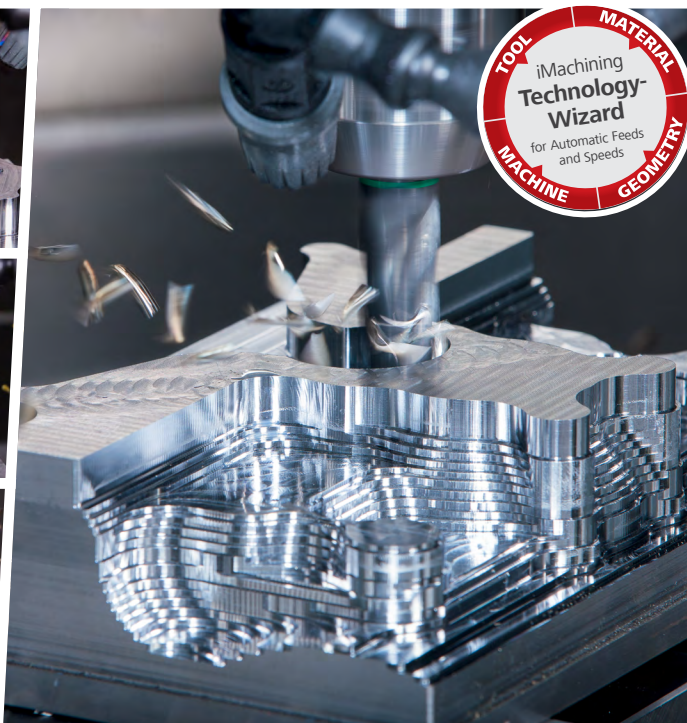
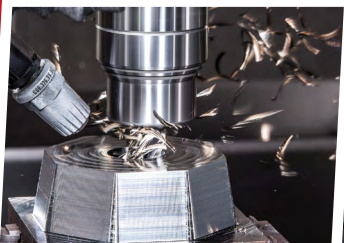


SolidCAM 2020

The unique, revolutionary Milling technology
iMachining[®]
patent by SolidCAM

TIME SAVINGS
70%
... AND MORE!



Getting **STARTED**

SolidCAM & iMachining



SolidCAM + SOLIDWORKS

The Complete Integrated Manufacturing Solution



SolidCAM

iMachining – The Revolution in CAM!

Contents

Introduction	iv
About this document	iv
SolidCAM	
The facts about SolidCAM	1
Getting Started with SolidCAM	3
Simple Cover Machining	7
iMachining	
The facts about iMachining	13
Getting Started with iMachining	15
iMachining Walkthrough	17
iMachining of an Enclosure	35
iMachining of a Bracket	39

Introduction

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 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 2020 and SolidCAM 2020.






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:

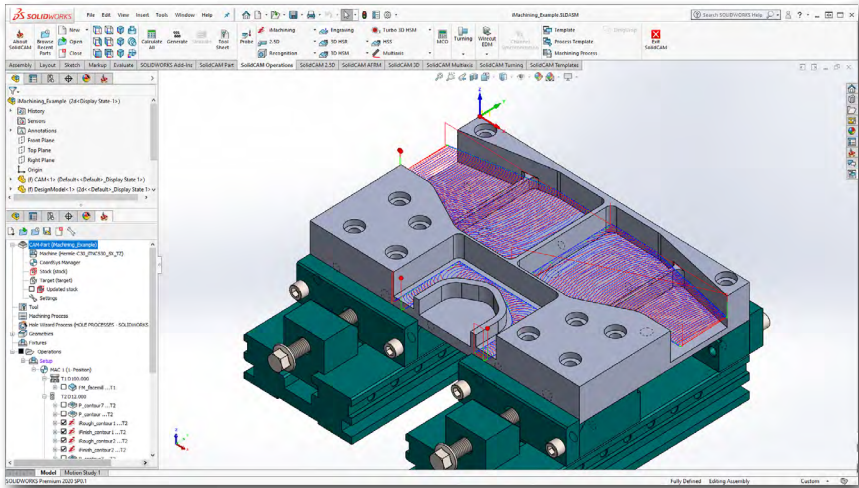
Bold	This style emphasizes SOLIDWORKS/ SolidCAM options and commands. For example, click the New button...
 3. Define the CAM-Part	The mouse icon and numbered bold text indicate the steps required to complete an exercise.
 Create and define...	The stairs icon indicates the start of the written steps 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.SLDPR	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 

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.

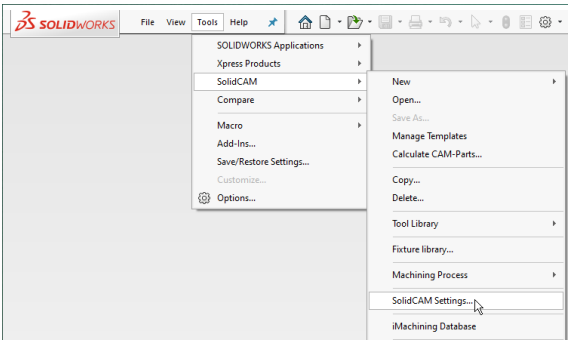
The successful manufacturing companies of today are using integrated CAD/CAM systems to get their products to market faster and reduce costs. With SolidCAM's gold-certified status and seamless single-window integration in SolidWorks, organizations of all sizes are realizing the benefits of SolidCAM +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

Getting Started with SolidCAM

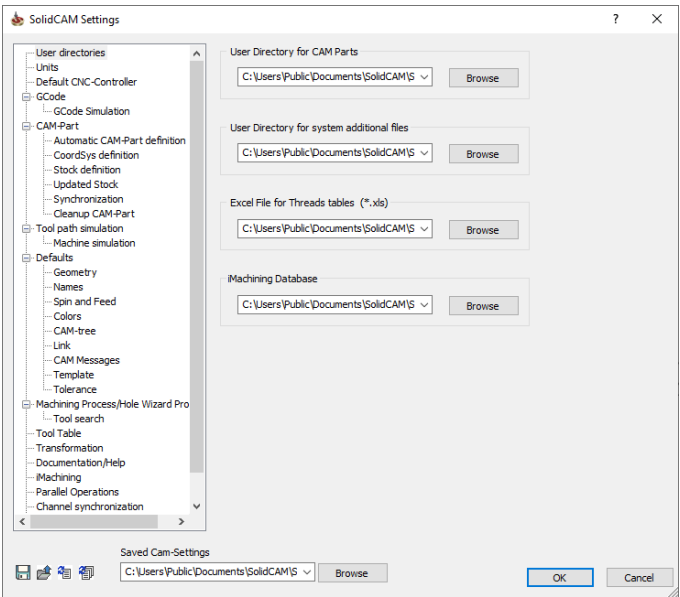
SolidCAM uses initial default settings to simplify certain tasks. For example, the CAM-Part is created and defined automatically. As a new user however, it is important to understand the full process of defining the CAM-Part and how the CAM-Part features are built. For this purpose, some of the default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

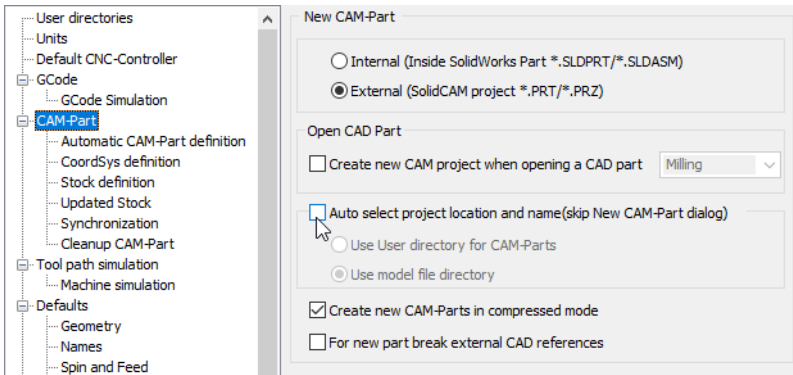


SolidCAM is started and the SolidCAM Settings dialog box is displayed.

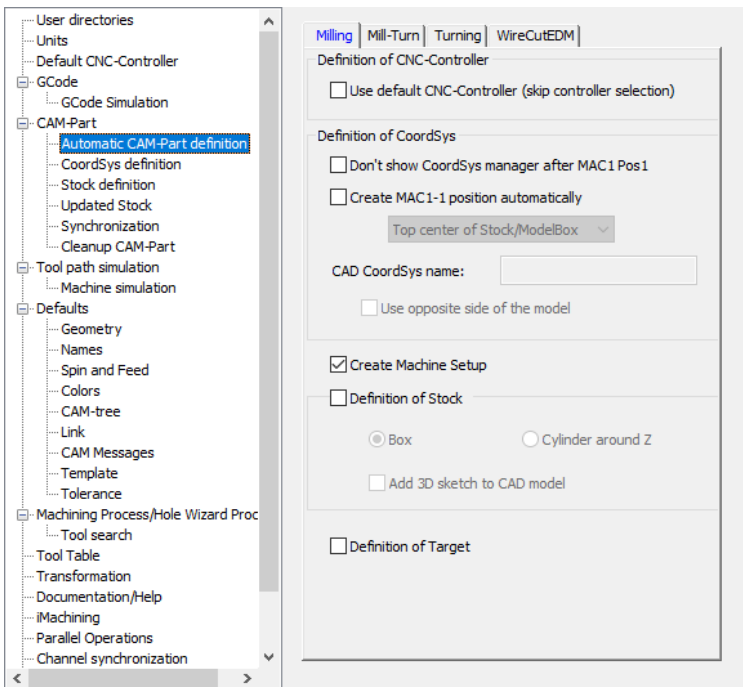




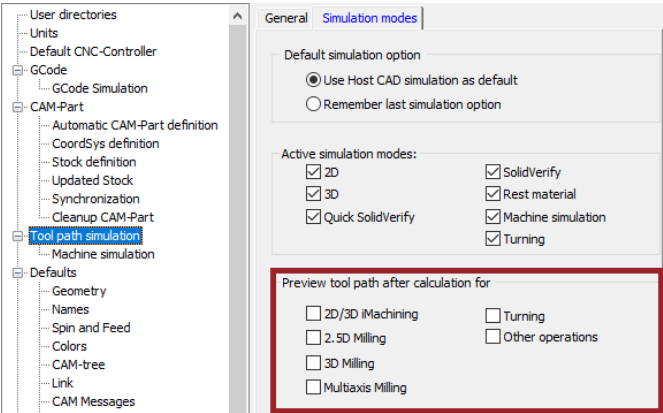
On the CAM-Part page, the option **Auto select project location and name** should be disabled; the CAM-Part creation will not be automated.



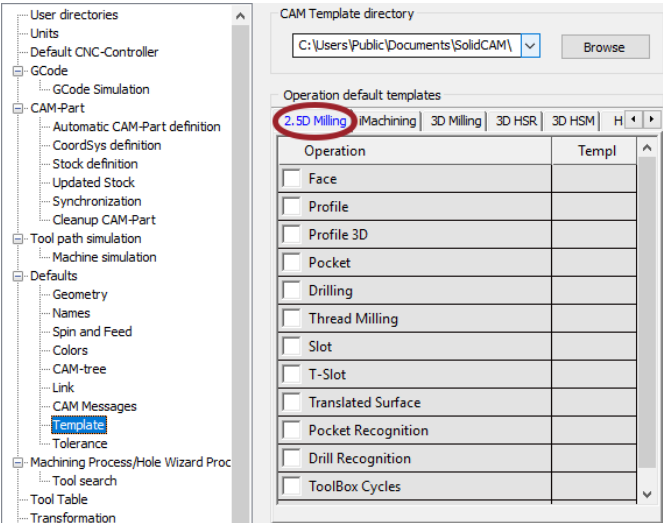
On the Automatic CAM-Part Definition subpage > Milling tab, disable any options relevant to **Definition of CNC-Controller**, **Definition of CoordSys**, **Definition of Stock** and **Definition of Target**.



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.



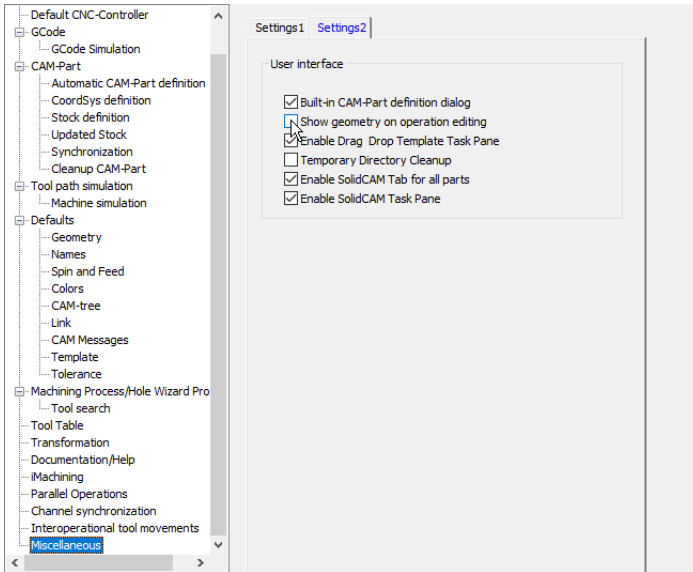
On the Defaults > Template subpage, Operation default templates for 2.5D Milling will not be used.



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



On the Miscellaneous page > Settings2 tab, you can also disable the option to **Show geometry on operation editing**.

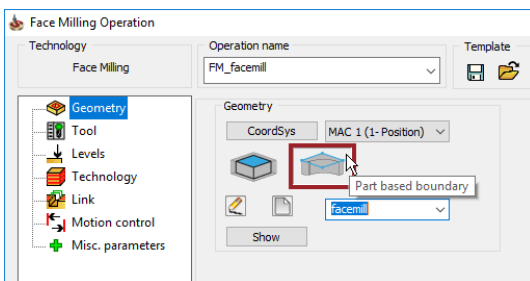


Most beneficial when using the Modify Geometry feature of SolidCAM, this setting is not needed in any of the Getting Started exercises.

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

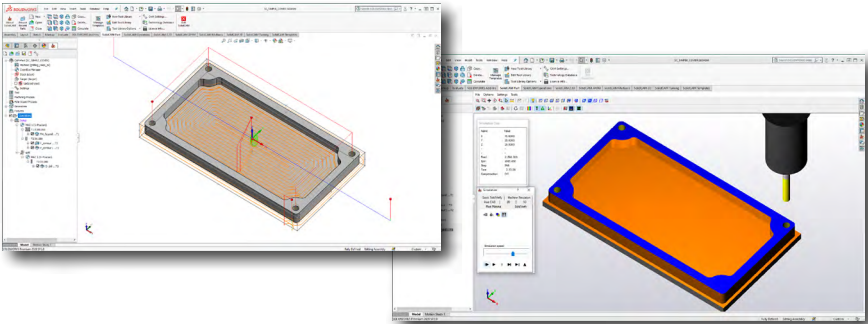
Click **OK** to confirm your changes.

Important Note: *The Jumpstart video series was recorded with previous versions of SolidCAM. In SolidCAM 2020, use **Part based boundary** for the Geometry definition in your Face Milling operation.*



There are no other notable differences.

Simple Cover Machining



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:



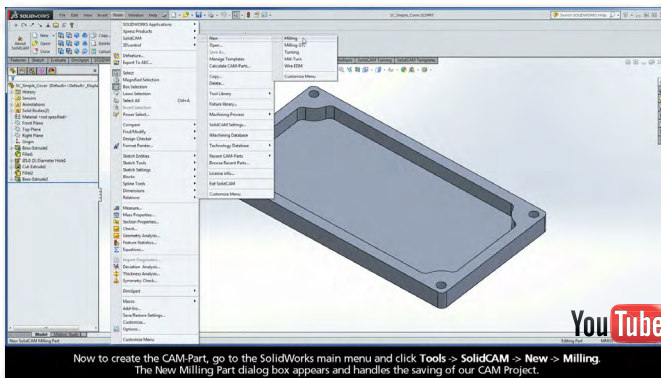
1. Load the SOLIDWORKS model

Click [SC_Simple_Cover.SLDPRT](#) to load the CAD model.



2. Create a New Milling CAM Project

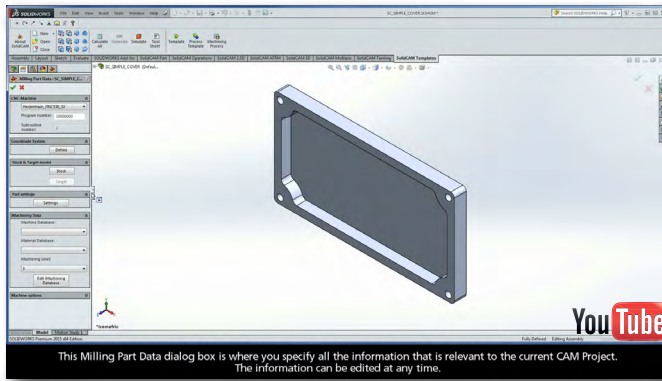
After the part file is loaded on your computer, the following video demonstrates how to start SolidCAM and create a New Milling CAM Project. Prior to creating the Project, the CAM Settings are customized to prepare for the upcoming tutorial videos. These include setting the defaults for the CAM-Part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





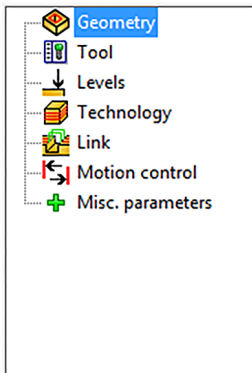
3. Define the CAM-Part

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

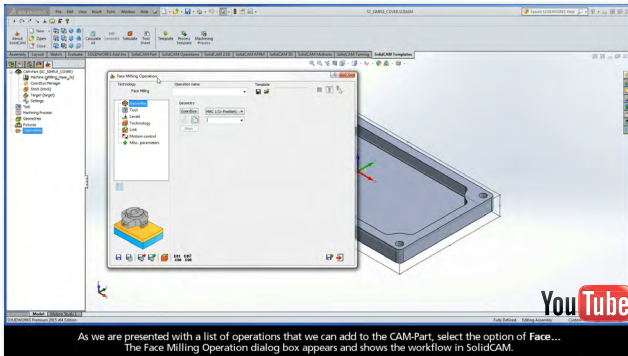
The part programming is completed in just four operations, which begin on the following page. As shown in the illustration below, the workflow in SolidCAM is displayed in each of the Operation dialog boxes. The Geometry is defined first, followed by creating and choosing a Tool, picking the Milling levels, 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.

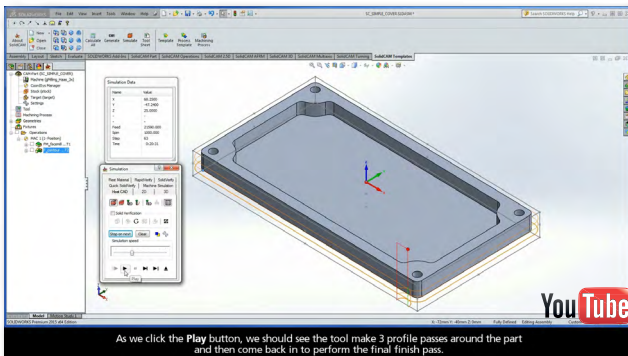
4.1 Add a Face Milling operation

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 $\varnothing 100$ 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

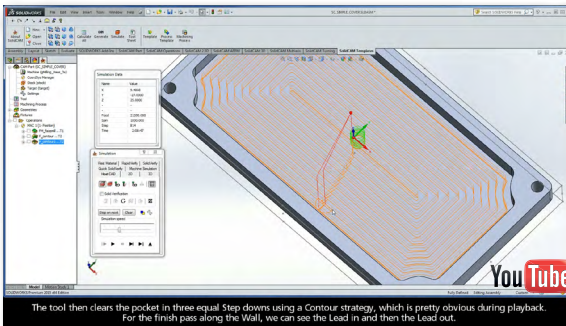
In this video, a Profile operation is added to the CAM-Part. The geometry is selected as a chain that runs along the outside contour of the Target model. A $\varnothing 6$ mm (0.25 in) end mill is defined for the operation. The tool is set to perform an Arc Lead in/out and climb cut at a Constant depth. 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...





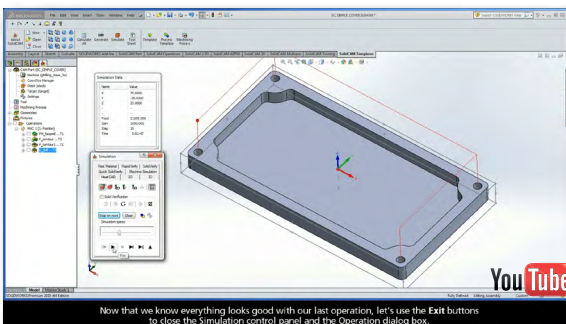
4.3 Add a Pocket operation

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 $\varnothing 6$ 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

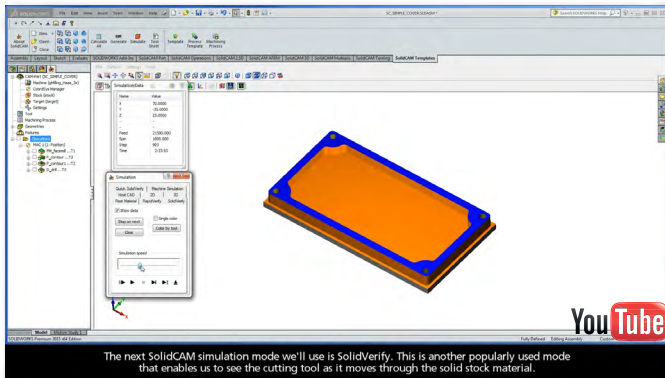
In this video, a Drilling operation is added to the CAM-Part to complete the part programming. SolidCAM finds the centers of all circle entities and defines their positions for the drill geometry. A drilling tool is defined and choosing a holder is also shown. The Levels are picked directly off the model, 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...



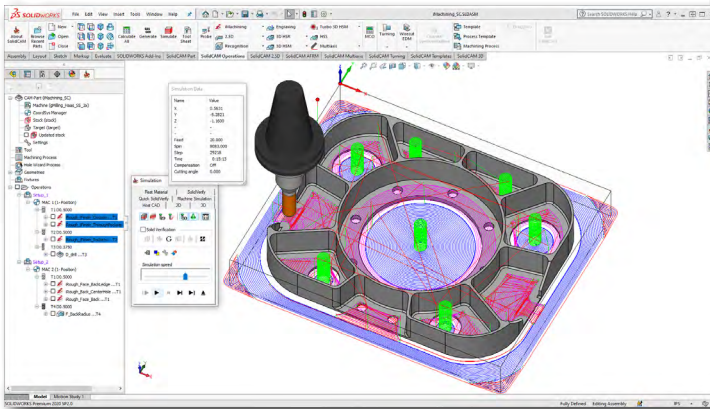
You have completed the SolidCAM Getting Started exercise.



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

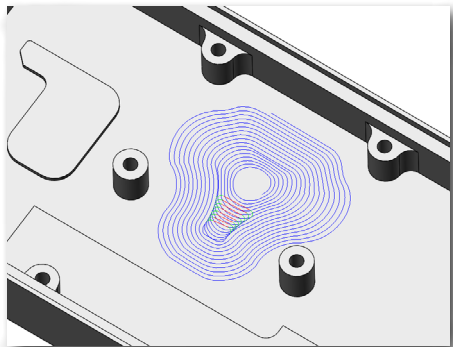
SolidCAM's iMaching™ is an intelligent high speed milling technology that is designed to produce fast and safe CNC programs to machine mechanical parts with *first part success* performance. The word *fast* meaning significantly faster than traditional machining at its best and the word *safe* meaning 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

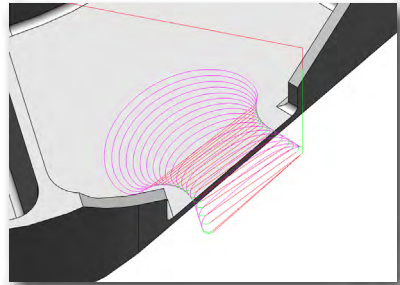
iMachining generates morphing spiral tool paths that spiral either outwardly from some central point of a walled area, gradually adopting the form of and nearing the contour of the outside walls, or inwardly from an outside contour of an open area to some central point or inner contour of an island. 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.



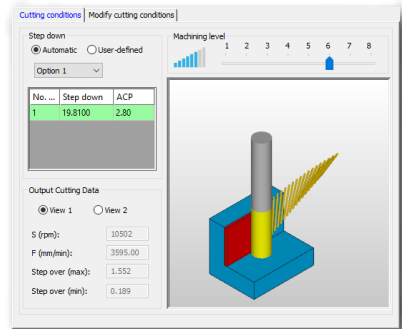
In some open areas, where the shape is too irregular to completely remove with a single spiral, proprietary topology analysis algorithms and channels are used to subdivide the area into a few large irregularly shaped sub-areas and then machines each of them by a suitable morphing spiral, achieving over 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).

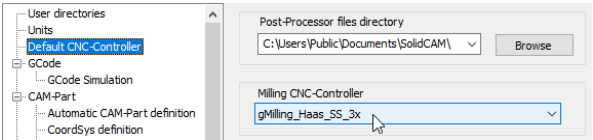


Another critical task performed by the Technology Wizard is dynamically adjusting the feed to compensate for the dynamically varying cutting angle – a by-product of the morphing spiral, thus achieving a constant load on the tool, which again maximizes tool life.

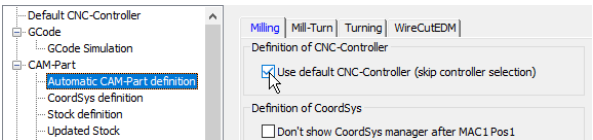
Getting Started with iMachining

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.

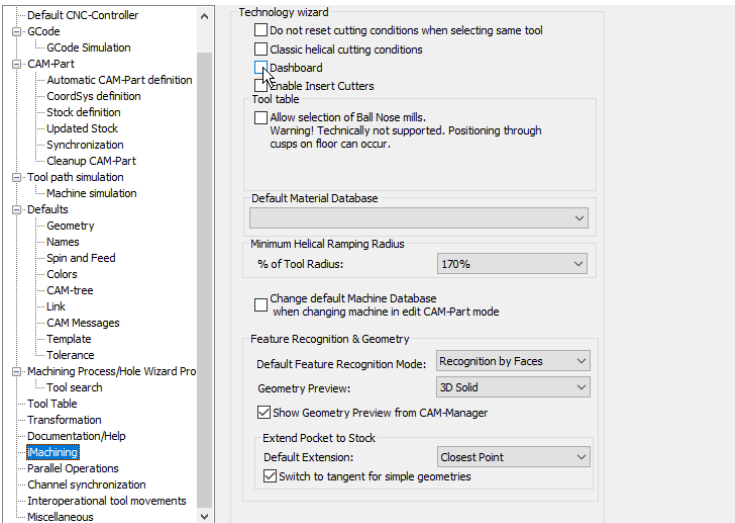


On the Automatic CAM-Part Definition subpage > Milling tab, enable the option **Use default CNC-Controller**.

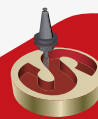


These settings will later simplify your selections since the remaining CAM-Parts use the gMilling_Haas_SS_3x controller.

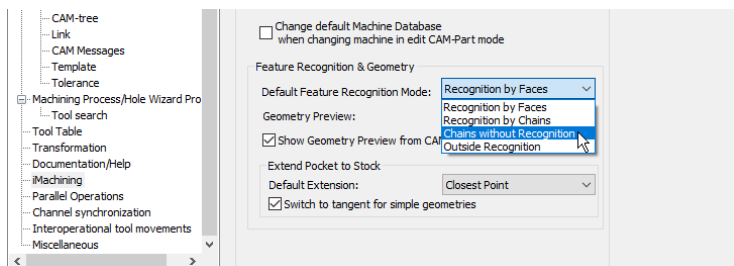
On the iMachining page, disable the **Dashboard** option.



Featured in the iMachining Operation dialog box when enabled, the Dashboard is primarily for experienced users.



In the Feature Recognition & Geometry section, select **Chains without Recognition** for the default mode. This is equivalent to SolidCAM's standard chaining method, which is widely used for 2.5D operations in SolidCAM.

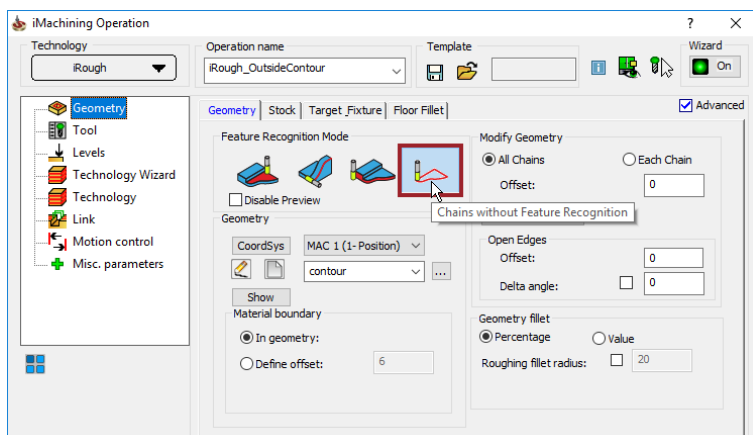


The iMachining Geometry Preview can also be turned **Off**.

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

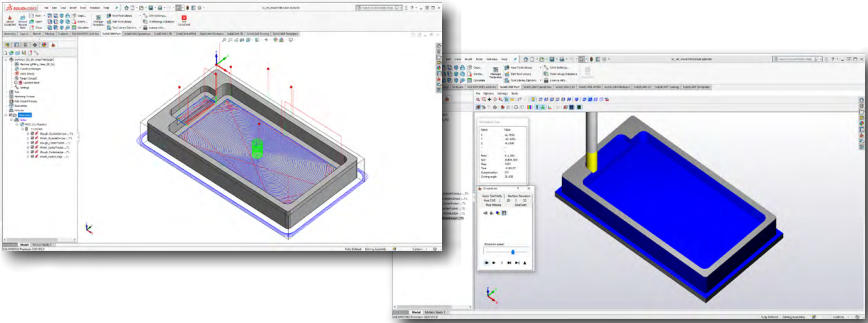
Click **OK** to confirm your changes.

Important Note: *The remaining videos were recorded with previous versions of SolidCAM. In SolidCAM 2020, use **Chains without Feature Recognition** for the Geometry definition in your iMachining operations.*



There are no other notable differences.

iMachining Walkthrough



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:



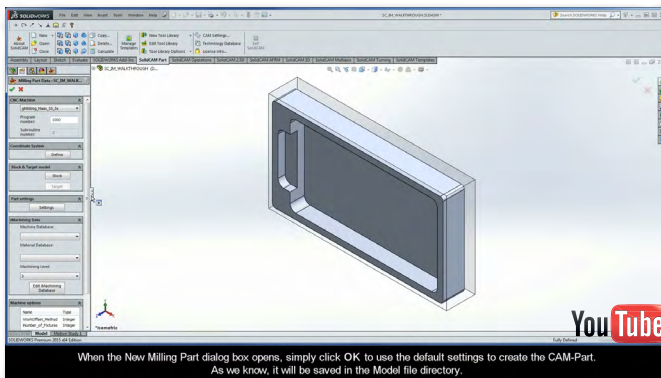
1. Load the SOLIDWORKS model

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.



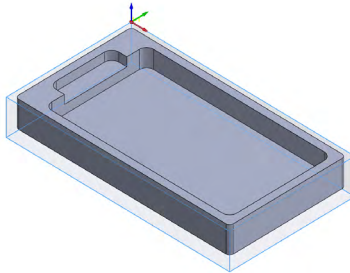
When the New Milling Part dialog box opens, simply click OK to use the default settings to create the CAM-Part. As we know, it will be saved in the Model file directory.



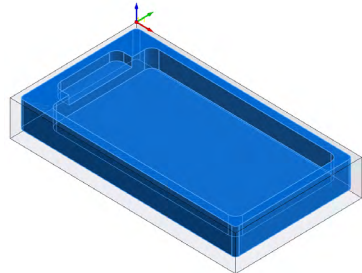
Create and define the CAM-Part.

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

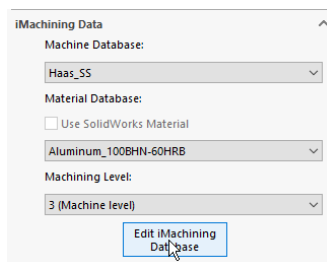
For this exercise, choose **Haas_SS** from the Machine Database list and choose **Aluminum_100BHN-60HRB** from the Material Database list.



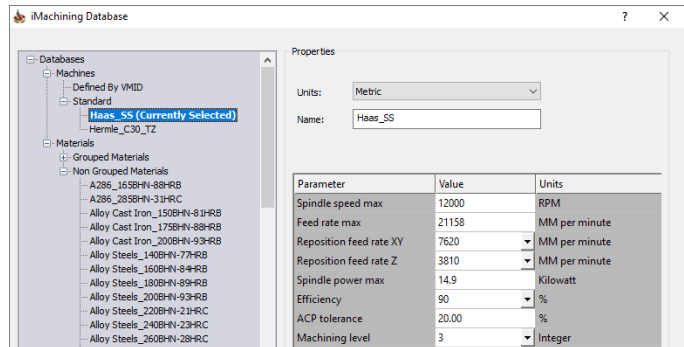
iMaching Database

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

Clicking the **Edit iMaching Database** button will display the iMaching Database dialog box.

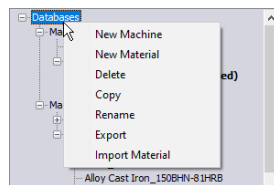


This dialog box is the interface that enables you to add new/edit existing machines and work materials contained in the iMaching Database.



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



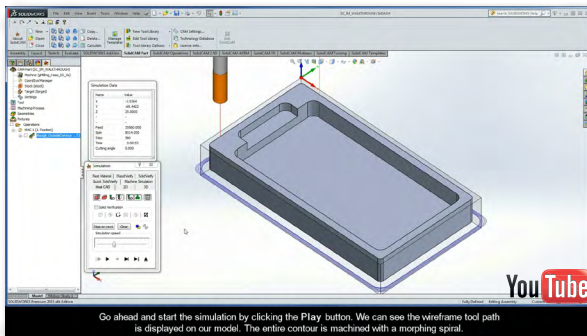
The following commands enable you to:

- **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.

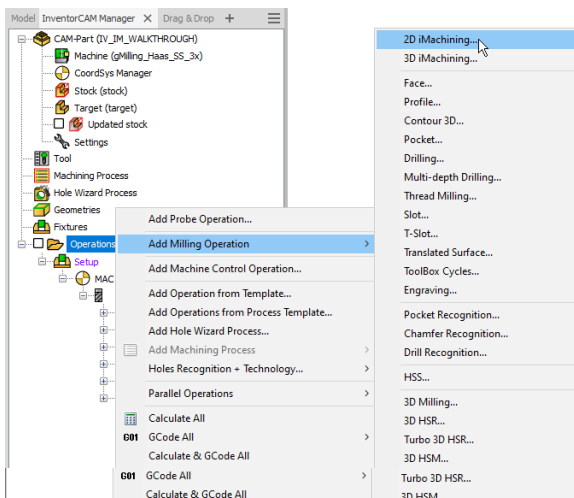


3. Define the machining of the outside contour

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.

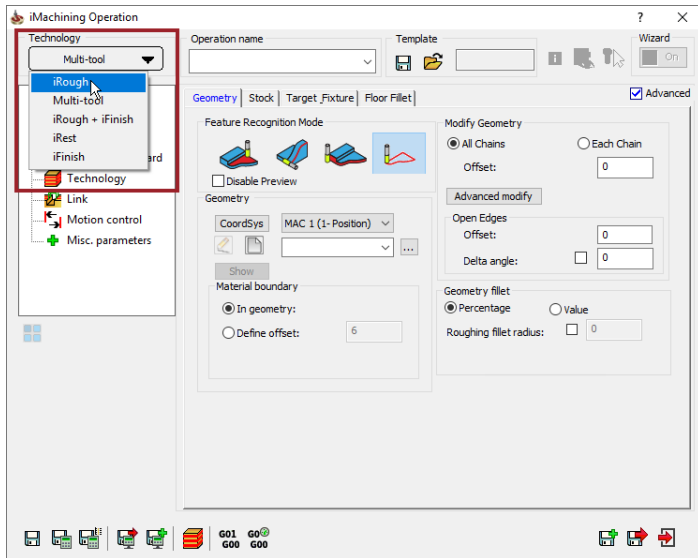


To add an iMachining operation to the CAM-Part, right-click the **Operations** header in the SolidCAM Manager and from the **Add Milling Operation** submenu, choose **2D iMachining**.

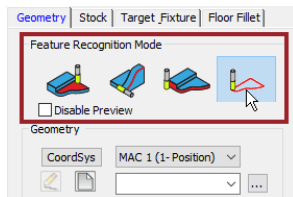


The iMachining Operation dialog box is displayed.

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



For the Geometry definition, **Chains without Feature Recognition** is used.



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



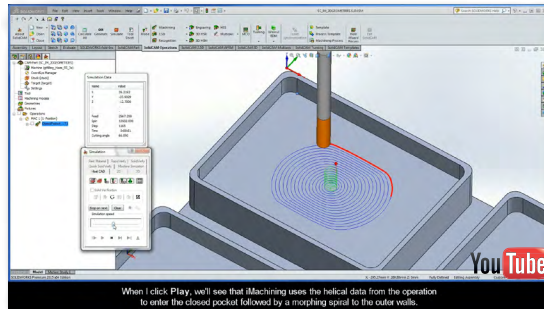
iMachining Geometry definitions

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

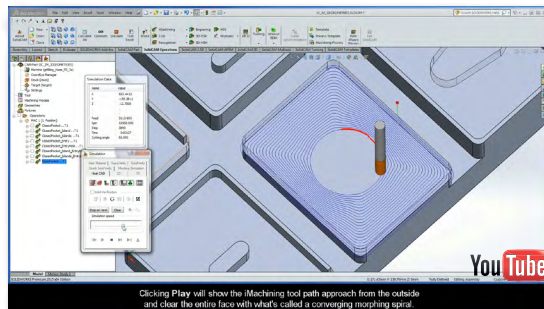
The videos on the following page demonstrate defining the different geometry types using SolidCAM's standard chaining method and the resulting iMachining tool path.



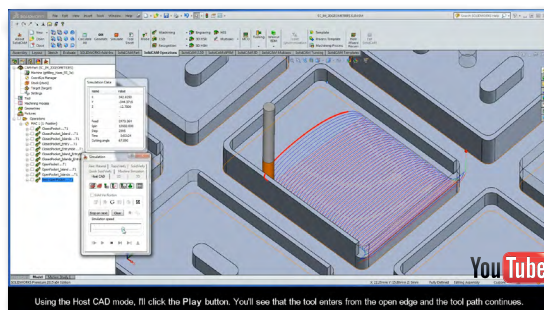
Closed pocket geometries in iMaching



Open pocket geometries in iMachining



Semi-open pocket geometries in iMachining

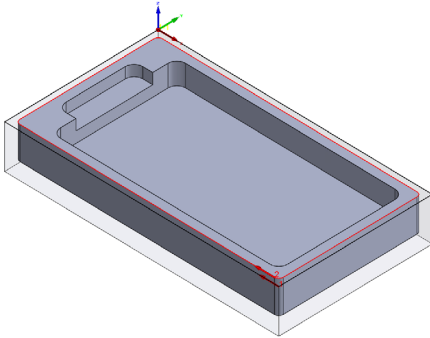


The completed CAM-Part is provided if you want to view the Geometry definitions and the iMachining tool path techniques in SolidCAM.

Click [SC_IM_GEOMETRIES.prz](#) to load the SolidCAM part.

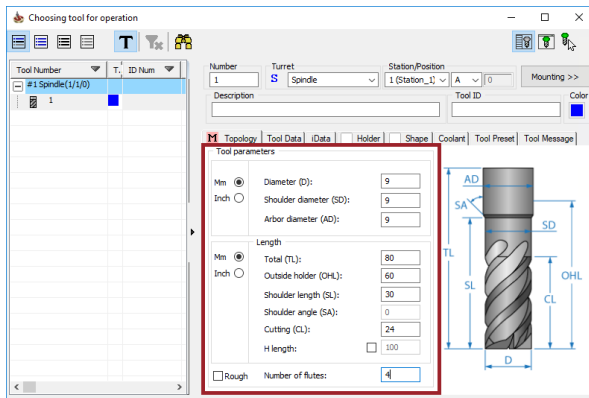
For this operation, the machining geometry is defined as an open pocket with island.

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 $\varnothing 9$ mm (0.3438 in).

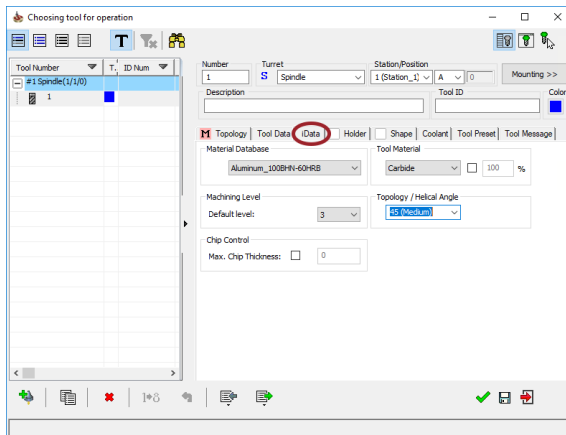
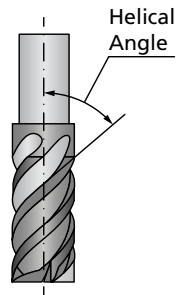



Define the remaining tool parameters as follows:

- 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.

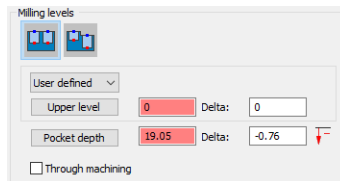


On the iData tab, the default 45 (Medium) value is used for the Helical Angle parameter. This parameter affects the Cutting conditions and Step down values generated by the iMachining Technology Wizard.

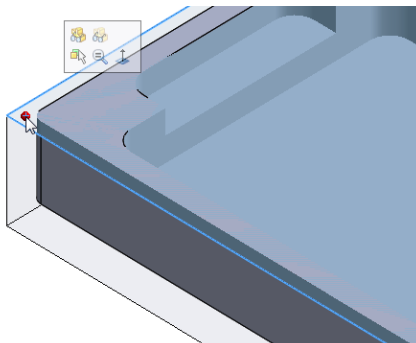


Click  to confirm the Tool definition.

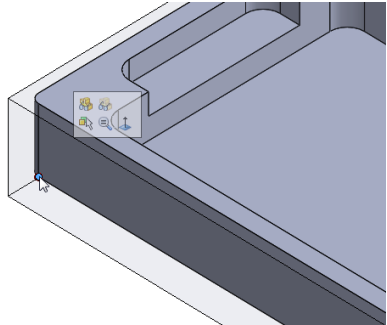
Switch to the **Levels** page to define the Milling levels for the operation.



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.



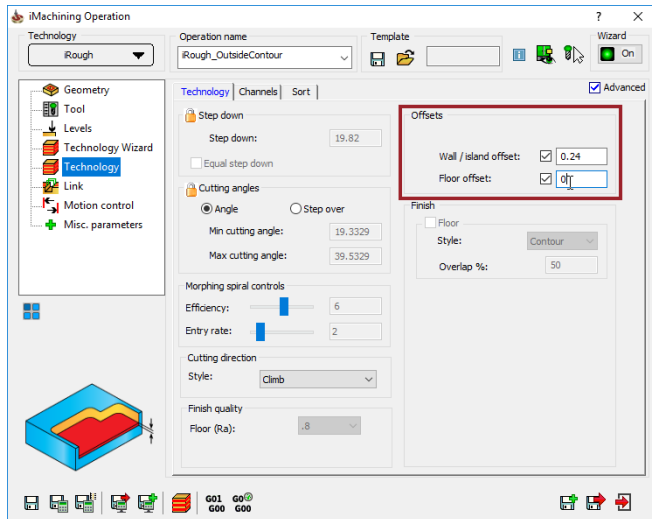
Click the **Pocket depth** button and pick on the bottom edge of the Target model to define the machining depth.



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).

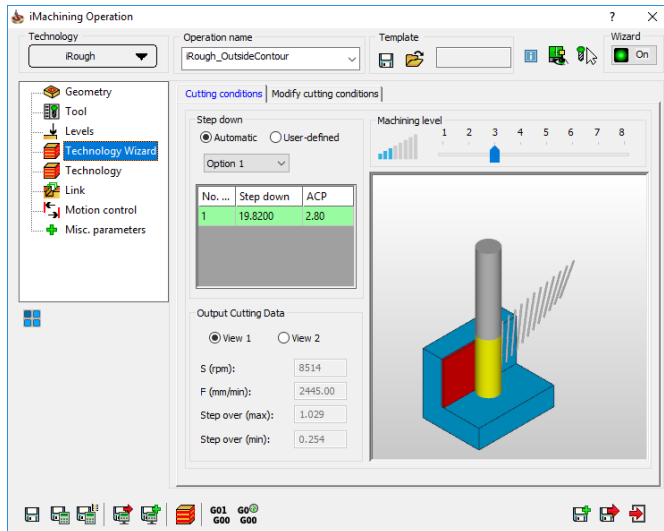
Switch to the **Technology** page and specify a Wall/island offset of 0.24 mm (0.01 in) and a Floor offset of 0 .



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).

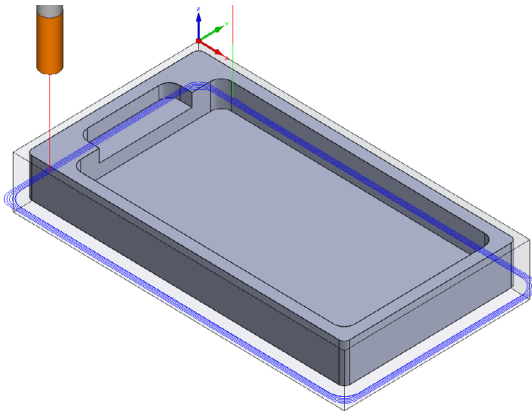
Machining level

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.



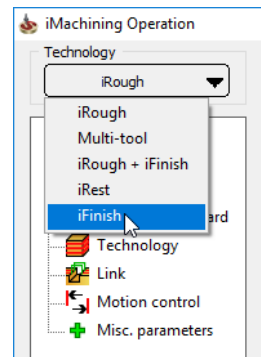
Define the finish machining of the outside contour.



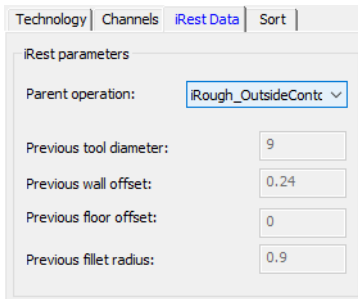
Click  to **Save & Copy** the current operation data and to create a new operation with the same parameters.

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

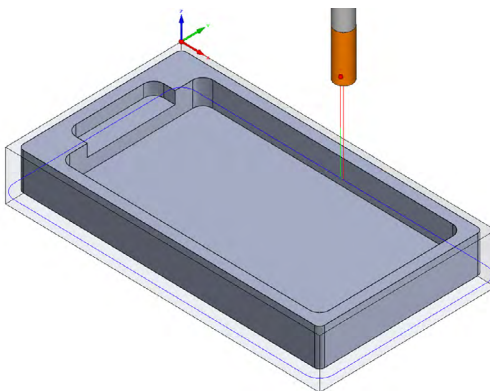
The copied Geometry, Tool and Levels definitions are used for finishing. The default Cutting conditions generated by the Wizard are also used.



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.



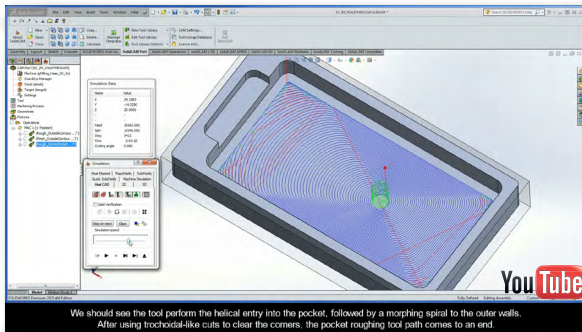
Save & Calculate the operation and then click **Simulate**. Run the operation simulation using the default Host CAD mode. The finishing tool path is performed in a single cutting pass.



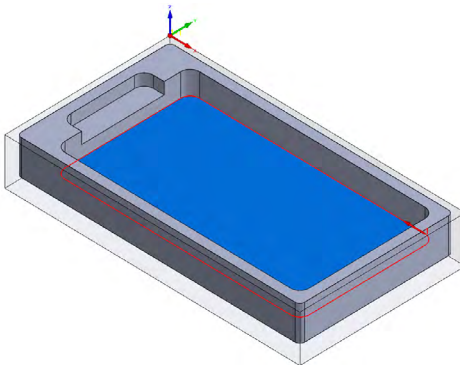


4. Define the machining of the center pocket

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.

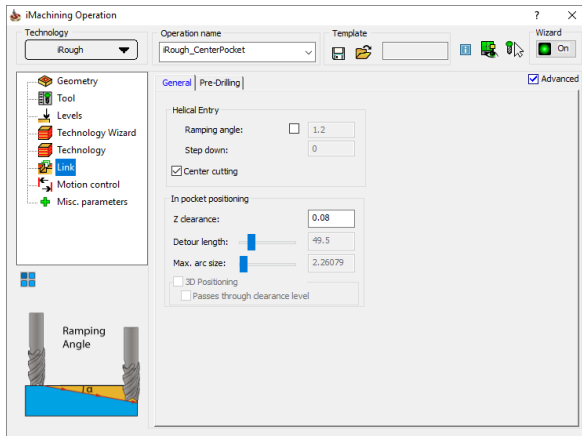


Use the $\varnothing 9$ mm (0.3438 in) end mill defined in the previous operation. Pick the top face of the Stock model for the Upper level definition and the lower face of the center pocket for the Pocket depth definition. Specify a Wall/island offset and a Floor offset of 0.24 mm (0.01 in).

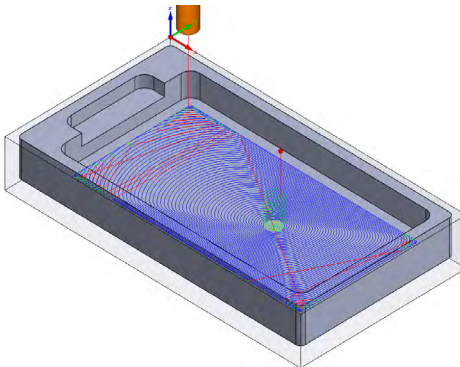


The default Cutting conditions generated by the Wizard are used.

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.



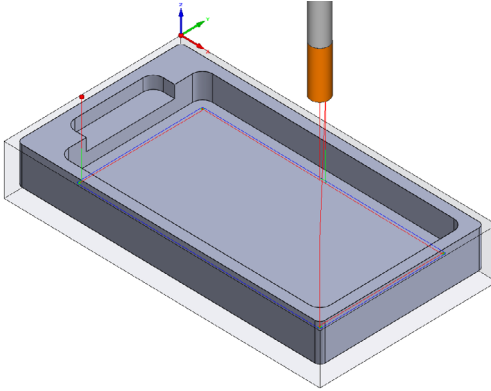
Save & Calculate and then **Simulate** the operation using the Host CAD mode. The tool performs a helical entry into the pocket followed by a morphing spiral to the outer walls.



Define the finish machining of the center pocket.

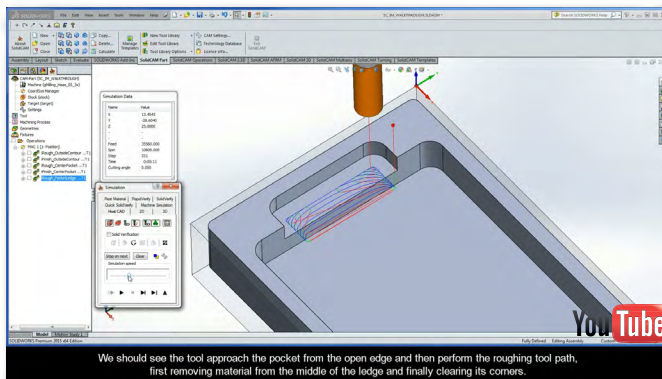
Save & Copy the operation and then choose **iFinish** for the Technology type. The previous **iRough_CenterPocket** operation is automatically selected as the Parent operation.

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.



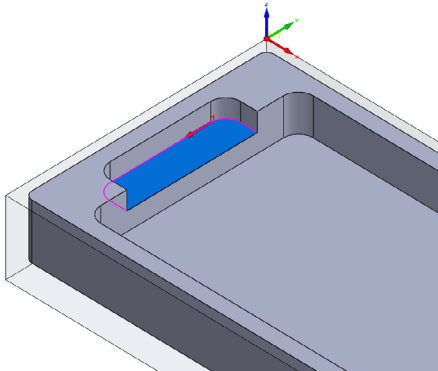
5. Define the machining of the pocket ledge

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.





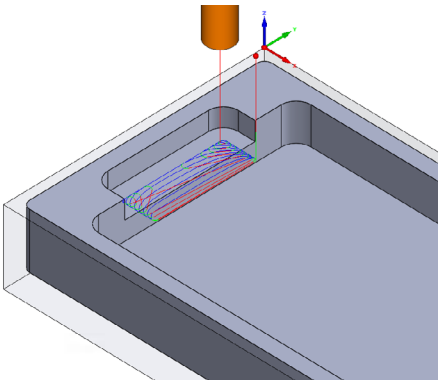
Add a new iMachining 2D operation. Use the **iRough** Technology type to define the rough machining of the pocket ledge feature. Select the lower contour of the pocket ledge and then mark the front edge as open using **Mark open edges**.



Use the $\varnothing 9$ 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).

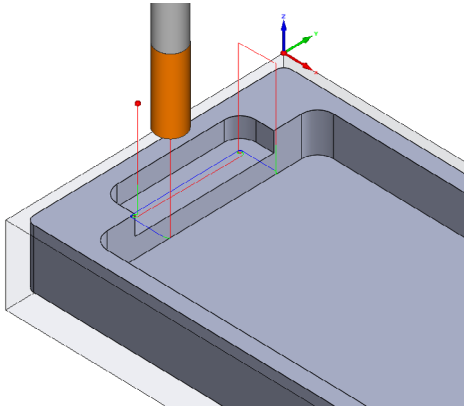
The default Cutting conditions generated by the Wizard are used.

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.



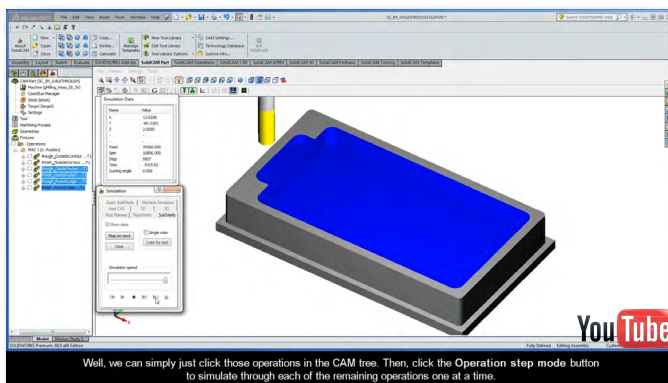
Define the finish machining of the pocket ledge.

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

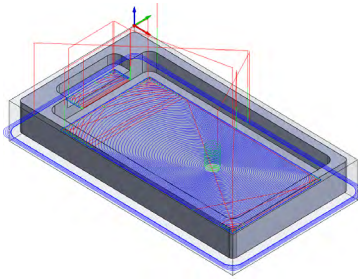
In this step, the iMachining tool path is verified. Included is an in-depth look at SolidCAM's Host CAD simulation options as well as using the SolidVerify simulation mode. A GCode file is also generated and the iMachining technology is shown managing the Feed rates with each cutting move. Click the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



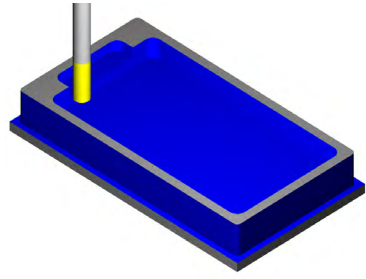
Well, we can simply just click those operations in the CAM tree. Then, click the Operation step mode button to simulate through each of the remaining operations one at a time.



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



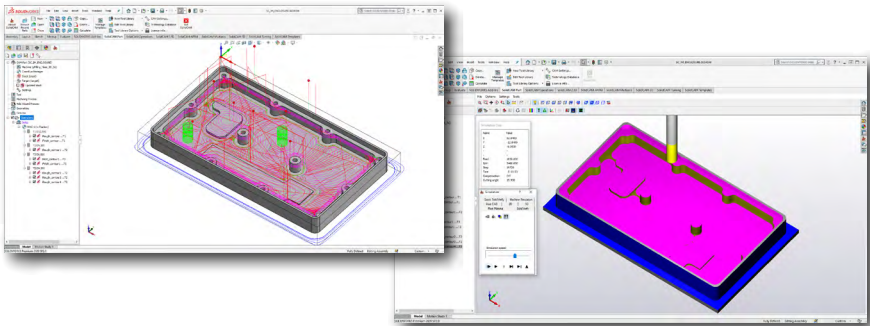
SolidVerify

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 - Notepad2
File Edit View Settings ?
1 %
2 O1000: (SC_IM_WALKTHROUGH)
3 N100 (COMPENSATION=WEAR)
4 N102 (REV=0, Y0)
5 N104 (NOV= 7-2018-1:23:52PM)
6
7
8 N106 (TOOL 1 - DIA 9.)
9
10
11 N1 G90 G17 G40 G80 G00
12 N108 M06 T1 ()
13 N110 (Rough-outs:idecontour)
14 N112 S8514 M03
15 N114 G00 G54 G90 X51.1381 Y4.8638
16 N116 G43 H1 Z120.
17 N118 Z25.
18 N120 Z2.
19 N122 G01 Z-19.73 F3800.
20 N124 G03 X52.0097 Y4.5003 Z-19.81 I0.8746 J0.8703 K2445.
21 N126 G02 X55.0097 Y4.472 I-0.5188 J-213.6944
22 N128 X58.0097 Y4.3902 I-5.81 Z-258.5057 K2745.
23 N130 X61.0097 Y4.2473 I-12.2077 J-2487.8332
24 N132 X63.6387 Y4.08 I-8.6799 J-156.9231
25 N134 G01 X64.0097 Y4.0328
26 N136 X64.7597 Y4.0092
27 N138 X65.8847 Y4.009
28 N140 X67.0097 Y4.0089
29 N142 X69.2597 Y3.9959
30 N144 X72.5788 Y3.9874
31 N146 X124.3288 Y3.9914
32 N148 X152.8288 Y3.9868
33 N150 X154.3490 Y3.9882
34 N152 G02 X157.0978 Y3.9201 I-1.0052 J-95.9356
35 N154 G01 X157.7707 Y3.8101
36 N156 X158.2851 Y3.6366
37 N158 X158.8282 Y3.3959
38 N160 X159.3532 Y3.0844
39 N162 X159.8441 Y2.7151
40 N164 X160.3005 Y2.2907
41 N166 X160.7103 Y1.8177
42 N168 X161.0722 Y1.3073
43 N170 X161.3874 Y0.7466
44 N172 X161.6565 Y0.2101
45 N174 X161.8765 Y0.0000
46 N176 X162.0000 Y0.0000
47 N178 X162.0000 Y0.0000
48 N180 X162.0000 Y0.0000
49 N182 X162.0000 Y0.0000
50 N184 X162.0000 Y0.0000
51 N186 X162.0000 Y0.0000
52 N188 X162.0000 Y0.0000
53 N190 X162.0000 Y0.0000
54 N192 X162.0000 Y0.0000
55 N194 X162.0000 Y0.0000
56 N196 X162.0000 Y0.0000
57 N198 X162.0000 Y0.0000
58 N200 X162.0000 Y0.0000
59 N202 X162.0000 Y0.0000
60 N204 X162.0000 Y0.0000
61 N206 X162.0000 Y0.0000
62 N208 X162.0000 Y0.0000
63 N210 X162.0000 Y0.0000
64 N212 X162.0000 Y0.0000
65 N214 X162.0000 Y0.0000
66 N216 X162.0000 Y0.0000
67 N218 X162.0000 Y0.0000
68 N220 X162.0000 Y0.0000
69 N222 X162.0000 Y0.0000
70 N224 X162.0000 Y0.0000
71 N226 X162.0000 Y0.0000
72 N228 X162.0000 Y0.0000
73 N230 X162.0000 Y0.0000
74 N232 X162.0000 Y0.0000
75 N234 X162.0000 Y0.0000
76 N236 X162.0000 Y0.0000
77 N238 X162.0000 Y0.0000
78 N240 X162.0000 Y0.0000
79 N242 X162.0000 Y0.0000
80 N244 X162.0000 Y0.0000
81 N246 X162.0000 Y0.0000
82 N248 X162.0000 Y0.0000
83 N250 X162.0000 Y0.0000
84 N252 X162.0000 Y0.0000
85 N254 X162.0000 Y0.0000
86 N256 X162.0000 Y0.0000
87 N258 X162.0000 Y0.0000
88 N260 X162.0000 Y0.0000
89 N262 X162.0000 Y0.0000
90 N264 X162.0000 Y0.0000
91 N266 X162.0000 Y0.0000
92 N268 X162.0000 Y0.0000
93 N270 X162.0000 Y0.0000
94 N272 X162.0000 Y0.0000
95 N274 X162.0000 Y0.0000
96 N276 X162.0000 Y0.0000
97 N278 X162.0000 Y0.0000
98 N280 X162.0000 Y0.0000
99 N282 X162.0000 Y0.0000
100 N284 X162.0000 Y0.0000
101 N286 X162.0000 Y0.0000
102 N288 X162.0000 Y0.0000
103 N290 X162.0000 Y0.0000
104 N292 X162.0000 Y0.0000
105 N294 X162.0000 Y0.0000
106 N296 X162.0000 Y0.0000
107 N298 X162.0000 Y0.0000
108 N300 X162.0000 Y0.0000
109 N302 X162.0000 Y0.0000
110 N304 X162.0000 Y0.0000
111 N306 X162.0000 Y0.0000
112 N308 X162.0000 Y0.0000
113 N310 X162.0000 Y0.0000
114 N312 X162.0000 Y0.0000
115 N314 X162.0000 Y0.0000
116 N316 X162.0000 Y0.0000
117 N318 X162.0000 Y0.0000
118 N320 X162.0000 Y0.0000
119 N322 X162.0000 Y0.0000
120 N324 X162.0000 Y0.0000
121 N326 X162.0000 Y0.0000
122 N328 X162.0000 Y0.0000
123 N330 X162.0000 Y0.0000
124 N332 X162.0000 Y0.0000
125 N334 X162.0000 Y0.0000
126 N336 X162.0000 Y0.0000
127 N338 X162.0000 Y0.0000
128 N340 X162.0000 Y0.0000
129 N342 X162.0000 Y0.0000
130 N344 X162.0000 Y0.0000
131 N346 X162.0000 Y0.0000
132 N348 X162.0000 Y0.0000
133 N350 X162.0000 Y0.0000
134 N352 X162.0000 Y0.0000
135 N354 X162.0000 Y0.0000
136 N356 X162.0000 Y0.0000
137 N358 X162.0000 Y0.0000
138 N360 X162.0000 Y0.0000
139 N362 X162.0000 Y0.0000
140 N364 X162.0000 Y0.0000
141 N366 X162.0000 Y0.0000
142 N368 X162.0000 Y0.0000
143 N370 X162.0000 Y0.0000
144 N372 X162.0000 Y0.0000
145 N374 X162.0000 Y0.0000
146 N376 X162.0000 Y0.0000
147 N378 X162.0000 Y0.0000
148 N380 X162.0000 Y0.0000
149 N382 X162.0000 Y0.0000
150 N384 X162.0000 Y0.0000
151 N386 X162.0000 Y0.0000
152 N388 X162.0000 Y0.0000
153 N390 X162.0000 Y0.0000
154 N392 X162.0000 Y0.0000
155 N394 X162.0000 Y0.0000
156 N396 X162.0000 Y0.0000
157 N398 X162.0000 Y0.0000
158 N400 X162.0000 Y0.0000
159 N402 X162.0000 Y0.0000
160 N404 X162.0000 Y0.0000
161 N406 X162.0000 Y0.0000
162 N408 X162.0000 Y0.0000
163 N410 X162.0000 Y0.0000
164 N412 X162.0000 Y0.0000
165 N414 X162.0000 Y0.0000
166 N416 X162.0000 Y0.0000
167 N418 X162.0000 Y0.0000
168 N420 X162.0000 Y0.0000
169 N422 X162.0000 Y0.0000
170 N424 X162.0000 Y0.0000
171 N426 X162.0000 Y0.0000
172 N428 X162.0000 Y0.0000
173 N430 X162.0000 Y0.0000
174 N432 X162.0000 Y0.0000
175 N434 X162.0000 Y0.0000
176 N436 X162.0000 Y0.0000
177 N438 X162.0000 Y0.0000
178 N440 X162.0000 Y0.0000
179 N442 X162.0000 Y0.0000
180 N444 X162.0000 Y0.0000
181 N446 X162.0000 Y0.0000
182 N448 X162.0000 Y0.0000
183 N450 X162.0000 Y0.0000
184 N452 X162.0000 Y0.0000
185 N454 X162.0000 Y0.0000
186 N456 X162.0000 Y0.0000
187 N458 X162.0000 Y0.0000
188 N460 X162.0000 Y0.0000
189 N462 X162.0000 Y0.0000
190 N464 X162.0000 Y0.0000
191 N466 X162.0000 Y0.0000
192 N468 X162.0000 Y0.0000
193 N470 X162.0000 Y0.0000
194 N472 X162.0000 Y0.0000
195 N474 X162.0000 Y0.0000
196 N476 X162.0000 Y0.0000
197 N478 X162.0000 Y0.0000
198 N480 X162.0000 Y0.0000
199 N482 X162.0000 Y0.0000
200 N484 X162.0000 Y0.0000
201 N486 X162.0000 Y0.0000
202 N488 X162.0000 Y0.0000
203 N490 X162.0000 Y0.0000
204 N492 X162.0000 Y0.0000
205 N494 X162.0000 Y0.0000
206 N496 X162.0000 Y0.0000
207 N498 X162.0000 Y0.0000
208 N500 X162.0000 Y0.0000
209 N502 X162.0000 Y0.0000
210 N504 X162.0000 Y0.0000
211 N506 X162.0000 Y0.0000
212 N508 X162.0000 Y0.0000
213 N510 X162.0000 Y0.0000
214 N512 X162.0000 Y0.0000
215 N514 X162.0000 Y0.0000
216 N516 X162.0000 Y0.0000
217 N518 X162.0000 Y0.0000
218 N520 X162.0000 Y0.0000
219 N522 X162.0000 Y0.0000
220 N524 X162.0000 Y0.0000
221 N526 X162.0000 Y0.0000
222 N528 X162.0000 Y0.0000
223 N530 X162.0000 Y0.0000
224 N532 X162.0000 Y0.0000
225 N534 X162.0000 Y0.0000
226 N536 X162.0000 Y0.0000
227 N538 X162.0000 Y0.0000
228 N540 X162.0000 Y0.0000
229 N542 X162.0000 Y0.0000
230 N544 X162.0000 Y0.0000
231 N546 X162.0000 Y0.0000
232 N548 X162.0000 Y0.0000
233 N550 X162.0000 Y0.0000
234 N552 X162.0000 Y0.0000
235 N554 X162.0000 Y0.0000
236 N556 X162.0000 Y0.0000
237 N558 X162.0000 Y0.0000
238 N560 X162.0000 Y0.0000
239 N562 X162.0000 Y0.0000
240 N564 X162.0000 Y0.0000
241 N566 X162.0000 Y0.0000
242 N568 X162.0000 Y0.0000
243 N570 X162.0000 Y0.0000
244 N572 X162.0000 Y0.0000
245 N574 X162.0000 Y0.0000
246 N576 X162.0000 Y0.0000
247 N578 X162.0000 Y0.0000
248 N580 X162.0000 Y0.0000
249 N582 X162.0000 Y0.0000
250 N584 X162.0000 Y0.0000
251 N586 X162.0000 Y0.0000
252 N588 X162.0000 Y0.0000
253 N590 X162.0000 Y0.0000
254 N592 X162.0000 Y0.0000
255 N594 X162.0000 Y0.0000
256 N596 X162.0000 Y0.0000
257 N598 X162.0000 Y0.0000
258 N600 X162.0000 Y0.0000
259 N602 X162.0000 Y0.0000
260 N604 X162.0000 Y0.0000
261 N606 X162.0000 Y0.0000
262 N608 X162.0000 Y0.0000
263 N610 X162.0000 Y0.0000
264 N612 X162.0000 Y0.0000
265 N614 X162.0000 Y0.0000
266 N616 X162.0000 Y0.0000
267 N618 X162.0000 Y0.0000
268 N620 X162.0000 Y0.0000
269 N622 X162.0000 Y0.0000
270 N624 X162.0000 Y0.0000
271 N626 X162.0000 Y0.0000
272 N628 X162.0000 Y0.0000
273 N630 X162.0000 Y0.0000
274 N632 X162.0000 Y0.0000
275 N634 X162.0000 Y0.0000
276 N636 X162.0000 Y0.0000
277 N638 X162.0000 Y0.0000
278 N640 X162.0000 Y0.0000
279 N642 X162.0000 Y0.0000
280 N644 X162.0000 Y0.0000
281 N646 X162.0000 Y0.0000
282 N648 X162.0000 Y0.0000
283 N650 X162.0000 Y0.0000
284 N652 X162.0000 Y0.0000
285 N654 X162.0000 Y0.0000
286 N656 X162.0000 Y0.0000
287 N658 X162.0000 Y0.0000
288 N660 X162.0000 Y0.0000
289 N662 X162.0000 Y0.0000
290 N664 X162.0000 Y0.0000
291 N666 X162.0000 Y0.0000
292 N668 X162.0000 Y0.0000
293 N670 X162.0000 Y0.0000
294 N672 X162.0000 Y0.0000
295 N674 X162.0000 Y0.0000
296 N676 X162.0000 Y0.0000
297 N678 X162.0000 Y0.0000
298 N680 X162.0000 Y0.0000
299 N682 X162.0000 Y0.0000
300 N684 X162.0000 Y0.0000
301 N686 X162.0000 Y0.0000
302 N688 X162.0000 Y0.0000
303 N690 X162.0000 Y0.0000
304 N692 X162.0000 Y0.0000
305 N694 X162.0000 Y0.0000
306 N696 X162.0000 Y0.0000
307 N698 X162.0000 Y0.0000
308 N700 X162.0000 Y0.0000
309 N702 X162.0000 Y0.0000
310 N704 X162.0000 Y0.0000
311 N706 X162.0000 Y0.0000
312 N708 X162.0000 Y0.0000
313 N710 X162.0000 Y0.0000
314 N712 X162.0000 Y0.0000
315 N714 X162.0000 Y0.0000
316 N716 X162.0000 Y0.0000
317 N718 X162.0000 Y0.0000
318 N720 X162.0000 Y0.0000
319 N722 X162.0000 Y0.0000
320 N724 X162.0000 Y0.0000
321 N726 X162.0000 Y0.0000
322 N728 X162.0000 Y0.0000
323 N730 X162.0000 Y0.0000
324 N732 X162.0000 Y0.0000
325 N734 X162.0000 Y0.0000
326 N736 X162.0000 Y0.0000
327 N738 X162.0000 Y0.0000
328 N740 X162.0000 Y0.0000
329 N742 X162.0000 Y0.0000
330 N744 X162.0000 Y0.0000
331 N746 X162.0000 Y0.0000
332 N748 X162.0000 Y0.0000
333 N750 X162.0000 Y0.0000
334 N752 X162.0000 Y0.0000
335 N754 X162.0000 Y0.0000
336 N756 X162.0000 Y0.0000
337 N758 X162.0000 Y0.0000
338 N760 X162.0000 Y0.0000
339 N762 X162.0000 Y0.0000
340 N764 X162.0000 Y0.0000
341 N766 X162.0000 Y0.0000
342 N768 X162.0000 Y0.0000
343 N770 X162.0000 Y0.0000
344 N772 X162.0000 Y0.0000
345 N774 X162.0000 Y0.0000
346 N776 X162.0000 Y0.0000
347 N778 X162.0000 Y0.0000
348 N780 X162.0000 Y0.0000
349 N782 X162.0000 Y0.0000
350 N784 X162.0000 Y0.0000
351 N786 X162.0000 Y0.0000
352 N788 X162.0000 Y0.0000
353 N790 X162.0000 Y0.0000
354 N792 X162.0000 Y0.0000
355 N794 X162.0000 Y0.0000
356 N796 X162.0000 Y0.0000
357 N798 X162.0000 Y0.0000
358 N800 X162.0000 Y0.0000
359 N802 X162.0000 Y0.0000
360 N804 X162.0000 Y0.0000
361 N806 X162.0000 Y0.0000
362 N808 X162.0000 Y0.0000
363 N810 X162.0000 Y0.0000
364 N812 X162.0000 Y0.0000
365 N814 X162.0000 Y0.0000
366 N816 X162.0000 Y0.0000
367 N818 X162.0000 Y0.0000
368 N820 X162.0000 Y0.0000
369 N822 X162.0000 Y0.0000
370 N824 X162.0000 Y0.0000
371 N826 X162.0000 Y0.0000
372 N828 X162.0000 Y0.0000
373 N830 X162.0000 Y0.0000
374 N832 X162.0000 Y0.0000
375 N834 X162.0000 Y0.0000
376 N836 X162.0000 Y0.0000
377 N838 X162.0000 Y0.0000
378 N840 X162.0000 Y0.0000
379 N842 X162.0000 Y0.0000
380 N844 X162.0000 Y0.0000
381 N846 X162.0000 Y0.0000
382 N848 X162.0000 Y0.0000
383 N850 X162.0000 Y0.0000
384 N852 X162.0000 Y0.0000
385 N854 X162.0000 Y0.0000
386 N856 X162.0000 Y0.0000
387 N858 X162.0000 Y0.0000
388 N860 X162.0000 Y0.0000
389 N862 X162.0000 Y0.0000
390 N864 X162.0000 Y0.0000
391 N866 X162.0000 Y0.0000
392 N868 X162.0000 Y0.0000
393 N870 X162.0000 Y0.0000
394 N872 X162.0000 Y0.0000
395 N874 X162.0000 Y0.0000
396 N876 X162.0000 Y0.0000
397 N878 X162.0000 Y0.0000
398 N880 X162.0000 Y0.0000
399 N882 X162.0000 Y0.0000
400 N884 X162.0000 Y0.0000
401 N886 X162.0000 Y0.0000
402 N888 X162.0000 Y0.0000
403 N890 X162.0000 Y0.0000
404 N892 X162.0000 Y0.0000
405 N894 X162.0000 Y0.0000
406 N896 X162.0000 Y0.0000
407 N898 X162.0000 Y0.0000
408 N900 X162.0000 Y0.0000
409 N902 X162.0000 Y0.0000
410 N904 X162.0000 Y0.0000
411 N906 X162.0000 Y0.0000
412 N908 X162.0000 Y0.0000
413 N910 X162.0000 Y0.0000
414 N912 X162.0000 Y0.0000
415 N914 X162.0000 Y0.0000
416 N916 X162.0000 Y0.0000
417 N918 X162.0000 Y0.0000
418 N920 X162.0000 Y0.0000
419 N922 X162.0000 Y0.0000
420 N924 X162.0000 Y0.0000
421 N926 X162.0000 Y0.0000
422 N928 X162.0000 Y0.0000
423 N930 X162.0000 Y0.0000
424 N932 X162.0000 Y0.0000
425 N934 X162.0000 Y0.0000
426 N936 X162.0000 Y0.0000
427 N938 X162.0000 Y0.0000
428 N940 X162.0000 Y0.0000
429 N942 X162.0000 Y0.0000
430 N944 X162.0000 Y0.0000
431 N946 X162.0000 Y0.0000
432 N948 X162.0000 Y0.0000
433 N950 X162.0000 Y0.0000
434 N952 X162.0000 Y0.0000
435 N954 X162.0000 Y0.0000
436 N956 X162.0000 Y0.0000
437 N958 X162.0000 Y0.0000
438 N960 X162.0000 Y0.0000
439 N962 X162.0000 Y0.0000
440 N964 X162.0000 Y0.0000
441 N966 X162.0000 Y0.0000
442 N968 X162.0000 Y0.0000
443 N970 X162.0000 Y0.0000
444 N972 X162.0000 Y0.0000
445 N974 X162.0000 Y0.0000
446 N976 X162.0000 Y0.0000
447 N978 X162.0000 Y0.0000
448 N980 X162.0000 Y0.0000
449 N982 X162.0000 Y0.0000
450 N984 X162.0000 Y0.0000
451 N986 X162.0000 Y0.0000
452 N988 X162.0000 Y0.0000
453 N990 X162.0000 Y0.0000
454 N992 X162.0000 Y0.0000
455 N994 X162.0000 Y0.0000
456 N996 X162.0000 Y0.0000
457 N998 X162.0000 Y0.0000
458 N1000 X162.0000 Y0.0000
459 N1002 X162.0000 Y0.0000
460 N1004 X162.0000 Y0.0000
461 N1006 X162.0000 Y0.0000
462 N1008 X162.0000 Y0.0000
463 N1010 X162.0000 Y0.0000
464 N1012 X162.0000 Y0.0000
465 N1014 X162.0000 Y0.0000
466 N1016 X162.0000 Y0.0000
467 N1018 X162.0000 Y0.0000
468 N1020 X162.0000 Y0.0000
469 N1022 X162.0000 Y0.0000
470 N1024 X162.0000 Y0.0000
471 N1026 X162.0000 Y0.0000
472 N1028 X162.0000 Y0.0000
473 N1030 X162.0000 Y0.0000
474 N1032 X162.0000 Y0.0000
475 N1034 X162.0000 Y0.0000
476 N1036 X162.0000 Y0.0000
477 N1038 X162.0000 Y0.0000
478 N1040 X162.0000 Y0.0000
479 N1042 X162.0000 Y0.0000
480 N1044 X162.0000 Y0.0000
481 N1046 X162.0000 Y0.0000
482 N1048 X162.0000 Y0.0000
483 N1050 X162.0000 Y0.0000
484 N1052 X162.0000 Y0.0000
485 N1054 X162.0000 Y0.0000
486 N1056 X162.0000 Y0.0000
487 N1058 X162.0000 Y0.0000
488 N1060 X162.0000 Y0.0000
489 N1062 X162.0000 Y0.0000
490 N1064 X162.0000 Y0.0000
491 N1066 X162.0000 Y0.0000
492 N1068 X162.0000 Y0.0000
493 N1070 X162.0000 Y0.0000
494 N1072 X162.0000 Y0.0000
495 N1074 X162.0000 Y0.0000
496 N1076 X162.0000 Y0.0000
497 N1078 X162.0000 Y0.0000
498 N1080 X
```


iMachining of an Enclosure



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:



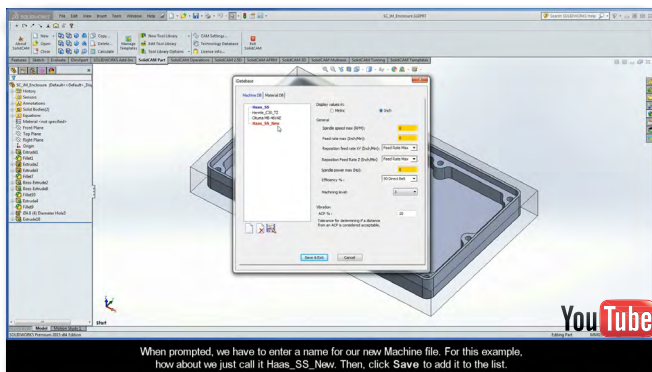
1. Load the SOLIDWORKS model

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...



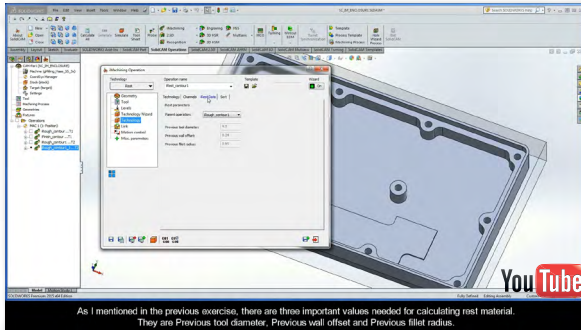
When prompted, we have to enter a name for our new Machine file. For this example, how about we just call it Haas_SS_New. Then, click Save to add it to the list.

[illegible][illegible]



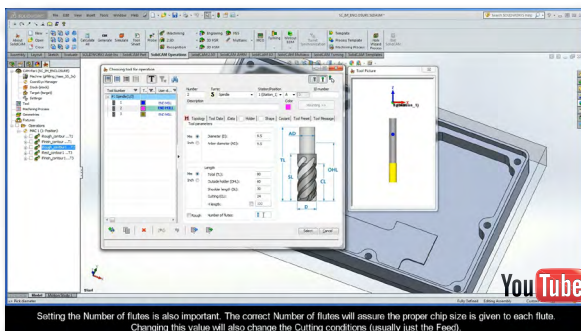
5. Using the iRest Technology type prior to finishing

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...

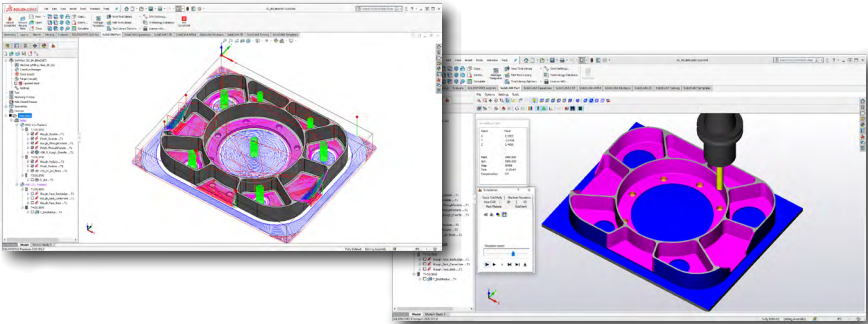


You have completed the iMachining Getting Started exercise.



If you would like to review SolidCAM's completed CAM-Part and compare it with yours, click [SC_IM_ENCLOSURE.prz](#).

iMachining of a Bracket



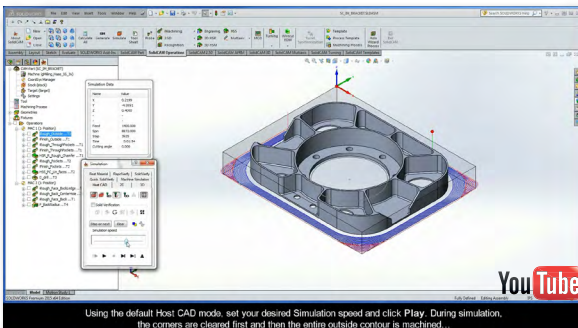
This example illustrates the use of SolidCAM's iMachining technology to define the machining of the part shown above. There are standard 2.5D tool paths and 3D tool paths to aid in the complete CNC program.

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)**

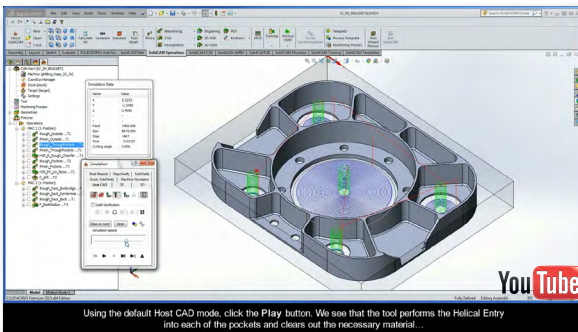
These iMachining operations define the machining of the outside shape of the part. Two chains are defined, with the first being the stock boundary and the second being the profile around the part. The stock chain is marked as open, which specifies the tool should start machining from that chain. A $\varnothing 12.5$ 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.





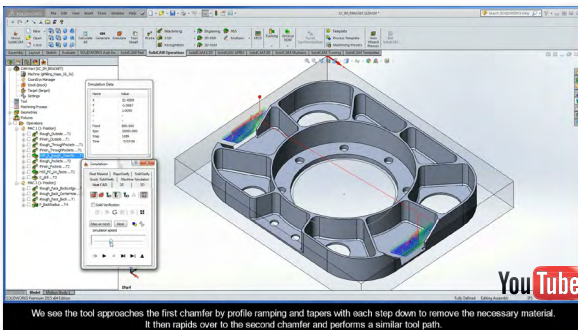
- **Through pockets machining (iRough_ThroughPockets; iFinish_ThroughPockets)**

These iMachining operations define the machining of the five circular through pockets. Five chains are defined, which represent the five through pockets. A $\varnothing 12.5$ 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.



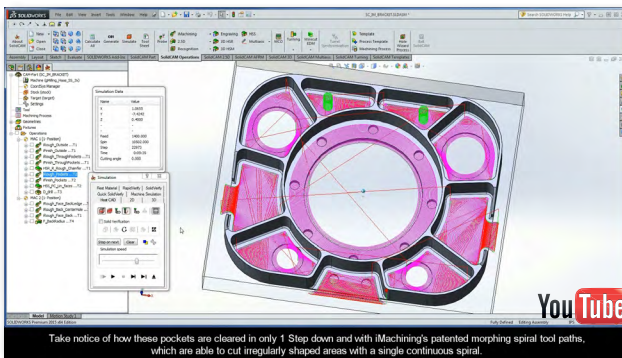
- **Rough machining of angled surfaces (HSR_R_Rough_Chamfer)**

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 $\varnothing 12.5$ 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.



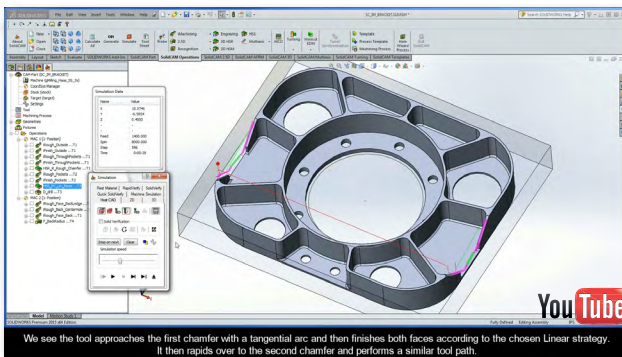
- Pockets machining (iRough_Pockets; iFinish_Pockets)**

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 $\varnothing 10$ 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)**

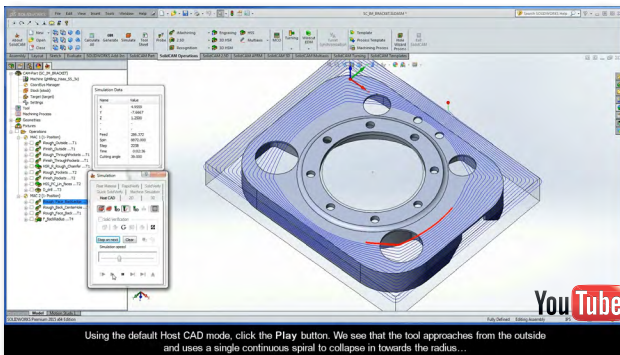
This HSS operation defines the finish machining of the four large chamfers on the ribs of the part. The $\varnothing 10$ mm (0.4063 in) bull nose mill is used and the Linear strategy generates Parallel cuts with a 0.5 mm (0.02 in) Maximum step over. Customized linking is used to allow short repositions and smooth transitions when starting each cut.





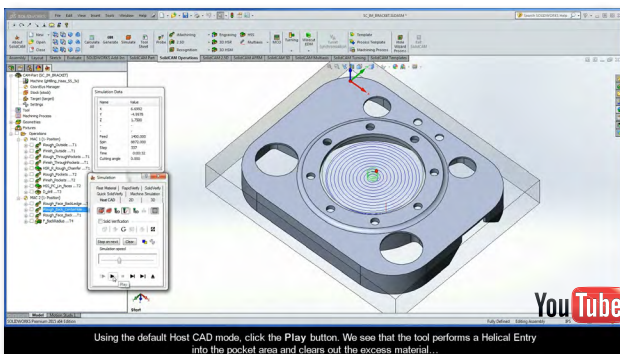
- **Bottom ledge machining (iRough_Face_BackLedge)**

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 $\varnothing 12.5$ 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.



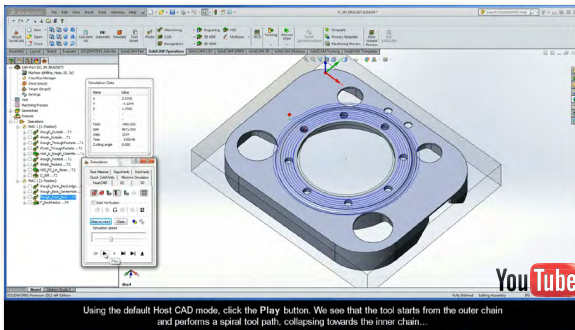
- **Cutting of excess material from through hole (iRough_Back_CenterHole)**

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 $\varnothing 12.5$ 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.



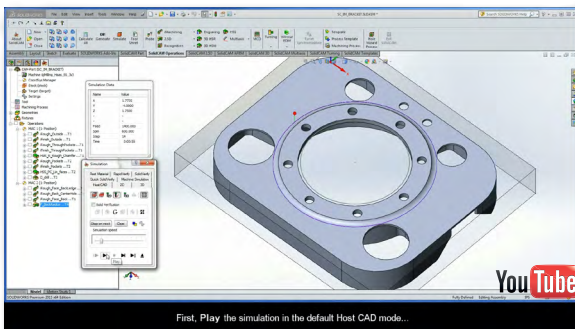
- **Bottom face machining (iRough_Face_Back)**

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 $\varnothing 12.5$ 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 $\varnothing 12.5$ 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.

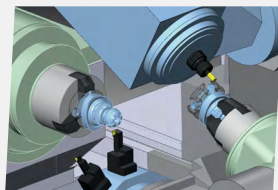
SolidCAM 2020

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.



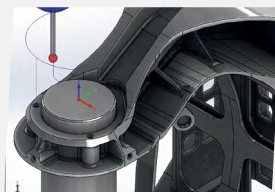
iMachining is driven by a knowledge-based Technology Wizard which considers the machine being used, the material being cut and the cutting tool data to provide optimal values for the Cutting conditions. With its Morphing spiral tool paths, controlled tool load at each point along the tool path, moating 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



solidcam.com/successes/imachining-successes

www.solidcam.com



SolidCAM
iMachining – The Revolution in CAM!