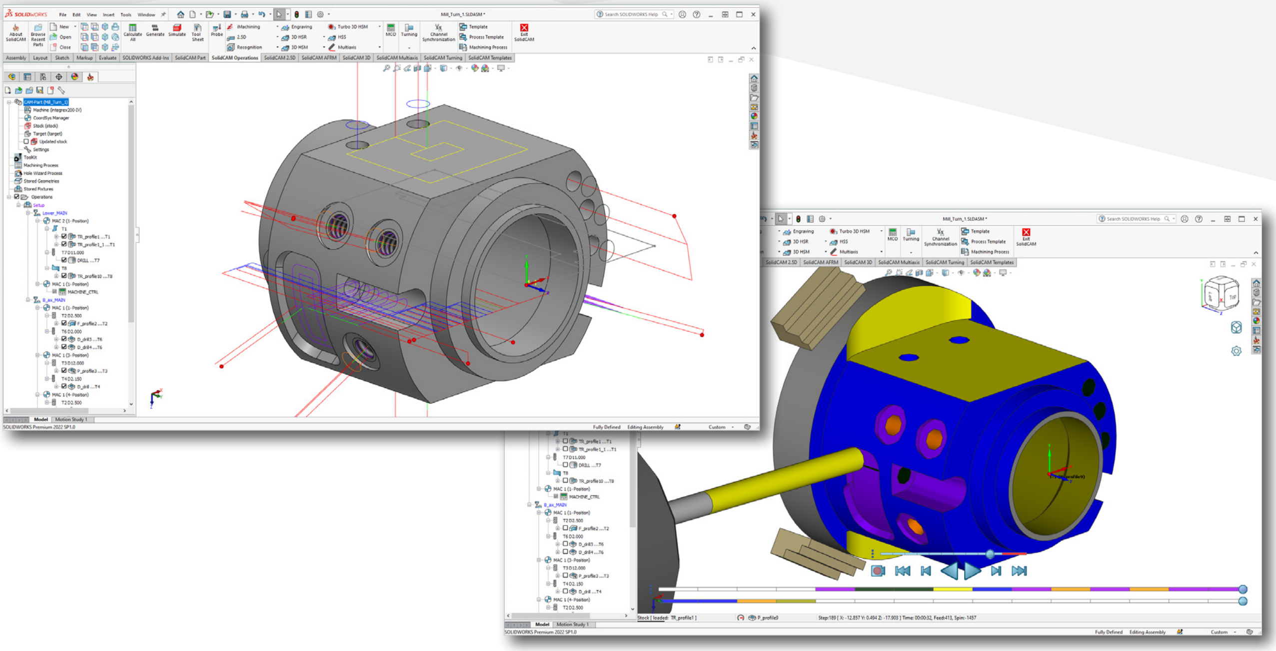
DMG Output

```
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

M902
M312
M302

M1
N1
T001.01 M6
M901
M200
M108
G90 G53 G0 B90.
M107
```

Mazak ISO Output

```
O0010(L32-1M12)
$1
(PROGRAMM-NR.: DCL32-L32-1M12)
(DATE: 9-MAY-2019)

G50 Z[#141-#142]
M52
M6
M9
M346
G0 X[#814+#815] Z-0.05
M51

G600

!2L110

(JOB-NR.2)
(MS-FACE)
T0202 Z-0.0867 (OD TURNING)
G18
G50 S1500
G96 M3 S300
M97
G0 G99 X1.436 Z-0.0867
Z0
G1 X-0.0315 F0.003
Z-0.08
G0 X1.4359
G97 M96

(JOB-NR.3)
(MS-OD)
G50 S4000
```

Mitsubishi / Fanuc
GCode Output



SolidCAM Part: Mill_Turn_1.prz

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

- **Contour and internal turning (TR_profile1; TR_profile1_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_profile2; 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.

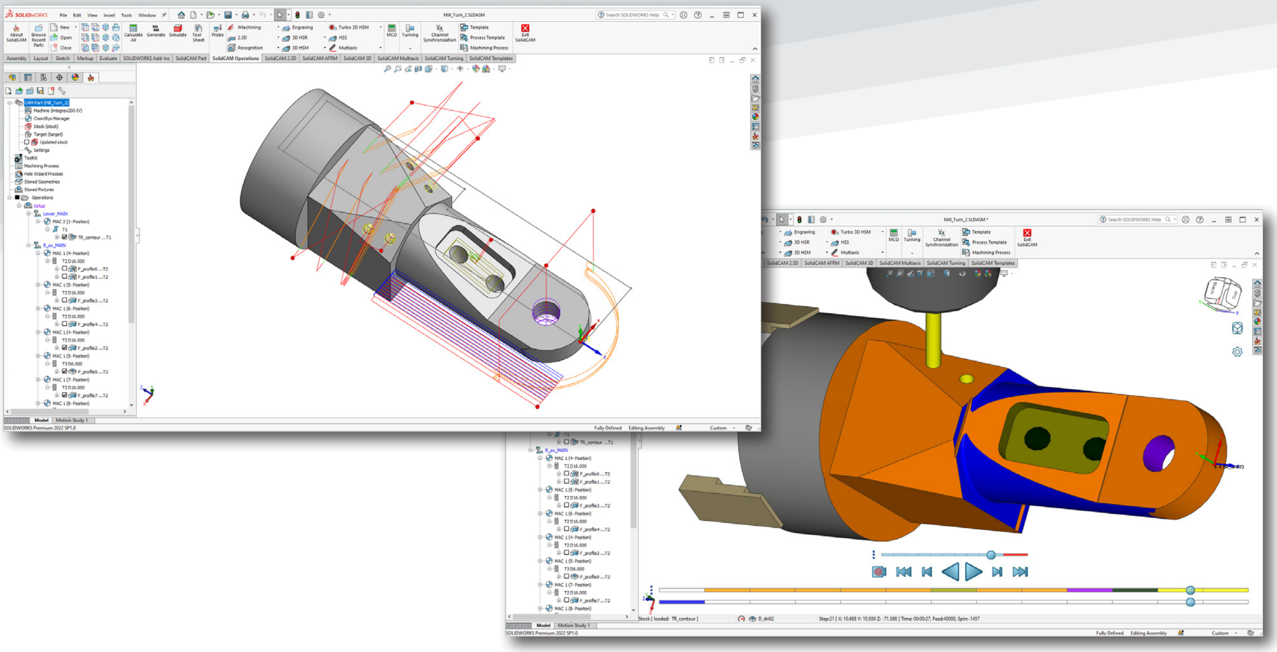
- **Radial holes machining (D_drill1; P_profile6; D_drill2; P_profile7)**

  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.



SolidCAM Part: Mill_Turn_2.prz

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

- **Contour turning (TR_contour)**

  This Turning operation defines the machining of the cylindrical surface.

- **Cube feature and horizontal faces machining (F_profile6; F_profile1)**

  These Profile operations define the indexial machining of the cube feature at the part base and the horizontal faces at the part front using Machine Coordinate System #1, Position #4 (MAC 1 (4- Position)). Both use the Transform option to create a circular pattern of operations around the revolution axis in order to machine all/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_profile9)**

  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)**

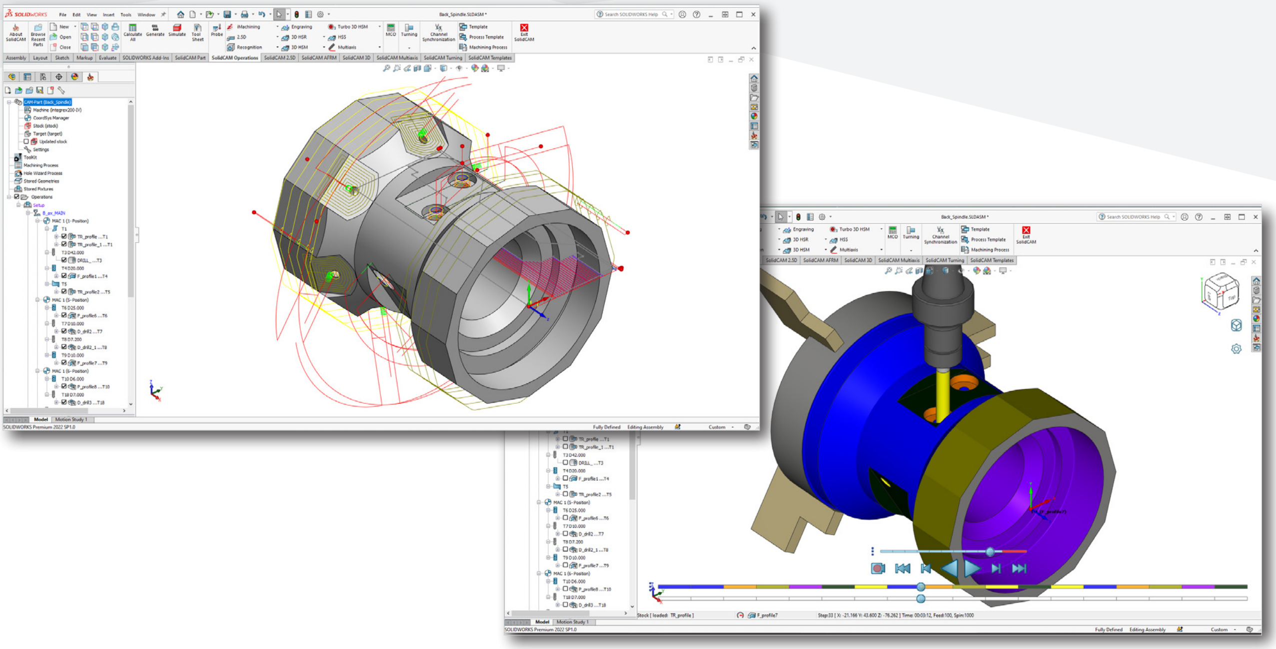
  These Profile operations define the machining of the inclined faces on the cube corners using MAC 1 (7 and 8- Positions) and the B-Axis tilting capabilities of the tool.

- **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.



SolidCAM Part: Back_Spindle.prz

The operations in this example illustrate the use of SolidCAM'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 SolidCAM 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 and medial pads machining (F_profile1; F_profile6)**

  The first 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 second Profile operation defines the indexial machining of the medial pads around the cylindrical surface 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 medial part holes machining (D_drill2; D_drill2_1; F_profile7)**

  These operations define the indexial machining of the counterbore holes located on the medial pads using MAC 1 (5- Position). The Drilling operations perform the center drilling and drilling of the holes. The Profile operation performs the counterbores machining. The Transform option is used to machine all the holes and counterbores.

- **Main spindle inclined pads machining/drilling (P_profile8; D_drill3)**

  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 is used to machine all the pads and holes.

- **Back spindle contour turning and facial milling (TR_profile9; F_profile10)**

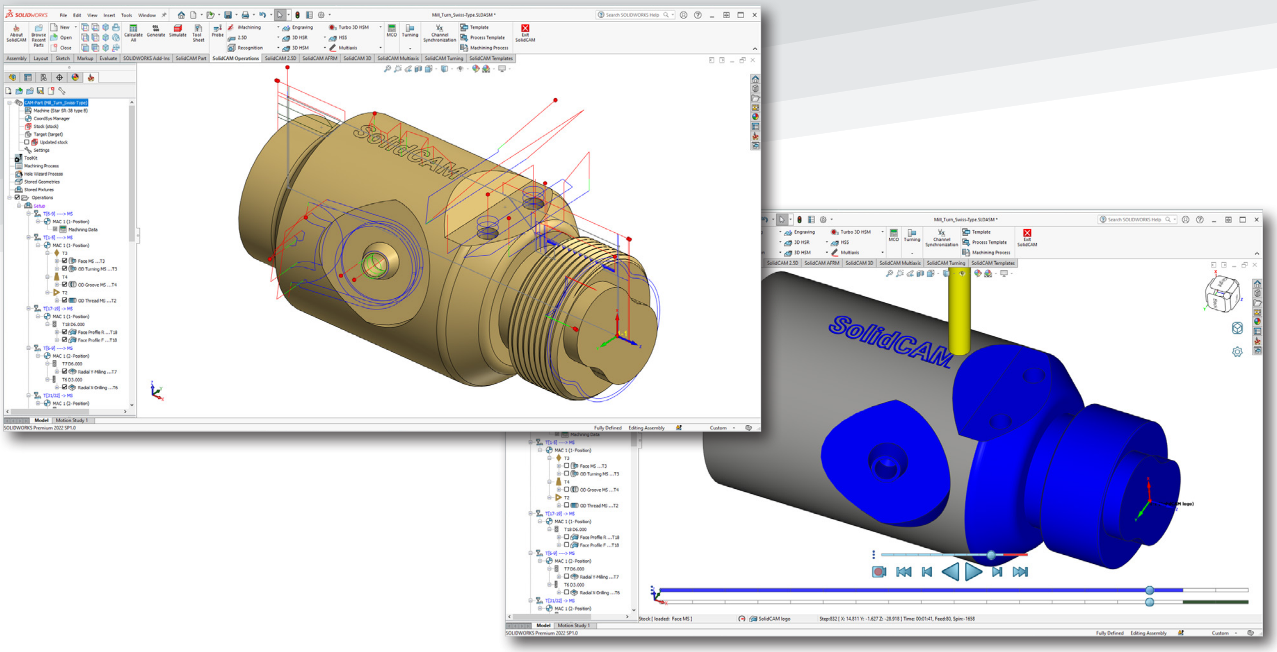
  These Turning and Profile operations define the initial machining of the external cylindrical surface on the back side and the final machining of the facets around the external surface using MAC 1 (1 and 4- Positions).

- **Back spindle internal turning (DRILL_1; TR_profile11)**

  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_profile12; D_drill4; D_drill4_1)**

  These operations define the machining of the features on the back face using MAC 1 (4- Position). The Profile operation performs the ledge machining. The Drilling operations perform the center drilling and drilling of the holes on the ledge corners.



SolidCAM Part: Mill_Turn_Swiss-Type.prz

The operations in this example illustrate the use of SolidCAM's Mill-Turn module with MCO cycles to machine the part shown above. The machining is performed on a Swiss-Type CNC-Machine using SolidCAM Milling and Turning technologies.

- **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 SolidCAM are similar to those required for SOLIDWORKS:

- Microsoft® Windows 10 or Windows 11, 64-bit operating system
- 64-bit; 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 (SOLIDWORKS certified recommended, visit solidworks.com/support/hardware-certification/ for certified cards and drivers)
- 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 SolidCAM user documentation