SolidCAM 2014

The unique, revolutionary Milling technology

iMachining®

patent by SolidCAM

# SolidCAM 2014 Modules Overview: Parts and Recordings

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

SOLIDWORKS Certified Gold Product

SolidCAM + SolidWorks

The complete integrated Manufacturing Solution

SolidCAM

The Leaders in Integrated CAM

# Table of Contents



**Document number: MOPRENG1402SP2**

The purpose of this booklet is to give you an overview of all available SolidCAM 2014 modules. There are SolidCAM Parts that demonstrate each of the modules, including a SolidCAM Professor recording about each Part.

These Parts are included in the SolidCAM installation and are located in your hard drive (full path: **C:\Users\Public\Documents\SolidCAM\SolidCAM2014\User\Getting_Started_Examples\SW**).

To enable correct loading of the Parts, make sure these Parts are always stored in this default location.

You can view the SolidCAM Professor recording of each Part in one of the following ways:

- click the **Play** button on the picture of the Part to open the recording in YouTube

  or

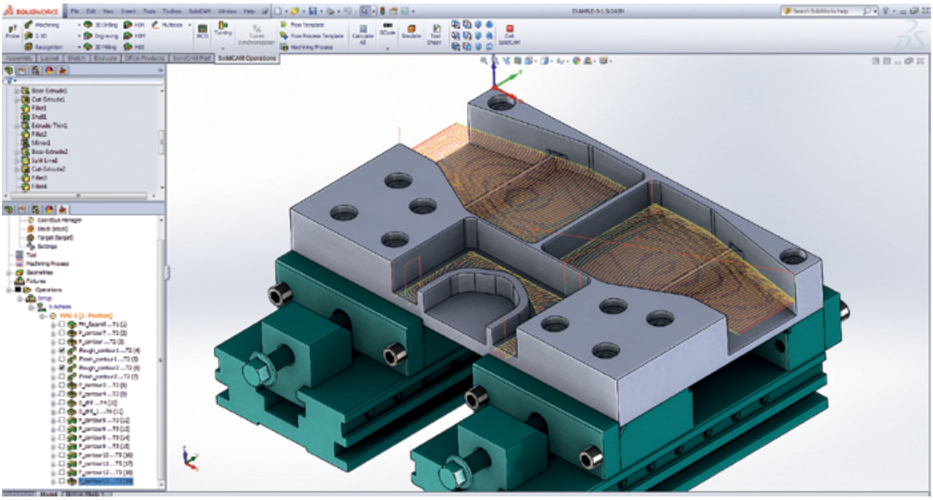
- click the **Download** button to download and store the recording in your hard drive

Click the **SolidCAM** icon to load the Part described in the recording.

To see many more Professor video tutorials that cover all features and uses of SolidCAM, visit us at www.solidcam.com/professor.

For the most updated PDF version of this booklet, visit our website at: http://www.solidcam.com/en/support/documentation/solidcam-2014/.

# SolidCAM – The leading CAM-Solution integrated in SolidWorks



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

SolidCAM, including the revolutionary iMachining, is seamlessly integrated in SolidWorks and with full tool path associativity to the SolidWorks model. With the single-window integration in SolidWorks, all machining operations can be defined, calculated and verified without leaving the parametric SolidWorks assembly environment.

SolidCAM+SolidWorks provide a powerful, easy-to-use integrated CAD/CAM solution that support the complete range of major manufacturing applications including iMachining 2D, iMachining 3D, 2.5D Milling, High Speed Surface milling, 3D Milling/ High-Speed Machining, Multisided Indexial 4/5-axis Milling, Simultaneous 5-axis Milling, Turning, Turn-Mill, WireEDM and Solid Probe.

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.

## Major Benefits of Using SolidCAM in SolidWorks:

- SolidWorks Look and Feel through seamless Single Window integration
- Full associativity: tool paths automatically update when SolidWorks model changes
- SolidCAM's 10+ years as SolidWorks Gold Partner
- SolidCAM works in the SolidWorks assembly mode to define Fixtures, Tooling and Vices
- SolidCAM + SolidWorks is scalable with packages for all CNC machine types and applications
- The integrated CAD/CAM-Solution SolidWorks + SolidCAM is available from SolidCAM at a competitive bundle-price

## SolidCAM + SolidWorks Automates IC Test System Manufacturing for Essai

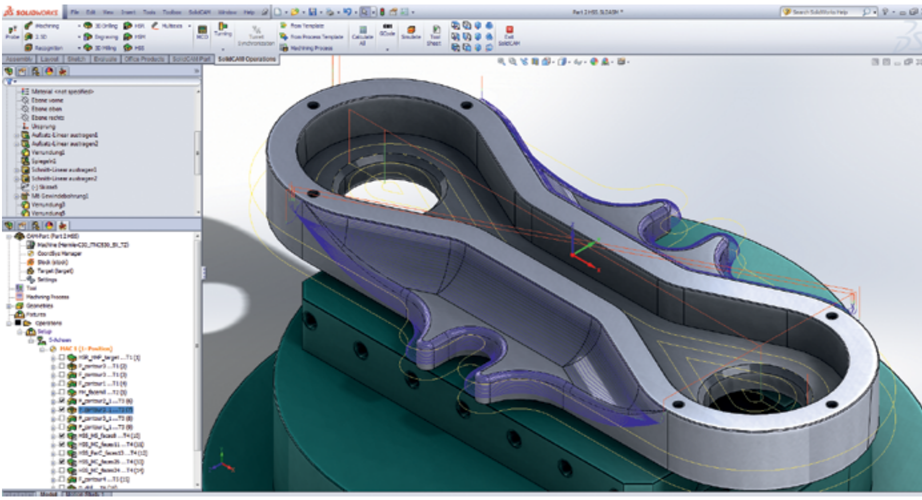
Deniz Valle, Essai Corporation Operations Manager:

- “Because SolidCAM is completely integrated with SolidWorks CAD software, we are actually building our CAM programming within 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.”
- “The integrated approach facilitates discussion and resolution of manufacturing issues because everyone is working with the same model and the same modeler. We communicate issues and features a lot better working with an integrated system.”

Barabi, Essai Corporation Founder & Manager:

- “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 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.”

## You Never Have to Leave the SolidWorks Window!



Larry Rehak, Intricate Metal Forming Co:

- "In the last 45 days, 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."

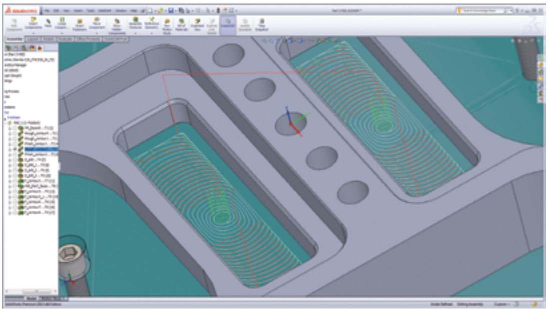
Terry Kramer, Kramer Design Corp.:

- "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."

# Imagine putting the knowledge and experience of hundreds of CAM and CNC masters in the palm of your hand – Experience iMachining Wizard & Tool path!

## Patented iMachining: “Simply Amazing”

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



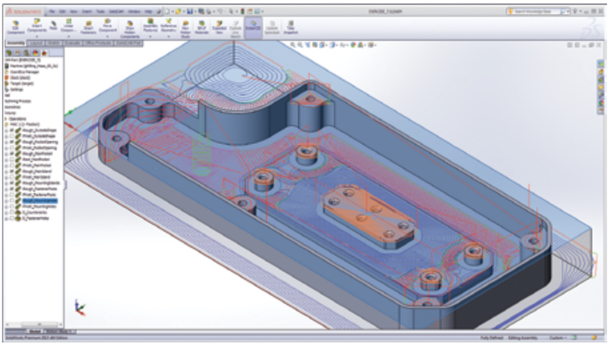
## The Revolution in CNC Machining

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

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

## Unique Technology Wizard

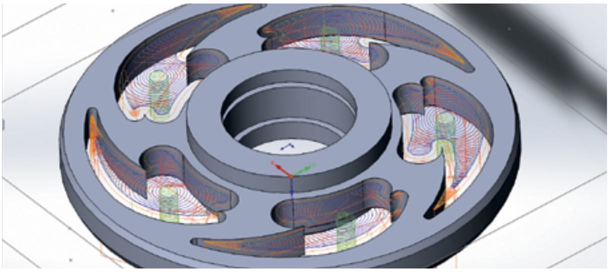
SolidCAM's iMachining has the exclusive patented iMachining Technology Wizard, the industry's first and only Tool Wizard that automatically calculates the cutting conditions for the iMachining tool path.



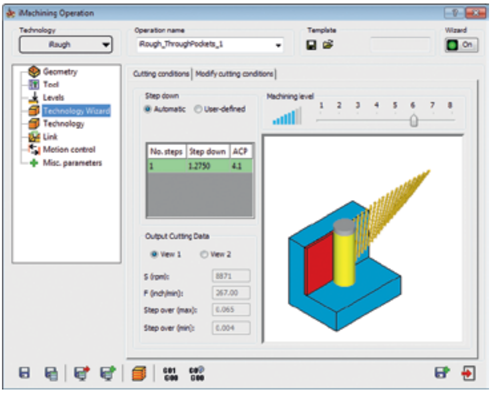
The unique Technology Wizard provides optimal feeds and speeds, taking into account the tool path, stock and tool material and machine specifications.

Using the "Controlled Step Over" technology, the iMachining tool path ensures that the cutting conditions set by the Wizard are strictly adhered to.

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



## iMachining Wizard + iMachining Tool path = the Ultimate Solution!




## SolidCAM's iMachining highlights:

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

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

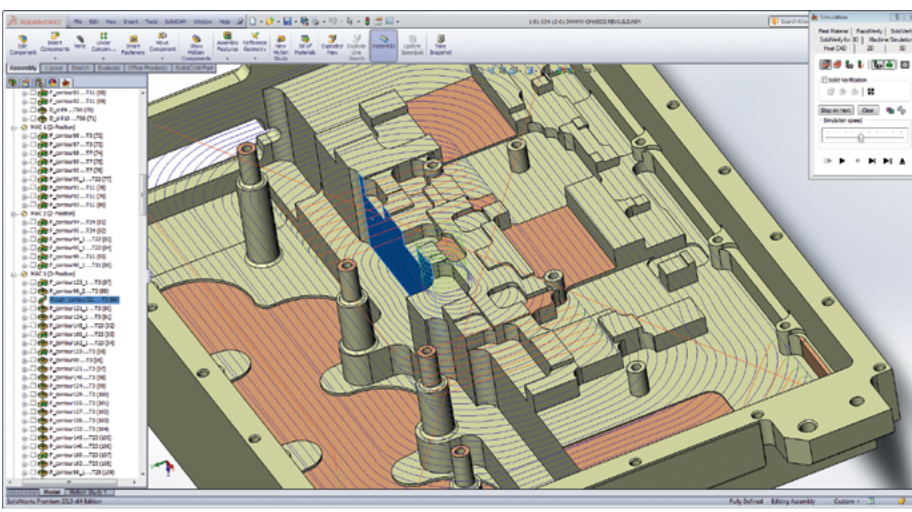
Jay Dixon, Dixons Surgical, UK

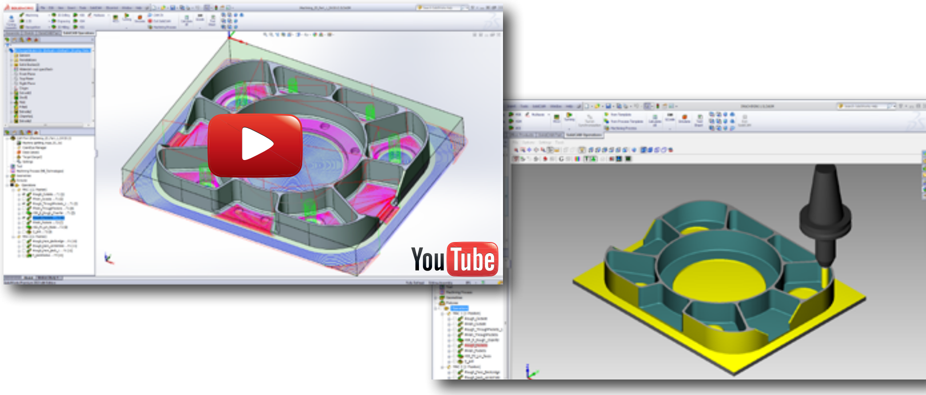
*"Every day we don't use SolidCAM's iMachining we are losing money!"*

Jason Near, Rotary Airlock , IL, USA

*"With iMachining, even on low-performance machines, we can reach very high metal removal rates"*

Dreiling Maschinenbau GmbH, Germany






Open SolidCAM Part: **iMachining_2D_1.prz**

Download the recording

This example illustrates the use of SolidCAM's iMachining technology to machine the part above. The machining is performed on a 3-axis CNC-machine in two setups, from both sides of the part.

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

  These iMachining operations perform the cutting of the outside shape of the part. An end mill of Ø12.7 mm (0.5") is used. 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. iRough has a 0.25 mm (0.01") allowance on the wall, and the iFinish operation finishes the profile.

- **Through pockets machining (iRough_ThroughPockets_1; iFinish_ThroughPockets)**

  These iMachining operations perform the cutting of the five circular through pockets. An end mill of Ø12.7 mm (0.5") is used. Five chains are defined to represent the five closed pockets. Since the pockets are closed, with no PreDrilling or EntryChain defined, helical ramping is used to enter the bottom of the pocket.

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

  This HSR operation performs the rough cutting of the four large chamfers on the ribs. An end mill of Ø12.7 mm (0.5") is used. Two boundaries are picked off the edges the make up the chamfer and the Tool Relations is set as centered. A 1.27 mm (0.05") step down is used and 0.127 mm (0.005") allowance on the surfaces.

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

  These iMachining operations perform the cutting on the three semi-open pockets and the 7 closed pockets. A bull nose mill of Ø10 mm (0.375") and corner radius of 1.6 mm (0.0625") is used. Since all the 10 pockets are located on the same Z-Level, they can be machined all in one operation. iRough has a 0.25 mm (0.01") allowance on the wall, and the iFinish operation finishes the profile.

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

  This HSS operation performs the finishing cut on the four large chamfers on the ribs. The same tool as in the previous operation is used.

- **Holes machining (D_drill)**

  This drilling operation performs drilling of the holes located on the lower surface of the part.

- **Bottom ledge machining (iRough_Face_BackLedge)**

  This iMachining operation finishes the bottom ledge on the underside of the part. An end mill of Ø12.7 mm (0.5") is used. The floor radius is not machined at this stage.

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

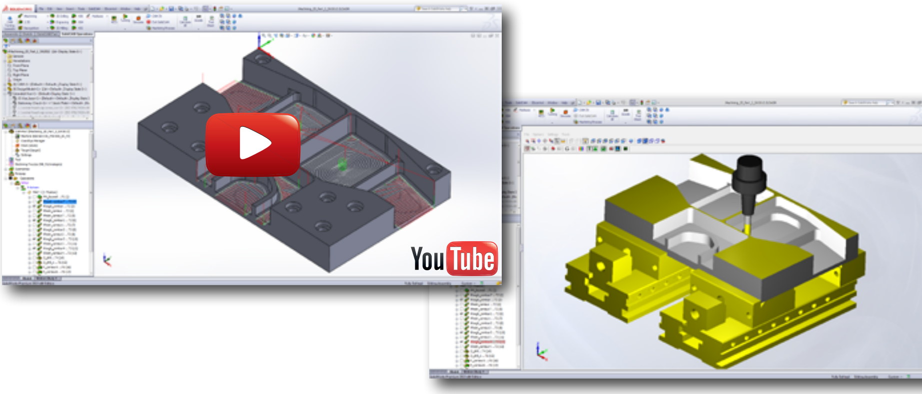
  This iMachining operation machines away the excess material from the center through hole of the part. This excess material was used for clamping from the first side.

- **Bottom face machining (iRough_Face_Back_1)**

  This iMachining operation finishes the circular face on the underside of the part. An end mill of Ø12.7 mm (0.5") is used. A spiral tool path is performed from the outside, collapsing towards the inner chain.

- **Floor radius finishing (F_backRadius)**

  This Profile operation finishes the 6.35 mm (0.25") floor radius on the underside of the part. A ball mill of Ø12.7 mm (0.5")t is used. The chain is the bottom edge of the radius and the Tool side is set to center. The 0.13 mm (0.005") floor offset is left after the first roughing pass and then removed with the finishing pass. A 0.25 mm (0.01") Lead in/out arc is used.



Open SolidCAM Part: **iMachining_2D_2.prz**

Download the recording

This example illustrates the use of SolidCAM's iMachining technology to machine the part above.

- **Upper face machining (FM_facemill)**

  This face milling operation machines the part face. A Ø100 mm end mill is used. Machining is performed using the hatch technology.

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

  These iMachining operations perform the cutting on the semi-open pocket. An end mill of Ø12 mm is used. iRough operations have a 0.24 mm allowance on the wall, and the iFinish operations finish the profiles.

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

  These iMachining operations perform the cutting on the semi-open and closed pocket. The first iRough and iFinish operations machine two larger open pockets. An end mill of Ø12 mm is used. iRough operations have a 0.24 mm allowance on the wall, and the iFinish operations finish the profiles. The second and third pair of operations machine the smaller open and closed pockets.

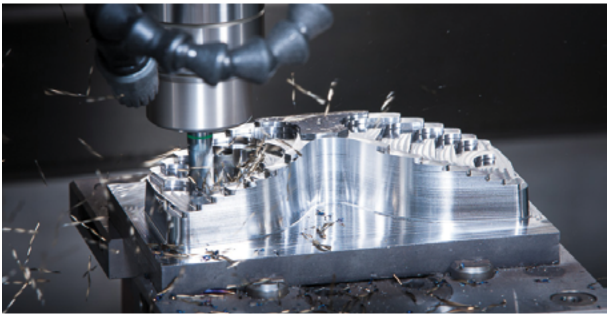
- **Holes machining (D_drill, D_drill_1, F_contour5)**

  The drilling operations perform center drilling and drilling of the holes located on the upper surface of the part. The profile operation finishes the walls of the holes.

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

  These profile operations are created to machine chamfers on the part edges using a Ø6 mm chamfer drill.
  The chamfer option is selected from Rest material/Chamfer section.

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



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

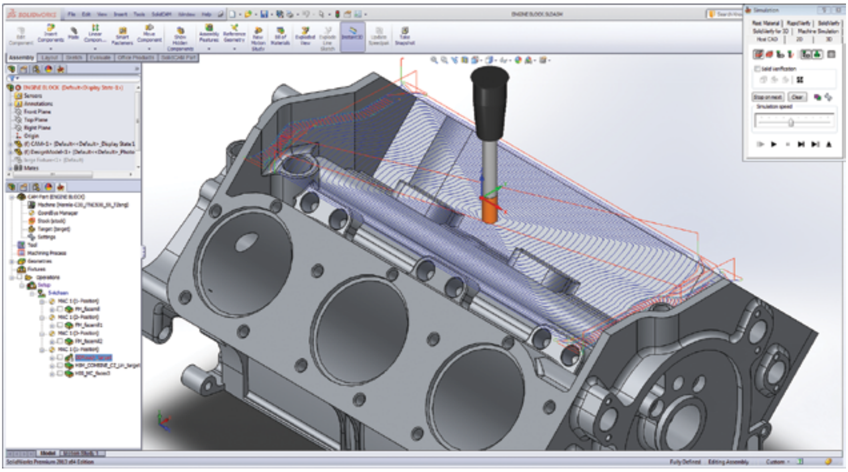
iMachining 3D automatically produces a complete, ready to run CNC program, with optimal cutting conditions achieved by the expert Knowledge-based Technology Wizard, to rough and rest rough a complete 3D part, all in a single operation, both for 3D surfaced and prismatic parts.

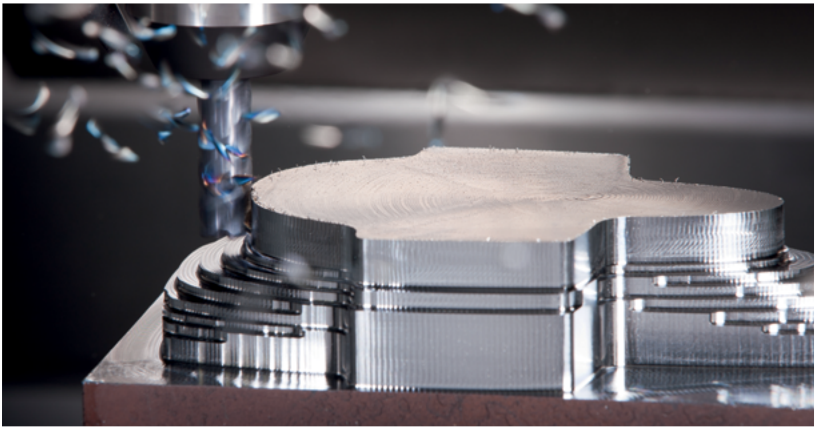
Combined with its full-depth step-down, intelligent step-up, localized machining and smart positioning, iMachining 3D eliminates almost all retracts, long positioning, and air cutting to produce the shortest optimal cycle times in the industry for roughing and semi-finish of molds, complex 3D parts and 3D prismatic parts.

Combined with SolidCAM HSM Finish, iMachining 3D provides a complete machining solution for 3D parts.

## Exclusive iMachining 3D Features:

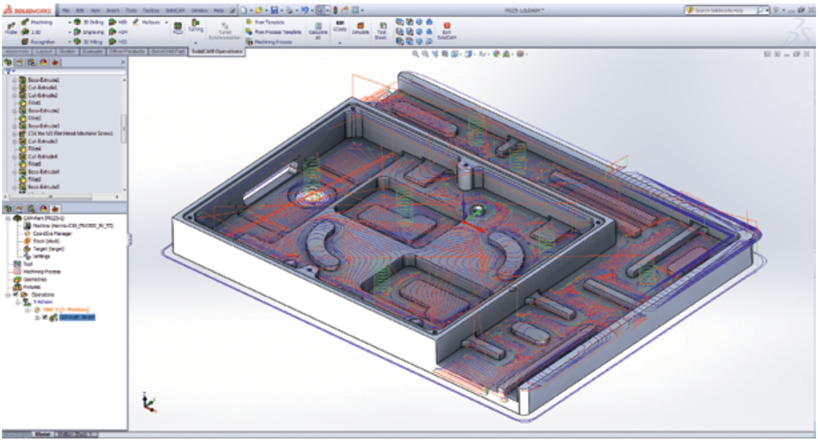
- Quick solid geometry selection
- Optimized machining of each Z-Step, using proven iMachining 2D technology
- Deep roughing utilizes the whole length of the flute, shortening cycle time and increasing tool life
- Rest material machining in small upward steps, optimized for constant scallop height, further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





## iMachining 3D for Prismatic Parts

With iMachining 3D, you can also mill prismatic parts, which include multiple pockets and islands. This is done in one operation, directly from the solid model of the part and the solid model of the stock, without the need for defining geometry chains. iMachining 3D will then calculate the tool path automatically and optimally – drastically reducing programming time.



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

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

Igor, Chief Programmer, Menes

## iMachining 3D Cuts Machining Time by 75 % against another CAM-System on Makino CNC:

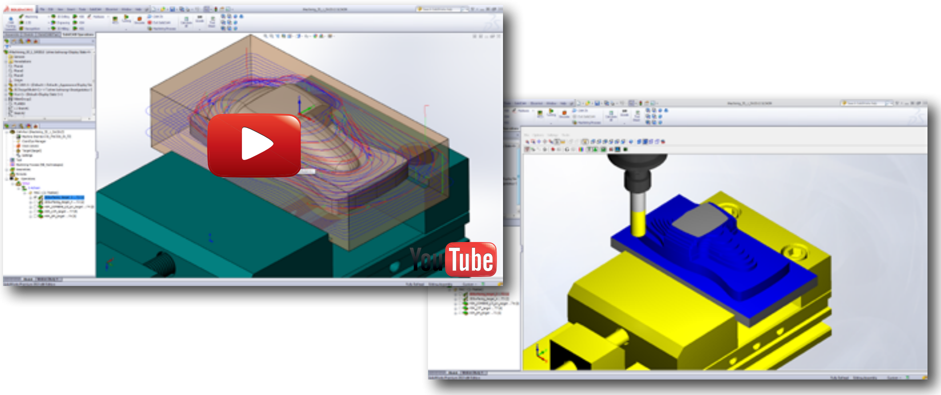
*"iMachining 3D cut machining time from 4 hours, with a Competitor's CAM, down to 58 minutes ... a 75 % time savings!"*

Galtronics, China

## SolidCAM Customer A.P.A. on iMachining 2D & 3D, mainly for Aluminum machining:

*"'Amazing', I have no words to describe my satisfaction from iMachining - I can't even imagine how much time it will save us in Aluminum!"*

David Franko, Owner, A.P.A.



Open SolidCAM Part: **iMachining_3D_1.prz**

Download the recording

This example illustrates the use of SolidCAM's iMachining 3D technology to machine the part above.

- **Roughing (i3DSurfacing_target_3)**

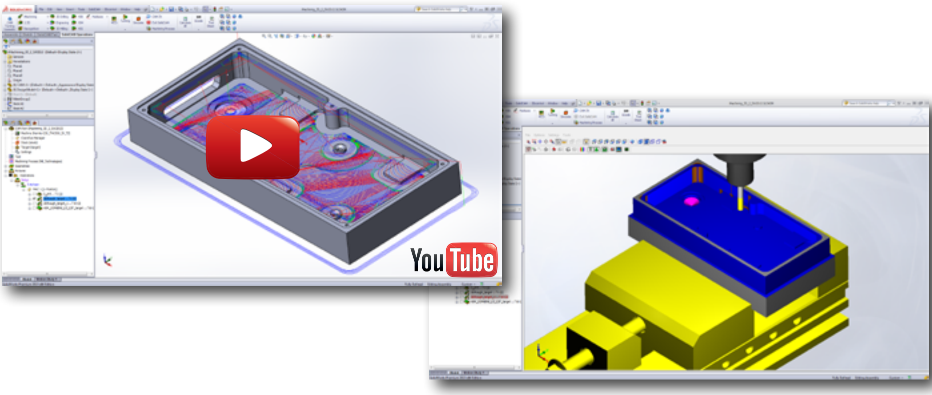
  This operation removes the bulk of material using the 3D Surfacing strategy. An end mill of Ø16 mm is used. The machining is performed in 2 large step down cuts till the bottom of the part. Then Rest Rough is performed in the step up manner leaving steps with the automatically defined value of Scallop. A machining allowance of 0.2 mm remains unmachined for further finish operations.

- **Roughing (i3DSurfacing_target_4)**

  This operation removes the material left after the previous operation. A bull nose mill of Ø12 mm with corner radius of 0.6 mm is used. The Rest Rough is performed in the step up manner leaving steps with the automatically defined value of Scallop. A machining allowance of 0.2 mm remains unmachined for further finish operations.

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

  These HSM operations use the Combined, Horizontal, and Rest Machining strategies for finishing different areas of the part, including floors and corners.




Open SolidCAM Part: **iMachining_3D_2.prz**

Download the recording

This example illustrates the use of SolidCAM's iMachining 3D technology to machine the electronic box shown above.

- **Hole machining (D_drill)**

  This drilling operation perform drilling of the through hole. The operation helps prepare for the tool entry in the following iMachining operation. A Ø16 mm drill is used.

- **Roughing (i3DRough_target)**

  This operation removes the bulk of material using the 3D Surfacing strategy. An end mill of Ø12 mm is used. The machining is performed in one step down cut till the bottom of the part. Then Rest Rough is performed in the step up manner leaving steps with the automatically defined value of Scallop. A machining allowance of 0.38 mm remains unmachined for further finish operations.

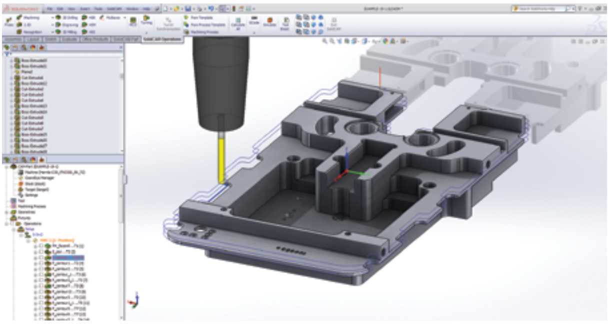
- **Roughing (i3DRough_target_1)**

  This operation removes the material left after the previous operation. An end mill of Ø6 mm is used. This operation cleans the floor and corners left unmachined. A machining allowance of 0.38 mm remains unmachined for further finish operations.

- **Finish machining (HSM_COMBINE_CZ_CZF_target))**

  This HSM operations combines the Constant Z and Horizontal strategies to finish different areas of the part, including floors and walls.

## The Most Powerful & Easiest to Create 2.5D CNC Milling Tool paths: Full Interactive Control + Feature Recognition!



The most straightforward, easy-to-use interface, 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.

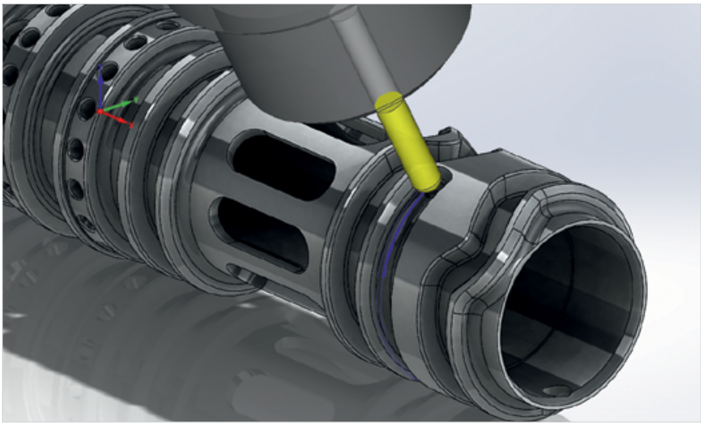
## Best of Both Worlds: Complete Interactive Control + Feature Recognition

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.

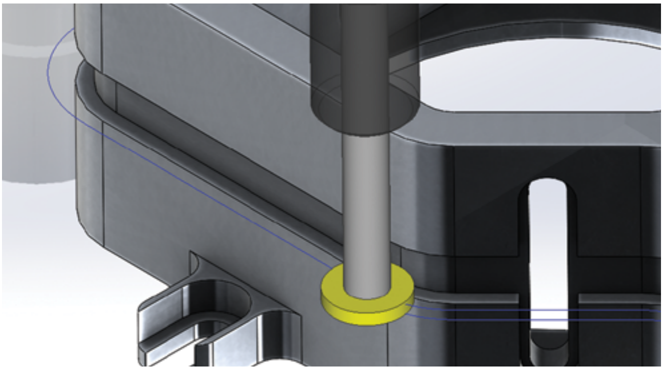
## Interactive 2.5D Mill Operations

Besides the standard 2.5D milling profiling, pocketing and drilling operations, SolidCAM offers:

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

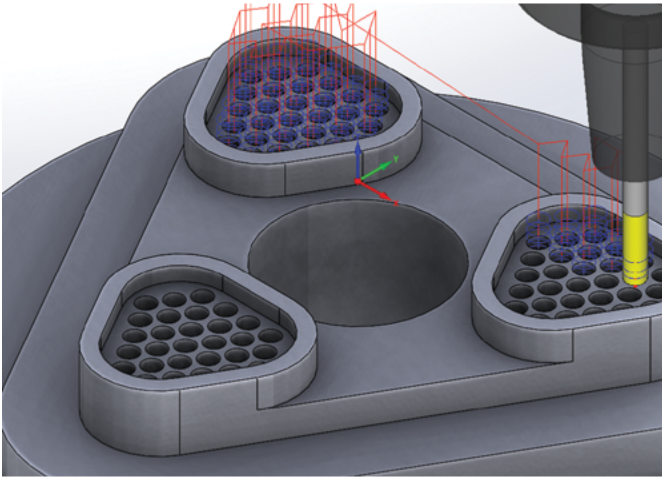


- Special operation for machining of the side slots with undercut by a T-slot tool



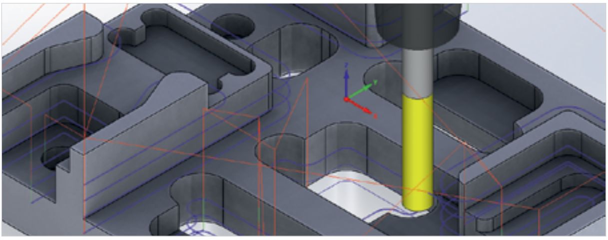
## Drill Recognition

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.



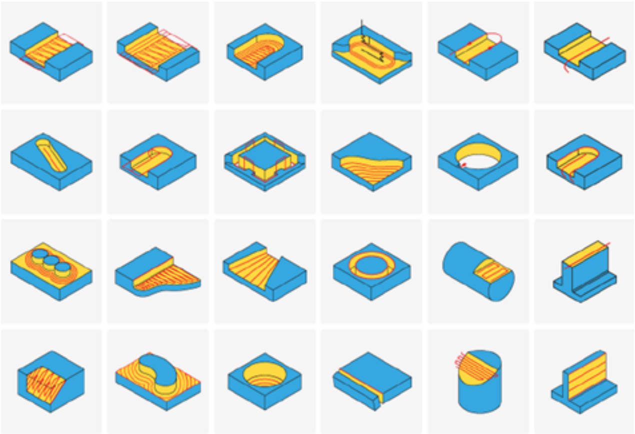
## Pocket Recognition

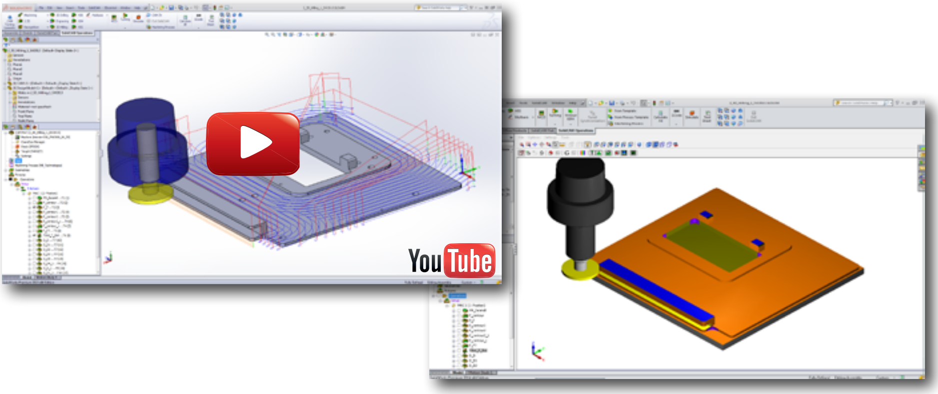
Takes SolidCAM's powerful pocketing operation to the next level, by automatically identifying all pockets on the CAD model. All strategies and options of the standard Pocket operation are available, combined with variable levels and depths recognized from the model faces.



## Cycle Toolbox

A very useful and convenient Cycle Toolbox provides additional specialized sub-operations for slots, corners, bosses, ruled surface, etc.






Open SolidCAM Part: **2_5D_Milling_1.prz**

Download the recording

This example illustrates the use of SolidCAM 2.5D Milling to machine the part shown in the video recording. The machining is performed on a 3-axis CNC machine in one setup.

The following SolidCAM operations are created to perform the machining:

- **Upper face machining (FM_facemill)**

  This face milling operation machines the part's face. A Ø40 mm end mill is used. Machining is performed using the hatch technology, rough removing 0.5 mm from the stock top face. The fillet option is used to finish the corners.

- **External contour machining (F_contour)**

  This operation machines the external contour of the part. A Ø16 mm end mill is used. The material is removed in three step down cuts, and an additional finish pass is made to remove the remaining 0.3 mm from the wall.

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

  These three operations machine the pocket and the islands in this part. The first operation creates the large island in the part using a Ø16 mm end mill. The open pocket technology is used with Use profile strategy and Approach from outside options, while 0.2 mm are left on the walls of the island for a finishing pass.

  The second operation machines the islands surface to create nubs. The same tool is used, and wall finish is applied to the nubs.

  The third operation machines a pocket inside the island, a Ø10 mm end mill is used and a helical ramping lead in is chosen to penetrate the surface.

- **Round corners rest machining (P_contour2_1; F_contour_1)**

  The first operation machines the corners of the pocket using a Ø3 mm end mill. The rest material option is selected to machine only the material left on the corners from the larger tool.
  The second operation machines only material left on the corners of the profile using the same Ø3 mm end mill.

- **Chamfer machining (F_F7)**

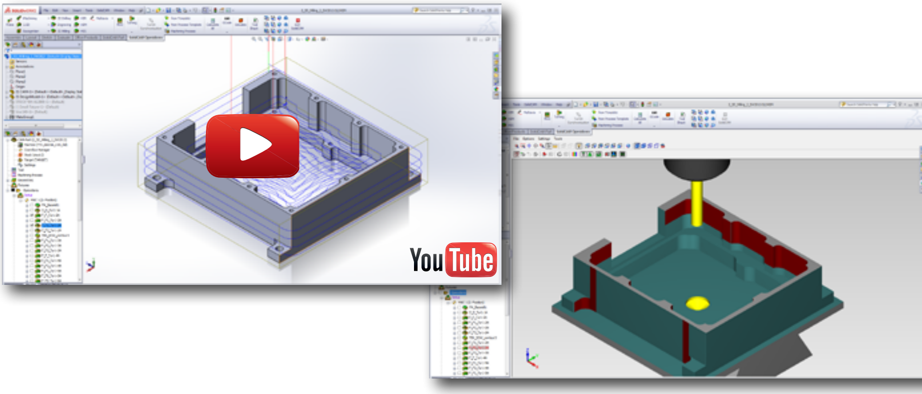
  To create a chamfer on the sharp edges left from previous operations, a profile operation is used with a Ø8 mm chamfer drill.
  The chamfer option is selected from Rest material/Chamfer section.

- **Slot machining (TSlot_T_Slot)**

  This slot milling operation machines a side groove. A Ø36 mm slot mill is used. Finish passes are made on the groove floor, ceiling and walls.

- **Holes machining (D_D - D_D2_1)**

  These drilling operations perform center drilling and drilling of the holes located on the upper surface of the part.




Open SolidCAM Part: **2_5D_Milling_2.prz**

Download the recording

This example illustrates the use of SolidCAM 2.5D Milling to machine the part shown above. The machining is performed on a 3-axis CNC machine in two setups, using two SolidCAM Coordinate Systems.

The following SolidCAM operations are created to perform the machining:

- **Upper faces machining (FM_facemill1)**

  The upper face of the part is machined in only one pass using a Ø100 mm face mill.

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

  To remove material from the part, a hole is drilled using a Ø10 mm drill (D_D_Tur1-1A) to make penetrating the surface easier for the rough mill used later. Two pocket operations (P_F2_Tur1-2A and P_F3_Tur1-2A) are used to machine the pocket in the part, working in a Contour strategy and using a Ø10 mm rough mill.

- **External contour machining (F_F_Tur1-2A; F_F1_Tur1-2A)**

  These operations rough the external contour of the part leaving 0.2 mm on the walls. A Ø10 mm rough mill is used.

- **Slot machining (TBX_SOW_contour3)**

  Using ToolBox cycles, a spiral tool path is created to penetrate this slot effectively using the whole cutting length of the Ø10 mm rough mill.

- **Round corners machining (F_F1_Tur1-3A; F_F1_Tur1-5A)**

  The round corners on the external contour are machined with a Ø5 mm rough mill to remove remaining material that could not be machined with a larger tool. Rest machining strategy is selected from the Rest Material\Chamfer section. The second operation finishes the corners with the same tool.

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

  These are semi-finish and finish operations applied on the part external and internal surfaces using different end mills. A smaller end mill is used for finish.

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

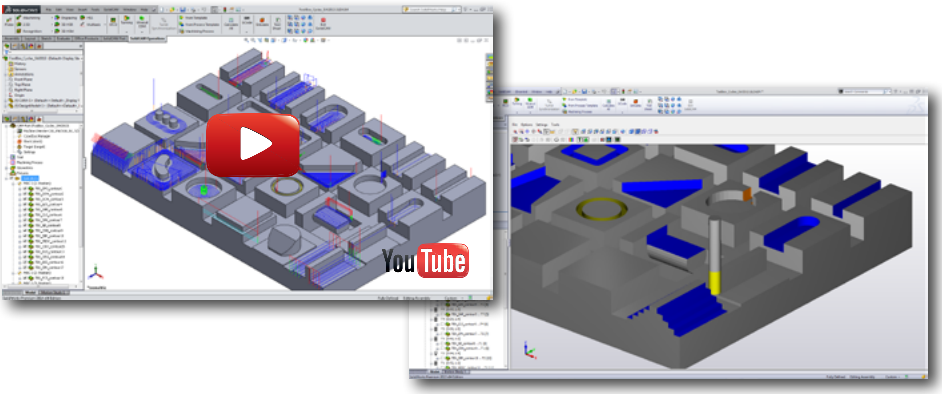
  These profile operations are created to machine a chamfer on the part edges using a Ø2 mm chamfer drill.

- **Holes machining (D_drill; D_drill1; D_drill2; D_drill; D_drill1; D_drill2)**

  These drilling operations perform center drilling and drilling of the holes located on the cover part faces.

- **Face and chamfer machining on the part lower surfaces (FM_facemill2; F_contour4)**

  The part is now rotated to a new setup and a Face milling operation is used to machine the part lower surface. Then, a profile operation is used to break the edges on the lower side of the part.



Open SolidCAM Part: **ToolBox_Cycles.prz**

Download the recording

This example illustrates the use of SolidCAM ToolBox cycles to machine the sample part shown above.

The following SolidCAM operations are created to perform the machining:

- **Closed slot machining (TBX_CLS_contour6)**

  In this cycle, machining of a closed slot with the width equal to the tool diameter is performed. The machining is done in the zigzag manner; the last cut is horizontal to clean the slot floor.

- **Spine slot machining (TBX_SPN_contour7)**

  SolidCAM automatically determines the spine of the slot and performs the machining according to the spine's shape.

- **Four nubs machining (TBX_NB_contour8)**

  In this cycle, the profile of a pocket is machined while preserving the rest of the material fixed to the wall using small thin bridges.

- **Simple corner machining (TBX_CRN_contour9)**

  In this cycle, machining of the open corner area is performed. The tool machines the corner in a number of cuts equidistant to the selected geometry.

- **Simple multi bosses machining (TBX_MBSC_contour11)**

  In this cycle, machining of several bosses is performed starting from one face. It is done in a number of equidistant Clear Offset passes parallel to the selected bosses geometry.

- **One side open slot machining (TBX_OSO_contour25)**

  In this cycle, machining of a one-sided slot is performed using two chains: a main chain and a drive chain. A spiral tool path is generated.

- **O-Ring machining (TBX_ORG_contour14)**

  In this cycle, you perform machining of slots having the O-ring shape. The geometry is defined as a pair of closed chains, the first one being an external chain and the second one – 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 step over or by angle parameters, ranging from minimal to maximal value.

- **Roll into open slot (TBX_ROS_contour21)**

  This cycle enables you to machine an open slot starting from the slot center to the walls, while rolling in the material with an arc movement.

- **Roll into closed slot (TBX_RCS_contour22)**

  This cycle enables you to machine a closed slot starting from the slot center to the walls, while rolling in the material with an arc movement.

- **Rib cut (TBX_RC_contour23)**

  This cycle enables you to machine a rib upper surface in a single pass shifted from the center line by a specified value. This is a safe method of cutting the upper face of a thin wall.

- **Thin wall machining (TBX_TWM_contour24)**

  This cycle enables you to machine thin walls removing gradually the excess material from either side of the wall. The strategy can be chosen from two options: Step support or Christmas tree tool path.

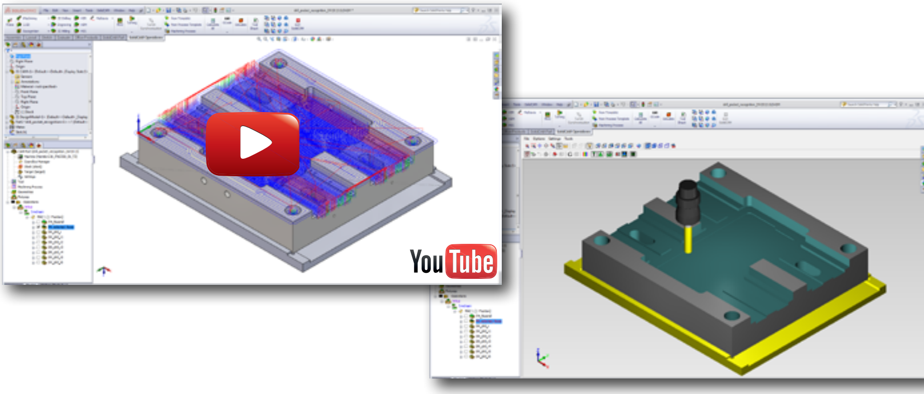
- **Wood cutting (TBX_SWC_contour20)**

  This cycle uses a special Saw tool to cut off a block of wood. The First Step down parameter defines the depth of cutting during the first pass. The second pass completes the cutting to the full depth.

- **Angled cylinder machining (TBX_AC_contour25)**

  In this cycle, the cylinder top is cut to create an angled surface. The cutting is performed by parallel cuts with a defined Step over.

# 2.5D Milling - Feature Recognition



Open Part: **Feature_Recognition.prz**

Download the recording

This example illustrates the use of SolidCAM's Automatic Feature Recognition to machine the mold base part shown above. The machining is performed on a 3-axis CNC machine.

The following SolidCAM operations are created to perform the machining:

- **Top face machining (FM_facemill)**

  This face milling operation performs the machining of the top face of the cover. A face mill of Ø40 mm is used.

- **Pockets machining (PR_selected_faces)**

  This pocket recognition operation automatically recognizes all the pocket areas in the model and performs their machining. An end mill of Ø20 mm is used. The open pocket machining is used to perform the approach movement from an automatically calculated point outside the material. The tool descends to the necessary depth outside the material and then moves horizontally into the material. A special machining strategy is applied to the through pockets; they are deepened in order to completely machine the pocket.

- **Center Drilling (DR_drill_r)**

  This drill recognition operation automatically recognizes all the hole features available for the machining with the current Coordinate System and performs the center drilling of all the holes in the mold base. A spot drill of Ø10 mm is used. The chamfer depth is applied automatically for each group of holes.

- **Drilling (DR_drill_r1; DR_drill_r2; DR_drill_r3; DR_drill_r4; DR_drill_r5; DR_drill_r6)**

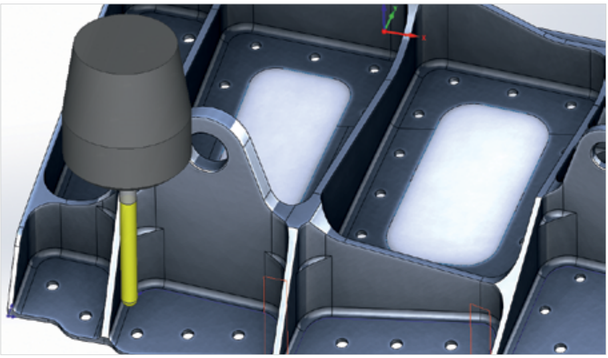
  These drill recognition operations perform the machining of all the hole features automatically recognized in the mold base. SolidCAM automatically recognized the upper level and drill depth from the model. The through holes are extended in order to completely machine the holes.

- **Chamfers machining (CHamfer_faces1)**

  This chamfer recognition operation performs machining of all the chamfer features automatically recognized in the mold base. A chamfer mill of Ø8 mm is used. The chamfer depth is set to 0.5 mm. The tool goes down and creates a chamfer cut on the part edges with a cutting diameter of 2 mm, maintaining a 3 mm safety offset from the walls.

## High Speed Surface Machining of Localized Surfaces - Important Module for Every Machine Shop!

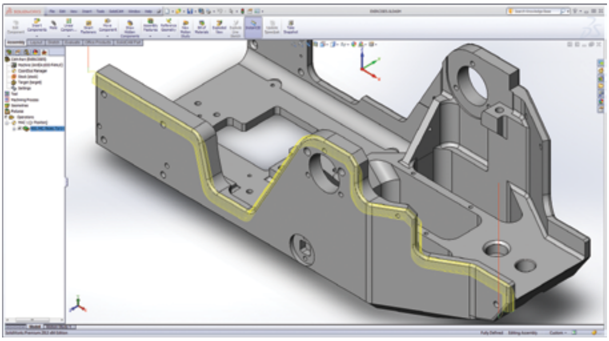
SolidCAM HSS is a high speed surface machining module for smooth and powerful machining of localized surface areas in the part, including undercuts. It provides easy selection of the surfaces to be machined, with no need to define the boundaries. It supports both standard and shaped tools.



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

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

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

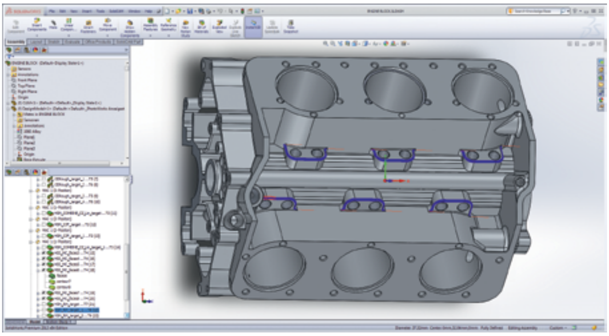


## Total Tool Control to Machine Only the Areas You Choose

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

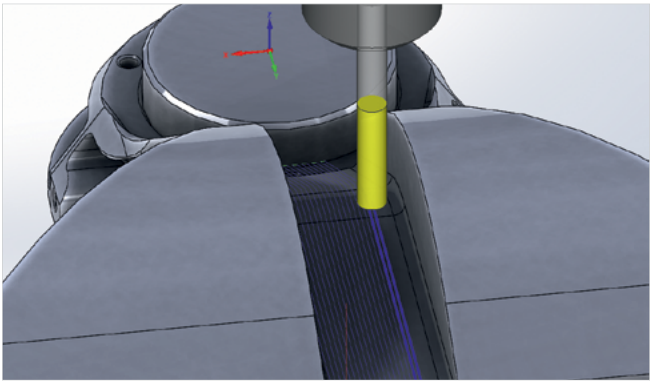
The HSS tool path is focused on single or multiple surfaces and excels in creating a flowing tool path on a group of surfaces that make up a complex 3D shape e.g. fillets.

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



## Advanced Gouge Control for Holder, Arbor and Tool

Complete Gouge Control is available for Holder, Arbor and Tool. Adjoining Check Surfaces that are to be avoided can be selected. Several retract strategies are available, under full user control.



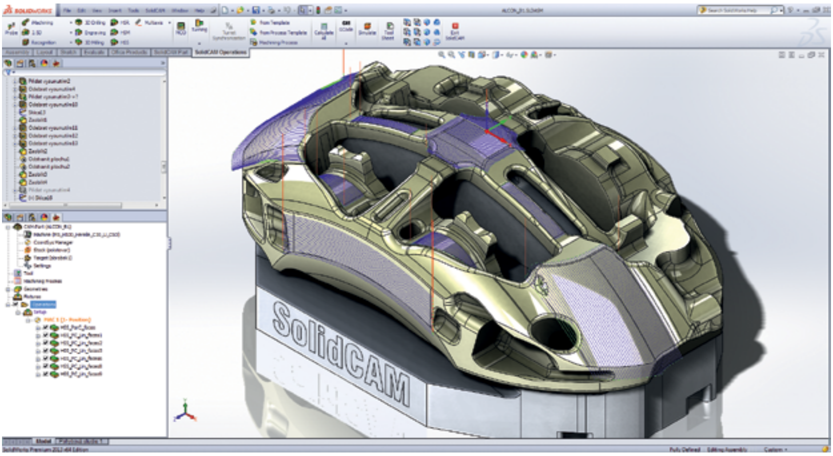
## Advanced Linking

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

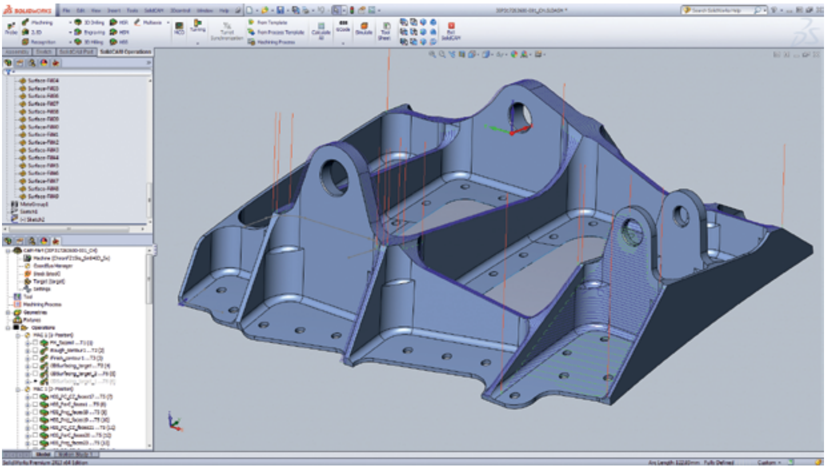
## Handling Undercut in HSS

Use Tapered, Lollipop, or T-Slot tools for undercuts or difficult to cut geometry.

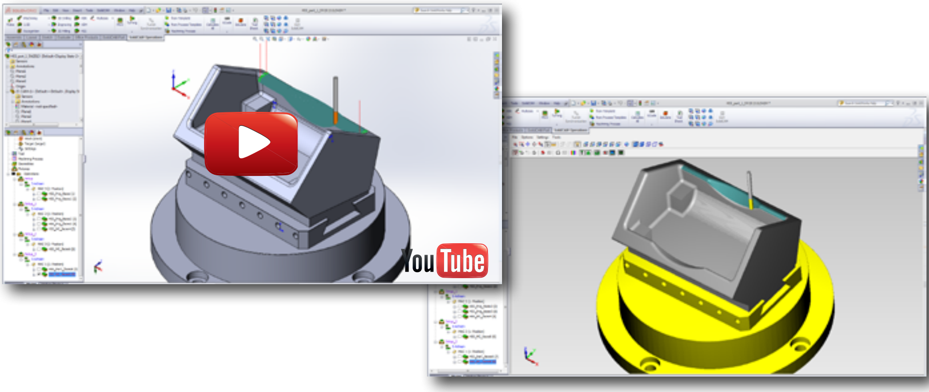
## Important Module for Every Machine Shop



The advantages of the SolidCAM HSS module translate to significantly increased surface quality. The SolidCAM HSS module is an important add-on for every machine shop for the machining of all types of parts.



# High Speed Surface Machining (HSS)



Open SolidCAM Part: **HSS_1.prz**

Download the recording

This example illustrates the use of several SolidCAM High Speed Surface Machining (HSS) strategies to machine the base part shown above.

The following SolidCAM operations are created to perform the machining:

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

  These operations utilize the HSS Projection strategy to perform the machining of four fillet areas. A ball nose mill of Ø10 mm is used with a corner radius of 5 mm to fit the corresponding surface radius. The depth cut option is used to machine the whole depth in several cutting passes.

- **Morphing machining (HSS_MC_faces4; HSS_MC_faces6)**

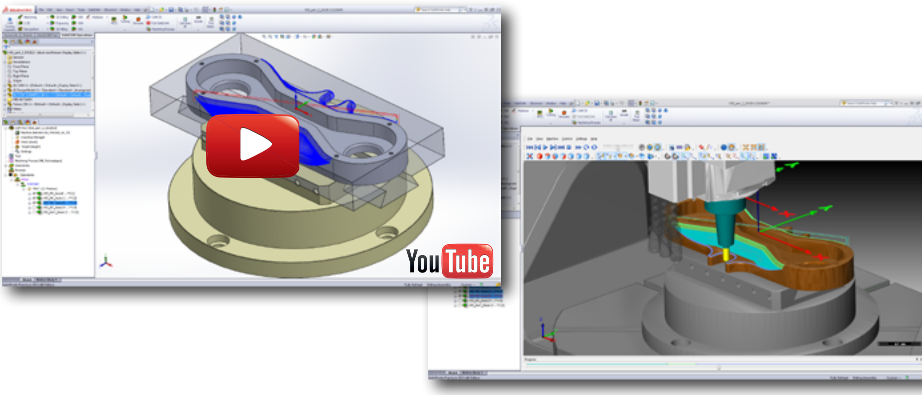
  This operation performs the machining of two internal fillet areas using the morph between two boundary curves strategy. This strategy is utilized to generate the tool path evenly distributed between the fillet boundaries. The gouge checking strategy is used to avoid possible gouges between the tool and the faces of the machining area. A ball nose mill of Ø10 mm is used.

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

  This operation performs the machining of the part bottom face. With this strategy, SolidCAM enables you to perform the machining of faces with cutting passes parallel to the selected curve. In this case, SolidCAM generates a pocket-style tool path enclosed within the boundaries of the selected face. An end mill of Ø10 mm is used.

- **Morphing between two curves (HSS_MC_faces9)**

  This operation performs the machining of the external fillet and an inclined face adjacent to the fillet. The Morphing between two boundary curves strategy is utilized to generate the tool path evenly distributed between the fillet boundaries. The tool path is generated using the scallop of 0.004 mm in order to obtain excellent surface quality. The gouge checking strategy is used to avoid possible gouges between the tool and the faces of the machining area. A ball nose mill of Ø6 mm is used.



Open SolidCAM Part: **HSS_2.prz**

Download the recording

This example illustrates the use of several SolidCAM High Speed Surface Machining (HSS) strategies to machine the base part shown above.

The following SolidCAM operations are created to perform the machining:

- **Morphing between surfaces (HSS_MS_faces8)**

  This operation performs machining of two external side areas using the Morph between two adjacent surfaces strategy. This strategy is utilized to generate the tool path evenly distributed between the fillet surfaces. A ball nose mill of Ø10 mm is used.

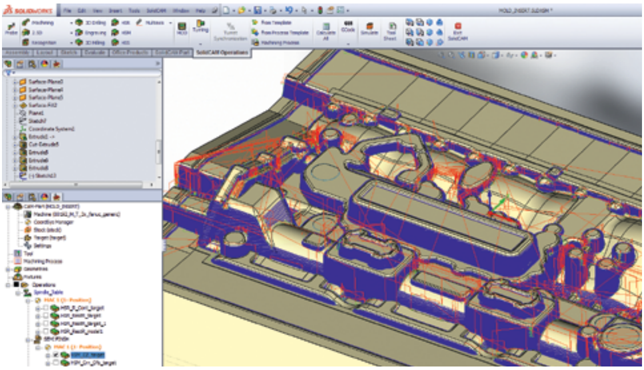
- **Morphing between curves (HSS_MC_faces11, HSS_MC_faces26, HSS_MC_faces24)**

  These operations perform machining of the external upper and bottom fillets and chamfers on the internal surfaces. The Morphing between two boundary curves strategy is utilized to generate the tool path evenly distributed between the boundaries. The gouge checking strategy is used to avoid possible gouges between the tool and the faces of the machining area. A ball nose mill of Ø10 mm is used.

- **Parallel to curves machining (HSS_ParC_faces13)**

  This operation performs machining of the part external face. With this strategy, SolidCAM enables you to perform machining of the face with cutting passes parallel to the selected curve. In this case, SolidCAM generates a tool path closing the large gaps between the edges by the Direct linking with a tangent arc.

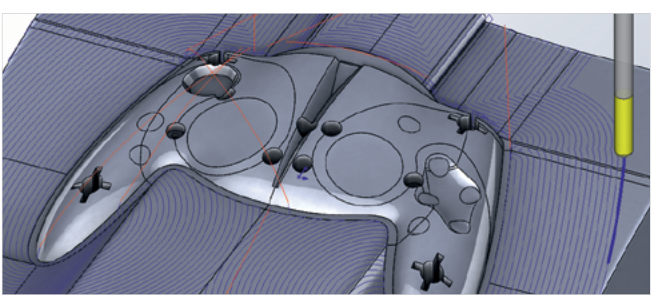
## Powerful Roughing and Finest Finish Tool paths Available for 3D Machining!



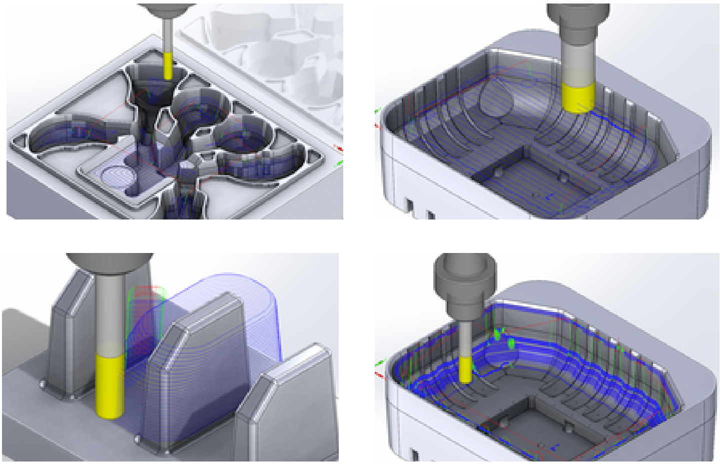
Experience 3D machining taken to an entirely new level of smoothness, efficiency and smart machining, with the finest finish tool paths available anywhere for 3D machining.

SolidCAM's HSR/HSM module is a very powerful and market-proven 3D high-speed machining module for complex 3D parts, aerospace parts, molds, tools and dies.

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



## HSR - High Speed Roughing



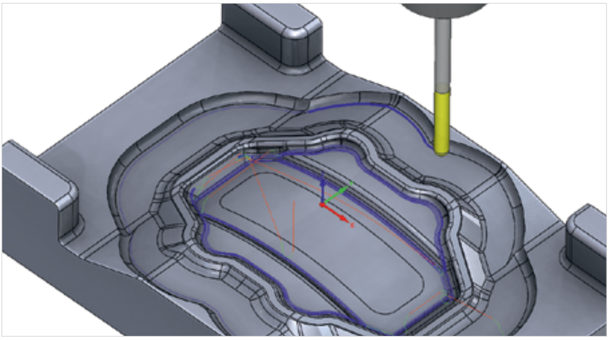
SolidCAM HSR provides powerful high-speed roughing strategies including contour, hatch, hybrid rib-roughing and rest roughing.

## HSM - High Speed Finishing

With SolidCAM's HSM module, retractions to high Z levels are kept to a minimum. Angled where possible and smoothed by arcs, retracts do not go any higher than necessary, minimizing air cutting and reducing machining time.

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

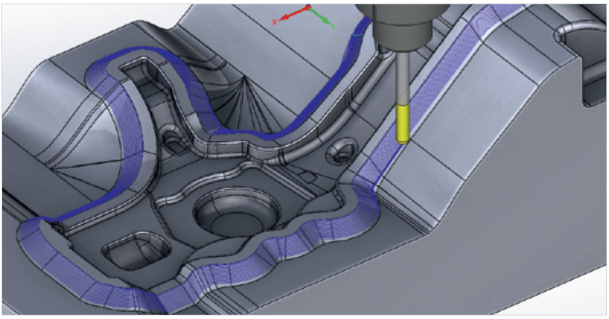
With demands for ever-shorter lead and production times, lower costs and improved quality, High Speed Machining is a must in today's machine shops.



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

Any HSM 3D machining strategy can be controlled by specifying the surface slope angle to be machined or by specifying the machining boundary.

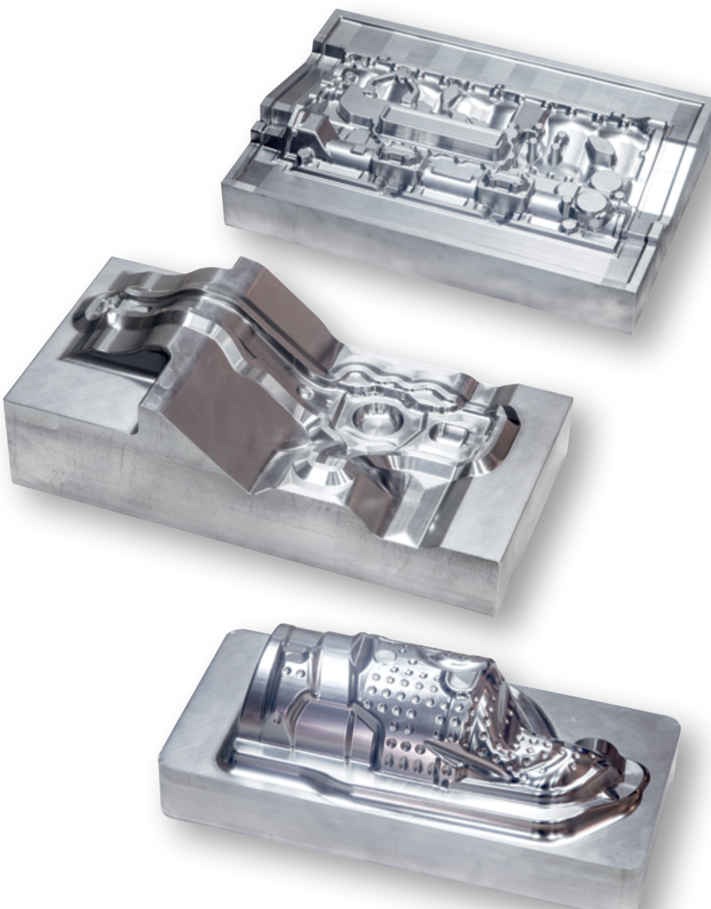
A comprehensive set of boundary creation tools is provided, including silhouette boundaries, cutter contact area boundaries, shallow boundaries, rest area boundaries, rest boundaries and user defined boundaries.

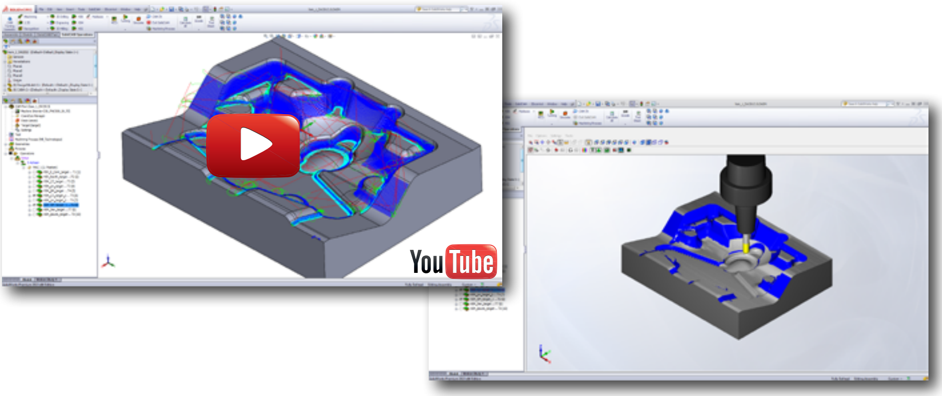


## HSM - 3D Machining to the Highest Level

SolidCAM 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 CNC's with reduced air cutting and smoothing arcs that maintain continuous tool motion.

Let us show you how HSM takes 3D Machining performance to the highest level – all with your current machines.






Open SolidCAM Part: **HSM_1.prz**

Download the recording

This example illustrates the use of several HSR and HSM strategies to machine the mold cavity shown above.

The following SolidCAM operations are created to perform the machining:

- **Rough machining (HSR_HMP_target)**

  The HM Roughing operation performs contour roughing of the cavity. An end mill of Ø16 and corner radius of 0.8 mm is used with a Step down of 1.5 mm. To machine the part's outside and inside areas, the HM spiral option is used as Step over type. Also, the Step down type is chosen as Constant & flats. A machining allowance of 0.5 mm remains unmachined for further semi-finish and finish operations.

- **Rest roughing (HSR_RestR_target)**

  This operation performs rest roughing of the cavity. A bull nose tool of Ø8 and corner radius of 2 mm is used with a Step down of 1 mm to remove the steps left after the roughing. The same machining allowance as in the previous roughing operation is used.

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

  This operation performs Constant Z semi-finishing of the steep faces (from 40° to 90°). A ball nose tool of Ø10 is used for the operation. The machining allowance of 0.25 mm remain unmachined for further finish operations. The Apply fillets option is used to add virtual fillets that will smooth the tool path at the corners.

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

  This operation performs Linear semi-finishing of the shallow faces (from 0° to 42°). The ball nose tool of Ø10 is used for the operation. The machining allowance of 0.25 mm remain unmachined for further finish operations. The Apply fillets option is used.

- **Corners rest machining (HSM_RM_target)**

  This operation uses the Rest Machining strategy for semi-finishing of the mold cavity corners. The semi-finishing of the model corners enables you to avoid tool overload in the corner areas during further finishing. A ball nose tool of Ø6 is used for the operation. A virtual reference tool of Ø16 is used to determine the model corners where the rest machining is performed. The machining allowance of 0.25 mm remain unmachined for further finish operations.

- **Steep faces finishing (HSM_CZ_target_1)**

  This operation performs Constant Z finishing of the steep faces (from 40° to 90°). A ball nose tool of Ø8 is used for the operation. The Apply fillets option is used.

- **Shallow faces finishing (HSM_Lin_target_1)**

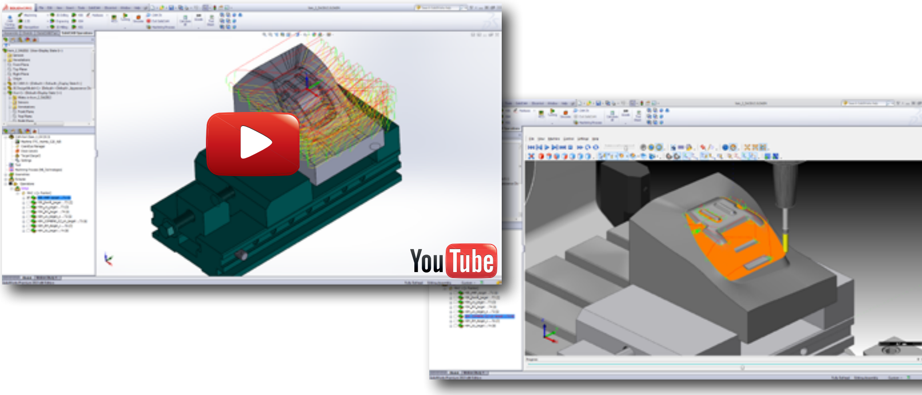
  This operation performs Linear finishing of the shallow faces (from 0° to 42°). A ball nose tool of Ø8 is used for the operation. The Apply fillets option is used.

- **Corners rest machining (HSM_RM_target_1)**

  This operation uses the Rest Machining strategy for finishing of the model corners. A ball nose tool of Ø4 is used for the operation. A virtual reference tool of Ø12 is used to determine the model corners where the rest machining is performed.

- **Chamfering (HSM_Bound_target)**

  This operation uses the Boundary Machining strategy for the chamfering of upper model edges. A chamfer drill tool is used for the operation. The chamfer is defined by the external offset of the drive boundary and by the Axial thickness parameter.



Open SolidCAM Part: **HSM_2.prz**

Download the recording

This example illustrates the use of several HSR and HSM strategies to machine the mold insert shown above.

The following SolidCAM operations are created to perform the machining:

- **Rough machining (HSR_HMP_target)**

  This operation performs the HM roughing of the part using the HM spiral step over. An end mill of Ø20 is used with a Step down of 5 mm to perform the roughing. A machining allowance of 0.3 mm remains unmachined for further semi-finish and finish operations.

- **Rest roughing (HSR_RestR_target)**

  This operation performs the rest roughing of the part. A ball nose tool of Ø12 and corner radius of 6 mm is used with a Step down of 2 mm to remove the steps left after the roughing. The same machining allowance as in the roughing operation is used.

- **Semi-finishing (HSM_Lin_target)**

  This operation performs Linear semi-finishing of the part. The ball nose tool of Ø10 is used with a Step down of 2 mm. The machining allowance of 0.1 mm remain unmachined for further finish operations. The Apply fillets option is used.

- **Corner rest machining (HSM_RM_target)**

  This operation uses the Rest Machining strategy for semi-finishing of the mold insert corners. The semi-finishing of the model corners enables you to avoid tool overload in the corner areas during further finishing. A ball nose tool of Ø6 is used for the operation. A virtual reference tool of Ø12 is used to determine the model corners where the rest machining is performed. The machining allowance of 0.1 mm remain unmachined for further finish operations.

- **Linear finishing (HSM_Lin_target_1)**

  This operation performs Linear finishing. A ball nose tool of Ø6 is used for the operation. The Apply fillets option is used. The Constraint boundaries are created manually to allow working on a separate area.

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

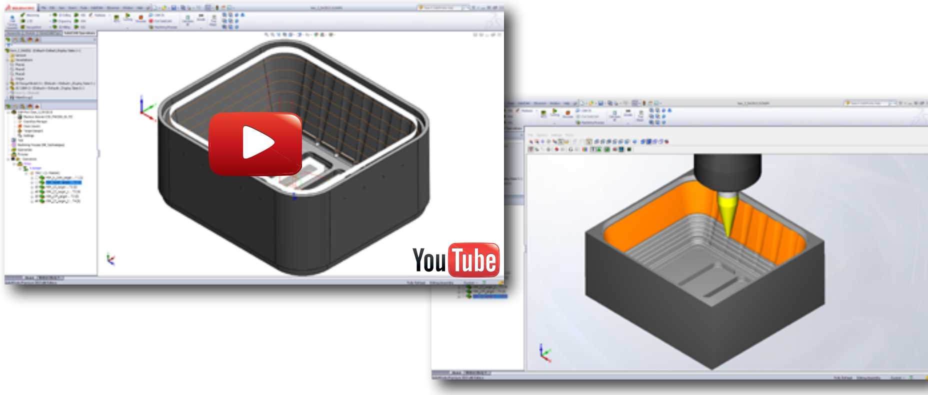
  This operation performs Constant Z finishing of steep areas (from 35° to 90°) and Linear finishing of shallow areas (from 0° to 38°). A ball nose tool of Ø6 is used for the operation. The Apply fillets option is used. The angle of linear passes is defined by the value of 90°.

- **Corner rest machining (HSM_RM_target1)**

  This operation uses the Rest Machining strategy for finishing of the corners. A ball nose tool of Ø4 is used for the operation. A virtual reference tool of Ø8.2 is used to determine the model corners where the rest machining is performed.

- **Concave areas machining (HSM_CS_target)**

  This operation performs Constant step over finishing of the concave model faces down to a certain depth. A ball nose tool of Ø4 and corner radius of 2 mm is used with the constant step over of 0.05 mm in horizontal and vertical directions.



Open SolidCAM Part: **HSM_3.prz**

Download the recording

This example illustrates the use of several HSR and HSM strategies to machine the electronic box shown above.

The following SolidCAM operations are created to perform the machining:

- **Rough machining (HSR_HMP_target)**

  The HM Roughing operation performs the contour roughing of the part. An end mill of Ø30 is used with a Step down of 10 mm to perform the roughing. To machine the internal part of the model, the Cavity option is used as Step over type. A machining allowance of 0.5 mm remains unmachined for further semi-finish and finish operations.

- **Rest roughing (HSR_RestR_target)**

  This operation performs the rest roughing of the part. A bull nose tool of Ø16 and corner radius of 1 mm is used with a Step down of 5 mm to remove the steps left after the roughing. The same machining allowance as in the roughing operation is used.

- **Upper faces machining (HSM_CZ_target)**

  This operation performs Constant Z finishing of the upper vertical model faces up to a certain depth. A bull nose tool of Ø12 and corner radius of 0.5 mm is used.

- **Bottom faces machining (HSM_CZ_target_1)**

  This operation performs Constant Z finishing of the bottom vertical model faces. A bull nose tool of Ø12 and corner radius of 0.5 mm is used.

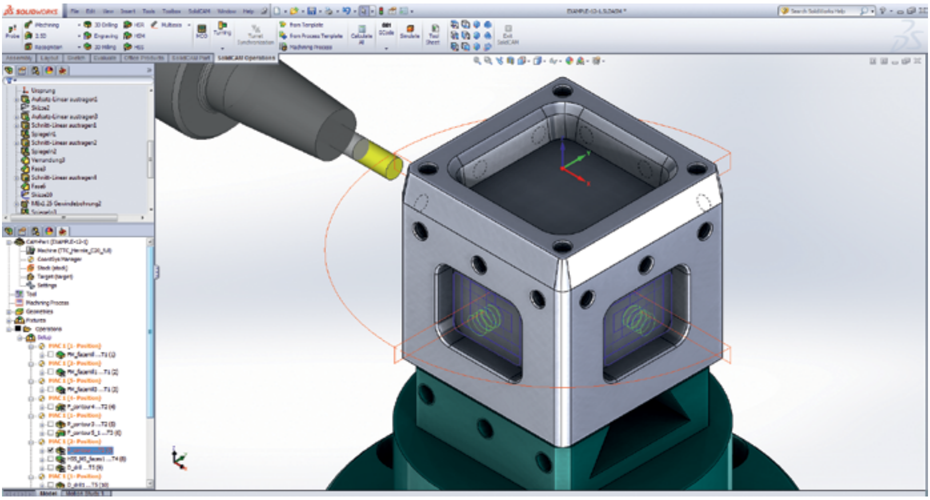
- **Flat faces machining (HSM_CZF_target)**

  This operation performs Horizontal Machining of the flat faces. A bull nose tool of Ø12 and corner radius of 0.5 mm is used.

- **Inclined faces machining (HSM_CZ_target_2)**

  This operation performs Constant Z Machining of the inclined faces. A taper mill of 12° angle is used to perform the machining of the inclined face with large stepdown (10 mm). Using such a tool enables you to increase the productivity of the operation.

## Powerful SolidCAM Indexial, Multi-Sided Machining - Easiest Coordinate System Definition!



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

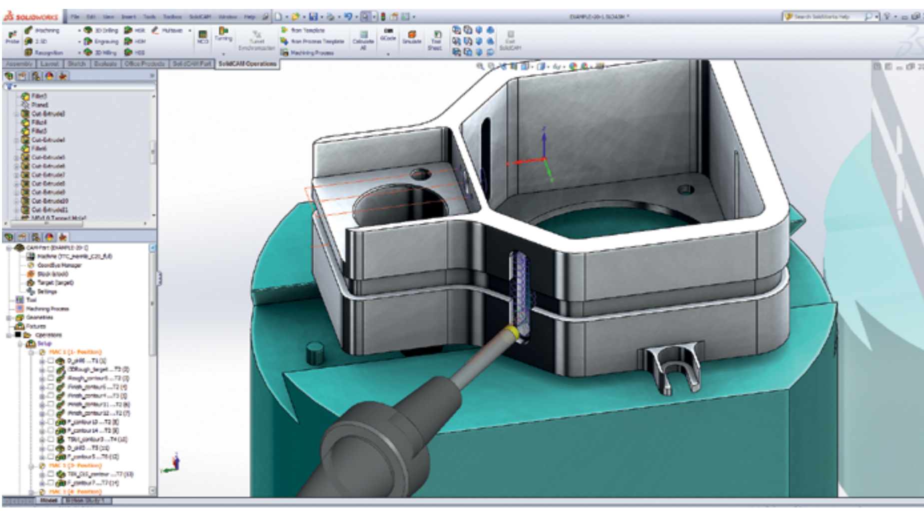
SolidCAM provides an effective and easy way to program on multiple sides of a part. SolidCAM is exceptionally strong in indexial 4/5-axis machining.

### Easiest Coordinate System Definition for Indexial 5-Axis!

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?

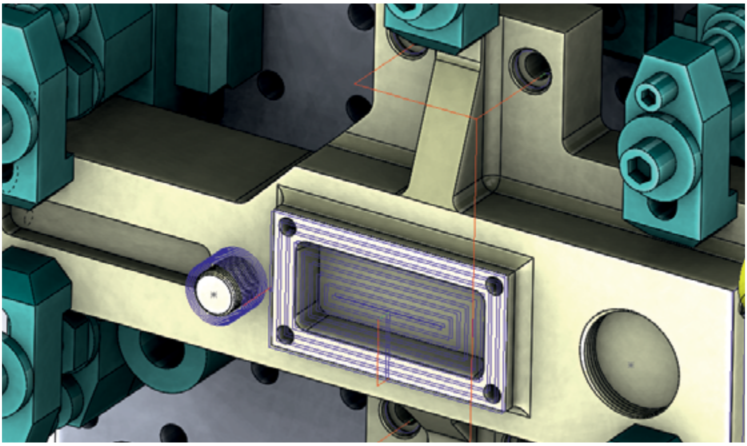
Experience single machine home position with One-click orientations for indexed setups - SolidCAM speeds up multi-sided machining by eliminating multiple coordinate system constructions. Define a CoordSys on the fly, by just picking a face, and continue programming your part.

- SolidCAM's "select a face and machine" is the fastest approach to indexial programming
- Our coordinate system manager keeps track of all necessary data for each tool orientation
- Solid Verify simulation shows tool holders and fixtures, together with material removal for all machining operations

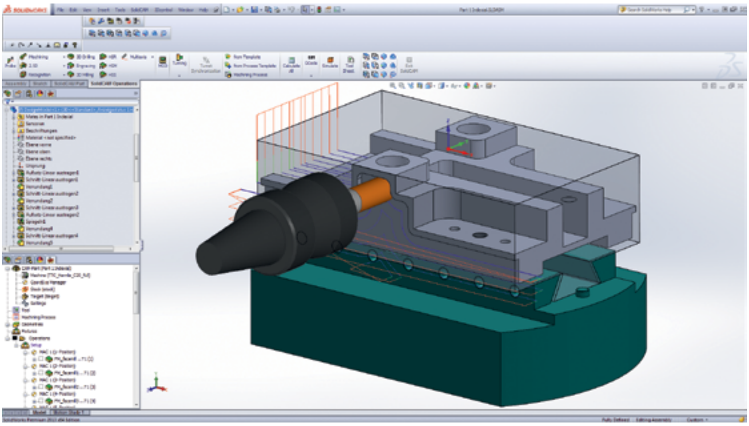


## Efficient, Edit-free GCode for Multi-Axis Machines

SolidCAM offers multiple options to get efficient GCode for multi-axis machines.

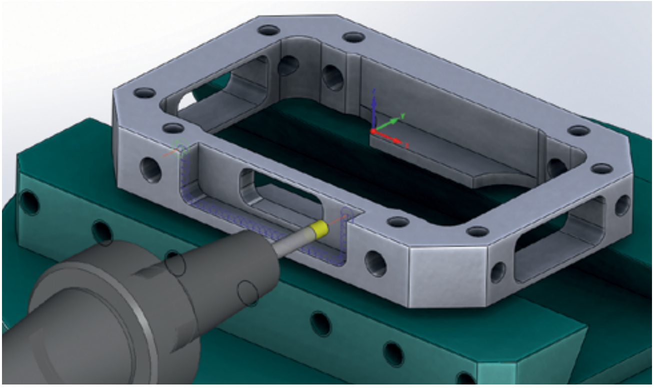


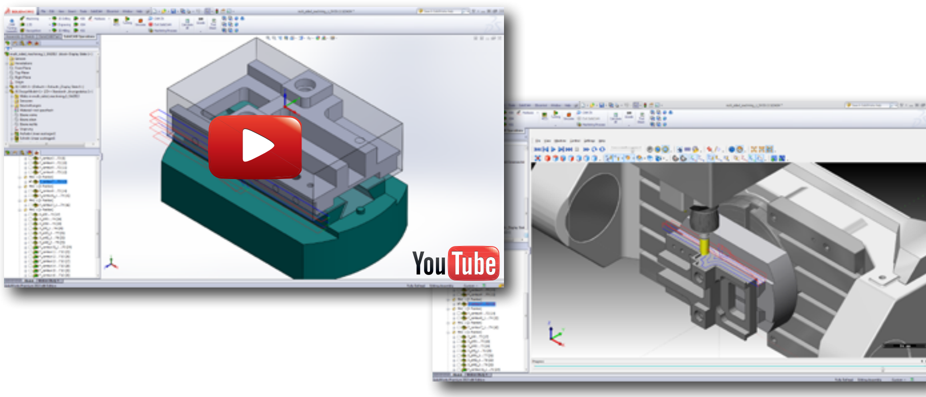
SolidCAM's post processor can be setup to handle all rotations and work offset shifting, to eliminate the need for setting up multiple work offsets at the machine. Whether your controller can calculate part rotations internally or it needs the post processor to handle rotations, SolidCAM has this covered.



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

Our philosophy to indexial milling is simple: from software to GCode - make the process for indexial milling the same as for single sided milling. No need for any special functions or tricks inside the software to machine multi-sided parts - it should just work!






Open SolidCAM Part: **Multi_Sided_Machining_1.prz** Download the recording

This example illustrates the use of SolidCAM Multi-sided machining to complete the machine fixture shown above, using a 5-axis CNC Machine.

The following SolidCAM operations are created to perform the machining:

- **Top face machining (FM_facemill)**

  This Face Milling operation machines the top face of the fixture. Machine Coordinate system #1 (Position #1) is used for the operation.

- **Side face machining (FM_facemill1)**

  This Face Milling operation machines the side face of the fixture. Machine Coordinate system #1 (Position #2) is used for the operation.

- **Other faces machining (FM_facemill2, FM_facemill3, FM_facemill4)**

  These Face Milling operations machine the rest of the faces: front, back, and side. Machine Coordinate system #1 (Positions #3, 4, 5) is used for the operations.

- **Profile rough and finish machining (F_contour, F_contour_1)**

  These Profile operations machine the step located on the top face of the fixture using a Ø20 mm end mill with 10 mm step down and clearing the offset and then making the final pass and removing the floor and wall offsets.

- **Opening machining (P_contour2)**

  This Pocket operation machines the open pocket located on the top face, using the Open pocket strategy with the Use profile strategy option. Machine Coordinate system #1 (Position #1) is used for the operation.

- **Opening machining (P_contour3)**

  This Pocket operation machines the closed pocket located on the top face of the fixture, using the Contour strategy. Machine Coordinate system #1 (Position #1) is used for the operation.

- **Openings machining (P_contour4, P_contour5, P_contour6)**

  These Pocket operations machine the rest of pockets located on the top face of the fixture, using strategies suitable for open and closed pockets. Machine Coordinate system #1 (Position #1) is used for the operations.

- **Side pockets machining (P_contour7, P_contour8)**

  These Pocket operations machine an open pocket located on one side face of the fixture and then another pocket located on the opposite side. Machine Coordinate system #1 (Positions #2, 3) is used for the operations.

- **Side pockets rest machining (P_contour8_1, P_contour7_1)**

  These Pocket operations perform rest machining of the pockets made in two previous operations cleaning the corners with a smaller tool. Machine Coordinate system #1 (Positions #2, 3) is used for the operations.

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

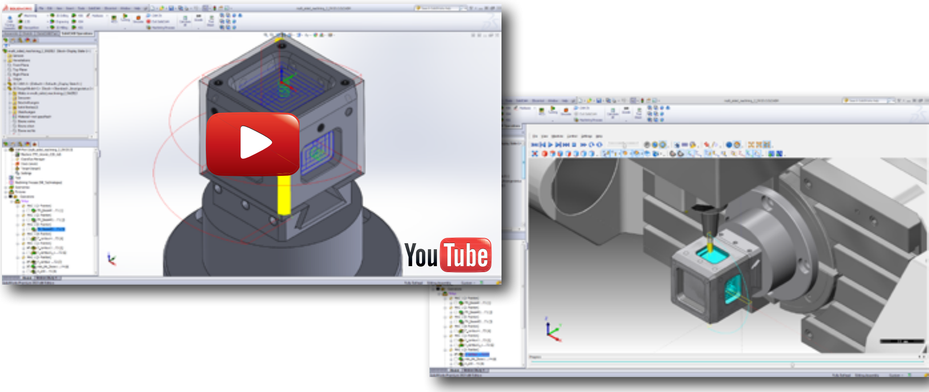
  These drilling operations perform center drilling, drilling, and tapping of the holes located on the top face of the part.

- **Hole machining (F_contour15_1)**

  This Profile operation completes machining of the holes located on the top face.

- **Chamfers machining (F_contour11–F_contour18)**

  These Profile operations break the edges of the different part faces. Machine Coordinate system #1 (Positions #1–5) is used for the operation.



Open SolidCAM Part: **Multi_Sided_Machining_2.prz**

Download the recording

This example illustrates the use of SolidCAM indexial 5-axis machining to complete the machine fixture shown above.

The following SolidCAM operations are created to perform the machining:

- **Top face machining (FM_facemill)**

  This Face Milling operation machines the top face of the fixture. Machine Coordinate system #1 (Position #1) is used for the operation.

- **Side faces machining (FM_facemill1)**

  This Face Milling operation machines the side faces of the fixture. Machine Coordinate system #1 (Position #2) is used for the operation. The Transform option is used to create a circular pattern of operations around the revolution axis in order to machine all side faces.

- **Inclined faces machining (FM_facemill3)**

  This Face Milling operation machines the inclined faces of the fixture. Machine Coordinate system #1 (Position #5) is used for the operation. The Transform option is used to create a circular pattern of operations around the revolution axis in order to machine all side faces.

- **Side edges machining (F_contour4)**

  This Profile operation machines the inclined faces of the fixture. Machine Coordinate system #1 (Position #5) is used for the operation. The Transform option is used to create a circular pattern of operations around the revolution axis in order to machine the side surfaces.

- **Top pocket machining (P_contour3, F_contour5_1)**

  The Pocket operation machines the closed pocket located on the top face, using the Contour strategy. Machine Coordinate system #1 (Position #1) is used for the operation.
  The Profile operation finishes machining of the pocket. A taper mill of 10° angle is used to perform the machining of the inclined face with 3 mm step down.

- **Side faces machining (P_contour, HSS_MS_faces1, D_drill)**

  These operations machine the side faces of the fixture in Machine Coordinate system #1 (Position #2), while the Transform option creates a circular pattern of operations around the revolution axis.
  The first operation is a pocket operation that machines closed pockets on the central part of each surface.
  The second operation is created to machine the radii on the edges using the Morph between two adjacent surfaces strategy. A Ø6 mm ball end mill is used.
  The third operation is a drilling operation that uses a Ø10 mm spot drill with a 90° angle.

- **Top holes drilling (D_drill1, D_drill1_1)**

  These operations perform drilling of the holes on the top surface in Machine Coordinate system #1 (Position #1). The first operation uses a Ø10 mm spot drill with a 90° angle. The second operation uses a Ø6.8 mm drill.

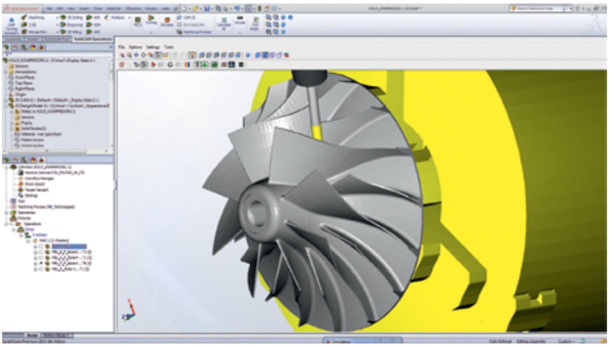
- **Side holes drilling (D_drill_1, D_drill_2)**

  These operations perform drilling of the holes on the side surface in Machine Coordinate system #1 (Position #2). The first operation uses a Ø6.8 mm drill. The second operation uses a Ø8 mm tap tool with a pitch of 1.25 mm.

- **Chamfers machining (F_contour5, F_contour1)**

  The first operation is created to machine a chamfer on the top surface edges. Machine Coordinate system #1 (Position #1) is used for the operation. A Ø12 mm chamfer drill is used, and **Chamfer** is selected in the Rest material\Chamfer section. The second operation machines chamfers on the side surface edges. Machine Coordinate system #1 (Position #2) is used for the operation. The same chamfer drill is used. The Transform option creates similar chamfers on all four sides.

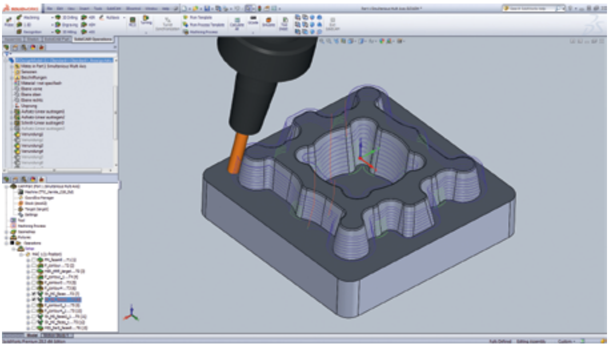
## Powerful Simultaneous 5-Axis Tool paths with a Very Friendly User Interface



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

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

SolidCAM 5-Axis machining supports all 5-Axis machine tools including Table/Table, Table/Head and Head/Head gantry machines as well as the latest Mill-Turn machining centers.



## Flexibility and Control

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

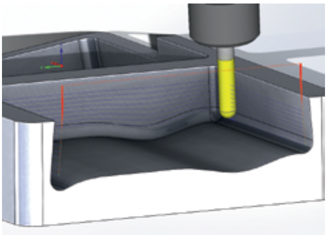
Link and approach moves are fully gouge protected and different strategies may be used depending on the distance of the link move. SolidCAM also provides options for control over lead/lag and side tilt angles to give complete control over the final tool path.

## Collision Avoidance for Tool and Holder

Collision Avoidance is supported for both the tool and holder, and a range of alternatives is offered to avoid collision. The machine simulation provides complete Cutter and Tool-holder gouge checking.

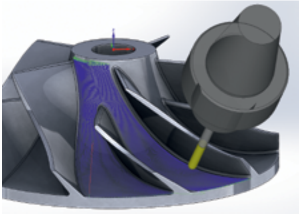
## SWARF Machining

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



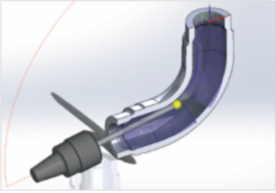
## Multi-Blade Machining

The Multi-blade machining operation easily handles impellers and bladed disks, with multiple strategies to efficiently rough and finish each part of these complex shapes. Multi-bladed parts are used in many industries and this operation is specifically designed to generate the necessary tool paths for the different multi-blade configurations.



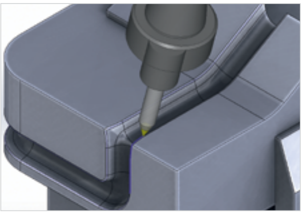
## Port Machining

The Port machining operation is an easy-to-use method for machining ports with tapered lollipop tools, and has collision checks for the entire tool. It provides both roughing and finishing tool paths to make ports from castings or billet.



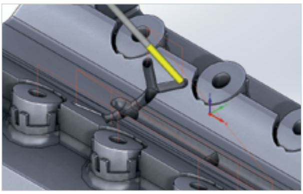
## Contour 5-Axis Machining

The Contour 5-axis machining strategy 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.



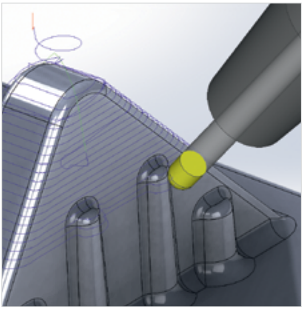
## Multi-Axis Drilling

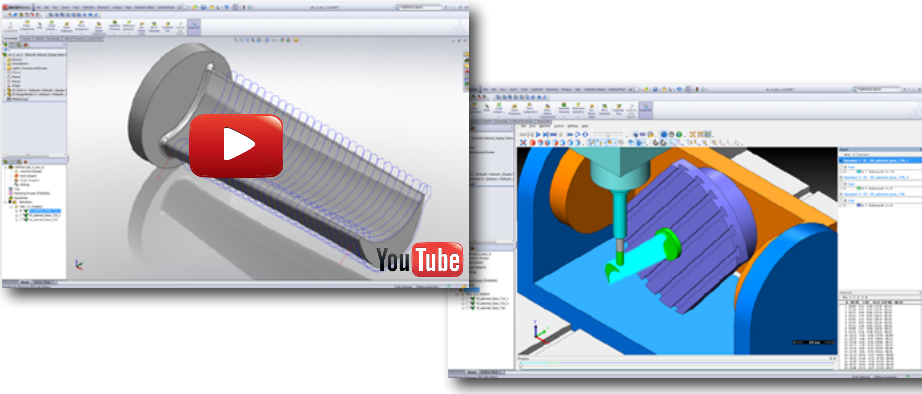
The Multi-axis Drilling operation uses SolidCAM's automatic hole recognition and then performs drilling, tapping or boring cycles, at any hole direction easily and quickly. All the advanced linking, tilting and collision avoidance strategies are available in this operation.



## Convert HSM to Sim. 5-Axis

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






Open SolidCAM Part: **Sim_5_axis_1.prz**

Download the recording

This example illustrates the use of the SolidCAM multiaxis simultaneous machining module for turbine blade machining.

The following Sim. 5-axis operations are used to perform the semi-finish and finish machining of the turbine blade:

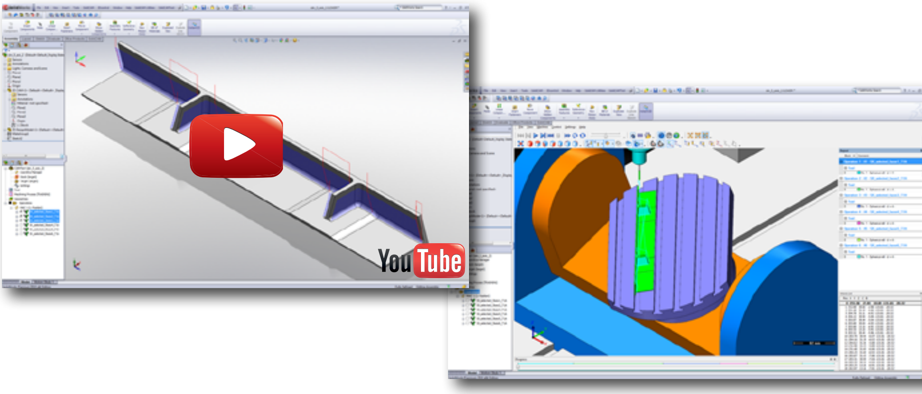
- **Blade Semi-finishing (5X_selected_faces; 5X_selected_faces_1)**

  The first operation provides the semi-finish of the turbine blade using a bull nose tool of Ø16 with a corner radius of 4 mm. The Parallel Cuts strategy is used to perform the spiral machining of the blade.
  The tool tilting is defined using the Tilted relative to cutting direction option with lag angle of 20°. The tool contact point is defined at the front tool face. This combination of parameters enables you to perform the machining by the trochoidal surface of the tool.
  Gouge checking is performed to avoid the possible collisions of the tool with the planar surface of the blade base. The remaining material will be machined at a later stage using a special tilting strategy.

  The second Sim. 5-axis operation provides semi-finishing of the blade area, close to the blade base. This area was not machined in the previous operation because of the gouge protection. A bull nose tool of Ø8 with a corner radius of 2 mm is used for the operation.
  The tool tilting is defined using the Tilted relative to cutting direction option with a lag angle of 20°. In addition to the lag angle, a side tilting angle of 10° is defined to avoid the gouging of the planar face of the blade base.

- **Blade finishing (5X_selected_faces_2)**

  This operation performs the finishing of the blade. A bull nose tool of Ø8 with a corner radius of 2.5 mm is used for the operation.
  The tool tilting is defined using the Tilted relative to cutting direction option with a lag angle of 20°. In addition to the lag angle, a side tilting angle of 10° is defined to avoid the gouging of the planar face of the blade base.




Open SolidCAM Part: **Sim_5_axis_2.prz**

Download the recording

This example illustrates the use of the Sim. 5-axis operation for an aerospace part machining.

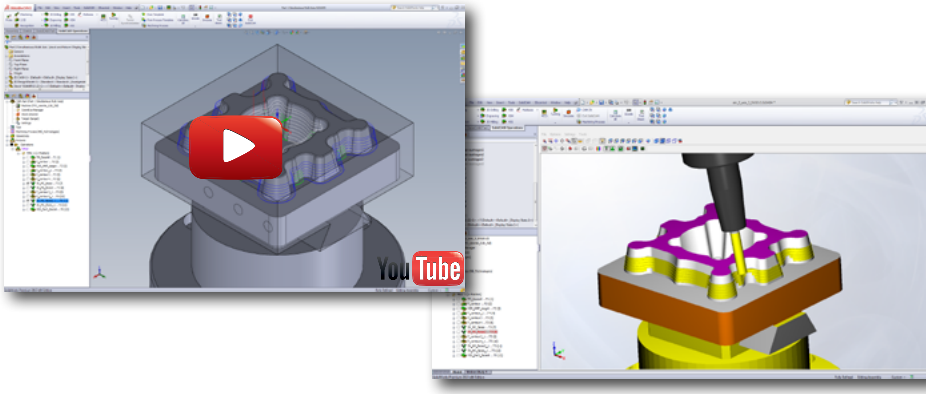
A number of Sim. 5-axis operations are defined in order to perform the finish machining of the inclined faces of the aerospace frame and their adjacent fillets. The inclined faces are forming an undercut area that cannot be machined using 3-axis milling; 5-axis milling has to be used with the appropriate tilting strategy, to machine the inclined faces.

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

  These operations perform the finish machining of the inclined walls.
  A ball nose tool of Ø4 is used for the operation.
  The Parallel Cuts strategy is used to generate a number of cuts parallel to the XY plane of the coordinate system.
  The tool tilting is defined using the Tilted relative to cutting direction option with a tilt angle of 90°. These parameters enable you to perform the machining with the side face of the tool.

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

  These operations perform the finish machining of the fillets adjacent to the walls.
  A ball nose tool of Ø4 is used for the operation.
  The Projection (User-defined) strategy is used to generate a single pencil milling pass, machining the fillets.
  The Tilted through curve tilting strategy is used to perform a smooth transition between different tool axis orientations.



Open SolidCAM Part: **Sim_5_axis_3.prz**

Download the recording

This example illustrates the use of the Sim. 5-axis operation for a machine part.

A number of Sim. 5-axis operations are defined in order to perform machining of the inclined internal and external walls of the part. The inclined faces cannot be machined using 3-axis milling; to machine them, 5-axis milling has to be used with the appropriate tilting strategy.

- **Rough machining (FM_Facemill, F_contour, HSR_HMP_target, F_contour_1, P_contour3, P_contour4)**

  These operations perform the rough machining of the part: face milling, profile machining, HM roughing, open and closed pocket machining.

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

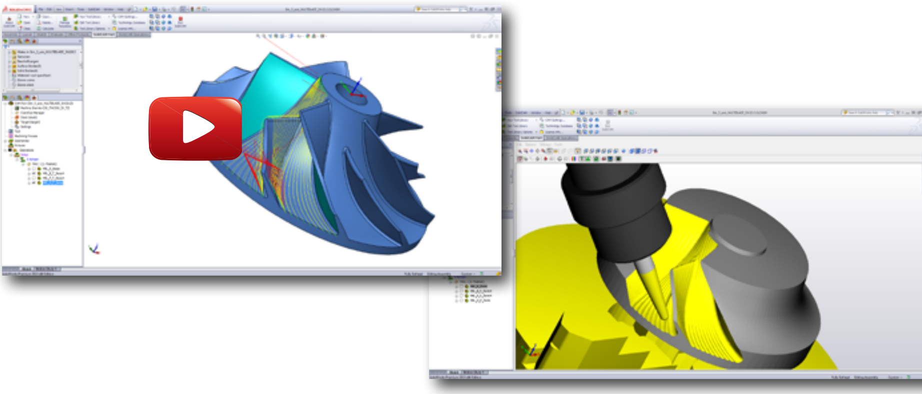
  This operation performs semi-finish machining of the internal inclined walls.
  An end mill tool of Ø8 is used for the operation.
  The Morph between two boundary curves strategy is used to generate a morphed tool path between two leading curves.
  The Drive surface offset is set to 0.2 mm to allow finish machining on a later stage.
  The tool tilting is defined using the Tilted relative to cutting direction option when the value of tilt angle at side of cutting is 90°. These parameters enable you to perform the machining with the side face of the tool.
  Gouge checking options are used with a check surface defined on the floor of the pocket.

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

  This operation performs finish machining of the external inclined walls.
  An end mill tool of Ø8 is used for the operation.
  The Morph between two adjacent surfaces strategy is used to generate a morphed tool path on a drive surface enclosed by two check surfaces.
  The tool tilting is defined using the Tilted relative to cutting direction option when the value of tilt angle at side of cutting is 90°. These parameters enable you to perform the machining with the side face of the tool.
  Gouge checking options are used with a check surface defined on the bottom surface of the part.

- **External and internal walls finishing (5X_MS_faces2_1, 5X_MC_faces_1)**

  These operations perform finish machining of the external and internal inclined walls. Two previous 5-axis operations are copied, and the Drive surface offset parameter is set to 0.
  The tool tilting is defined using the Tilted relative to cutting direction option when the value of tilt angle at side of cutting is 90°. These parameters enable you to perform the machining with the side face of the tool.
  Gouge checking options are used with a check surface defined on the bottom surface of the part.



Open Part: **Sim_5_axis_Multiblade.prz**

Download the recording

This example illustrates the use of the Multiblade Machining operation for machining surfaces of a part with multiple blades.

A number of Multiblade machining strategies are used in order to perform roughing and finishing machining of the various part surfaces.

- **Roughing (MBL_R_faces)**

  This operation performs the rough machining of external surfaces of the part. A taper mill tool of Ø12 mm is used for the operation. The maximum number of layers is 10, and the maximum number of slices is 10. The tool tilting is applied with preferred lead angle of 5°, while the maximum lead angle is set to 45°. The tool path is calculated for the area between two blades, gradually cleaning off the material by layers and slices.

- **Blade finishing (MBL_B_F_faces4)**

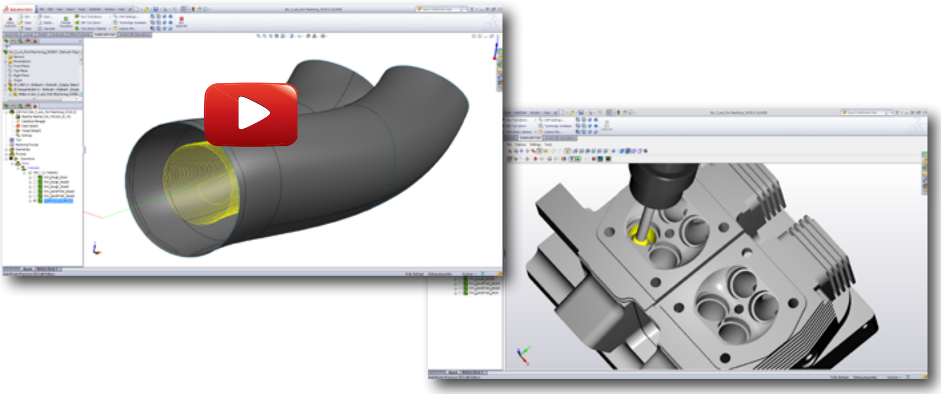
  This operation performs finishing machining of blades. The tool goes over the blade on a full contour trimming the trailing edges and removes maximum 15 layers.

- **Fillet finishing (MBL_F_F_faces4)**

  This operation performs finishing machining of fillets. The tapered mill tool works on full contour trimming the trailing edges. The blade side area is defined by the number of cuts, which is set to 8, and the number of cuts on the hub side is set to 5. The side step for both sides is 0.3.

- **Floor finishing (MBL_H_F_faces)**

  This Multiblade operation performs finishing machining of the hub area. The tapered mill tool cuts a maximum number of 18 slices on the surface. The tool axis control options allow to set the tool tilting to achieve a smooth tool path along all machined surfaces.



Open Part: **Sim_5_axis_Port_Machining.prz**

Download the recording

This example illustrates the use of the Port Machining operation for machining internal surfaces of intakes and curved tubes.

A number of Port machining strategies are used in order to perform roughing and finishing machining of the various internal surfaces.

- **Rough machining (Port_Rough_faces)**

  This operation performs the rough machining of the bottom part of the port. A lollipop tool of Ø16 mm with corner radius of 8 mm is used for the operation. The maximum step over is set to 5 mm, and the step down is 2 mm. The tool path is calculated for the top area up until the Maximum from bottom distance.

- **Rough machining (Port_Rough_faces8, Port_Rough_faces9)**

  These operations perform the rough machining of the top part of each tube. A lollipop tool of Ø16 mm with corner radius of 8 mm is used for the operation. The maximum step over is set to 3 mm, and the step down is 2 mm. The tool path is calculated for the top area down until the Maximum from top distance.

- **Finishing machining (Port_SpiralFinish_faces9, Port_SpiralFinish_faces8)**

  These operations perform finishing spiral cuts on the internal top surfaces. A lollipop tool of Ø16 mm with corner radius of 8 mm is used for the operation. The maximum step over is set to 1 mm. The tool path is calculated for the top area down until the Maximum from top distance. Gouge checking option is used with check surfaces defined on the second side of the port.

- **Finishing machining (Port_SpiralFinish_faces)**

  This operation performs finishing spiral cuts on the internal bottom surfaces. A lollipop tool of Ø16 mm with corner radius of 8 mm is used for the operation. The tool path is calculated for the bottom area up until the Maximum from bottom distance. Gouge checking option is used with check surfaces defined on the opposite side of the port.

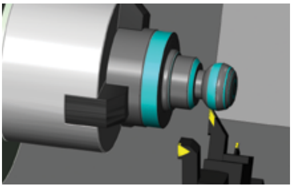
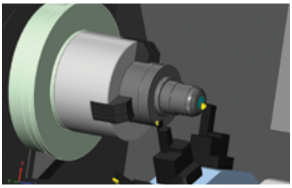
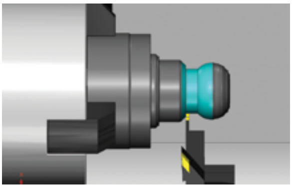
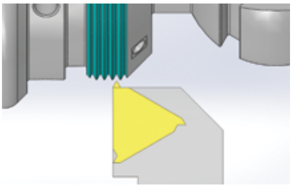
## SolidCAM Module for Fast and Efficient Turning

SolidCAM provides a most comprehensive turning package with powerful tool paths and techniques for fast and efficient turning.

SolidCAM Turning provides functionality for a wide range of machine tools, including 2-Axis lathes, multi-turret configurations, sub-spindle turning centres and mill-turn machines.

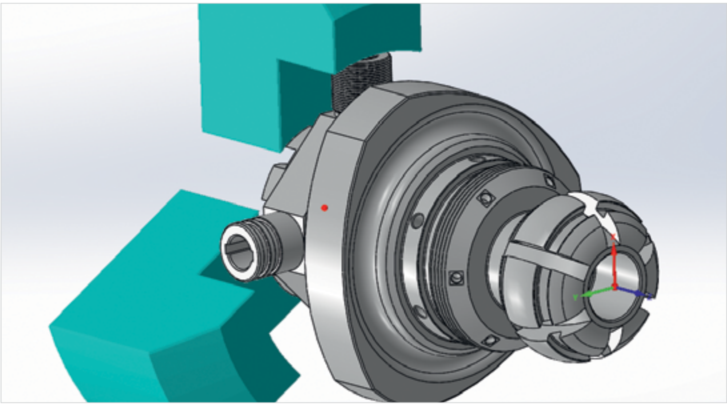
On a mill-turn machine, C-, Y- and B-Axis milling and drilling take place within the same program as the turning, providing a fully integrated and associative programming solution.

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

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

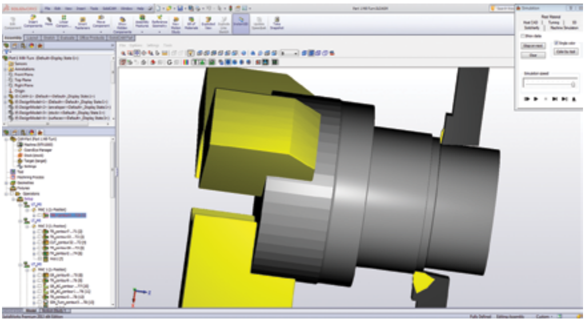
Standard fixture libraries are available and specialized fixtures can be added.



## Updated Stock

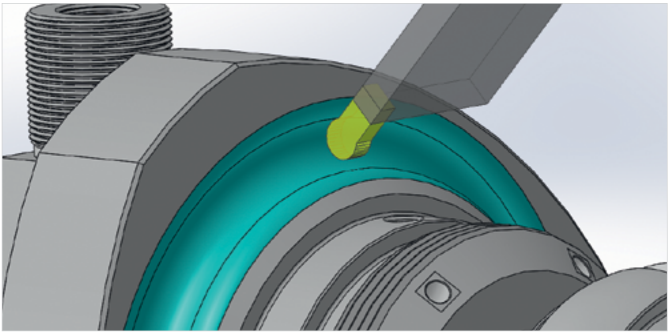
SolidCAM has the ability to keep the stock updated live within the operations tree. Updated stock is supported from the most basic 2-axis turning center, right through to a CYB multi-turret, sub-spindle Mill- Turn CNC machine.

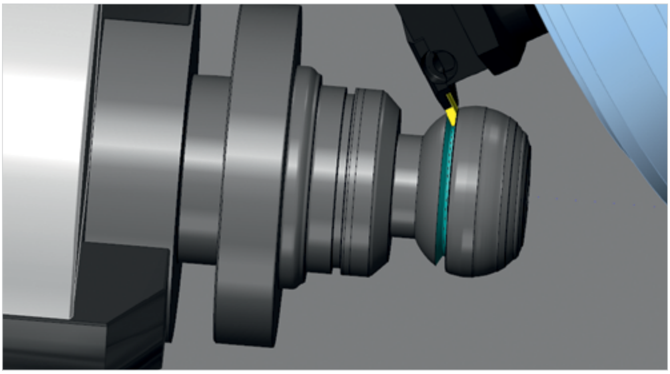
On a sub-spindle turning center, when a component is transferred from the main spindle to the sub spindle, the updated stock transfers with it. Any subsequent machining on the sub-spindle will detect the stock in the state that it left the main spindle, ultimately providing the most efficient machining sequence possible.

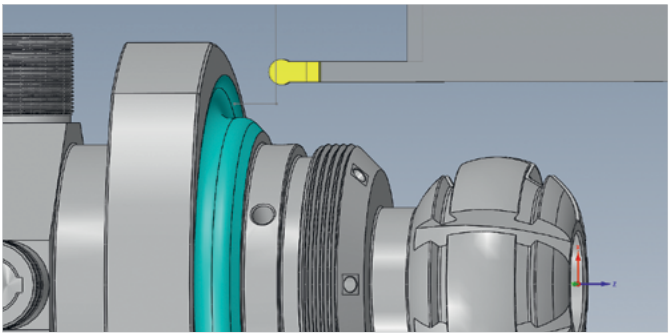


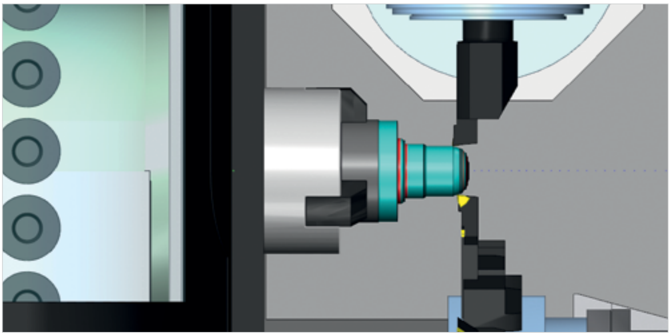
## Advanced Turning Operations

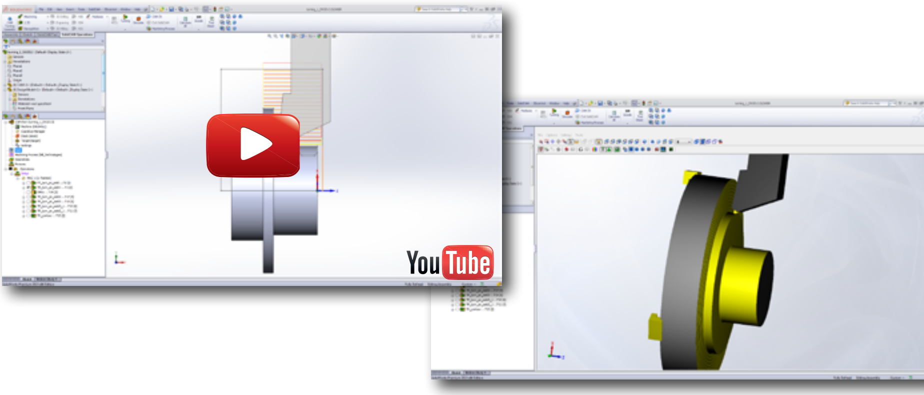
- Balanced Roughing: two tools working simultaneously, or in trailing mode, to perform roughing turning of long and large parts
- Angled Grooving: performs internal or external inclined grooves, at any defined angle
- Manual Turning: performs turning according to user-defined geometry, regardless of stock and target model
- 4-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
- Multi-Turret Synchronization: powerful capability to synchronize multiple turret operations along a machining time line












Open SolidCAM Part: **Turning_1.prz**

Download the recording

This example illustrates the use of the SolidCAM Turning for the machining of the flange shown above.

The following Turning operations are used to perform the machining of the part:

- **Face Turning (FT_turn_on_solid)**

  This operation is used to generate tool path for front face machining. An External roughing tool is used for the operation. The offset of 0.1 mm is left on the face for further operations.

- **Contour Turning (TR_turn_on_solid1)**

  This operation is used to generate the tool path for the external faces roughing. An External roughing tool is used for the operation. The Rough option is selected as a Work type for this operation to perform machining in a number of equidistant passes.

- **Drilling (DRILL)**

  This Drill operation is used to drill a hole in the center of the part. A Ø23 mm flat drill is used for this operation.

- **Internal Turning (TR_turn_on_solid2)**

  To rough the internal faces of the part and create chamfers on the hole, an internal turning operations is used along with an internal roughing tool. The Non-descending motions strategy is used here to avoid collisions.

- **Internal Grooving (TR_turn_on_solid3)**

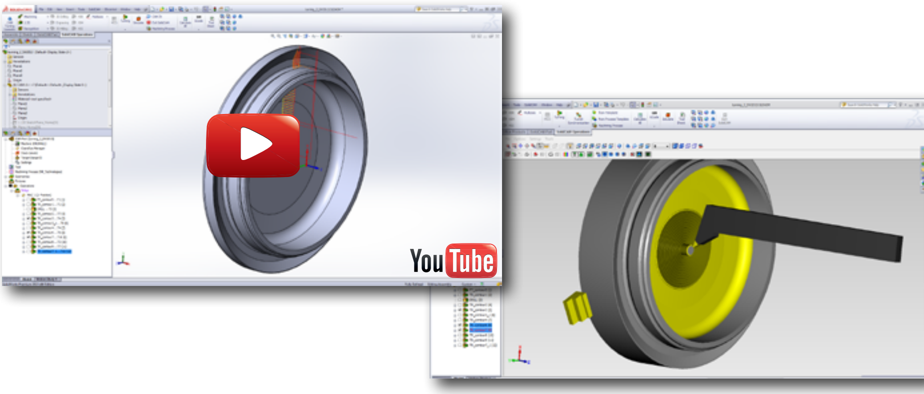
  This turning operation is used to rough the part's internal groove. An Internal grooving tool is used for the operation. Roughing type is set to Smooth for a smooth finish.

- **Internal finishing (TR_turn_on_solid3_1, TR_turn_on_solid2_1)**

  These operations are used to finish the internal surfaces of the part using internal grooving tools. The Work type is set to Finish only to perform only finishing cuts.

- **Internal Threading (TH_profile6)**

  This Threading operation is used to machine an internal thread in the part with a major diameter of 28.7 mm and pitch of 16 TPI (thread per inch). An Internal threading tool is used for the operation.




Open SolidCAM Part: **Turning_2.prz**

Download the recording

This example illustrates SolidCAM's functionality for rest material machining, during longitudinal and facial rough/finish turning operations, performed on the part shown above.

The following Turning operations are used to perform the machining of the part:

- **Face turning (FT_contour5)**

  This operation is used to rough the part's face using an External roughing tool. To perform machining on the face of the part, Face front is selected as the Mode. The Rough work type is selected for this operation to rough the part in a number of equidistant passes.

- **Contour turning (TR_contour1)**

  This turning operation is used to rough the external contour of the part. The Mode is set to Long external to create passes along the part's rotational axis.

- **Drilling (DRILL)**

  This drilling operation is used to make machining the internal contour easier and more efficient. A Ø33 mm drill is used.

- **Internal roughing (TR_contour2)**

  This operation performs roughing of the internal contour of the part using an internal roughing tool. This operation uses Smooth as a rough type to create a smoother surface.

- **External contour rest machining and finishing (TR_contour3; TR_contour3_1, TR_contour4)**

  These external turning operations finish the contour roughed in the Contour turning operation. The external roughing tool with additional angle is used in the first operation to remove the material left unmachined. In the next two operations, the Work type is set to Finish only. The third external finishing operation is performed on the selected contour using Smooth as the roughing type to create a smoother surface.

- **Removing internal rest material (TR_contour6; TR_contour7)**

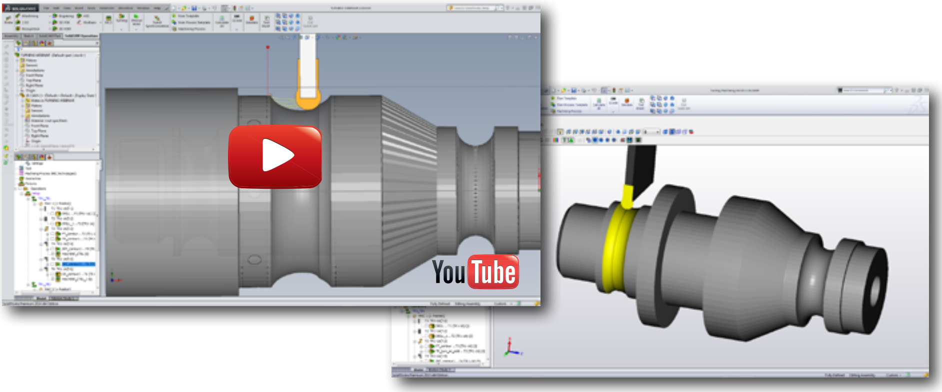
  The first operation is used to remove the material left from internal roughing operation. To reach a particular location, the tool is mounted in a different orientation.

- **External contour finishing (TR_contour8)**

  This operation is used to remove the material left from external roughing operation.

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

  The first operation is used to remove the material left from internal roughing operation. To reach a particular location, the tool is mounted in a different orientation. Similarly, the other operation are used to finish the remaining surface.




Open SolidCAM Part: **Turning_iMachining.prz**

Download the recording

This example illustrates using iMachining Turning operation for roughing separate segments of the part shown above.

The following Turning operations are used to perform the machining of the part:

- **Front face drilling (DRILL, DRILL_1)**

  These drilling operations are used to drill a hole in the center of the part.

- **Front face turning (FT_contour)**

  This turning operation is used to rough the part's face using an External roughing tool. The Mode is set to Front to create passes along the part's front face.

- **Contour turning (TR_turn_on_solid)**

  This turning operation is used to rough the external contour of the part. The Mode is set to Long external to create passes along the part's rotational axis.

- **Grooving (IMT_contour1)**

  This iMachining turning operation is used to rough the external groove. This operation can use only a groove tool with a round insert. The Mode is set to Long external, and the Cutting angles are chosen in the range from 30° to 45°.

- **Rough grooving (IMT_contour2)**

  This iMachining turning operation is used to rough the external groove. It uses the same groove tool with a round insert as in the previous operation. The Mode is set to Long external, and the Cutting angles are chosen in the range from 30° to 45°. When the tool descends to the bottom of the groove, it cleans the corners performing iMachining movements in each corner separately.

- **Finish grooving (GR_contour2)**

  This grooving operation is used to complete rough grooving performed in the previous operation. The operation uses the same geometry to finish the corners with a regular grooving tool. The work type is set to Finish only, and the operation is performed on the Rest material only.

- **Back face drilling (DRILL_2, DRILL_3)**

  These drilling operations are used to drill a hole in the back face of the part.

- **Back face turning (FT_contour3)**

  This turning operation is used to rough the back face using an External roughing tool. The Mode is set to Back to create passes along the part's back face.

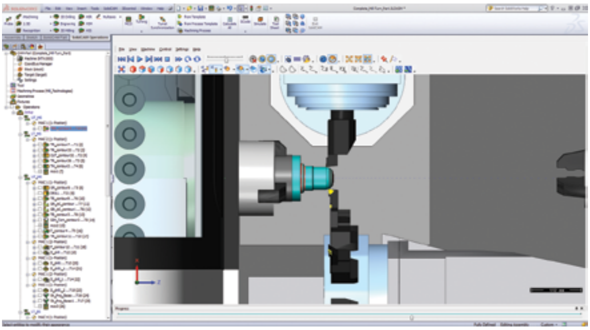
- **Contour turning (TR_contour4)**

  This turning operation is used to rough the external contour of the part. The Mode is set to Long external to create passes along the part's rotational axis.

- **Back side grooving (IMT_contour5)**

  This iMachining turning operation is used to rough another groove. This operation also uses the groove tool with a round insert. The Mode is set to Long external, and the Cutting angles are chosen in the range from 30° to 45°.

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



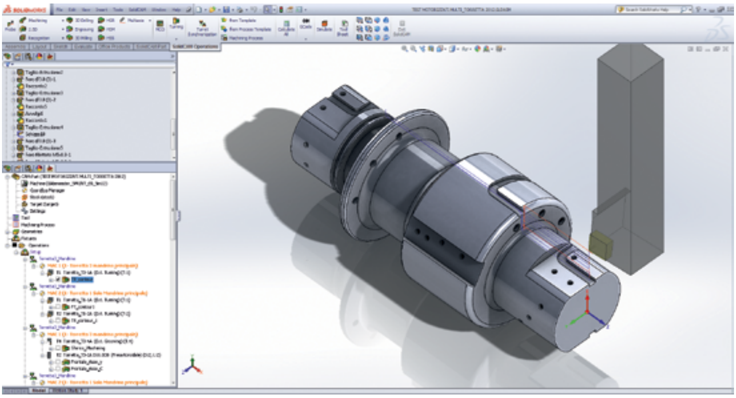
The fastest growing and most demanding class of CNC machines on the market today are multi-task machines that combine several capabilities into one machine – multiple spindles, multiple turrets, material being machined in multiple stages, transferring from spindle to spindle without handling, stock inserted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

4/5-Axis Simultaneous Mill-Turn machines have many uses and allow much more flexibility and capabilities not offered from other machine configurations. With this in mind, many of these have multi-axes, upper turrets, lower turrets, CYB and Sub Spindles.

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

- The use of tail stocks, steady rest, sub-spindles, rotary and linear turrets along with C-Axis, CY-Axis and B-Axis, are regular features on today's Mill-Turn machine tools. In this collision rich environment, the programming of these machines is made simple and safe by utilizing SolidCAM's turning and milling operations in a single environment.

- Support for multi-turret and multi-spindle programming, with turret synchronization and full machine simulation, is seamlessly integrated into one extremely powerful package.
- All SolidCAM milling and turning operations, including the powerful, revolutionary iMachining operations, are available for the programming of mill-turn machines. All ancillary devices can also be defined and taken into account for simulation and gouge checking.



## Easy Programming for Complex Mill-Turn CNC Machines

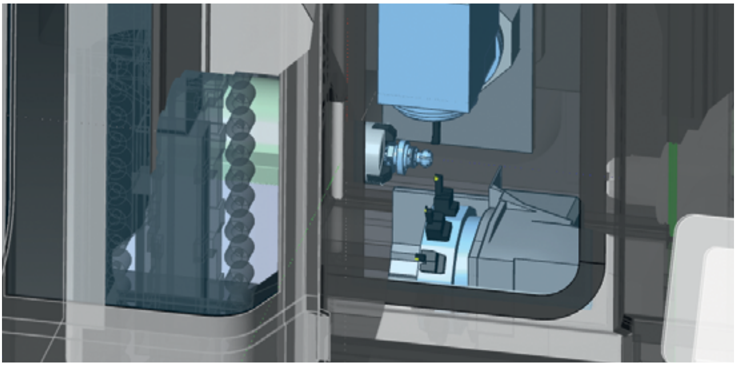
SolidCAM goes beyond just programming these complex machines, with intelligent management of rest material between Milling and Turning operations, for the most efficient tool paths and reduced cycle times, ensuring the highest possible productivity.

## Machine ID

Defines the CNC machine components and their kinematics, enabling users to setup and support the most complicated mill-turn machines easily and effectively.

## MCO (Machine Control Operation)

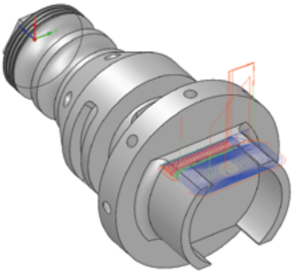
MCO (Machine Control Operation) enables the user to insert various control operations while manufacturing a part. These operations control the CNC machine and activate different options and devices such as: opening or closing fixtures, activating coolants, rotating part, moving part from one table to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



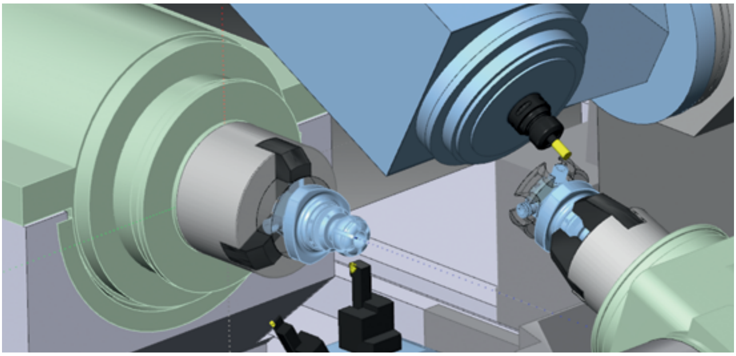
## Transfer Between Spindles

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

## iMachining in Mill-Turn



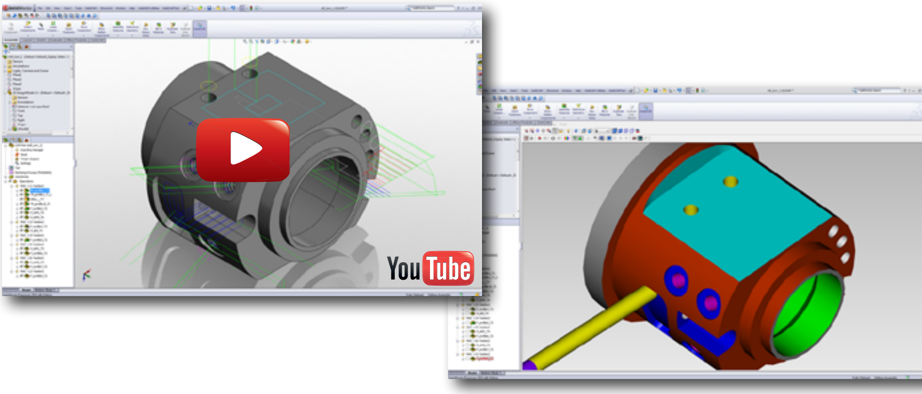
In a mill-turn part, using iMachining 2D & 3D saves you programming and cycle time. Additionally, iMachining has the very important advantage of exerting smaller cutting forces, eliminating vibrations and excessive tool wear, even in situations of non-rigid workpiece holding.



## Mill-Turn Machine Simulation

Mill-Turn machine simulation in SolidCAM offers a full kinematic simulation package, supporting simulation of all turning and milling operations and of all CNC machine components and devices. The simulator offers full collision detection between machine components, workpiece, fixtures and tool holders.

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




Open SolidCAM Part: **Mill_Turn_1.prz**

Download the recording

This example illustrates the use of the SolidCAM Mill-Turn module for the machining of the optical part shown above on a 4-Axis Mill-Turn CNC-Machine.

The following Turning and Milling operations are used to perform the machining of the part:

- **Turning (TR_profile1; TR_profile1_1; DRILL; TR_profile10)**

  These turning operations are used to generate the tool paths for the rough and finish machining of the external and internal cylindrical faces.

- **Facial Milling (F_profile2; D_drill3; D_drill4)**

  These operations perform the machining of the screw slot and four holes using SolidCAM capabilities for facial milling. Position #1 of Coordinate System #1 is used to perform the facial machining.

- **Machining of the side faces (P_profile3)**

  This Pocket operation is used to perform the machining of the side faces of the model. The Contour strategy is used in combination with a negative Wall offset value in order to generate an overlapping tool path that completely machines the faces.

  Position #3 of Coordinate System #1 is used for the operation. The Transform option is used to create a circular pattern of operations around the revolution axis.

- **Drilling on the side face (D_drill)**

  This Drill operation is used to perform the machining of two holes located on the side face of the model. Position #3 of Coordinate System #1 is used for the operation.

- **Slot machining (F_profile5)**

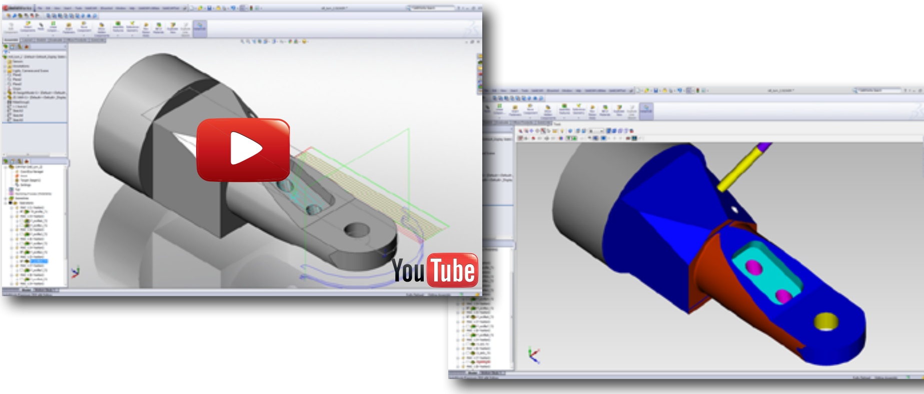
  This Profile operation is used to perform the machining of the slot using indexial 4-axis milling.
  Position #4 of Coordinate System #1 is used for the operation.
  An end mill of Ø2.5 is used for the operation.

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

  These Drill and Pocket operations are used to perform the machining of three counter bore holes located on the cylindrical face.
  Position #5 and Position #6 of Coordinate System #1 are used for the operations.

- **Pocket machining (P_profile9)**

  This Pocket operation is used to perform the simultaneous 4-axis machining of the pocket, wrapped on the external face of the part. Position #2 of Coordinate System #1 is used to perform the pocket machining. An end mill of Ø2.5 is used for the operation.
  The Wrap option, chosen during the machining geometry definition, enables you to define the wrapped geometry of the pocket directly on the solid model.
  The Contour strategy is chosen for the pocket machining.




Open SolidCAM Part: **Mill_Turn_2.prz**

Download the recording

This example illustrates the use of the SolidCAM Mill-Turn module for the machining of the console part shown above on a 5-axis Mill-Turn CNC-Machine.

The following Turning and Milling operations are used to perform the machining of the part:

- **Turning (TR_contour)**

  This turning operation is used to generate the tool path for the rough and finish machining of the external cylindrical faces.

- **Indexial milling (F_profile6)**

  This Profile operation is used to perform the machining of the cube sides using the SolidCAM indexial milling capabilities. Position #4 of Coordinate System #1 is used for the operation. The Transform option is used to create a circular pattern of operations around the revolution axis in order to machine all the cube faces. An end mill of Ø16 is used for the operation.

- **Horizontal faces machining (F_profile1)**

  This Profile operation is used to perform the indexial milling of the horizontal faces at the front part of the console. Position #4 of Coordinate System #1 is used for the operation.
  The Transform option is used to create a circular pattern of operations around the revolution axis in order to machine both sides of the console's front part.

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

  These Profile operations are used to perform the machining of the inclined faces using the B-axis. Positions #5 and #6 of Coordinate System #1 are used for these operation.
  An end mill of Ø16 is used for the operations.

- **Cylindrical face machining (F_profile2)**

  This Profile operation is used to perform the machining of the cylindrical face at the front part of the console. Position #4 of Coordinate System #1 is used for the operation.
  An end mill of Ø16 is used for the operation.

- **Pocket machining (P_profile9)**

  This Pocket operation is used to perform the machining of the pocket located on the inclined faces using the B-axis. Position #5 of Coordinate System #1 is used for the operation.
  An end mill of Ø6 is used for the operation.

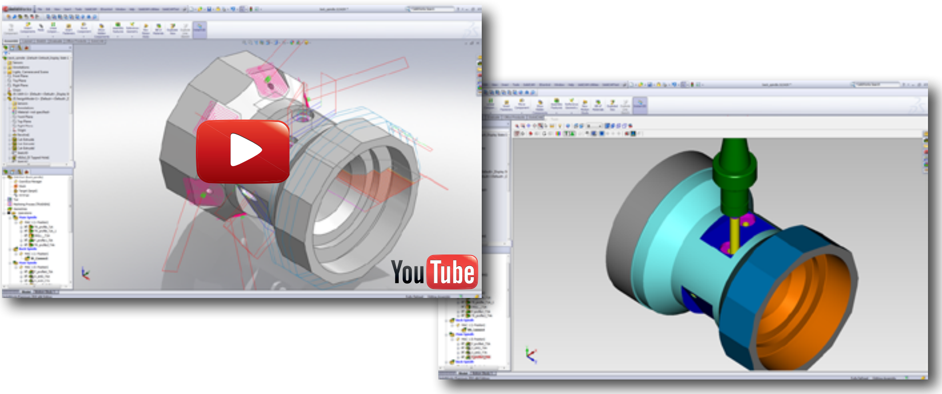
- **Inclined faces machining (F_profile7; F_profile8)**

  These Profile operations are used to perform the machining of the inclined faces on the cube using the B-axis. Positions #7 and #8 of Coordinate System #1 are used for the operation.
  An end mill of Ø16 is used for the operations.

- **Hole machining (D_drill; D_drill1; D_drill2; D_drill3)**

  These Drill operations are used to perform the machining of the inclined faces on the cube using the B-axis. Positions #4, #6, #7 and #8 of Coordinate System #1 are used for the operations.

# Mill-Turn - 2 Spindles




Open SolidCAM Part: **Back_Spindle.prz**

Download the recording

This example illustrates the use of the SolidCAM's front and back spindles functionality for machining of the connector part shown above on a 5-axis Mill-Turn CNC-Machine.

The following Turning and Milling operations are used to perform the machining of the part:

- **Turning and front side milling**
  **(TR_profile; TR_profile_1; DRILL; F_profile1; TR_profile2)**

  These operations are used to perform turning and facial milling of the front faces of the connector. Position #1 of Coordinate System #1 is used for the operation. The back spindle is not used in these operations; only the main spindle is used.

- **Indexial machining of the middle part**
  **(F_profile6; D_drill2; D_drill2_1; F_profile7)**

  These Profile and Drill operations are used to perform the machining of the pads and holes located around the cylindrical surface, in the middle part of the connector. Position #5 of Coordinate System #1 is used for the operation.

- **Indexial machining of the back part (P_profile8; D_drill3)**

  These Profile and Drill operations are used to perform the machining of the pads and holes located around the conical surface, in the middle part of the connector. Position #6 of Coordinate System #1 is used for the operation.

- **Turning and back side milling (TR_profile9; F_profile10; DRILL_1; TR_profile11; F_profile12; D_drill4; D_drill4_1)**

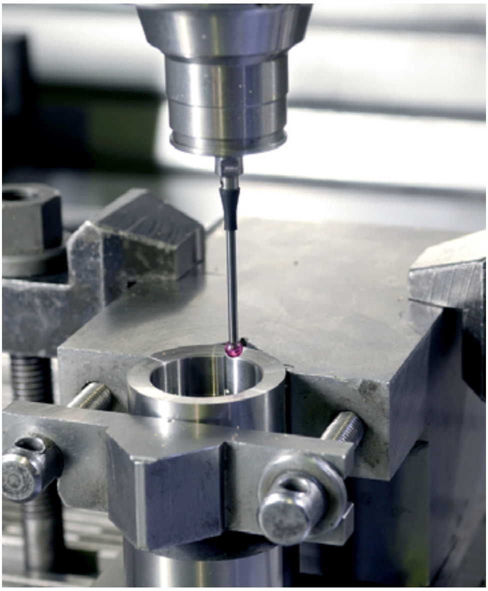
  These operations perform turning and facial milling of the back faces of the connector using the back spindle. Position #1 of Coordinate System #1 is used for the turning operations. Position #4 of Coordinate System #1 is used for milling operations. Separate setup settings are used for machining of the part clamped in the back spindle.

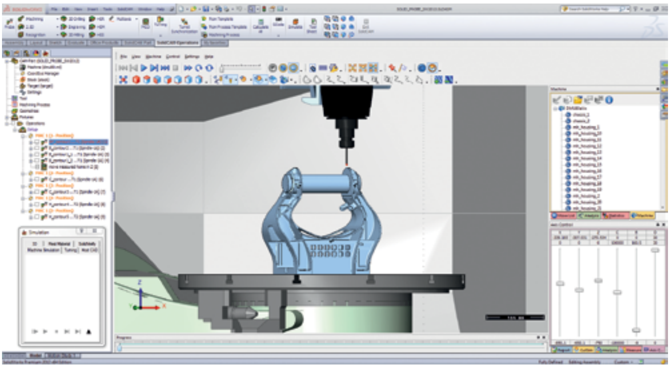
# New Module for Home Definition and On-Machine Verification

## SolidCAM Solid Probe Module

SolidCAM is bringing you Solid Probe, a new SolidCAM module that provides capabilities for Home definition and On-Machine Verification, using probes on the CNC machine, to do 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 Module for Every Machinist using Probes:

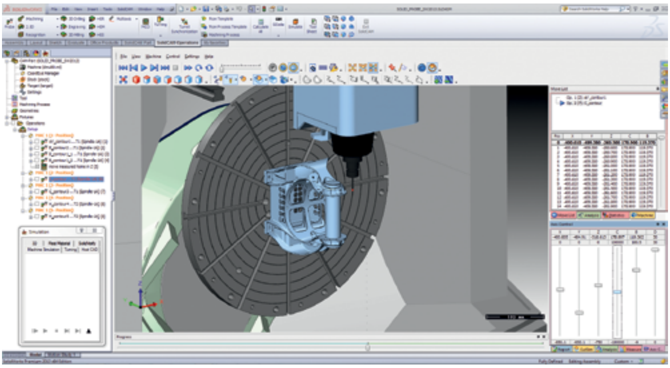
- Easy Home definition
- On-Machine Verification
- Tool Presetter support
- Easy geometry selection on solid model
- Supports wide range of probe cycles
- Visualization of all the Probe tool movements
- Support of different Probe controllers

## Combined Probe and Machining Operations

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

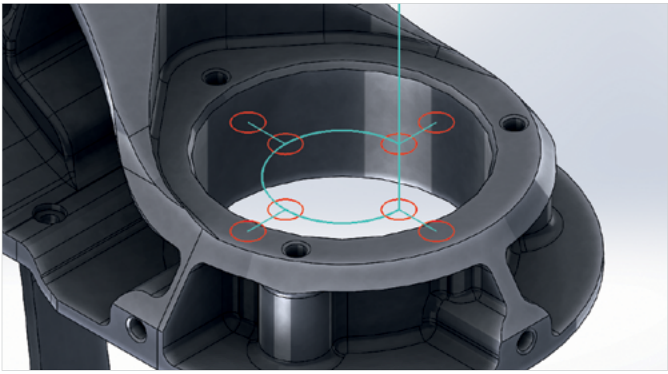
## Home Definition

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



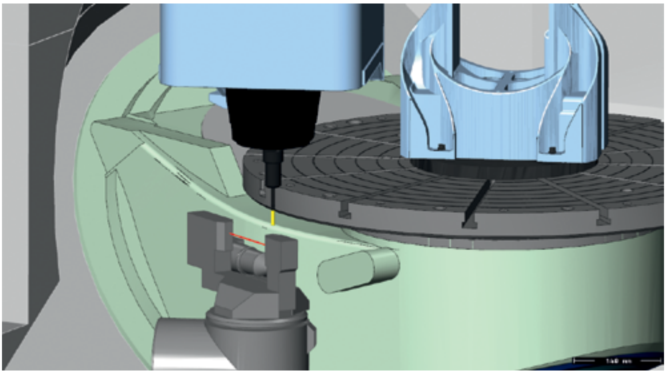
## On-Machine Verification

Solid Probe cycles are used for measuring of machined surfaces without transferring the part to a CMM machine - the part can be inspected on the machine tool itself.



## Preview of Cycle Movements

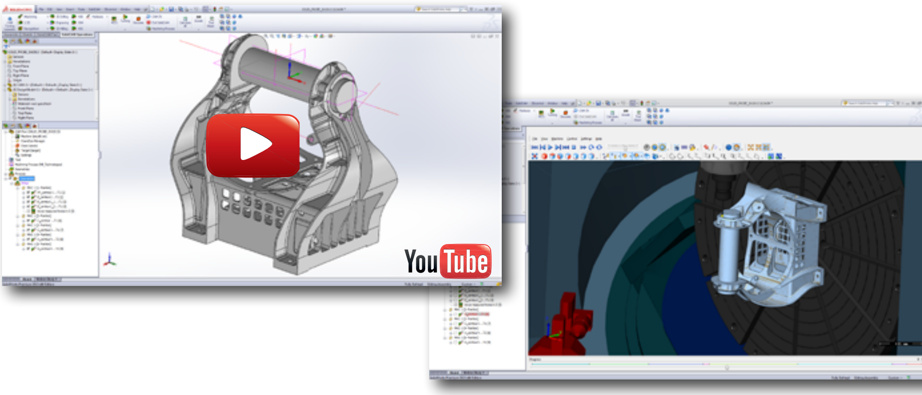
Solid Probe uses the same geometry as the 2.5D milling operations. Full control over tolerances, different sorting options and direct preview of cycle movements are provided.



## Tool Presetter Support

The Solid Probe module includes support of Tool Presetter.

This option enables checking milling or turning tools between machining operations. It is a useful option for tool checking, after every operation or every tool change event. It enables tool breakage detection, providing safe machining.




Open SolidCAM Part: **Solid_Probe.prz**

Download the recording

This example illustrates the use of the SolidProbe Cycles in multiple positions to check the precision of machining for different areas of the part.

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

  This Angle Y cycle is used to check if the cylinder surface is straight. Position #1 of Coordinate System #1 is used for the cycle.

- **Cylinder width checking (B_contour2)**

  This Boss cycle is used to check the width of the cylinder. Position #1 of Coordinate System #1 is used for the cycle.

- **Cylinder length checking (B_contour1_1)**

  This Pocket cycle is used to check the length of the cylinder, measuring the distance between two boundary surfaces. Position #1 of Coordinate System #1 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. Position #1 of Coordinate System #1 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. Position #2 of Coordinate System #1 is used for the first cycle and Position #3 is used for the second one.

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

  These Hole cycles are used to check the diameters of the holes. Position #4 of Coordinate System #1 is used for the first cycle and Position #5 is used for the second one.

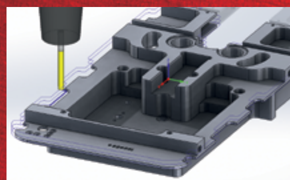
# System Requirements

- Microsoft® Windows 7 (minimum 32-bit, recommended 64-bit), Microsoft® Windows 8 64-bit
- Intel® Core™2 Duo, or AMD dual-core processor with SSE2, 2 GHz or greater recommended, or Intel® Xeon® E3 or Core i7 or equivalent with SSE2, 3.0 GHz or greater
- 4 GB RAM (8 GB is recommended for large CAM-Parts machining)
- 15 GB free disk space for installation files
- 1,280 x 1,024 or higher screen resolution
- Mouse or other pointing device
- Microsoft® Direct3D 9® or compatible graphics card (Microsoft® Direct3D 11® or higher recommended)
- CD drive
- Internet Explorer version 6 if you are using the SolidCAM online help
- For viewing SolidCAM User Guides and Training Courses, Adobe Acrobat version 9 or higher is recommended

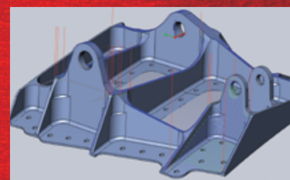
# SolidCAM 2014

## SolidCAM 2014 Modules Overview:
## Parts and Recordings

iMachining 2D

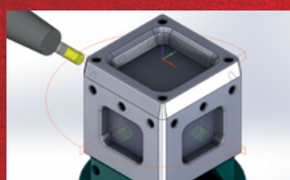


2.5D Milling



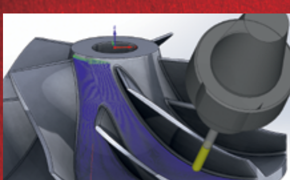
HSS (High-Speed Surface Machining)

iMachining 3D

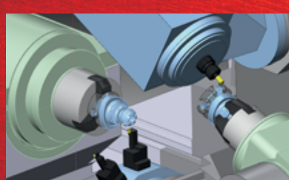


Indexed Multi-Sided Machining

HSM (High-Speed Machining)



Simultaneous 5-Axis Machining



Turning & Advanced Mill-Turn

Solid Probe