SolidCAM 2019

The unique, revolutionary Milling technology

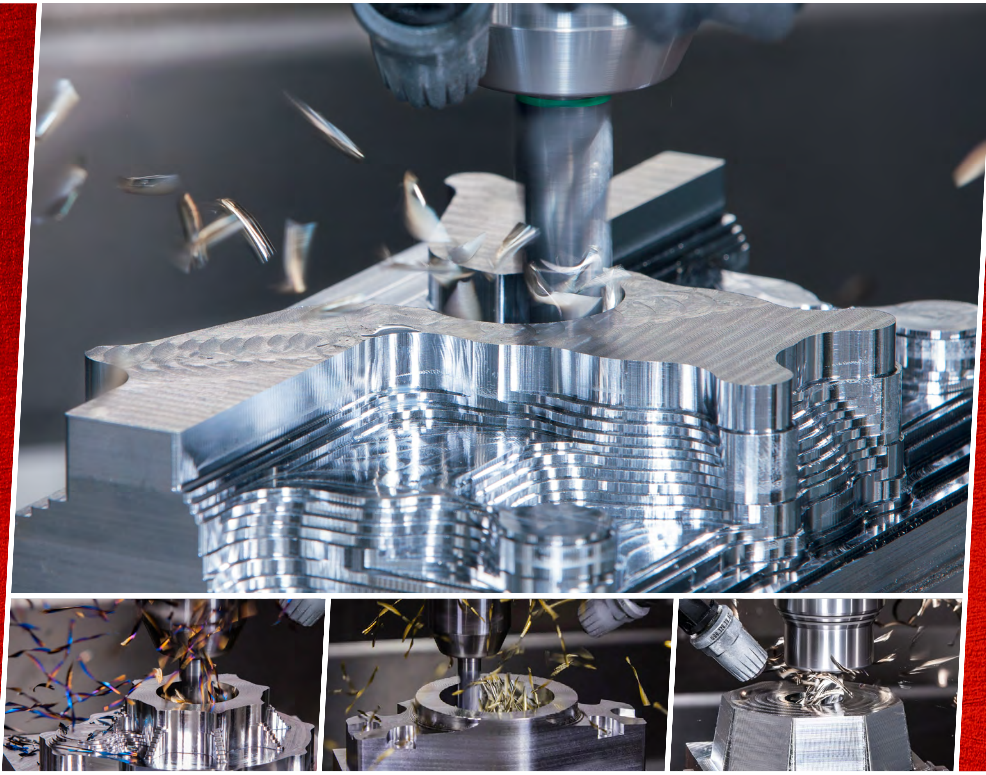
iMachining®

patent by SolidCAM

TIME SAVINGS

70%

... AND MORE!



Getting STARTED

SolidCAM & iMachining

TOOL
MATERIAL
iMachining
Technology-Wizard
for Automatic Feeds and Speeds
MACHINE
GEOMETRY

CERTIFIED Gold Product
SOLIDWORKS

SolidCAM + SOLIDWORKS

The Complete Integrated Manufacturing Solution

SolidCAM

iMachining – The Revolution in CAM!

Contents

Document number: SCiMGSENG1-2019

About this document

This Getting Started document is an interactive guide consisting of two main parts. The goal of the first part is to jump-start your basic knowledge of SolidCAM by using several of its 2.5D Milling technologies. The second part is geared to get you started using SolidCAM's revolutionary iMachining technology.

Document design

This document is primarily designed around a task-based approach to learning. Being interactive, it uses virtual guided exercises to quickly and easily help you get started with SolidCAM and iMachining.

Using this document

This document is intended for new users to be used as self-study material. It is recommended to complete the exercises in order, from beginning to end. The various parts can be automatically opened by clicking the file names. The videos demonstrating the steps to complete the exercises can be viewed by simply clicking the play buttons.

Software versions used for this document

Screenshots for this document were created using previous and current versions of SolidCAM integrated with SOLIDWORKS, running on Windows 10. The videos, on the other hand, were made using previous SOLIDWORKS/SolidCAM versions. Any major feature differences will be described ahead of time so you can easily follow along. If you are running a different version of Windows, you may also notice differences in the appearance of menus and dialog box windows. These differences do not affect the performance of the software.

About the *.zip file

The *.zip file contains this document as well as copies of the various part files that are required for the exercises/examples, which are located in the **SC_GS_Examples** subfolder. When extracting the *.zip file, it is important to extract the whole folder and not just its contents – this document links to the locations of the part files contained in the folders. In the **Completed_CAM-Parts** subfolder, the SolidCAM final manufacturing projects are provided for checking the integrity of your completed CAM-Parts. The various part files used throughout this course were prepared with SOLIDWORKS 2019 and SolidCAM 2019.

Adobe Reader

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

Conventions used in this document

This document uses the following typographic conventions:

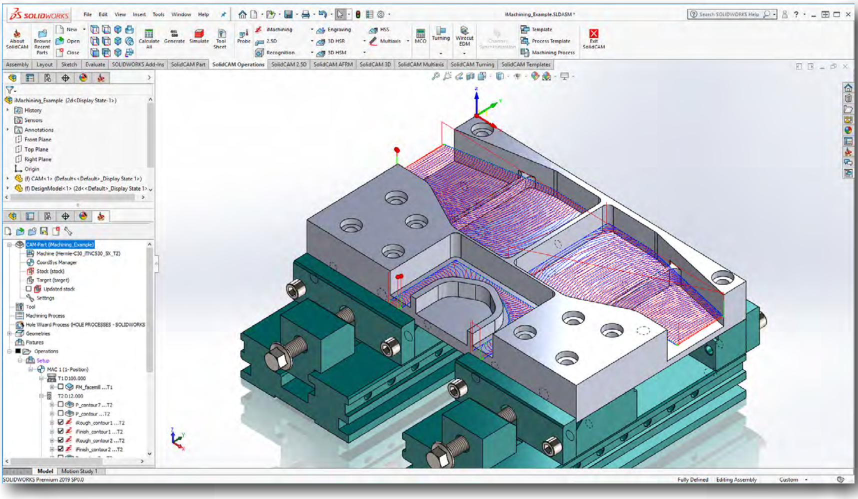
Bold	This style emphasizes SOLIDWORKS/ SolidCAM options and commands. For example, click the **New** button...
3. Define the CAM-Part	The mouse icon and numbered bold text indicate the steps required to complete an exercise.
Create and define...	The stairs icon indicates the start of the written steps that accompany the iMachining Walkthrough videos.
Explanation	This style combined with the lamp icon is used for explanations and notes.

The following buttons are used in this document as the interactive features for opening part files and connecting to URLs:

SC_Simple_Cover.SLDPRT	Similar to a hyperlink function, this style signifies a button that can be clicked to automatically open the corresponding part file.
	Click this play button to start playing the video. Your internet browser will automatically launch and connect to YouTube

The facts about SolidCAM

SolidCAM + SOLIDWORKS is the most complete and powerful, easy-to-use integrated CAD/CAM solution for design and manufacturing. SolidCAM is the leading, best-in-class CAM suite for profitable CNC programming in SolidWorks. SolidCAM, including the revolutionary iMachining, is seamlessly integrated in SolidWorks with full tool path associativity to the SolidWorks model.



With the single-window integration, all machining operations can be defined, calculated and verified without leaving the parametric SolidWorks assembly environment. All 2D and 3D geometries used for machining are fully associative to the SolidWorks model. If you make any changes to your CAD model, all of your CAM operations are updated automatically.

SolidCAM supports the complete range of major manufacturing applications including iMachining 2D, iMachining 3D, 2.5D Milling, High Speed Surface Machining (HSS), 3D Mill High Speed Machining (HSR/HSM), Indexial Multiaxis Machining, Simultaneous 5-Axis Machining, Turning, Mill-Turn and even Solid Probe.

SolidCAM is widely used in the mechanical manufacturing, electronics, medical, consumer products, machine design, automotive and aerospace industries, as well as in mold and die and rapid prototyping shops.

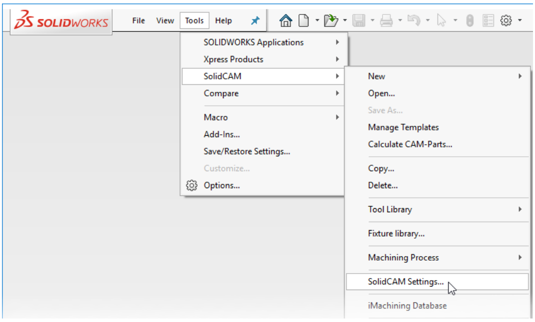
The successful manufacturing companies of today are using integrated CAD/CAM systems to get their products to market faster and reduce costs. With SolidCAM's gold-certified status and seamless single-window integration in SolidWorks, organizations of all sizes are realizing the benefits of SolidCAM + SOLIDWORKS.

Getting Started with SolidCAM

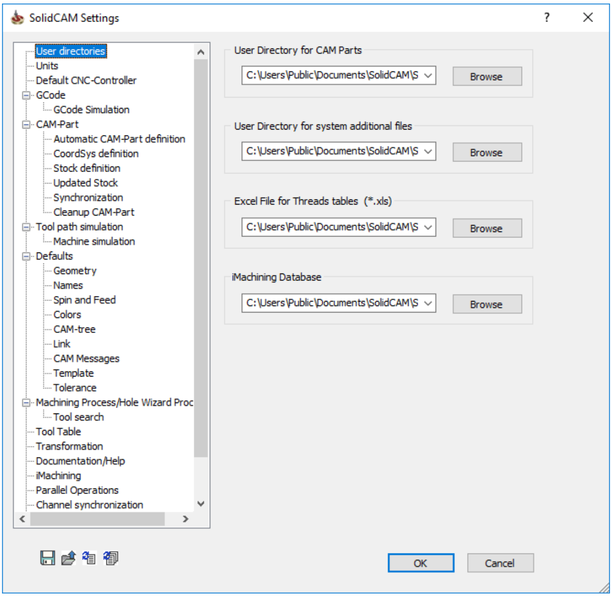
SolidCAM uses initial default settings to simplify certain tasks. For example, the CAM-Part is created and defined automatically. As a new user however, it is important to understand the full process of defining the CAM-Part and how the CAM-Part features are built. For this purpose, some of the default SolidCAM Settings should be changed before you begin.

Start by opening SOLIDWORKS.

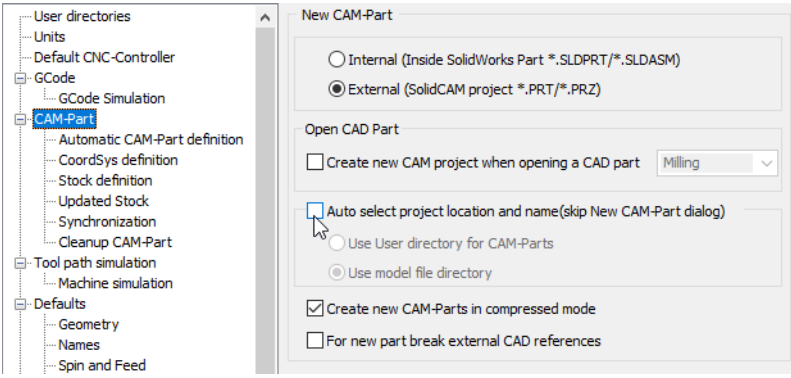
In the SOLIDWORKS Menu Bar, click **Tools** to access the **SolidCAM** main menu and then choose **SolidCAM Settings**.



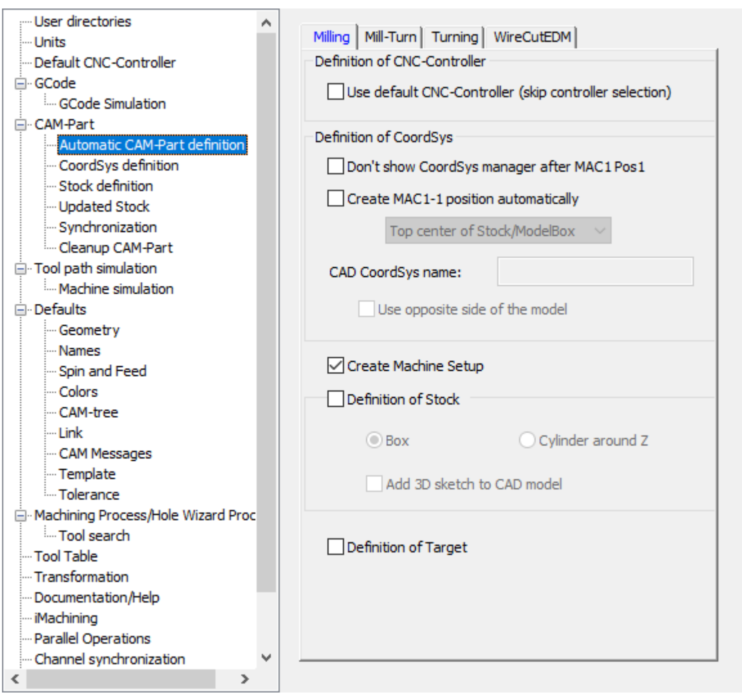
SolidCAM is started and the SolidCAM Settings dialog box is displayed.



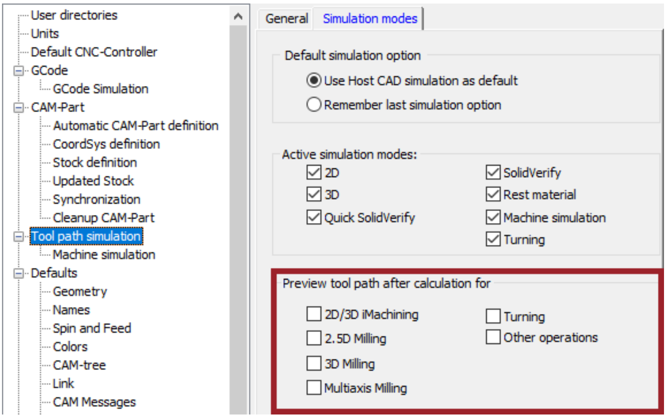
On the CAM-Part page, the option **Auto select project location and name** should be disabled; the CAM-Part creation will not be automated.



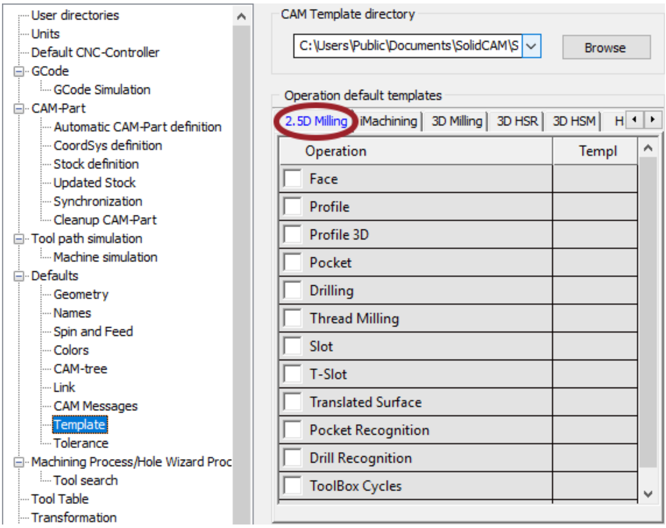
On the Automatic CAM-Part Definition subpage > Milling tab, disable any options relevant to **Definition of CNC-Controller**, **Definition of CoordSys**, **Definition of Stock** and **Definition of Target**.



On the Tool path simulation page > Simulation modes tab, all the options in the **Preview tool path after calculation for** section can be disabled; SolidCAM Simulation will be used extensively to view and check the generated tool path.

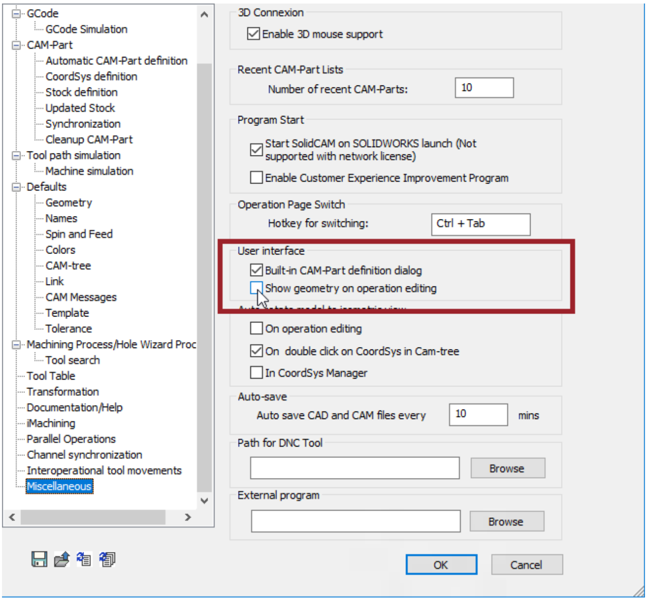


On the Defaults > Template subpage, Operation default templates for 2.5D Milling will not be used.



Changes to some of these settings are duplicated and reviewed in the upcoming Simple Cover Machining exercise.

On the Miscellaneous page in the User interface section, you can also disable the option **Show geometry on operation editing**.

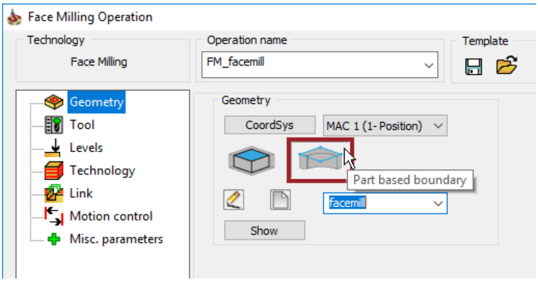


Most beneficial when using the Modify Geometry feature of SolidCAM, this setting is not needed in any of the Getting Started exercises.

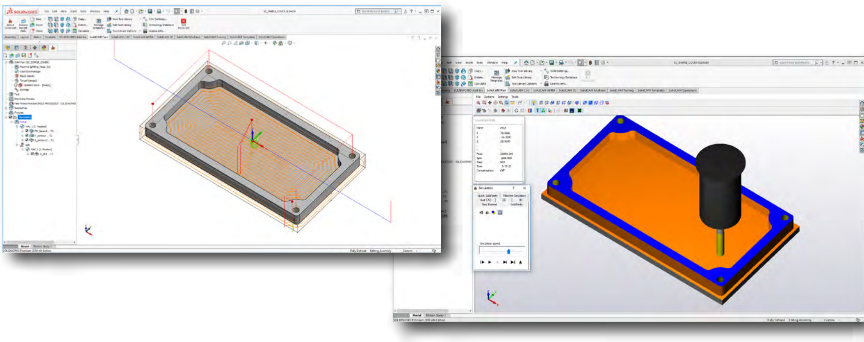
These settings can be reverted or changed again at any time.

Click **OK** to confirm your changes.

Important Note: *The Jumpstart video series was recorded with previous versions of SolidCAM. In SolidCAM 2019, use* **Part based boundary** *for the Geometry definition in Face Milling operations.*



There are no other notable differences.



This exercise is based on the SolidCAM Professor video series called Jumpstart – the easy way to learn SolidCAM. The machining of a simple cover is defined using several SolidCAM 2.5D Milling technologies.

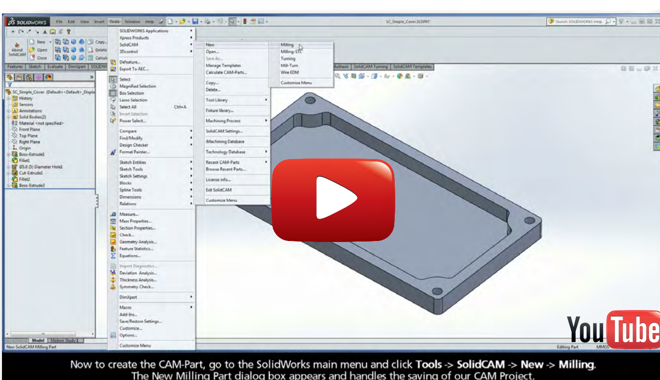
The following steps have to be implemented in order to reach the final CAM-Part:

1. Load the SOLIDWORKS model

Click **SC_Simple_Cover.SLDPRT** to load the CAD model.

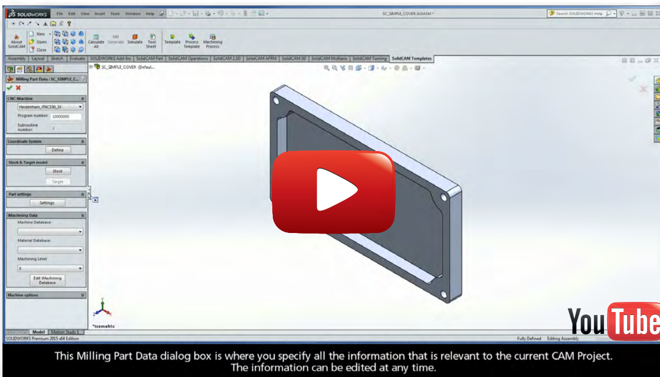
2. Create a New Milling CAM Project

After the part file is loaded on your computer, the following video demonstrates how to start SolidCAM and create a New Milling CAM Project. Prior to creating the Project, the CAM Settings are customized to prepare for the upcoming tutorial videos. These include setting the defaults for the CAM-Part creation as well as disabling the Automatic CAM-Part definition options. Click the play button below...



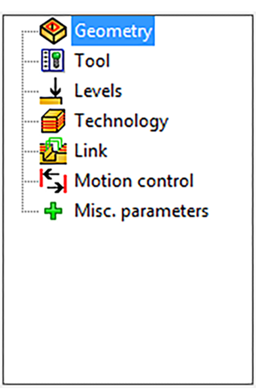
3. Define the CAM-Part

In the following video, the CAM-Part is defined using the Milling Part Data dialog box. The CAM-Part Definition includes selecting the CNC-Machine Controller, defining the Coordinate System for all machining operations of the CAM-Part, and finally defining the Stock and Target models. Click the play button below...



4. Complete the part programming

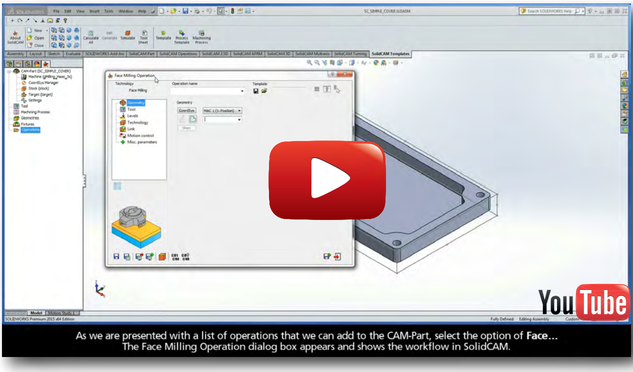
The part programming is completed in just four operations, which begin on the following page. As shown in the illustration below, the workflow in SolidCAM is displayed in each of the Operation dialog boxes. The Geometry is defined first, followed by creating and choosing a Tool, picking the Milling levels, defining the type of Technology to use, and finally choosing the Lead in and Lead out tool Link movements.



Each operation is saved and the calculated tool path is quickly shown in the SolidCAM Simulation.

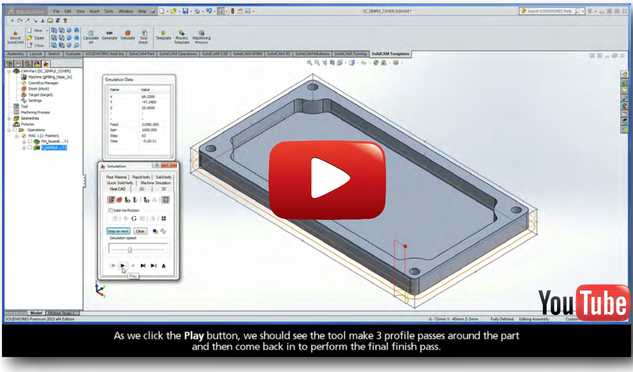
4.1 Add a Face Milling operation

In this video, a Face Milling operation is added to the CAM-Part. The target geometry is selected, which automatically creates a chain used for the machining boundary. A Ø100 mm (4 in) face mill is defined for the operation. The tool is set to machine the 0.5 mm (0.02 in) of stock material off the top of the part using the One Pass technology. Click the play button below...



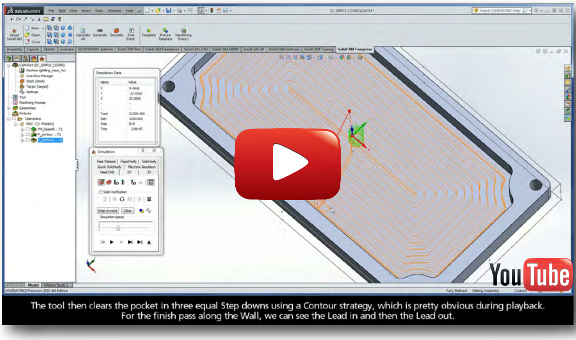
4.2 Add a Profile operation

In this video, a Profile operation is added to the CAM-Part. The geometry is selected as a chain that runs along the outside contour of the Target model. A Ø6 mm (0.25 in) end mill is defined for the operation. The tool is set to perform an Arc Lead in/out and climb cut at a Constant depth. After the full profile depth is reached, the tool will remove the 0.24 mm (0.01 in) allowance with a single finish pass. Click the play button below...



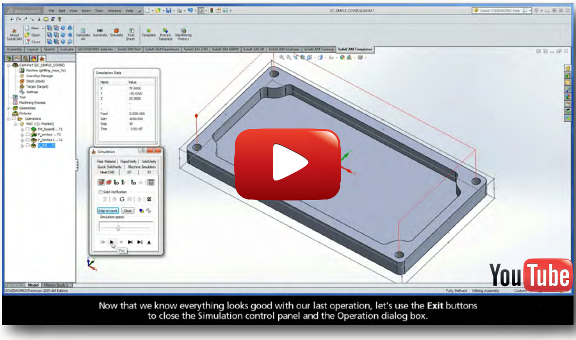
4.3 Add a Pocket operation

In this video, a Pocket operation is added to the CAM-Part. Like the previous Profile operation, a single edge is picked during the geometry selection. The chain is closed automatically. The Ø6 mm (0.25 in) end mill is chosen for the operation from the Part Tool Table. The tool is set to perform a Helical entry into the pocket and a Contour strategy is used for cutting. After the roughing, the tool will take a finish pass on both the Wall and Floor to remove the excess offset material. Click the play button below...



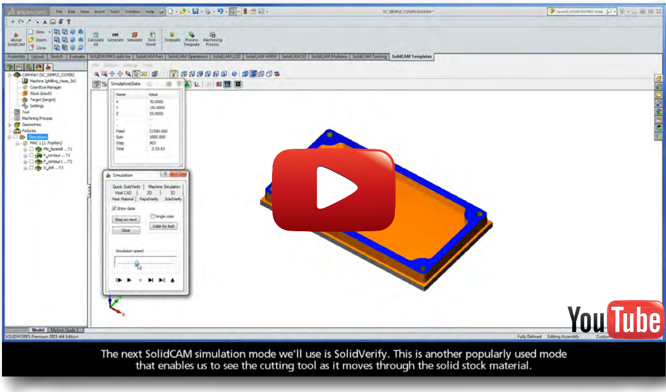
4.4 Add a Drilling operation

In this video, a Drilling operation is added to the CAM-Part to complete the part programming. SolidCAM finds the centers of all circle entities and defines their positions for the drill geometry. A drilling tool is defined and choosing a holder is also shown. The Levels are picked directly off the model, like in the previous operations. The standard drilling method of G81 is defined for the Drill cycle type. Click the play button below...



5. Simulate the tool path and generate GCode

In the following video, the tool path for the entire CAM Project is simulated. Included is an in-depth look at SolidCAM's Host CAD simulation options as well as using the SolidVerify simulation mode. Afterwards, the GCode commands are used to generate GCode for the entire CAM Project as well as for a single selected operation. The GCode files output per the specified CNC-Machine Controller. Click the play button below...

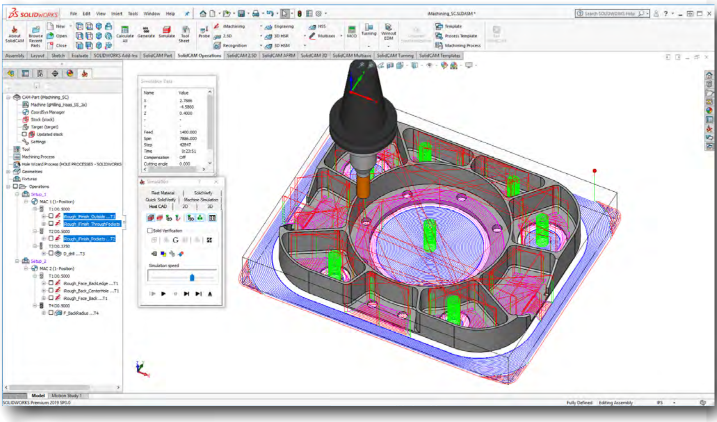


You have completed the SolidCAM Getting Started exercise.

If you would like to review SolidCAM's completed CAM-Part and compare it with yours, click **SC_SIMPLE_COVER.prz**.

The facts about iMachining

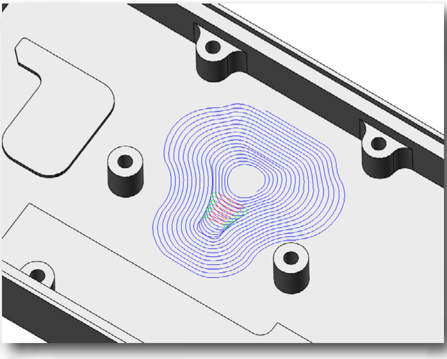
SolidCAM's iMachining™ is an intelligent high speed milling technology that is designed to produce fast and safe CNC programs to machine mechanical parts with *first part success* performance. The word *fast* meaning significantly faster than traditional machining at its best and the word *safe* meaning without the risk of breaking tools or subjecting the machine to excessive wear, all while maximizing tool life.



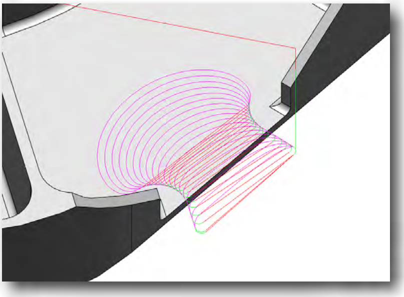
To achieve these goals, the iMachining technology uses advanced, patented algorithms to generate smooth tangent tool paths, coupled with matching conditions, that together keep the mechanical and thermal load on the tool constant, while cutting thin chips at high cutting speeds and deeper than standard cuts (up to 4 times diameter).

iMachining tool paths

iMachining generates morphing spiral tool paths that spiral either outwardly from some central point of a walled area, gradually adopting the form of and nearing the contour of the outside walls, or inwardly from an outside contour of an open area to some central point or inner contour of an island. As a result, iMachining manages to cut irregularly shaped areas with a single continuous spiral.



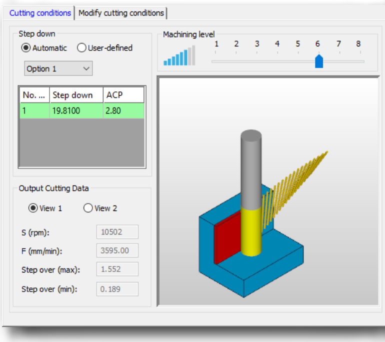
To machine narrow passages, separating channels and tight corners, iMachining uses proprietary constant load one-way tool paths.



In some open areas, where the shape is too irregular to completely remove with a single spiral, proprietary topology analysis algorithms and channels are used to subdivide the area into a few large irregularly shaped sub-areas and then machines each of them by a suitable morphing spiral, achieving over 80% of the volume being machined by spiral tool paths. Since spiral tool paths have between 50% and 100% higher Material Removal Rate (MRR) than one-way tool paths, and since iMachining has the only tool path in the industry that maintains a constant load on the tool, it achieves the highest MRR in the industry.

iMachining also performs an automatically optimized tool path when finishing 2.5D features. The finish tool path is executed in several consecutive steps with intelligent tool movements, all of which would be programming intensive and difficult to achieve using traditional machining methods. Because of its highly systematic approach to finishing and dedication to eliminate over engagement, the iMachining technology is able to further maximize tool life.

iMachining Technology Wizard



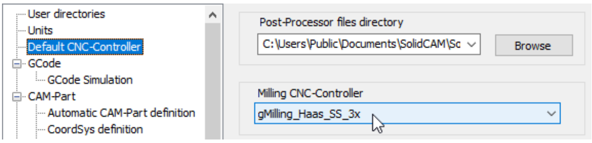
A significant part of the iMachining system is devoted to calculating synchronized values of feed rate, spindle speed, axial depth of cut, cutting angles and (undeformed) chip thickness based on the mechanical properties of the workpiece and cutting tool, while also keeping within the boundaries of the machine capabilities (maximum feeds and spindle speed, power and rigidity). The iMachining Technology Wizard, which is responsible for these calculations, provides the user with the means of selecting the level of machining aggressiveness most suitable to the specific machine and setup conditions and to their production requirements (quantity, schedule and tooling costs).

Another critical task performed by the Technology Wizard is dynamically adjusting the feed to compensate for the dynamically varying cutting angle – a by-product of the morphing spiral, thus achieving a constant load on the tool, which again maximizes tool life.

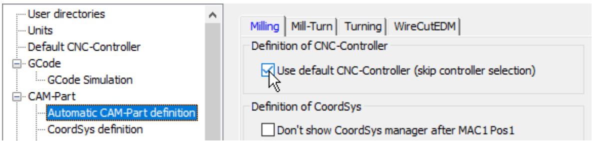
Getting Started with iMachining

Before you begin the upcoming exercises, additional changes should be made to the default SolidCAM Settings.

Open the SolidCAM Settings dialog box and on the Default CNC-Controller page, select **gMilling_Haas_SS_3x** from the Milling CNC-Controller drop-down list.

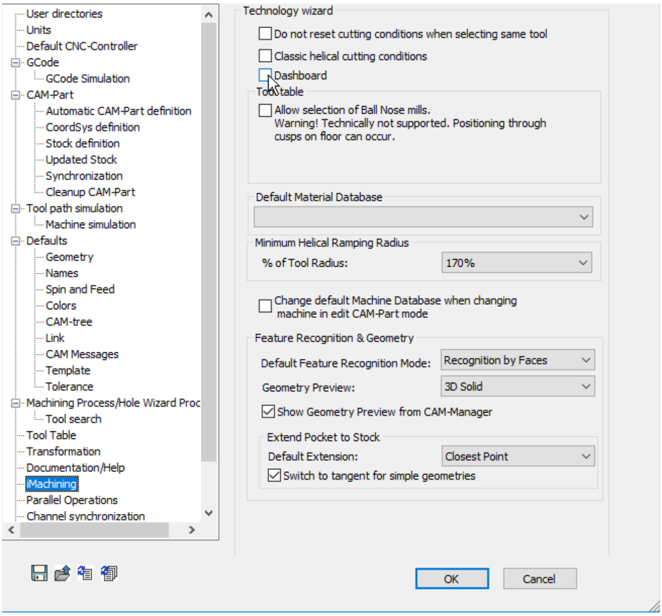


On the Automatic CAM-Part Definition subpage > Milling tab, enable the option **Use default CNC-Controller**.



These settings will later simplify your selections since the remaining CAM-Parts use the gMilling_Haas_SS_3x controller.

On the iMachining page, disable the **Dashboard** option.

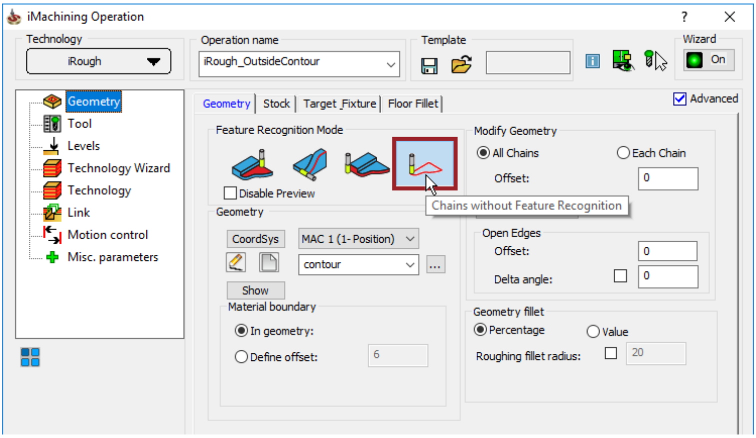


When enabled, the Dashboard page is featured in the iMachining Operation dialog box. The iMachining Dashboard was designed only for iMachining 2D operations and is primarily for experienced users.

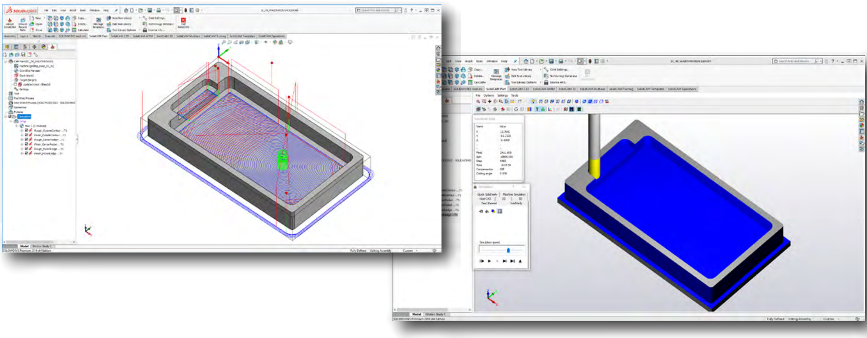
These settings can be reverted or changed again at any time.

Click **OK** to confirm your changes.

Important Note: *The remaining videos were recorded with previous versions of SolidCAM. In SolidCAM 2019, use* **Chains without Feature Recognition** *(SolidCAM's standard chaining method) for the Geometry definition in iMachining operations.*



There are no other notable differences.



This exercise is based on a SolidCAM Professor video series, which provides a step-by-step guide on the definition process of the iMachining technology to machine the part shown above. The videos demonstrating the steps are duplicated and accompanied by a written walkthrough. The roughing and finishing of the outside contour, center pocket and pocket ledge are defined.

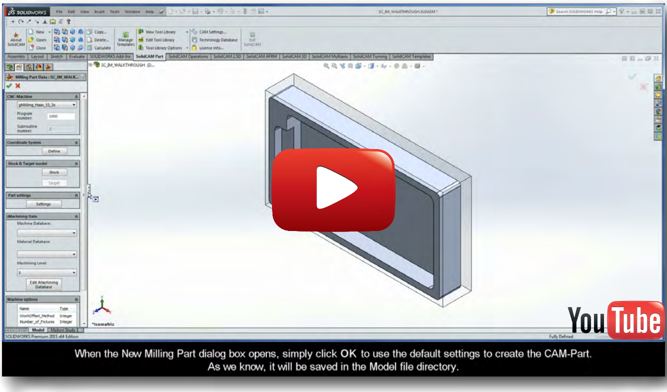
The following steps have to be implemented in order to reach the final CAM-Part:

1. Load the SOLIDWORKS model

Click **SC_iM_Walkthrough.SLDPRT** to load the CAD model.

2. Create and define the CAM-Part

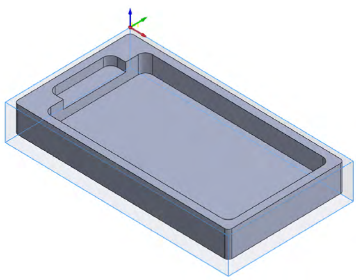
In this step, a New Milling CAM Project is created and the CAM-Part is defined. In addition, the machine and work material parameters are defined for iMachining. Click the play button below to watch the video. Following the video is also a written walkthrough to complete this step.



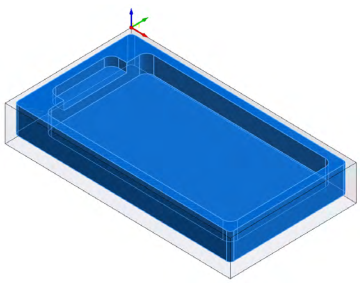
Create and define the CAM-Part.

Using the Milling Part Data dialog box, define the CNC-Machine Controller (gMilling_Haas_SS_3x), the Machine Coordinate System, the Stock model and the Target model.

The Stock model and the Target model should be defined as shown.



3D Model Stock



Target

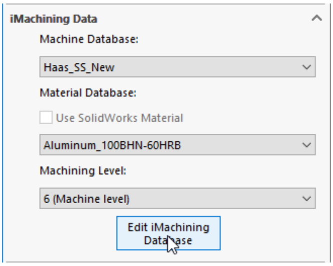
The iMachining Database selections (machine and work material) must also be defined for the CAM-Part. The corresponding properties are used by the Technology Wizard in iMachining calculations.

For this exercise, choose **Haas_SS** from the Machine Database list and choose **Aluminum_100BHN-60HRB** from the Material Database list.

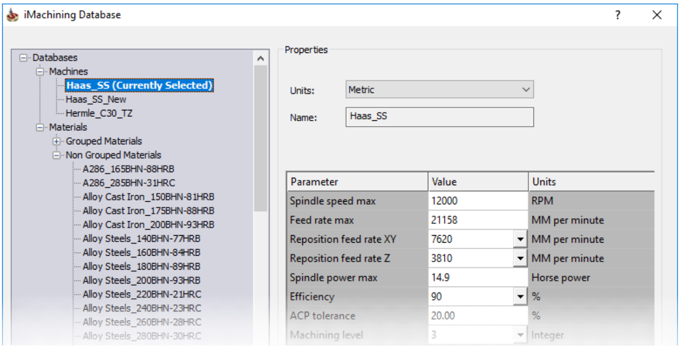
iMachining Database

The iMachining Database contains all the machines and work materials available for selection in the CAM-Part Definition.

Clicking the **Edit iMachining Database** button displays the iMachining Database dialog box.

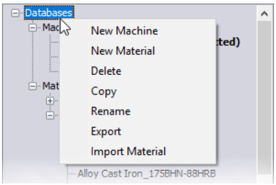


This dialog box is the interface that enables you to add new/ edit existing machines and work materials contained in the iMachining Database.



On the left is the Databases tree with Machines and Materials lists. On the right is displayed the properties by which the selected machine or material is defined.

The iMachining Database managing commands are displayed in the right-click menu of the Databases tree.

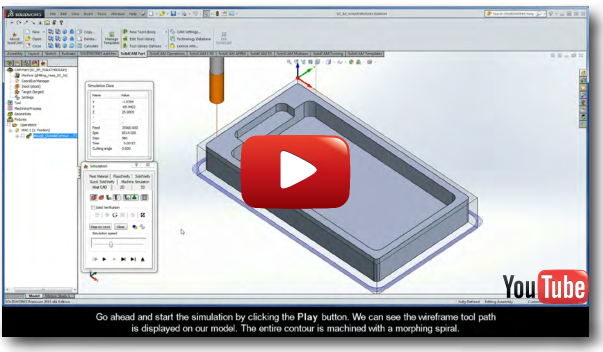


The following commands enable you to:

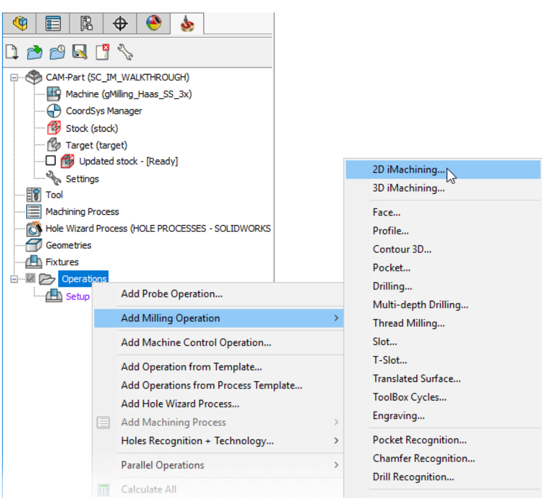
- **New Machine** – add a new machine.
- **New Material** – add a new material.
- **Delete** – delete the selected machine/material.
- **Copy** – copy the selected machine/material.
- **Rename** – rename the selected machine/material.
- **Export** – export the selected machine/material to a specified location on your computer.
- **Import Material** – import saved material(s) from a specified location on your computer.

3. Define the machining of the outside contour

In this step, the machining of the outside contour is defined. For this example, the geometry is defined as an open pocket with island. The Geometry, Tool and Levels are defined and the Offsets are specified; the iMachining Technology Wizard automatically produces the optimal Cutting conditions. The roughing operation is then copied and the iFinish Technology type is used to define the finishing. Click the play button below to watch the video. Following the video is also a written walkthrough to complete this step.

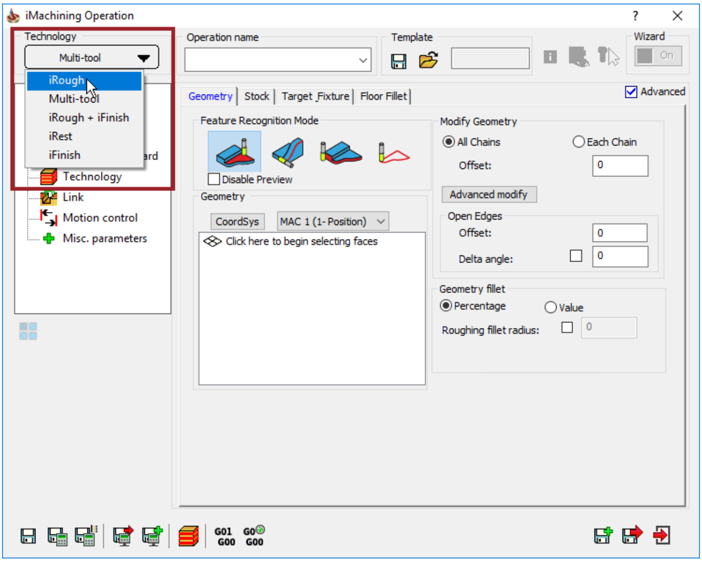


To add an iMachining operation to the CAM-Part, right-click the **Operations** header in the SolidCAM Manager and from the **Add Milling Operation** submenu, choose **2D iMachining**.

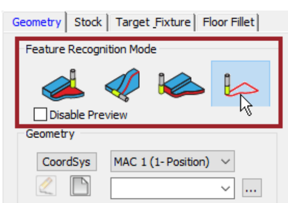


The iMachining Operation dialog box is displayed.

For the Technology type, use **iRough** to define the rough machining of the outside contour feature.



On the **Geometry** page, choose **Chains without Feature Recognition**.



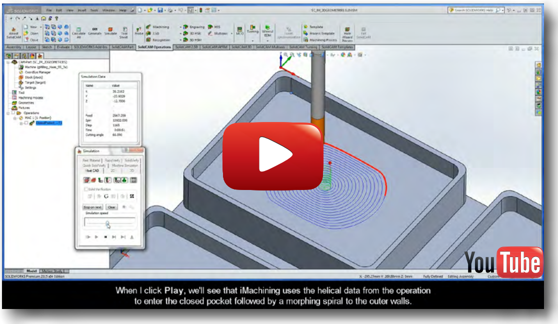
The relevant options appear in the Geometry section. Click  to start the Geometry definition.

iMachining Geometry definitions

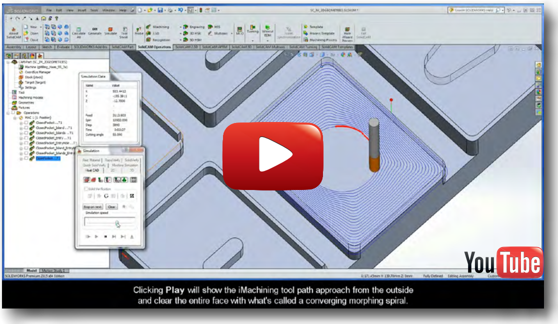
The geometry in iMachining is represented by a pocket that can be closed, open or semi-open (containing open edges).

The videos on the following page demonstrate defining the different geometry types using SolidCAM's standard chaining method and the resulting iMachining tool path.

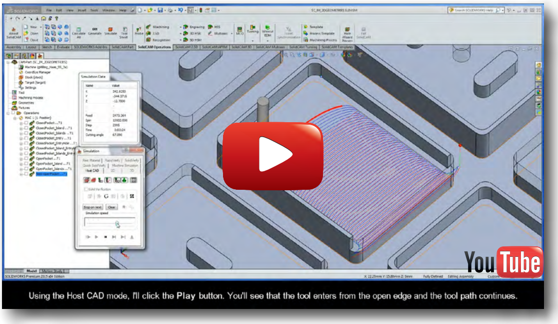
Closed pocket geometries in iMachining



Open pocket geometries in iMachining



Semi-open pocket geometries in iMachining

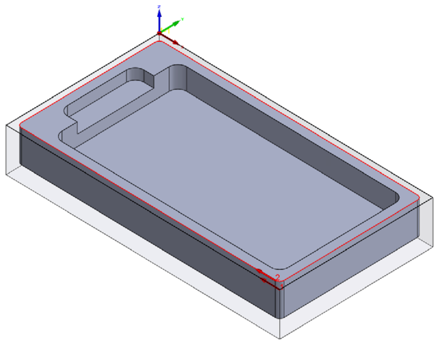


The completed CAM-Part is provided if you want to view the Geometry definitions and the iMachining tool path techniques in SolidCAM.

Click **SC_IM_GEOMETRIES.prz** to load the SolidCAM part.

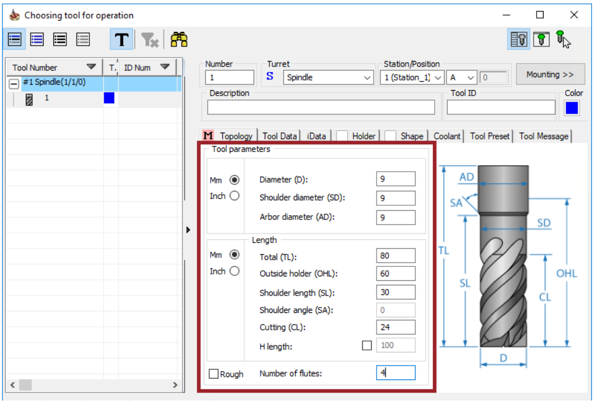
For this operation, the machining geometry is defined as an open pocket with island.

Select the two chains as shown.



Mark the outer chain (1-Chain) as open to enable the tool to approach from the outside.

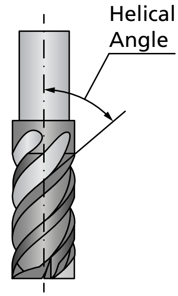
Switch to the **Tool** page and add an end mill of Ø9 mm (0.3438 in).

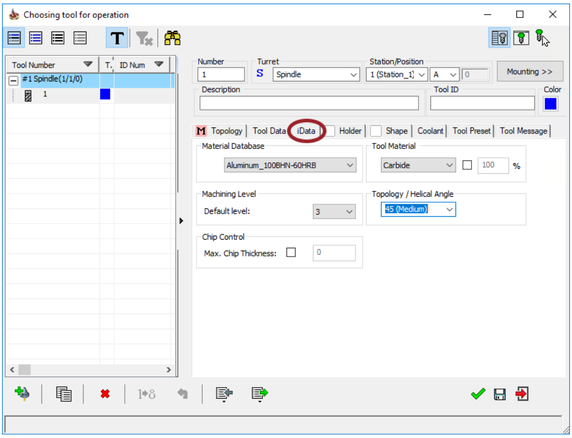


Define the remaining tool parameters as follows:

- Keep the Total length of 80 mm (3.125 in);
- Keep the Outside holder length of 60 mm (2.375 in);
- Keep the Shoulder length of 30 mm (1.25 in);
- Keep the Cutting length of 24 mm (1 in);
- Set the Number of flutes to 4.

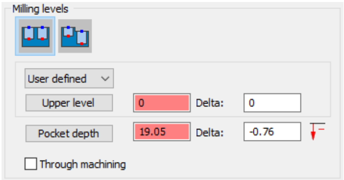
On the iData tab, the default 45 (Medium) value is used for the Helical Angle parameter. This parameter affects the Cutting conditions and Step down values generated by the iMachining Technology Wizard.



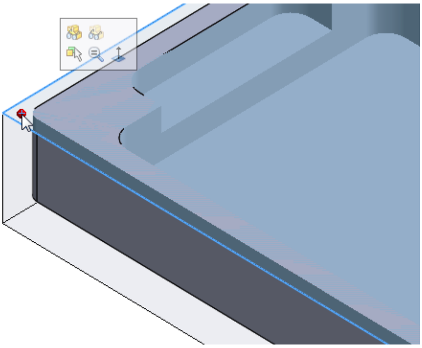


Click  to confirm the Tool definition.

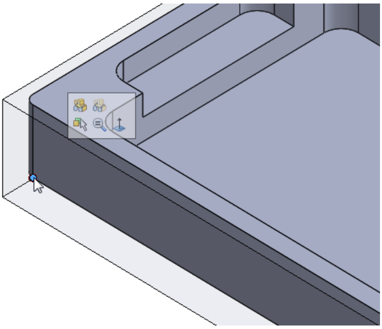
Switch to the **Levels** page to define the Milling levels for the operation.



Click the **Upper level** button and pick on the top face of the Stock model to define at what Z-level to start the machining.



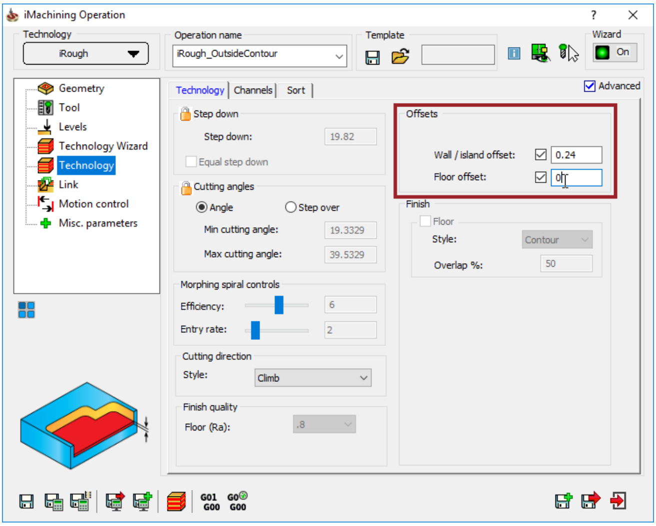
Click the **Pocket depth** button and pick on the bottom edge of the Target model to define the machining depth.



Note that the Milling levels fields are highlighted – this is because the values are associative to the picked entities and if the model changes, the corresponding values will update automatically.

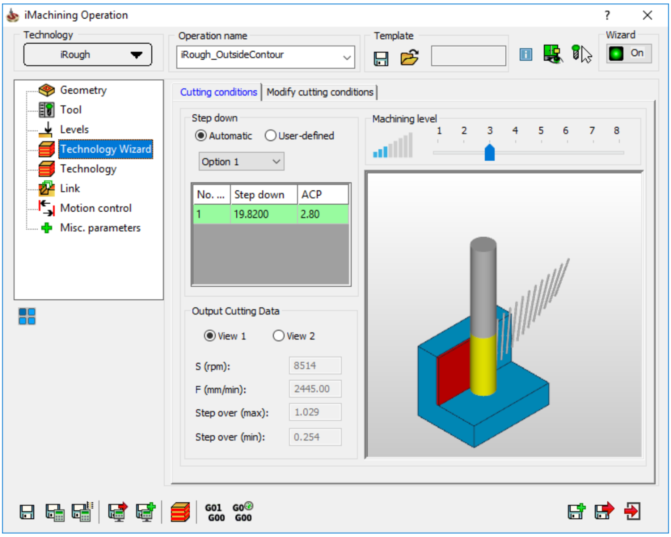
In addition to the picked depths, define a Delta depth to perform machining deeper than the part bottom edge. Set the value to −0.76 mm (−0.03 in).

Switch to the **Technology** page and specify a Wall/island offset of 0.24 mm (0.01 in) and a Floor offset of 0.



Switch to the **Technology Wizard** page of the iMachining Operation dialog box.

This Wizard automatically calculates the optimal Cutting conditions specific to the current operation according to the Geometry, Tool and Levels definitions in addition to the machine and work material defined for the CAM-Part.



Step down

There are two modes for selecting the way in which the Wizard calculates Step down:

Automatic mode automatically provides you with optimal step downs, taking into account the Tool and Levels definitions as well as the Floor offset (if any) specified for the operation.

User-defined mode enables you to manually choose the method used for calculating step downs, either by No. steps or by a specified value.

The output grid displays the No. steps, the Step down value and the number of Axial Contact Points (ACPs) automatically calculated by the Wizard.

Rows are created for each Step down value that is not the same.

Output Cutting Data

Displayed are two sets of data relative to the current Cutting conditions (max feed rate and spindle speed, step over range, cutting speed, chip thickness, and cutting angle range).

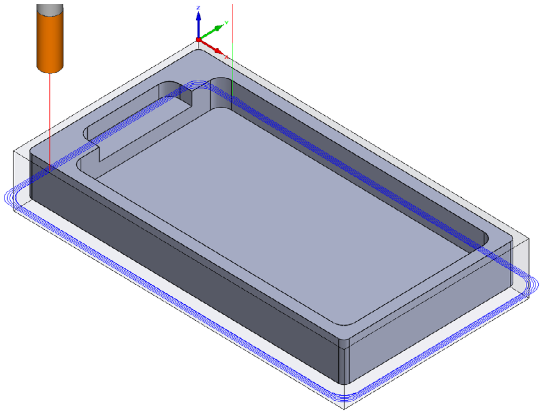
Machining level

The Machining level slider enables you to select from calculated sets of Cutting conditions. Moving the slider up in machining levels provides you with a convenient and intuitive way to control the Material Removal Rate (MRR). Increasing the position of the slider increases MRR and machining aggressiveness.

For this operation, use the Cutting conditions generated by the Wizard based on the default position of the Machining level slider (**3**).

Click to **Save & Calculate** the operation.

Then click to **Simulate**. Run the operation simulation using the default Host CAD mode to view the wireframe tool path. The simulated tool path is performed as follows: the entire contour is machined with a morphing spiral.

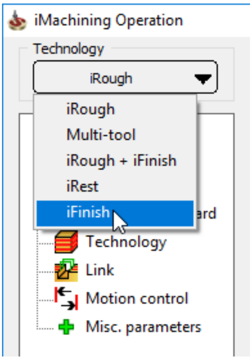


Define the finish machining of the outside contour.

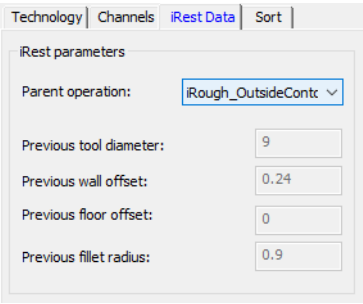
Click to **Save & Copy** the current operation data and to create a new operation with the same parameters.

After the new operation opens for editing, choose **iFinish** for the Technology type.

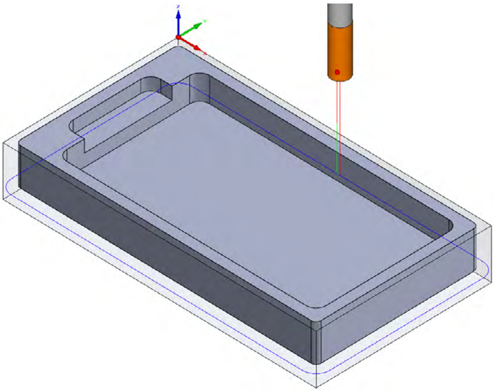
The copied Geometry, Tool and Levels definitions are used for finishing. The default Cutting conditions generated by the Wizard are also used.



On the iRest Data tab of the Technology page, note that the previous **iRough_OutsideContour** operation is selected as the Parent operation by default and the data needed for calculating rest material is populated to the parameters fields automatically.

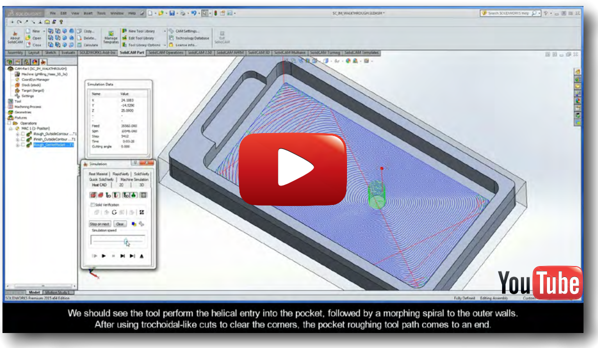


Save & Calculate the operation and then click **Simulate**. Run the operation simulation using the default Host CAD mode. The finishing tool path is performed in a single cutting pass.

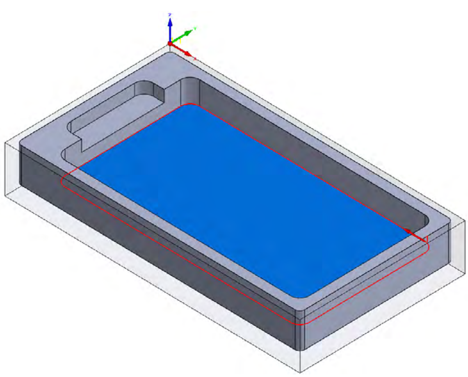


4. Define the machining of the center pocket

In this step, the machining of the center pocket is defined. For this example, the geometry is defined as a closed pocket. The Geometry, Tool and Levels are defined and the Offsets are specified; the iMachining Technology Wizard automatically produces the optimal Cutting conditions. The roughing operation is then copied and the iFinish Technology type is used to define the finishing. Click the play button below to watch the video. Following the video is also a written walkthrough to complete this step.



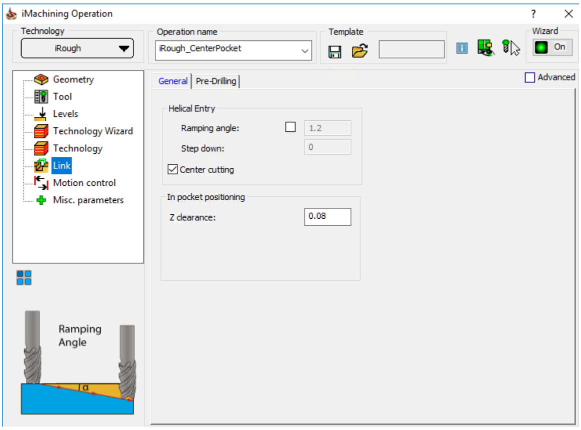
Add a new iMachining 2D operation. Use the **iRough** Technology type to define the rough machining of the center pocket feature. Select the lower contour of the pocket for the Geometry definition.



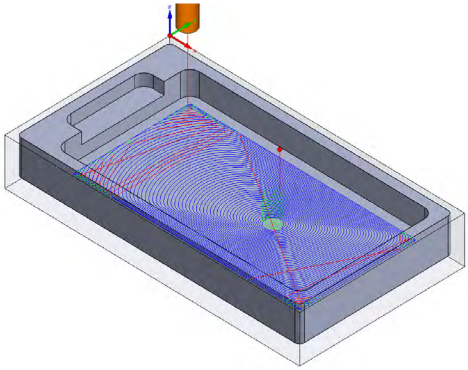
Use the Ø9 mm (0.3438 in) end mill defined in the previous operation. Pick the top face of the Stock model for the Upper level definition and the lower face of the center pocket for the Pocket depth definition. Specify a Wall/island offset and a Floor offset of `0.24` mm (`0.01` in).

The default Cutting conditions generated by the Wizard are used.

The Link page displays the Helical Entry parameters for the operation. The Ramping angle is calculated automatically according to the material hardness and the Machining level aggressiveness. An override check box is provided in the instance you want to manually enter a preferred value.



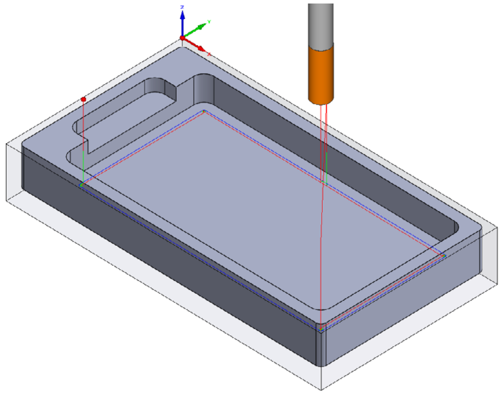
Save & Calculate and then **Simulate** the operation using the Host CAD mode. The tool performs a helical entry into the pocket followed by a morphing spiral to the outer walls.



Define the finish machining of the center pocket.

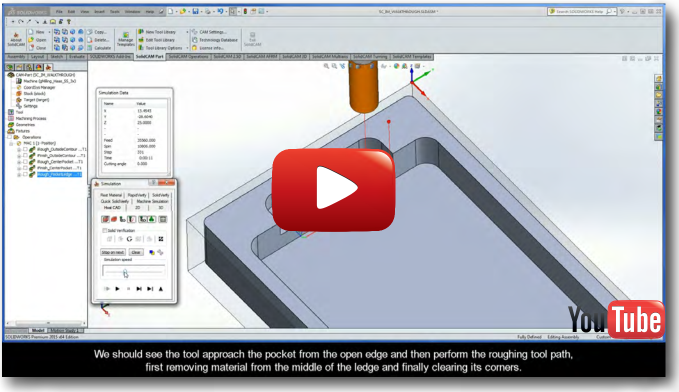
Save & Copy the operation and then choose **iFinish** for the Technology type. The previous **iRough_CenterPocket** operation is automatically selected as the Parent operation.

Save & Calculate and then **Simulate** the operation using the Host CAD mode. The pocket corners are cleared before a final pass is taken along the walls.

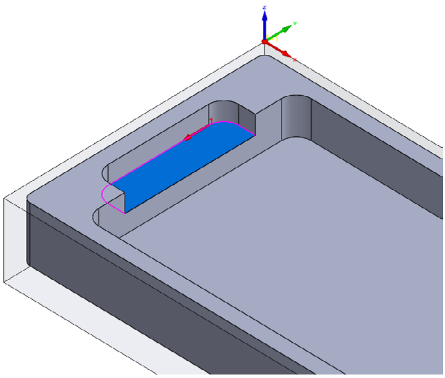


5. Define the machining of the pocket ledge

In this step, the machining of the pocket ledge is defined. For this example, the geometry is defined as a semi-open pocket. The Geometry, Tool and Levels are defined and the Offsets are specified; the iMachining Technology Wizard automatically produces the optimal Cutting conditions. The roughing operation is then copied and the iFinish Technology type is used to define the finishing. Click the play button below to watch the video. Following the video is also a written walkthrough to complete this step.



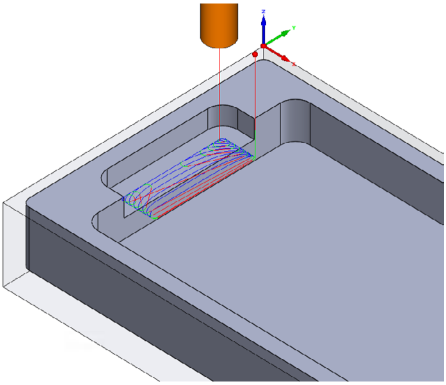
Add a new iMachining 2D operation. Use the **iRough** Technology type to define the rough machining of the pocket ledge feature. Select the lower contour of the pocket ledge and then mark the front edge as open using **Mark open edges**.



Use the Ø9 mm (0.3438 in) end mill. Pick the top face of the Stock model for the Upper level definition and the lower face of the pocket ledge for the Pocket depth definition. Specify a Wall/island offset and a Floor offset of `0.24` mm (`0.01` in).

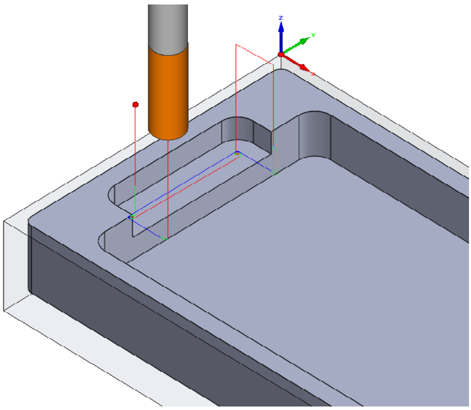
The default Cutting conditions generated by the Wizard are used.

Save & Calculate and then **Simulate** the operation using the Host CAD mode. The tool approaches from the open edge and then performs the roughing tool path, first removing material from the middle of the ledge and then clearing its corners.



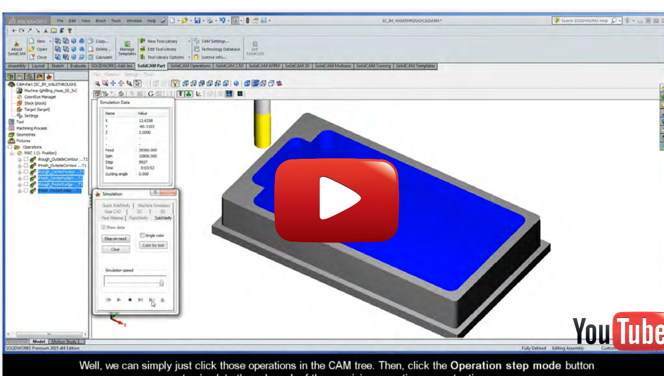
Define the finish machining of the pocket ledge.

Save & Calculate and then **Simulate** the operation using the Host CAD mode. After the corners are cleared, the tool finishes the walls of the pocket ledge in a single cutting pass.

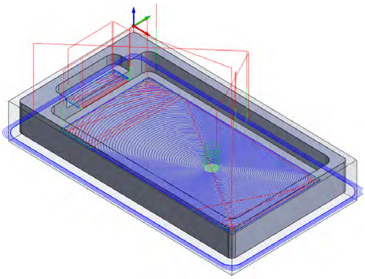


6. Verify the tool path and generate GCode

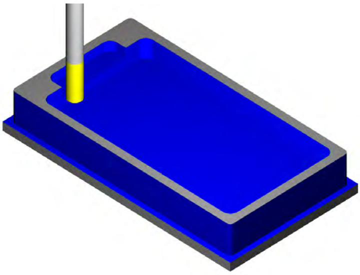
In this step, the iMachining tool path is verified. Included is an in-depth look at SolidCAM's Host CAD simulation options as well as using the SolidVerify simulation mode. A GCode file is also generated and the iMachining technology is shown managing the Feed rates with each cutting move. Click the play button below to watch the video. Following the video is also a written walkthrough to complete this step.



To verify the iMachining tool path for all operations at once, right-click the **Operations** header in the SolidCAM Manager and choose the **Simulate** command. Run the operation simulation using both Host CAD and SolidVerify modes.



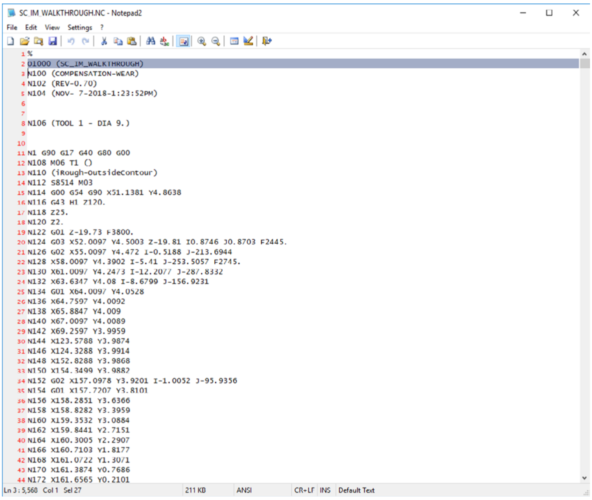
Host CAD



SolidVerify

Now generate the GCode.

Right-click the **Operations** header in the SolidCAM Manager and choose the **Generate** command from the **GCode** All submenu.


SC_IM_WALKTHROUGH.NC - Notepad2

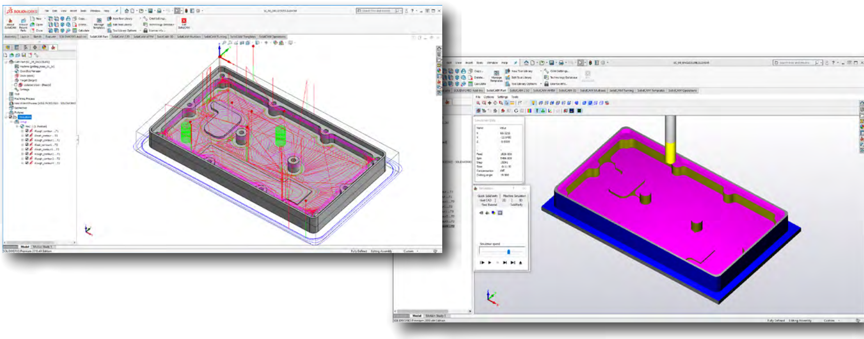
File Edit View Settings ?

```
%
O1000 (SC_IM_WALKTHROUGH)
N100 (COMPENSATION-WEAR)
N102 (REV-0.70)
N104 (NOV- 7-2018-1:23:52PM)


N106 (TOOL 1 - DIA 9.)


N1 G90 G17 G40 G80 G00
N108 M06 T1 ()
N110 (iRough-OutsideContour)
N112 S8514 M03
N114 G00 G54 G90 X51.1381 Y4.8638
N116 G43 H1 Z120.
N118 Z25.
N120 Z2.
N122 G01 Z-19.73 F3800.
N124 G03 X52.0097 Y4.5003 Z-19.81 I0.8746 J0.8703 F2445.
N126 G02 X55.0097 Y4.472 I-0.5188 J-213.6944
N128 X58.0097 Y4.3902 I-5.41 J-253.5057 F2745.
N130 X61.0097 Y4.2473 I-12.2077 J-287.8332
N132 X63.6347 Y4.08 I-8.6799 J-156.9231
N134 G01 X64.0097 Y4.0528
N136 X64.7597 Y4.0092
N138 X65.8847 Y4.009
N140 X67.0097 Y4.0089
N142 X69.2597 Y3.9959
N144 X123.5788 Y3.9874
N146 X124.3288 Y3.9914
N148 X152.8288 Y3.9868
N150 X154.3499 Y3.9882
N152 G02 X157.0978 Y3.9201 I-1.0052 J-95.9356
N154 G01 X157.7207 Y3.8101
N156 X158.2851 Y3.6366
N158 X158.8282 Y3.3959
N160 X159.3532 Y3.0884
N162 X159.8441 Y2.7151
N164 X160.3005 Y2.2907
N166 X160.7103 Y1.8177
N168 X161.0722 Y1.3071
N170 X161.3874 Y0.7686
N172 X161.6565 Y0.2101
```

Ln 3 : 5,568 Col 1 Sel 27 | 211 KB | ANSI | CR+LF | INS | Default Text

You have completed the iMachining Getting Started exercise.

If you would like to review SolidCAM's completed CAM-Part and compare it with yours, click **SC_IM_WALKTHROUGH.prz**.



This exercise is based on another SolidCAM Professor video series that uses the iMachining technology to define the machining of the part shown above. During the definition process, the most common need-to-know topics about iMachining are covered in detail.

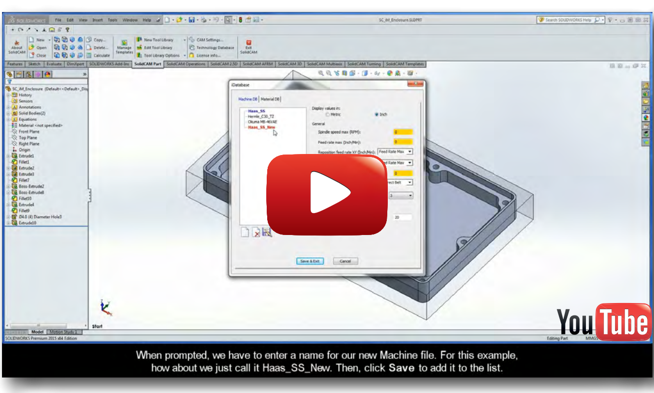
The following steps have to be implemented in order to reach the final CAM-Part:

1. Load the SOLIDWORKS model

Click **SC_iM_Enclosure.SLDPRT** to load the CAD model.

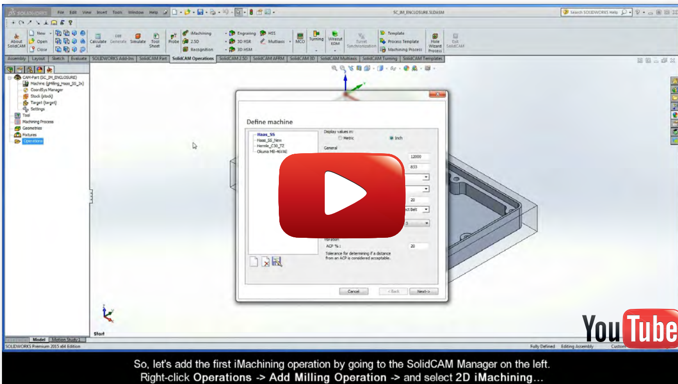
2. Adding a new Machine and Material to the iDatabase

After the part file is loaded on your computer, the following video demonstrates adding a new Machine and Material to the iMachining Database as well as defining the important parameters that are required by the iMachining technology. Click the play button below...



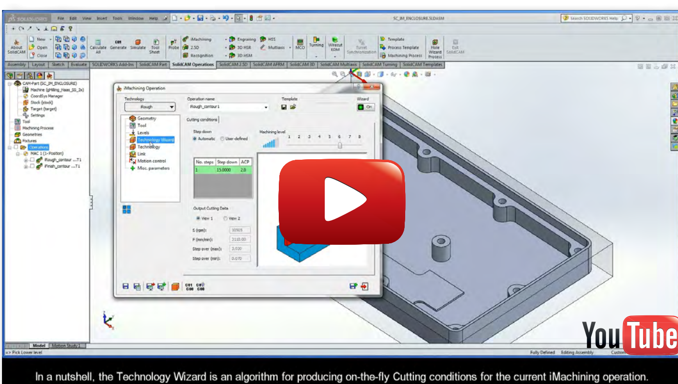
3. Defining the CAM-Part & Adding an iMachining operation

In the following video, a New CAM Project is started and the CAM-Part is defined. The iMachining Data is selected when the first iMachining operation is added to the CAM-Part. Finally, the machining of the outside contour is defined using the iRough and iFinish Technology types in iMachining. Click the play button below...



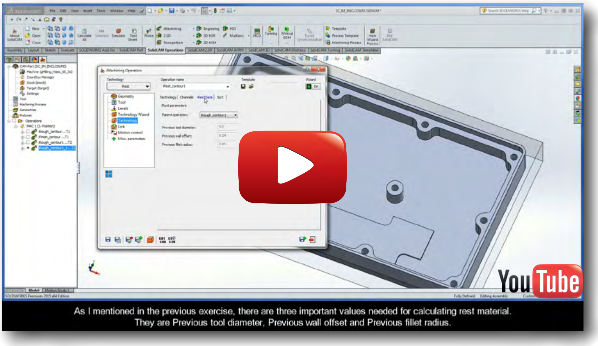
4. Using the iMachining Technology Wizard

In the following video, the iMachining Technology Wizard is discussed in detail and some of the different settings are used to control the Cutting conditions calculations. Click the play button below...



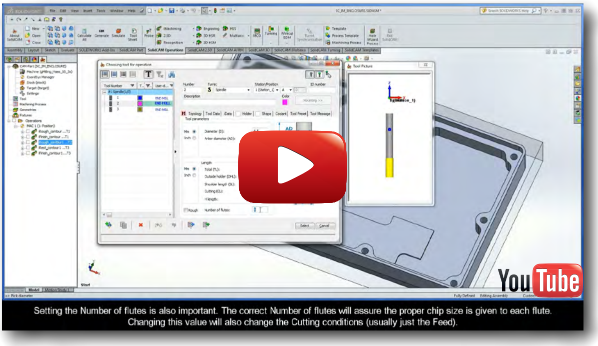
5. Using the iRest Technology type prior to finishing

With iMachining, it is possible to use iFinish directly after iRough. In the following video however, there are narrow areas and corners inside the pocket where the roughing tool cannot fit. In such cases, the iRest Technology type is then used to remove the rest material prior to finishing. The importance of the iRest Data is also explained in detail. Click the play button below...



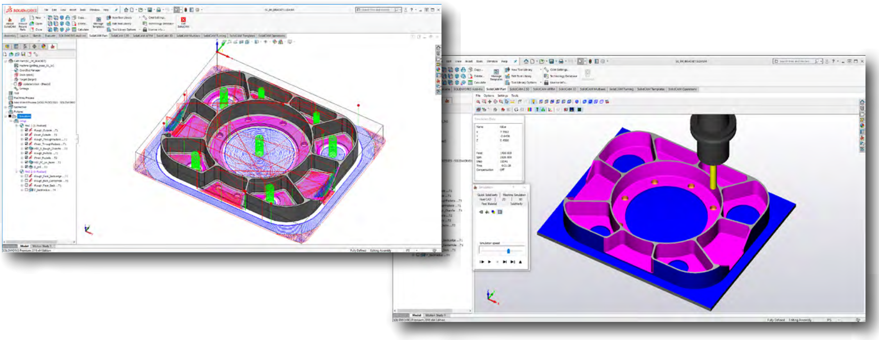
6. The Tool definition and its effects on iMachining

In the following video, the Tool definition and its important parameters related to iMachining are covered in detail. Also shown is how the Wizard calculates the depths and what the importance of ACPs are when machining. Click the play button below...



You have completed the iMachining Getting Started exercise.

If you would like to review SolidCAM's completed CAM-Part and compare it with yours, click **SC_IM_ENCLOSURE.prz**.



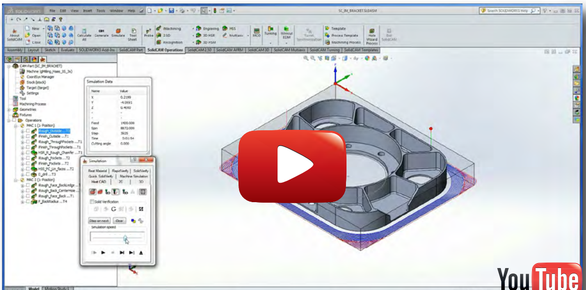
This example illustrates the use of SolidCAM's iMachining technology to define the machining of the part shown above. There are standard 2.5D tool paths and 3D tool paths to aid in the complete CNC program.

Click **SC_IM_BRACKET.prz** to load the SolidCAM part. If your CAM package does not contain the required 3D modules, click **SC_IM_BRACKET_No3D.prz**.

The following operations are defined to perform the machining on a 3-Axis CNC-Machine in two setups, using two SolidCAM Coordinate Systems:

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

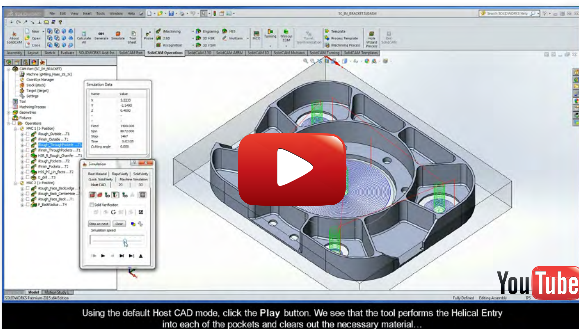
  These iMachining operations define the machining of the outside shape of the part. Two chains are defined, with the first being the stock boundary and the second being the profile around the part. The stock chain is marked as open, which specifies the tool should start machining from that chain. A Ø12.5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



Using the default Host CAD mode, set your desired Simulation speed and click **Play**. During simulation, the corners are cleared first and then the entire outside contour is machined...

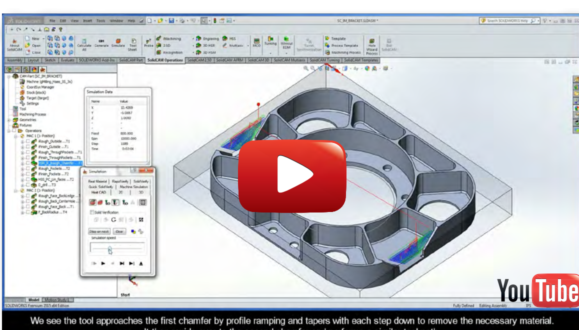
- **Through pockets machining (iRough_ThroughPockets; iFinish_ThroughPockets)**

  These iMachining operations define the machining of the five circular through pockets. Five chains are defined, which represent the five through pockets. A Ø12.5 mm (0.5 in) end mill is used and since the pockets are closed with no Pre-Drilling data or entry chains defined, the tool performs helical ramping to enter into the pockets. iRough has a 0.25 mm (0.01 in) allowance on the walls, which is then removed by the iFinish operation. A Delta depth is specified for both operations, so the tool machines deeper than the part bottom edge.



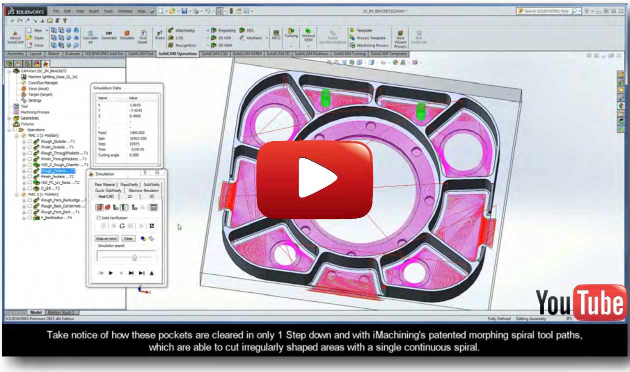
- **Rough machining of angled surfaces (HSR_R_Rough_Chamfer)**

  This HSR operation defines the rough machining of the four large chamfers on the ribs of the part. Two boundaries are picked off the edges the make up the chamfers. A Ø12.5 mm (0.5 in) end mill is used with the Tool Relation set to Centered. A 1.27 mm (0.05 in) Step down is specified with a 0.127 mm (0.005 in) allowance to remain on the surfaces.



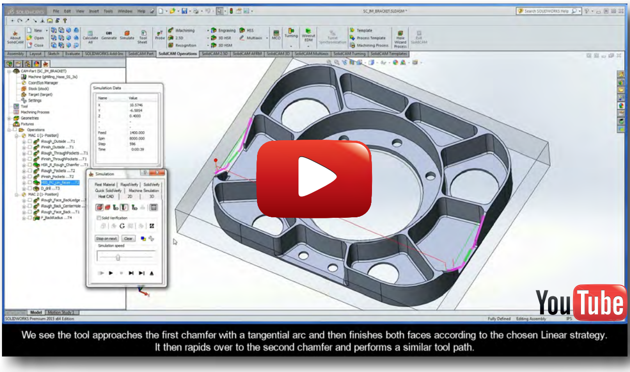
- **Pockets machining (iRough_Pockets; iFinish_Pockets)**

  These iMachining operations define the machining of the three semi-open pockets and the seven closed pockets. Since all ten pockets have the same depth, they can all be machined in one operation. A Ø10 mm (0.4063 in) bull nose mill with a corner radius of 1.6 mm (0.0625 in) is used entering the semi-open pockets through the open edges. The tool enters the remaining pockets using helical ramping and the defined entry geometry. iRough has a 0.25 mm (0.01 in) allowance on the walls, which is then removed by the iFinish operation.



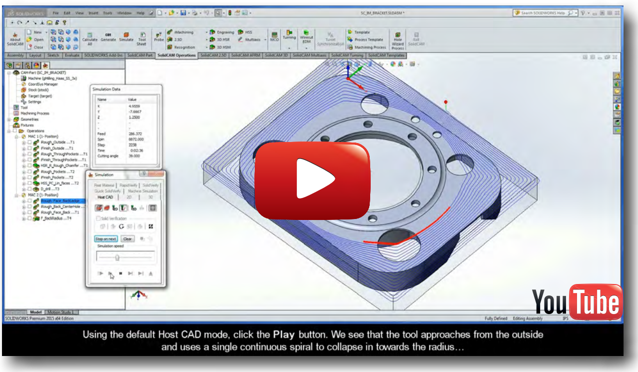
- **Finish machining of angled surfaces (HSS_PC_Lin_faces)**

  This HSS operation defines the finish machining of the four large chamfers on the ribs of the part. The Ø10 mm (0.4063 in) bull nose mill is used and the Linear strategy generates Parallel cuts with a 0.5 mm (0.02 in) Maximum step over. Customized linking is used to allow short repositions and smooth transitions when starting each cut.



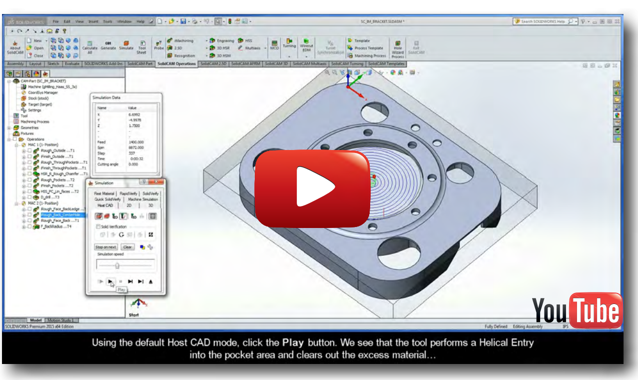
- **Bottom ledge machining (iRough_Face_BackLedge)**

  This iMachining operation defines the bottom ledge machining on the underside of the part. Two chains are defined, with the first being the stock boundary and the second being the bottom of the floor radius. Using a Ø12.5 mm (0.5 in) end mill, the tool starts machining from the stock chain (marked as open) and collapses in towards the radius. The floor radius is not machined at this time.



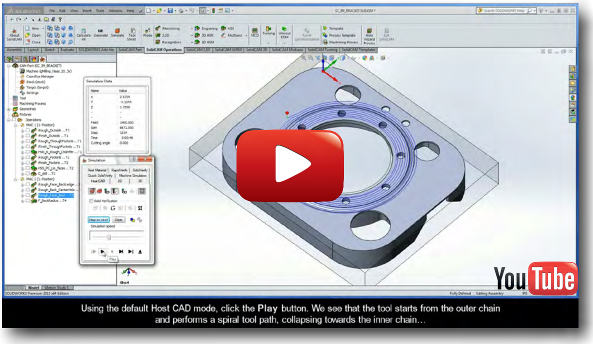
- **Cutting of excess material from through hole (iRough_Back_CenterHole)**

  This iMachining operation is defined to machine away the excess material from the center through hole of the part. This excess material was used for clamping in the first setup. A single closed chain is defined and using a Ø12.5 mm (0.5 in) end mill, the excess material is removed. In this case, the default allowance can be used since the desired wall was finished during the top side machining.



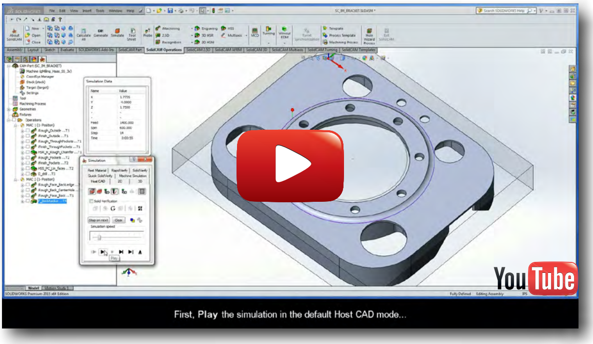
- **Bottom face machining (iRough_Face_Back)**

  This iMachining operation defines the bottom face machining on the underside of the part. Two chains are defined, with the first being the outside boundary of the face and the second being an offset edge created in SOLIDWORKS. Using a Ø12.5 mm (0.5 in) end mill, a morphing spiral tool path is performed from the outer open chain, collapsing towards the inner closed chain.



- **Floor radius finishing (F_BackRadius)**

  This Profile operation defines the 6.35 mm (0.25 in) floor radius finishing on the underside of the part. The chain is defined as the bottom edge of the radius. A Ø12.5 mm (0.5 in) ball nose mill is used with the Tool side set to Center. A 0.13 mm (0.005 in) offset remains on the floor after the first roughing pass and is then removed with a single finishing pass. A 0.25 mm (0.01 in) arc is used for the Lead in/out.



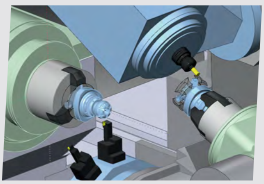
Congratulations! The SolidCAM & iMachining Getting Started Guide is now completed. For additional documentation and many more Professor videos, visit us on the web at www.solidcam.com.

SolidCAM 2019

Getting STARTED

SolidCAM & iMachining

The Complete Range of Manufacturing Applications Inside SOLIDWORKS®



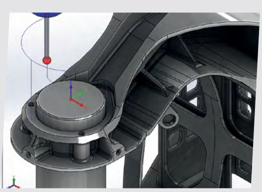
SolidCAM is the leading and fastest growing developer of integrated CAM software solutions for the manufacturing industry. SolidCAM supports the complete range of major manufacturing applications in Milling, Turning, Mill-Turn and WireEDM totally integrated inside SOLIDWORKS.

The Revolutionary iMachining® Module

SolidCAM's iMachining module is a giant leap forward in CNC machining technology, reducing cutting times by up to 70% and increasing tool life dramatically. iMachining achieves these advantages by using a patented "Controlled Stepover" technology and managing feed rates throughout the entire tool path, ensuring constant tool load while allowing much deeper and more efficient cutting.

iMachining is driven by a knowledge-based Technology Wizard which considers the machine being used, the material being cut and the cutting tool data to provide optimal values for the Cutting conditions. With its Morphing spiral tool paths, controlled tool load at each point along the tool path, moating of islands to enable continuous spiral cuts (even with multiple islands), and its automatic thin wall avoidance, iMachining brings efficiency to a new level for CAM users.

The Highest Level of SOLIDWORKS Integration



SolidCAM provides the highest level of CAD integration with its seamless, single-window integration and full associativity to SOLIDWORKS, ensuring automatic tool path updates for your CAD revisions.

SolidCAM Turns Your SOLIDWORKS System into the Best CAD/CAM Solution.

youtube.com/SolidCAMProfessor
youtube.com/SolidCAMiMachining

facebook.com/SolidCAM
facebook.com/iMachining

iMachining Successes
solidcam.com/successes/imachining-successes

www.solidcam.com

SolidCAM
iMachining – The Revolution in CAM!