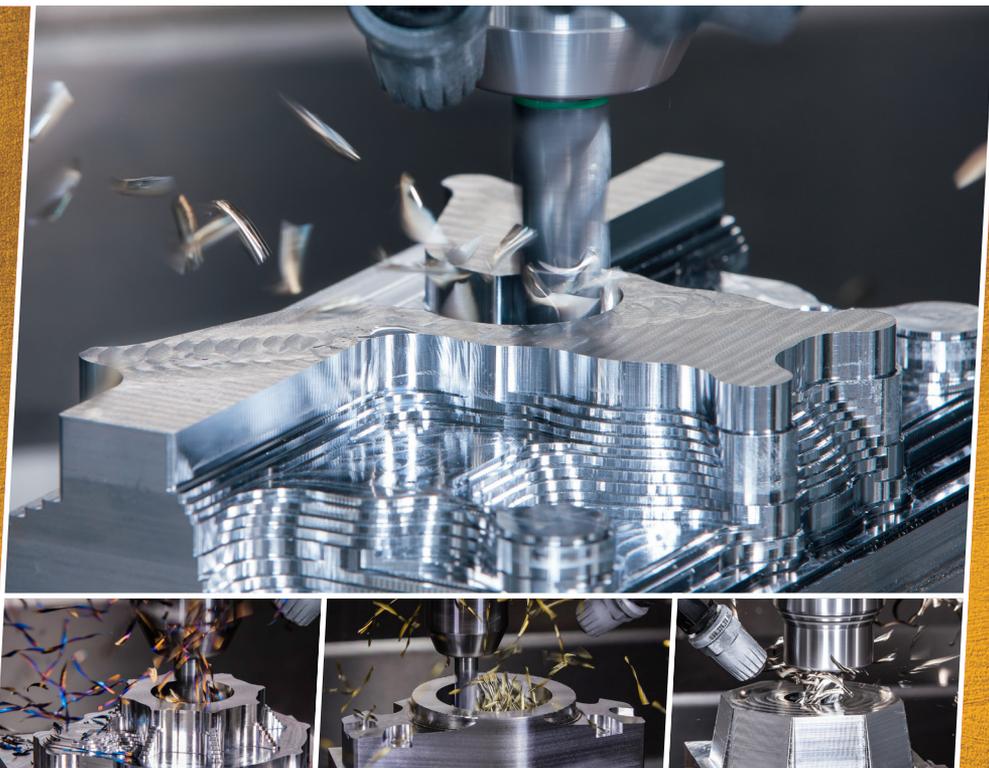


InventorCAM 2019

The unique, revolutionary Milling technology | TIME SAVINGS
imachining® **70%**
patent by SolidCAM ... AND MORE!



What's **NEW**

iMachining **2D** | iMachining **3D**



InventorCAM + Inventor

The Complete, Integrated Manufacturing Solution



InventorCAM

iMachining – The Revolution in CAM!

Contents

New Features and Improvements

| | |
|------------------------------------|----|
| Feature Recognition technology | 1 |
| Floor fillet option moved | 24 |
| Semi-open as closed... | 25 |
| Enter all open edges first... | 27 |
| STL Holder collision protection... | 28 |
| Limit depth to one ACP... | 32 |

New Features and Improvements

1. iMachining 2D Feature Recognition technology

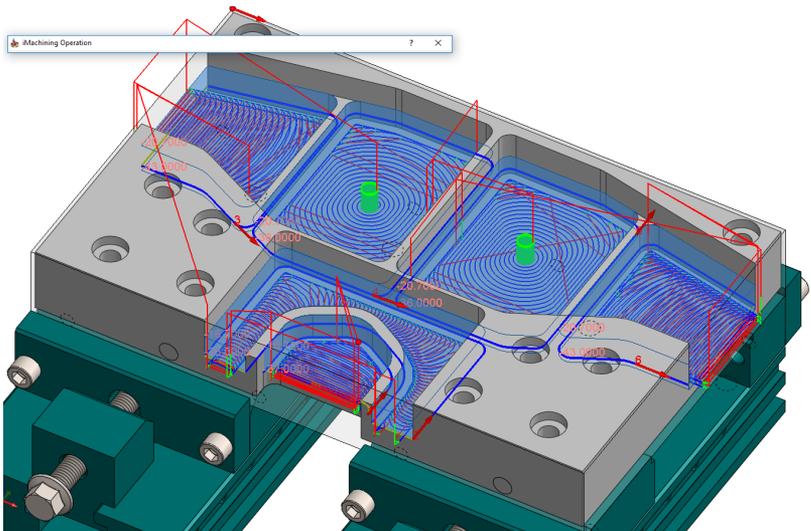
It's no news that iMachining 2D operations can be used to machine nearly all the 2.5D features of your parts. When defining a feature, its machining geometry is always represented by a pocket that is either closed, open or semi-open (containing open edges).

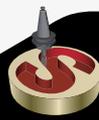
Since its original release, iMachining 2D has required the use of chain geometries that are defined by selecting CAD elements such as edges of models, 2D sketch segments and points on a contour. The Geometry definition can consist of one or more chains, each chain being composed of one or more entities. In some cases, the chain picking process can get quite time-consuming, especially when you have to define an increased number of chains.

When InventorCAM added its Smart Face feature, the Geometry definition process was simplified by a considerable extent. One or more chains can be defined at once with just the click of a model face. The edges contacting the selected floor face are used to automatically create the chain(s).

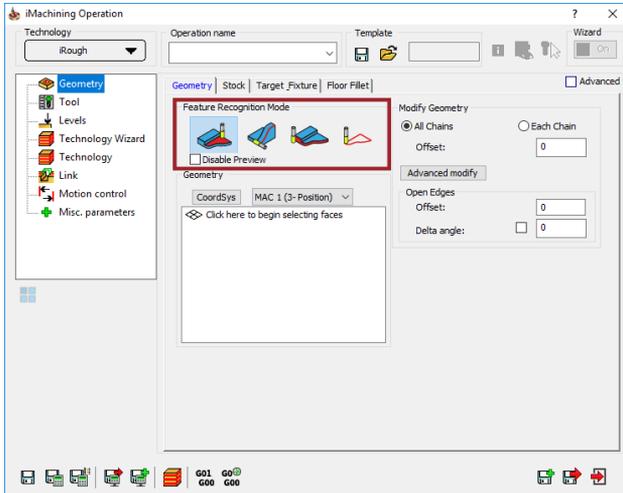
In the most recent release, the iMachining developers have further simplified and improved the Geometry definition process for iMachining 2D operations, this time by a remarkable extent.

NEW Added to InventorCAM 2019 is iMachining's Feature Recognition technology – a revolutionary way to define your machining geometry.





Several Feature Recognition Modes now appear on the Geometry page.



These modes can detect and define the machinable features of your parts based on your selections in combination with the solid model data.

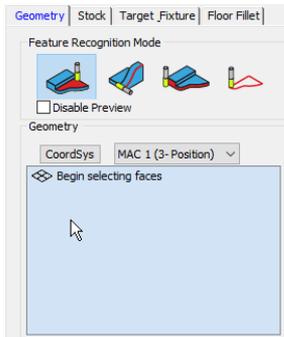


Because iMachining's Feature Geometry Recognition technology analyzes the Stock and Target models, it is important they are accurately defined in the CAM-Part Definition.

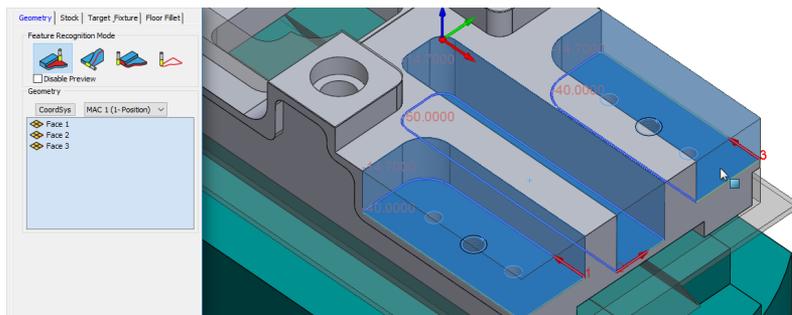
To start the Geometry definition process, select the mode in which you want to work.

Feature Recognition by Faces (default mode)

When using this mode, the Geometry section displays the Faces Selection box. Upon clicking it, you are prompted to *Begin selecting faces*.

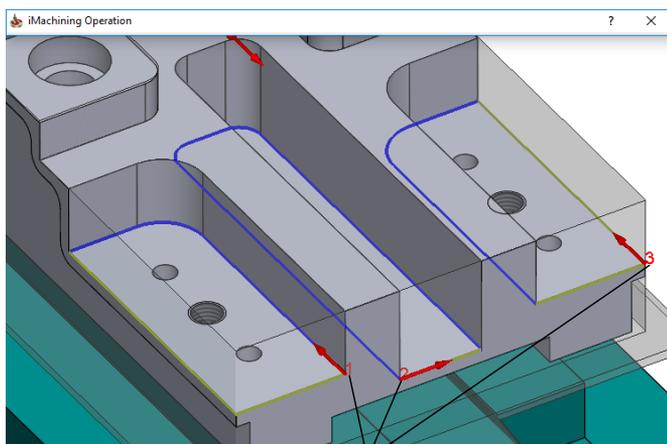


With the Faces Selection box activated, simply click on the floor faces of the pocket features that you want machined by the operation. The faces can be at different depths.



In the example above, three faces are selected. Upon picking each face, it appears in the Faces list and two major processes are performed:

1. The Smart Face feature builds and defines the chain geometries.



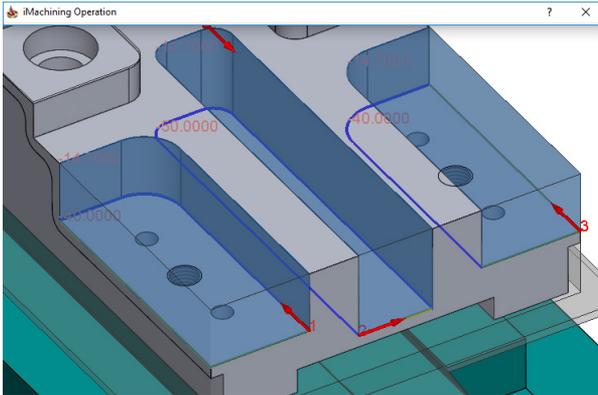
Chains with color-coded entities

The chains are displayed in the Autodesk Inventor graphics window. Walls and open edges are identified and marked accordingly. The hole features are detected but ignored.

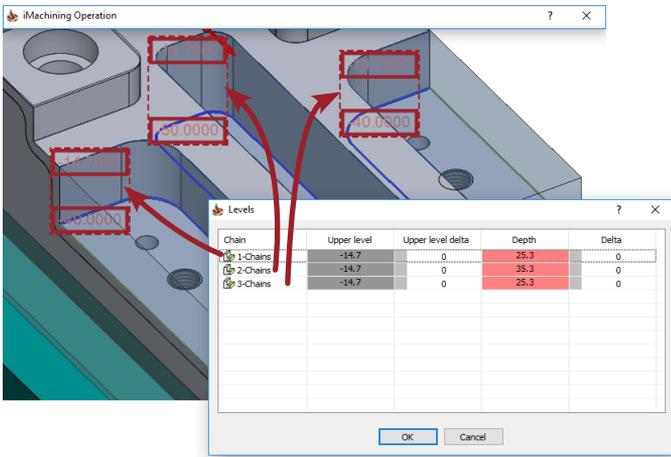
2. According to the Updated Stock model (USM) and the selected faces of the Target model, iMachining automatically detects and defines the corresponding pocket features and their levels.



Highlighted and displayed in the Autodesk Inventor graphics window are the generated iMachining regions, which are a visual representation of the material that will be removed by the operation.



Also displayed for each iMachining region are both the Upper level and Lower level values.

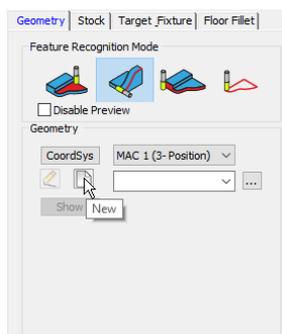


The associated data is automatically written to the Milling levels of the operation. Since the pockets have varying depths in this case, their Upper level and calculated Depth values can be seen in the Variable Levels dialog box.

With just a single click, you can define for each pocket feature both the machining geometry and levels. Apart from the Tool definition, this fulfills the minimum requirements for defining an iMachining operation.

Feature Recognition by Chains

When using this mode, the Geometry section displays the familiar options relevant to InventorCAM's standard chaining method.

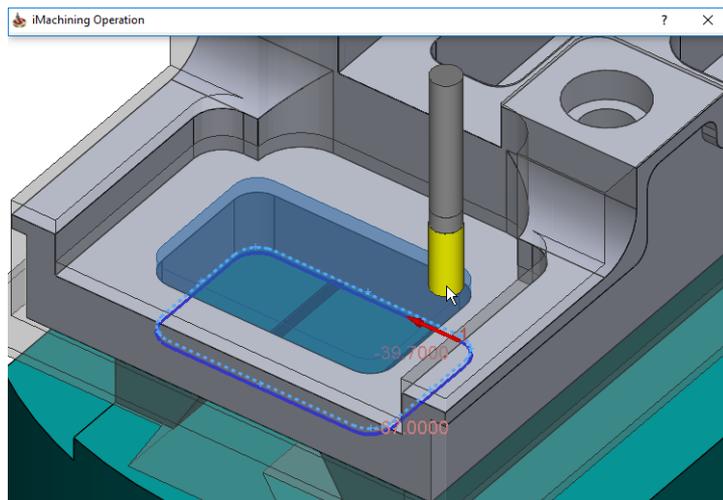


You can use the appropriate options to define new, edit existing or select from/browse for predefined chain geometries.

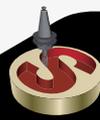
The geometry, which can consist of closed and/or open chains, is best suited for features that cannot be defined via floor face selection. Such features can include through pockets and side profiles.

Example 1: Through pockets milling

In the example below, the geometry chain is defined by selecting the lower contour of the through pocket feature.

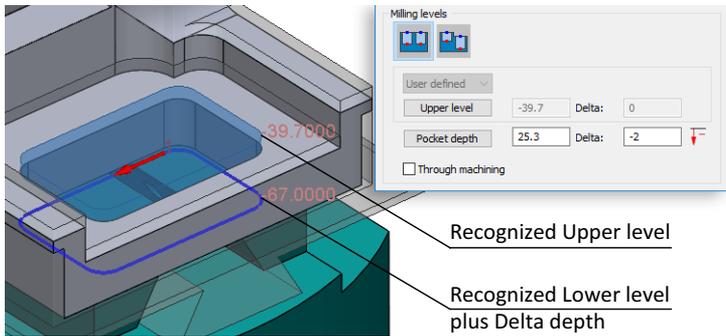


According to the chain definition and the USM, the exact area that requires machining is detected automatically.



In this case, the Upper level is extended in the positive Z-direction to the adjusted depth of the previously machined semi-open pocket.

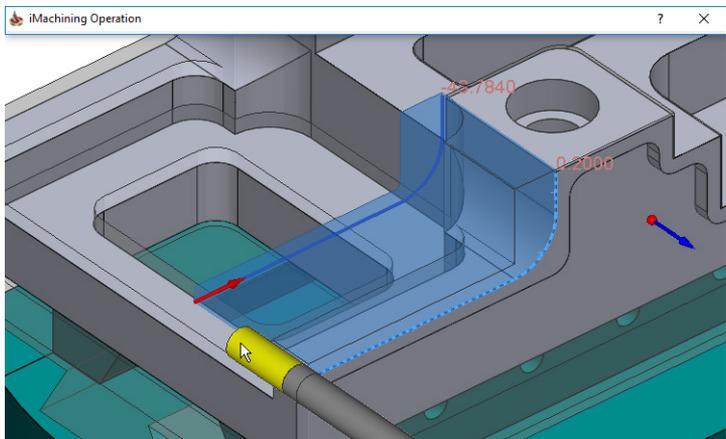
The Pocket depth is defined according to the chain selection depth. For this operation, a Delta depth is specified so the tool can perform the machining deeper than the part bottom edge.



When specifying a Delta depth, the chain geometry is projected to the adjusted depth.

Example 2: Side milling of profiles

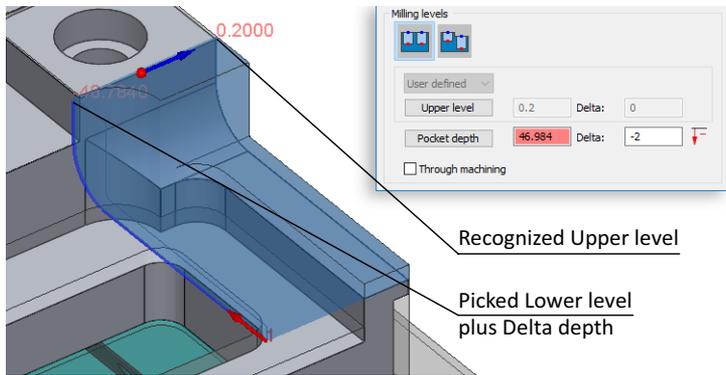
In the example below, using a different Coordinate System Position, the geometry chain is defined by selecting, along the side of the part, the contour of a profile feature.



According to the chain definition, USM and one simple modification, the area that requires machining is defined.

In this case, the Upper level is extended in the positive Z-direction to the roughing offset that remains on the adjacent side wall after the outside shape machining.

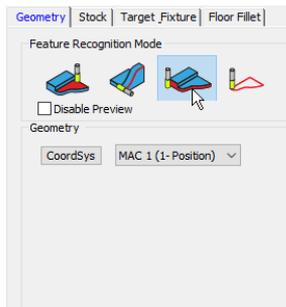
The Pocket depth is defined by selecting an appropriate point on the Target model plus specifying a Delta depth.



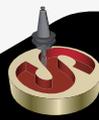
This mode, although requiring one or two user inputs, can significantly reduce your programming time in addition to the savings already provided by iMachining 2D operations. While it is true in the two examples shown, it's especially true in cases of complex geometries that would otherwise require custom sketching in Autodesk Inventor.

Outside Feature Recognition

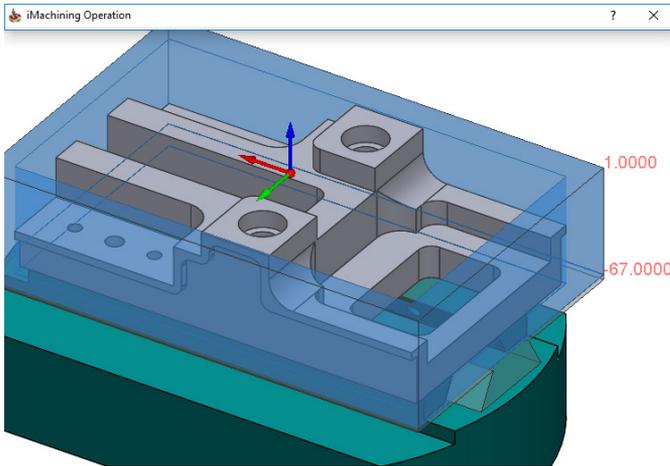
When using this mode, other than the Coordinate System selection, the Geometry section is intentionally left blank because no other user inputs are required for the Geometry definition.



The only requirement is to click the corresponding button when you want to machine the entire outside shape of your part.



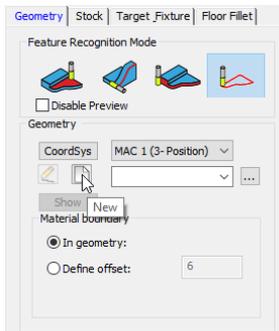
After analyzing the solid model data, iMachining automatically detects and defines the machinable surrounding stock and its levels.



As shown in this example, the Upper level is defined according to the top of the Stock model and the Pocket depth is defined according to the bottom of the Target model. The displayed Lower level value includes a negative Delta depth measurement.

Chains without Feature Recognition

When using this mode, the Geometry section displays the familiar options relevant to InventorCAM's standard chaining method.

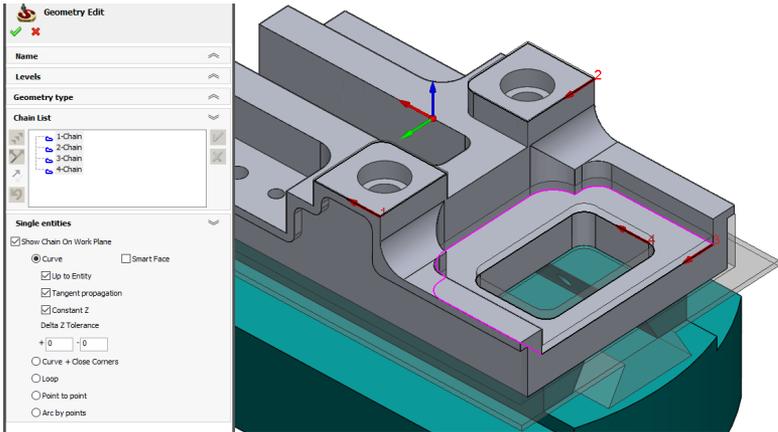


Also displayed in the Geometry section is the Material boundary feature, which is not available for use with any of the other modes.



For more information about the Material boundary feature in iMachining, refer to the InventorCAM Milling Help documentation.

By default, this mode enables you to define your machining geometry exactly as you did prior to InventorCAM 2019 and without the capabilities of iMachining's Feature Recognition technology.

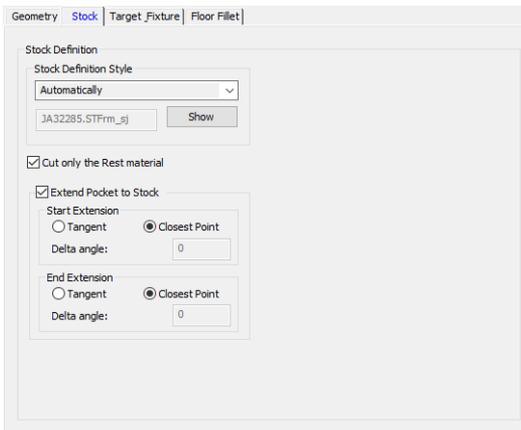


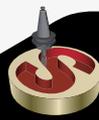
Feature Recognition Capabilities

iMachining performs the functions of its Feature Recognition technology by utilizing the solid models of the stock and target. During the process of analyzing the data, iMachining also protects the 3D Model geometries against possible collisions with the selected tool.

Stock model data utilization

The Stock tab enables you to view/modify options relevant to the Stock/Updated Stock model (USM).

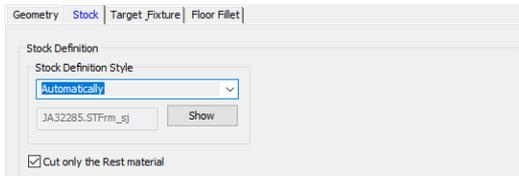




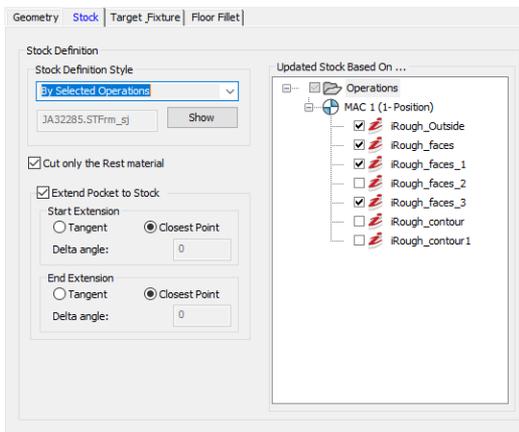
Stock Definition Style

This option specifies how the starting Stock model is defined for the current operation.

The default selection is **Automatically**, which uses the USM that is generated after processing all previous operations.



Also available for selection is **By Selected Operations**. When chosen, the Updated Stock Based On... window appears.



Contained in the window is the Operations tree showing all the operations preceding the current one. Any number of previous operations, whether calculated or not, can be selected for the USM definition.

The USM generated after processing the selected operations is used as the starting Stock model for the current operation.

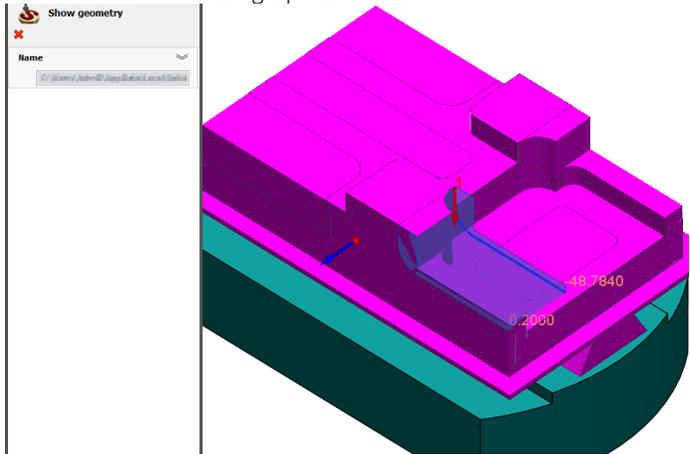


By Selected Operations might be used when you want to ignore non-important operations (e.g., drilling of holes) or those that are not prerequisites to the current operation.

The field below the Stock Definition Style drop-down list displays the file name of the USM used in the rest material calculation. All *.STFrm files are stored within the CAM-Part.



Clicking the **Show** button displays the USM/starting Stock model in the Autodesk Inventor graphics window.



Stock material existing outside the iMachining regions is protected at every stage of the operation machining.

Cut only the Rest material



After every operation in InventorCAM is calculated the exact shape of the remaining stock, which is reflected in the Updated Stock model (USM). This option uses the corresponding USM data in comparison with the Target model to limit the machining to only the rest material geometry.

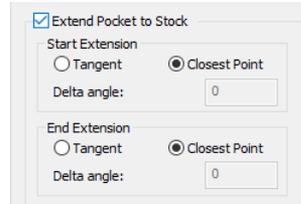


During calculation of the iMachining tool path, no time will be wasted on the air cutting of empty volumes such as those that are already machined.



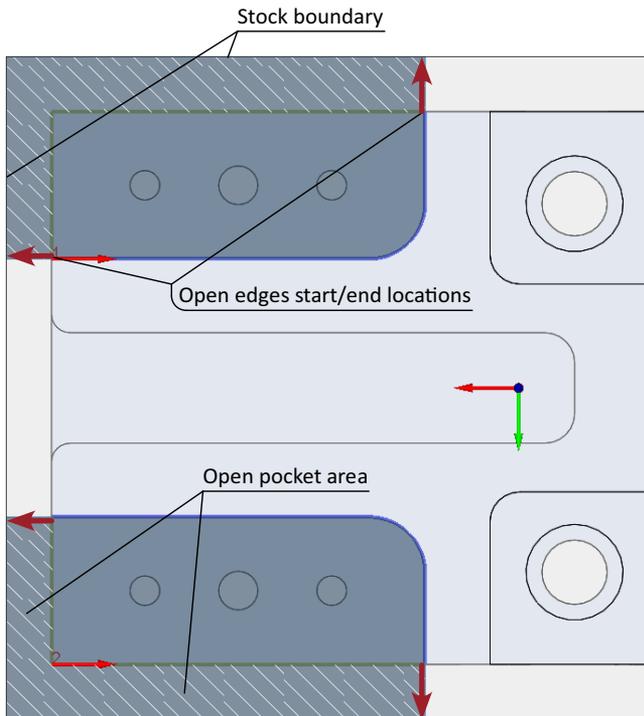
Extend Pocket to Stock

In the case of a semi-open pocket geometry, this option extends the iMachining region beyond the open edge(s) if stock material is detected in the open pocket area.



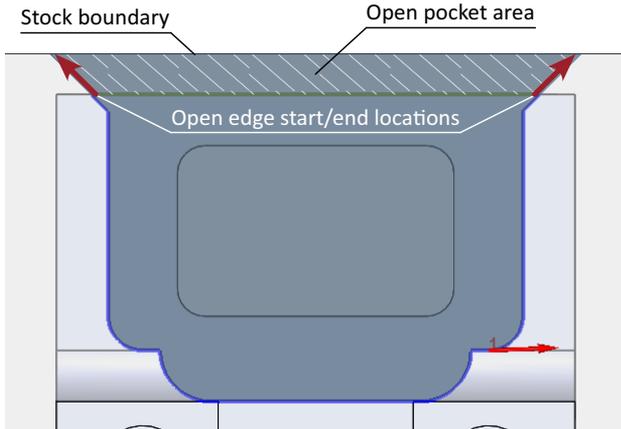
Start and End Extensions enable you to define the parameters of the pocket geometry extension at the start and end locations of the open edge(s).

Closest Point (default selection) extends the pocket geometry by connecting the adjacent closed edges to the nearest points on the stock boundary.

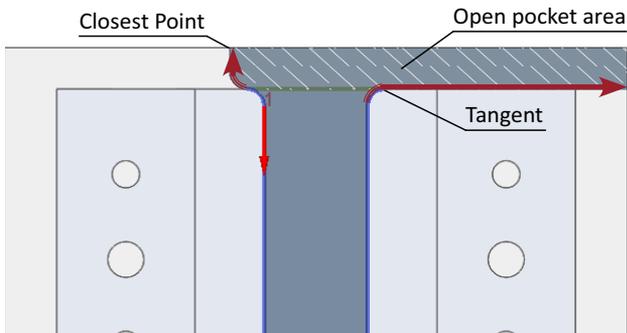


Tangent extends the pocket geometry by tangentially connecting the adjacent closed edges to the stock boundary.

In cases of simple geometries, such as when the open edge is intersected by a chamfer- or fillet-type feature, the selection is switched to **Tangent** automatically.

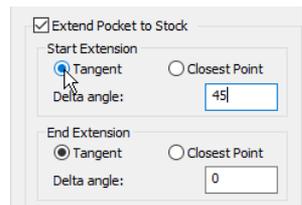


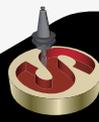
If you instead choose **Closest Point**, an initial tangent extension is performed in accordance with the tool diameter before connecting the geometry normal to the stock boundary.



The **Delta angle** parameter is opened for editing only when the selection is set to **Tangent**.

It can be used to apply an angle to the pocket geometry extension at the open edge start/end locations.

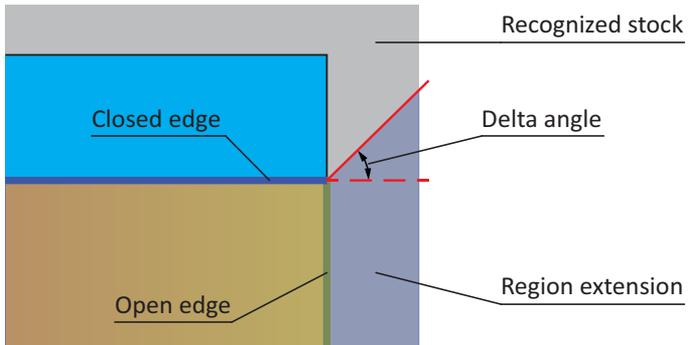




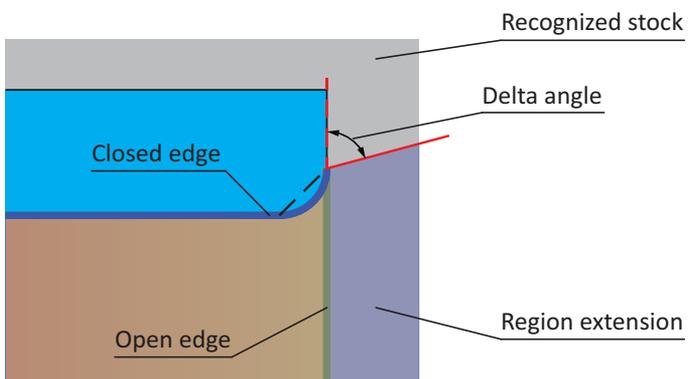
InventorCAM automatically determines the direction to which the angle is measured according to the geometric relationship of closed to open edges.

Take, for instance, the two examples described below.

If the closed edge intersects the open edge perpendicularly, the Delta angle is measured away from the open area, consequently increasing the machinable region.



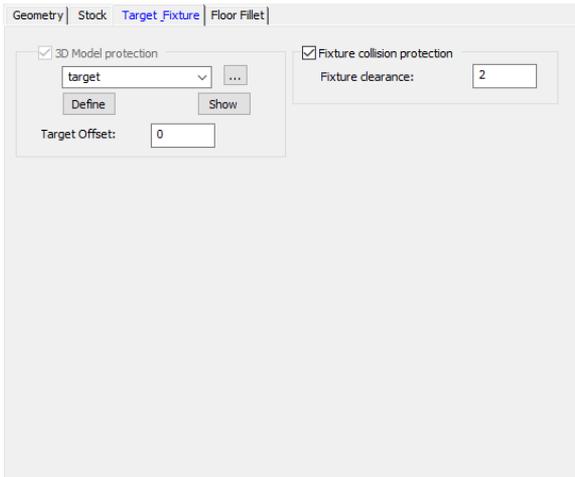
If the closed edge intersects the open edge by either a chamfer- or fillet-type feature, the Delta angle is measured towards the open area, consequently decreasing the machinable region.



In most cases, you will not need to modify the options/parameters appearing on the Stock tab.

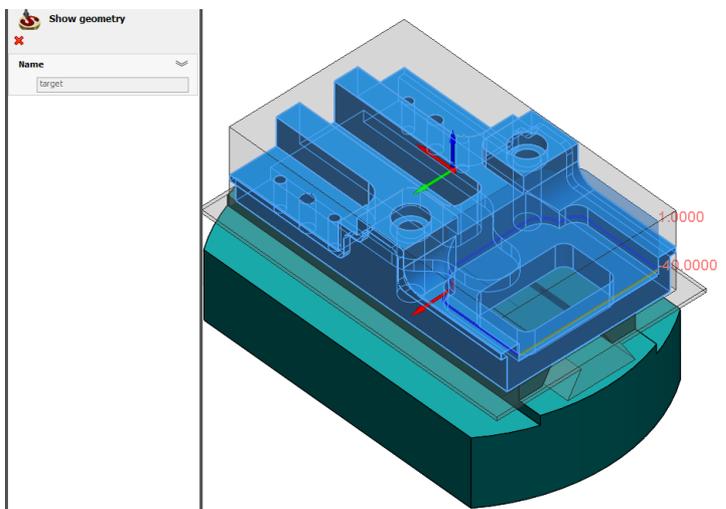
Target model data utilization

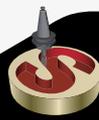
The Target_Fixture tab enables you to view/modify options relevant to the Target model as well as any fixtures you may have defined.



3D Model protection

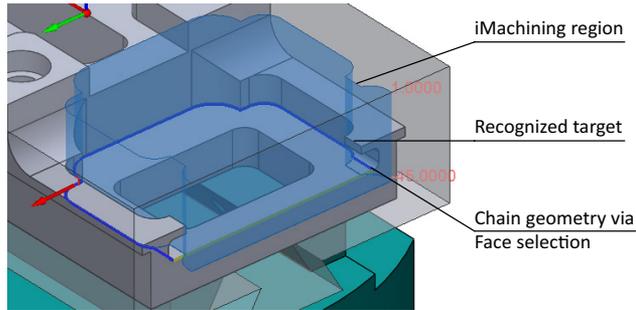
This option defines the target solid model used in the recognition calculations and that which is protected at every stage of the operation machining.



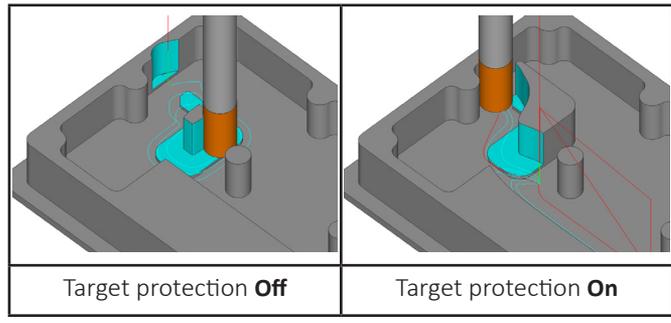


Among many other benefits, you can now:

Easily avoid features that make undercut areas without having to perform custom sketching in Autodesk Inventor.



Safely use large tools in confined spaces without having to worry about plunging down on or gouging the workpiece.



By default, **target** appears in the selection box. If necessary, you can use the options to edit/show the current 3D Model geometry or select from/browse for predefined ones.

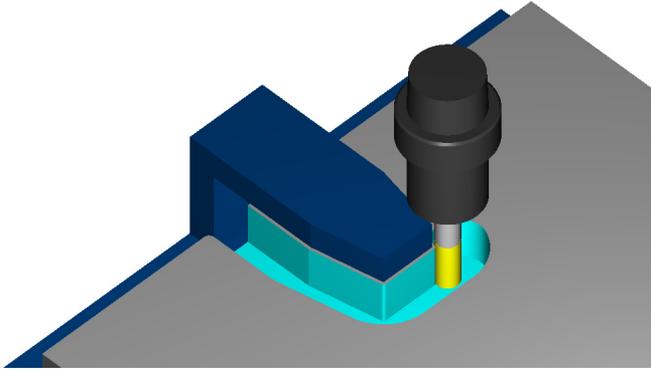
The **Target Offset** parameter enables you to offset all the surfaces of the target solid model for the current operation.

Fixture collision protection



If your Setup definition contains fixture geometry, this option will protect it against possible collisions with the selected tool.

The **Fixture clearance** parameter defines the distance by which the tool can approach the fixture during the machining.



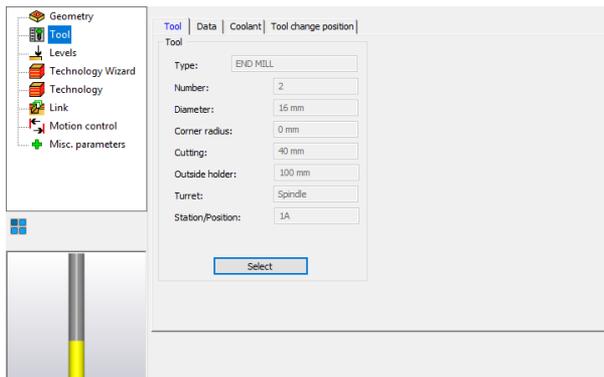
The iMachining 2D tool path is automatically adjusted taking into account the fixture geometry plus the Fixture clearance value.

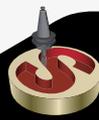
In most cases, you will not need to modify the options/parameters appearing on the Target_Fixture tab.

Dynamic display of iMachining regions

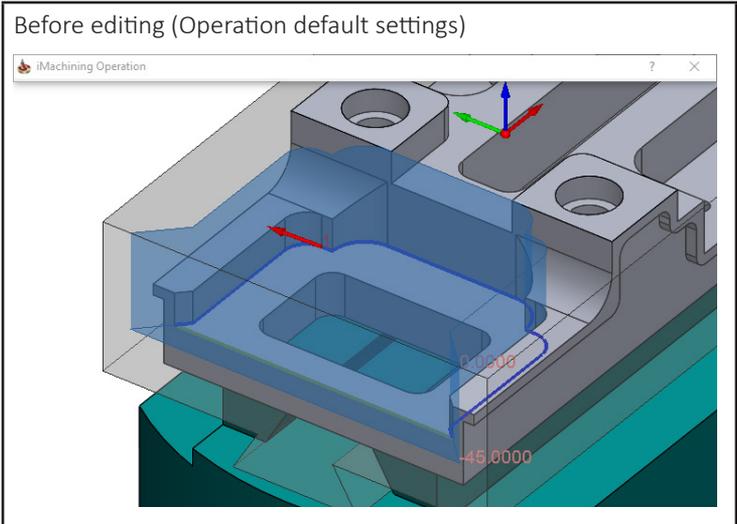
As shown earlier, iMachining generates and displays a preview of the machinable regions and their levels. This is partially accomplished by utilization of the solid model data.

If, for some reason, the iMachining regions are not appearing in the Autodesk Inventor graphics window, confirm that your tool is defined and/or selected for the operation. The tool information is also used in the machinable regions calculations.

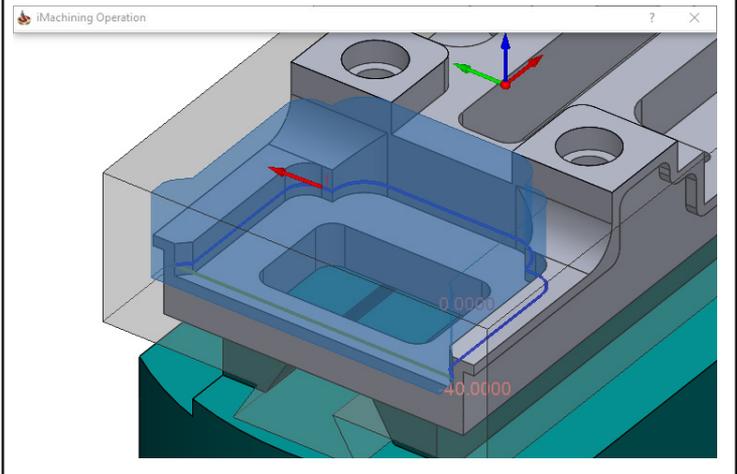




The displayed preview is dynamically updated on operation editing, enabling you to instantly visualize the effect of your modifications on the iMachining regions.

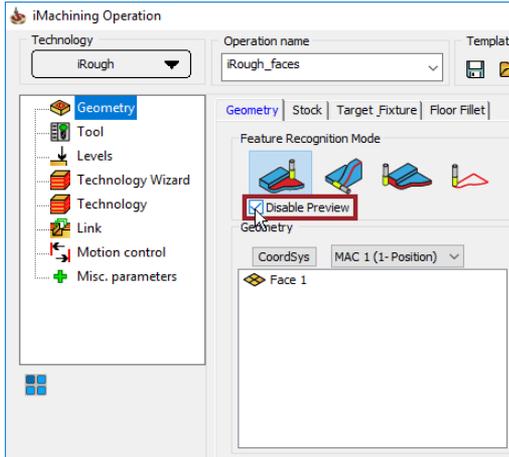


After editing (Extend Pocket to Stock is changed to Closest Point and a +5 mm (+0.1969 in) Delta depth is added)

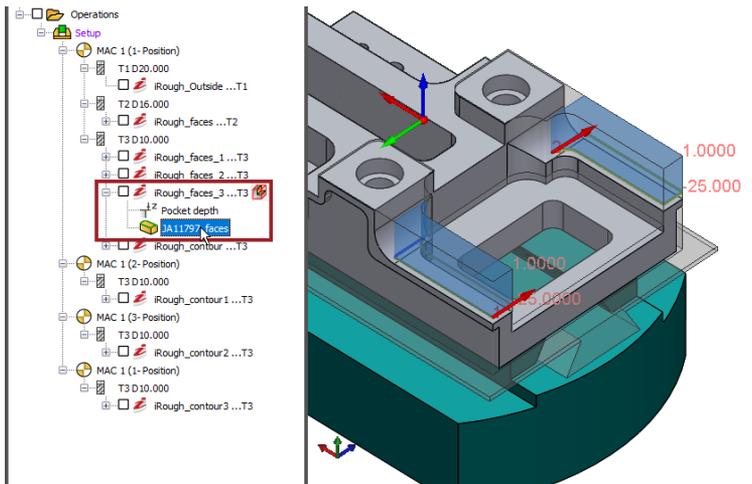


By default, while actively working in the operation, the preview is always visible in the Autodesk Inventor graphics window.

If necessary, you can turn off the display of the iMachining regions by selecting the **Disable Preview** option on the Geometry tab of the Geometry page.



It is also possible, when not editing an operation, to preview the iMachining regions from the CAM tree. Use the **+** button to expand the iMachining Feature Recognition operation and click its geometry item in the list.



Displayed in the Autodesk Inventor graphics window are the corresponding iMachining regions.



Operation data management and flow

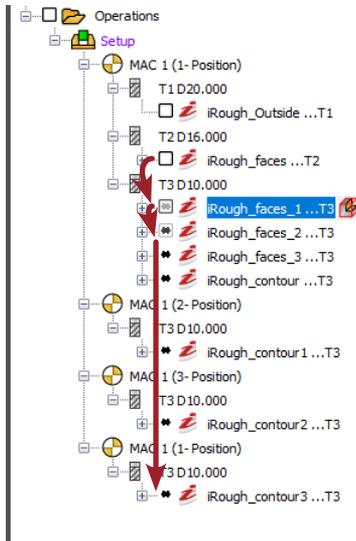
iMachining is considered unique for many reasons. For one, it can comprehensively manage data within and between operations. Over the past several years, the management of data has been enhanced to further improve programming performance and efficiency.

This year happens to be no different now that iMachining 2D utilizes the solid model data. Any edits to an existing operation will ultimately affect the Updated Stock model (USM), thus affecting all subsequent iMachining Feature Recognition operations. Upon recalculation of a modified operation, the latter ones will need to be recalculated.

In such cases, use **Save & Calculate with all related operations** at the bottom of the iMachining Operation dialog box window.



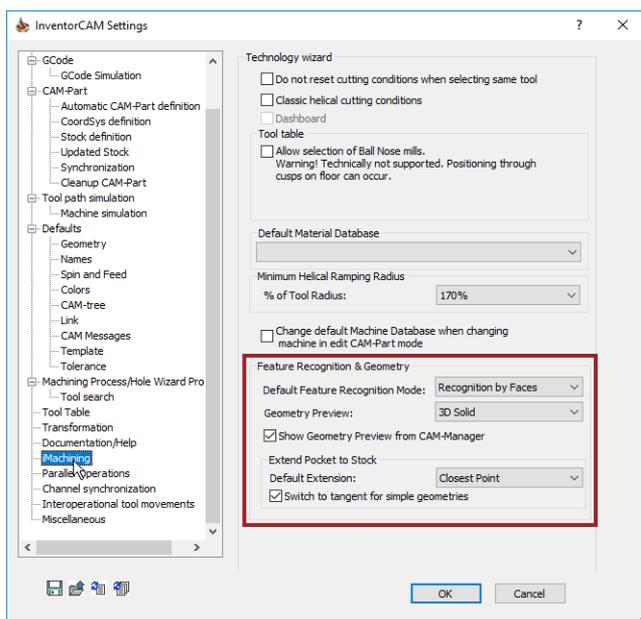
All operations are quickly and easily recalculated in sequence.



Since the USM is shared as a common geometry, each operation automatically receives, updates and recalculates the changes in data without the need for user input.

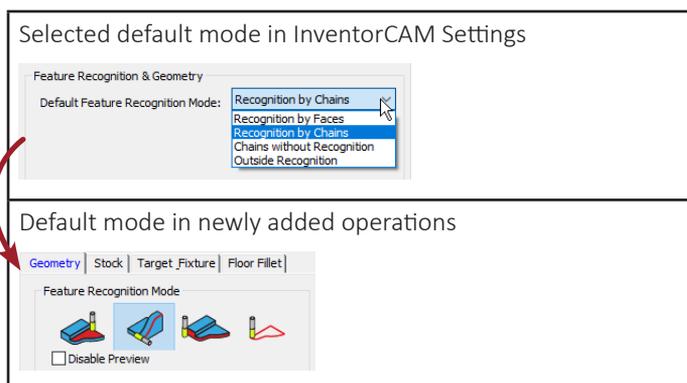
Feature Recognition Settings

If you have preferred settings relevant to iMachining's Feature Recognition technology, you can modify the defaults in the InventorCAM Settings.



In the Feature Recognition & Geometry section on the iMachining page, the following options enable you to modify the default settings:

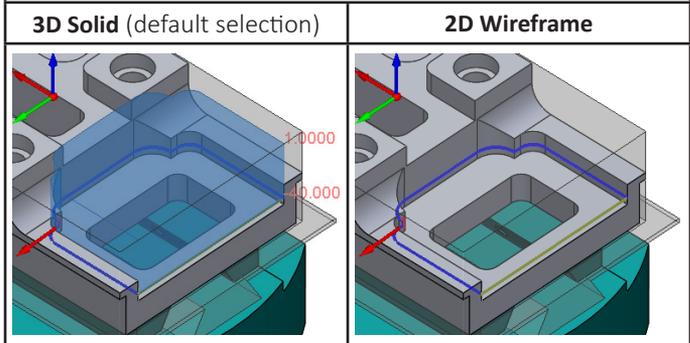
- **Default Feature Recognition Mode** – specifies the default mode used in iMachining 2D operations.



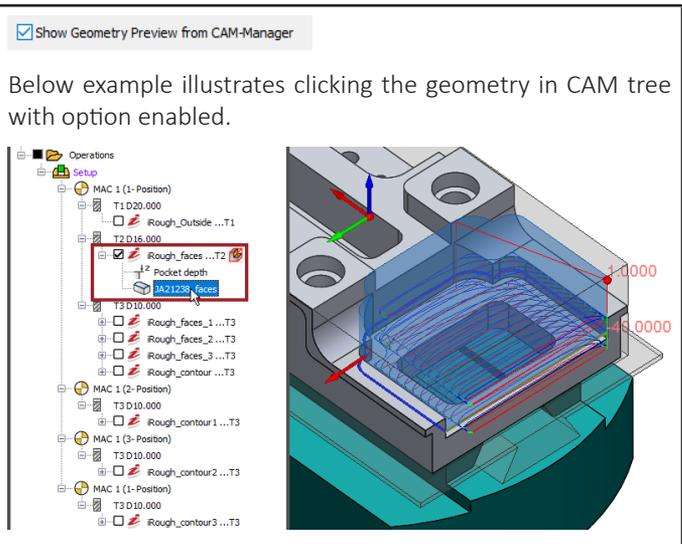


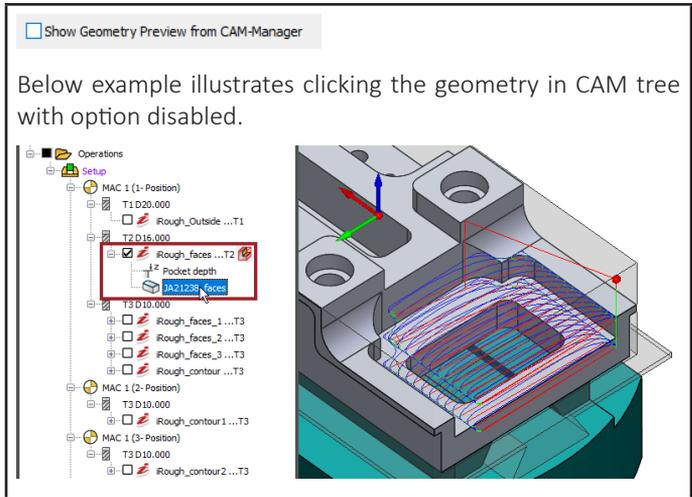
- **Geometry Preview** – specifies how the iMaching regions are displayed in the Autodesk Inventor graphics window.

Choosing **Off** disables the preview visibility for all operations. There are otherwise two available options: 2D Wireframe only or 3D Solid with 2D Wireframe.

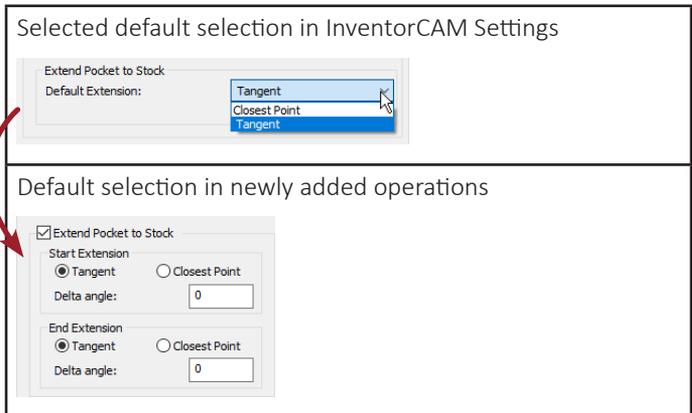


- **Show Geometry Preview from CAM-Manager** – turns off and on the display of the iMaching regions from the CAM tree.

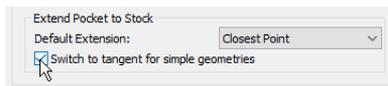




- **Default Extension** – specifies the default Start and End Extension selections used for Extend Pocket to Stock.



- **Switch to tangent for simple geometries** – turns off and on the automatic switching to **Tangent** for Extend Pocket to Stock.



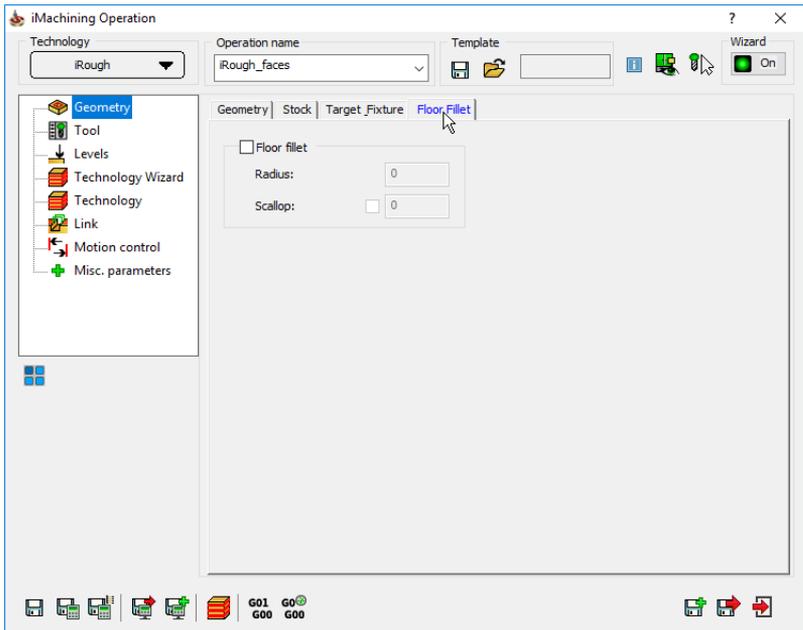
Disabling this option retains a default selection of Closest Point for all operations regardless of the geometry composition.



2. iMachining 2D Floor fillet option moved



With all the aforementioned enhancements added to InventorCAM 2019, the **Floor fillet** option moved to its own tab on the Geometry page.



For more information about the Floor fillet option in iMachining, refer to the InventorCAM Milling Help documentation.

3. Semi-open as closed in iMachining 2D

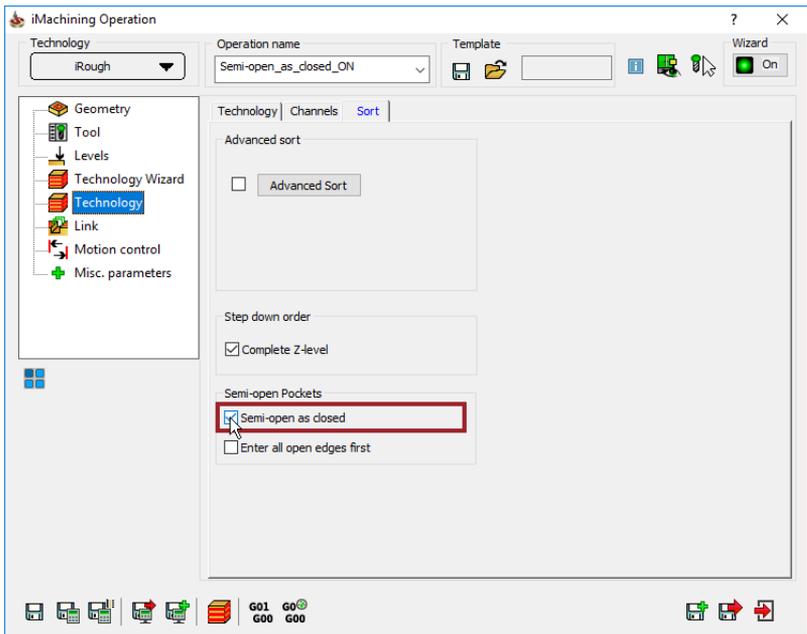
In iMachining 2D operations, the geometry is defined by either closed, open or semi-open pockets.

The Geometry definition determines not only the resulting tool path but also the method used for entering the pocket.

For closed pocket geometries, the tool performs a helical entry into the material. For open and semi-open pocket geometries, the tool approaches the material from outside air. For semi-open pockets specifically, the tool leads into the material through an open edge.

The pocketing tool path techniques are then adapted to each geometry.

NEW In InventorCAM 2019, appearing on the Sort tab of the Technology page, is the **Semi-open as closed** option.



When this option is enabled, all semi-open geometries are instead treated as closed and the iMachining 2D tool path is adjusted automatically.

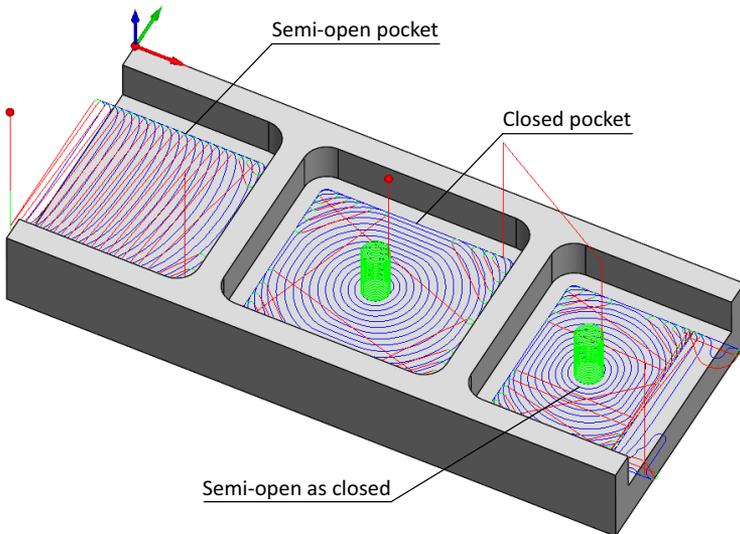


If clicking **Semi-open as closed** does not produce the intended result, it may be that the tool diameter and/or the Minimum Helical Ramping Radius is too large for the tool to fit in the designated area(s).



For more information about Minimum Helical Ramping Radius, refer to the InventorCAM Milling Help documentation.

Illustrated below is the adjusted tool path (far right), which resembles the machining of a closed pocket.

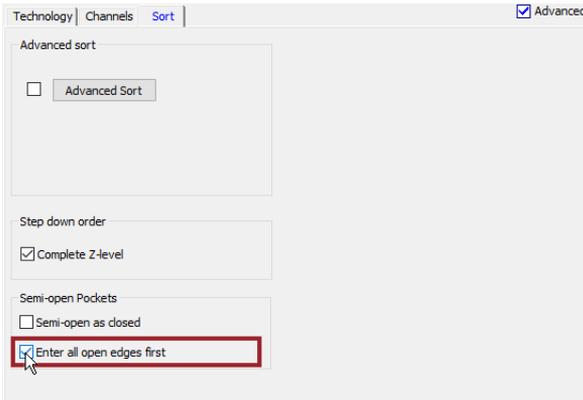


Rather than lead into the material through the open edge, the tool performs a helical entry into the pocket as if it were closed. The pocketing tool path is also characterized by that which is used for a closed geometry. The material open to air is addressed carefully and subsequently removed.

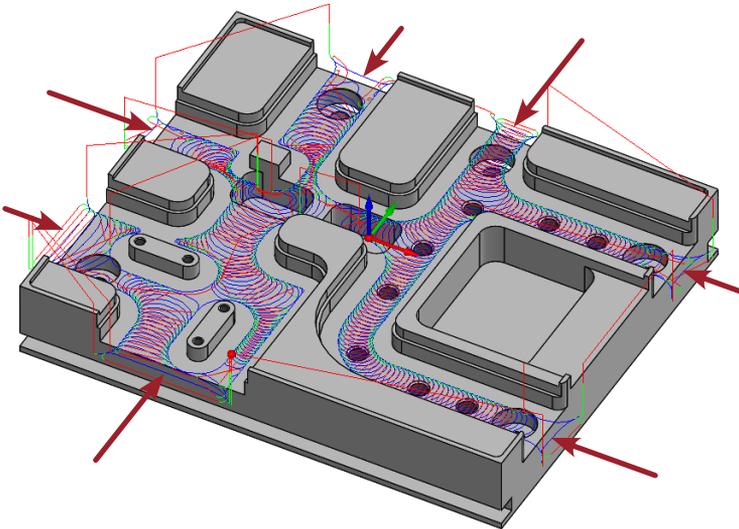
4. Enter all open edges first in iMachining 2D



In InventorCAM 2019 is the **Enter all open edges first** option, also appearing on the Sort tab of the Technology page.



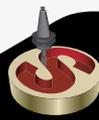
When this option is enabled, the iMachining 2D tool path is automatically adjusted for semi-open pockets having multiple open edges.



Rather than break out from material into air, the tool enters all open edges from outside air before it performs the pocketing tool path.



In the case of brittle materials, entering through the open edges first is preferred to avoid possible breakage of the edges.



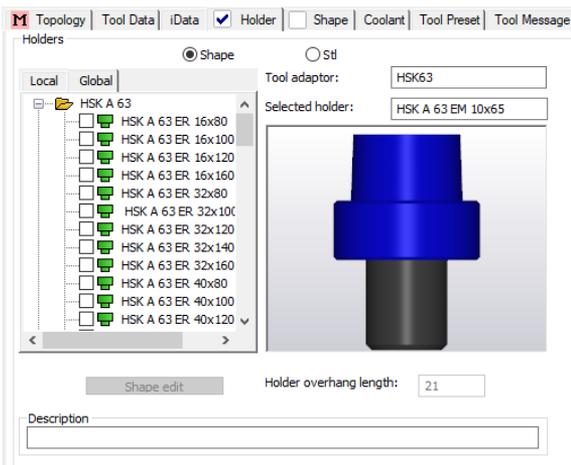
5. STL Holder collision protection in iMachining 3D

InventorCAM enables you to define, edit and use a variety of tool holders.

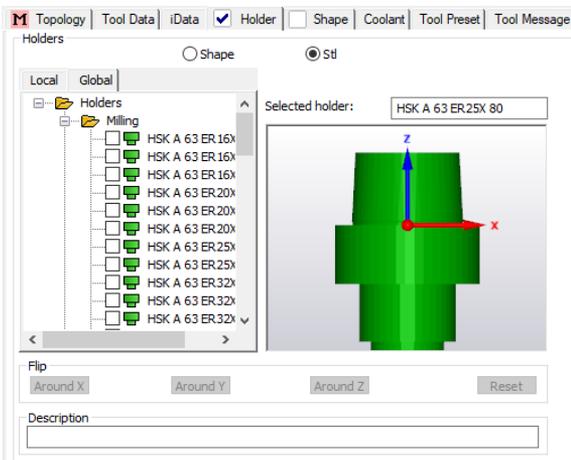
The tool holding system of InventorCAM provides a more realistic visualization of the machining simulation as well as aids in the checking and prevention of all possible collisions between your tool holder and the workpiece.

You can use one of two tool holder classifications:

- Standard/Shape Holders



- STL Holders



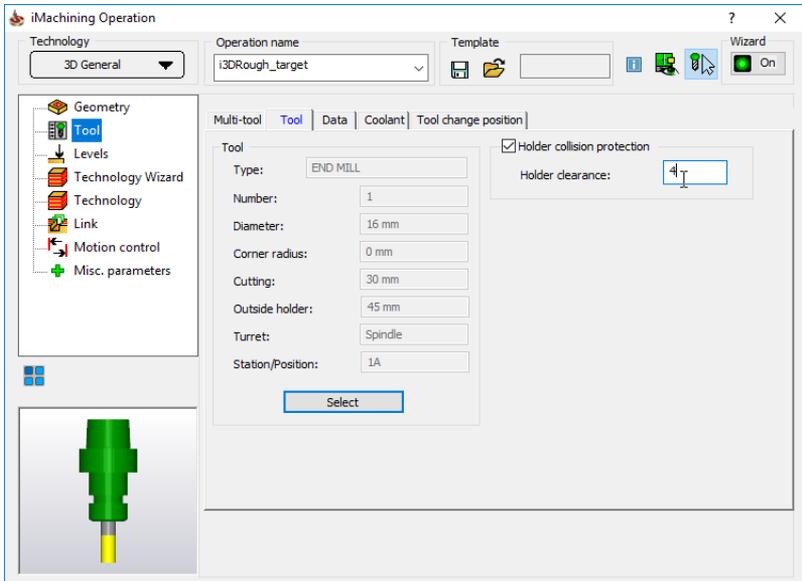
A Standard Holder is comprised of precise geometric data from which InventorCAM generates and uses a 3D Model representation.

A STL Holder, on the other hand, is comprised of a STL file that contains data on the shape and the visual properties of the actual tool holder. InventorCAM uses its STL Model representation.



For more information about the tool holding system, refer to the InventorCAM Milling Help documentation.

Since its original release, iMachining 3D has provided Holder collision protection. This option appears on the Tool tab of the Tool page and is useful when machining deep pockets and/or steep areas.



When **Holder collision protection** is enabled, the iMachining 3D tool path is automatically adjusted taking into account the tool holder geometry plus the Holder clearance value.

The **Holder clearance** parameter defines the distance by which the tool holder can approach the workpiece during the machining.

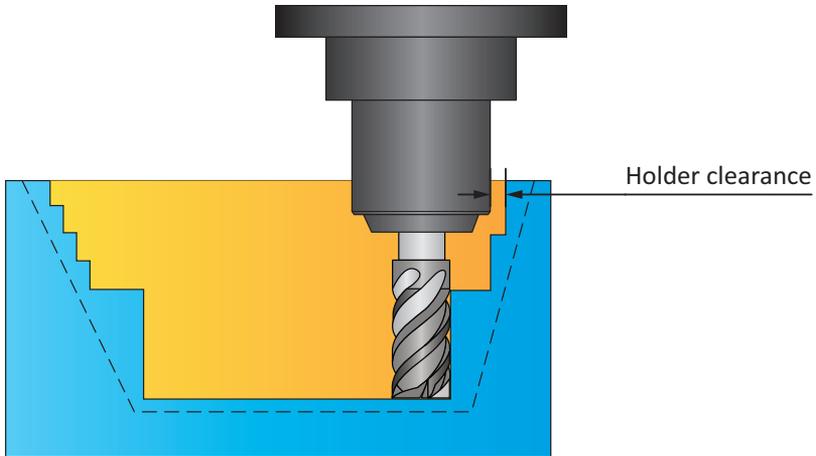


The Tool definition must include a tool holder to take advantage of Holder collision protection in iMachining 3D operations.

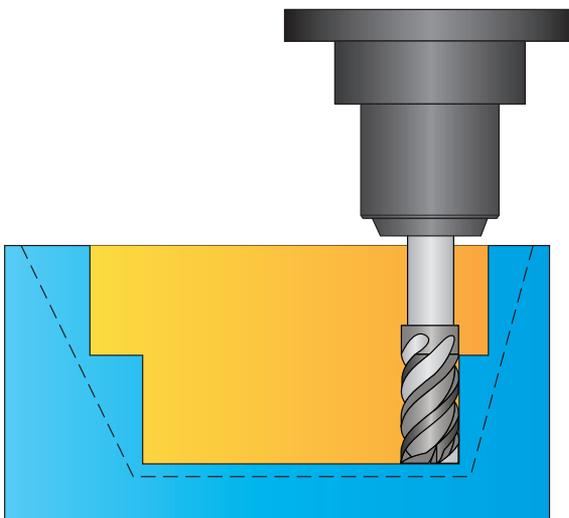


Contact between your defined tool holder and the workpiece is prevented at every stage of the operation machining.

With holder consideration, the outside holder length of the tool can be short and strong, enabling it to run faster with a higher aggressiveness. However, because the tool path is limited, the operation may not be able to remove the extent of material that could otherwise be removed.



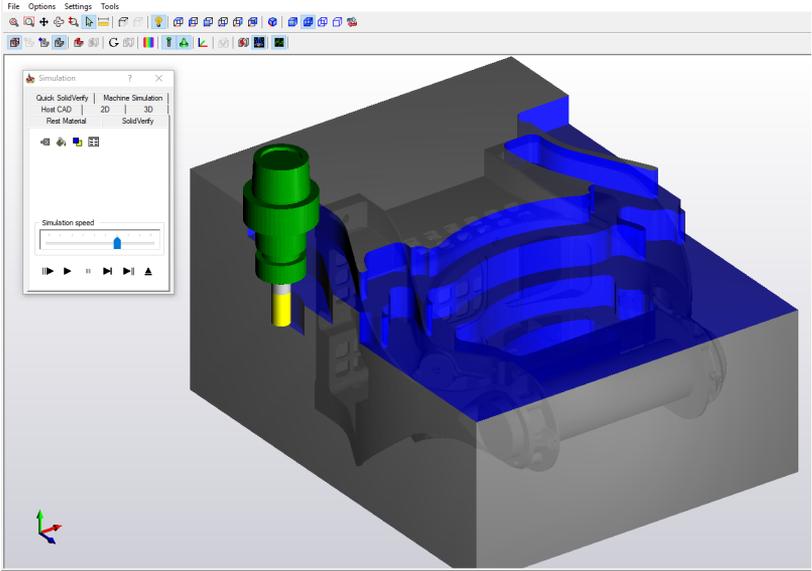
Without holder consideration, the outside holder length of the tool may instead need to be long, making it inherently less stable.



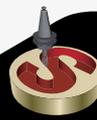
In InventorCAM 2018 and earlier, only protection of Standard Holders geometry was supported in iMachining 3D operations.



In InventorCAM 2019, iMachining 3D adds support of STL Holders geometry when using Holder collision protection.

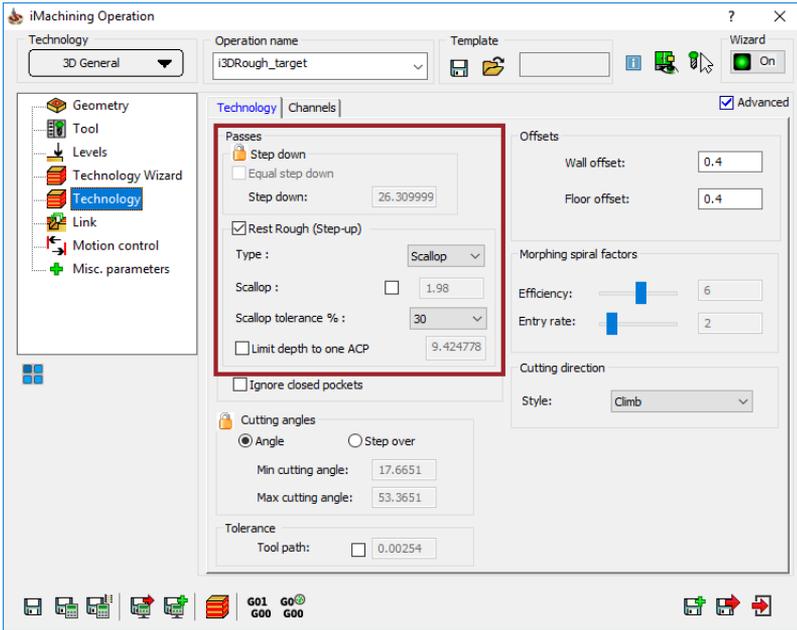


Any possible collisions between your STL tool holder and the workpiece are prevented automatically.



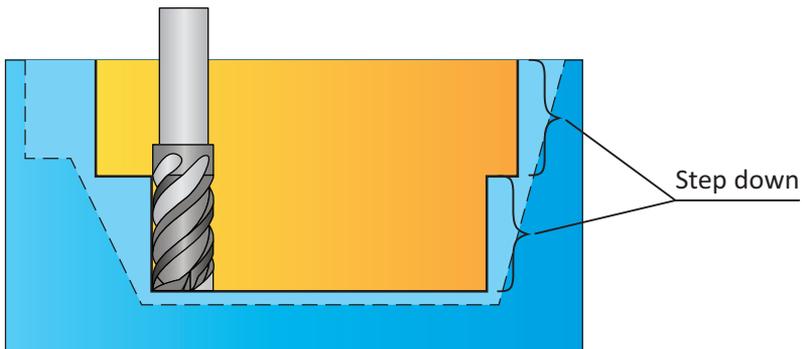
6. Limit depth to one ACP in iMachining 3D

iMachining 3D uses the Step down and the Rest Rough (Step-up) parameters for calculation of its tool path passes.

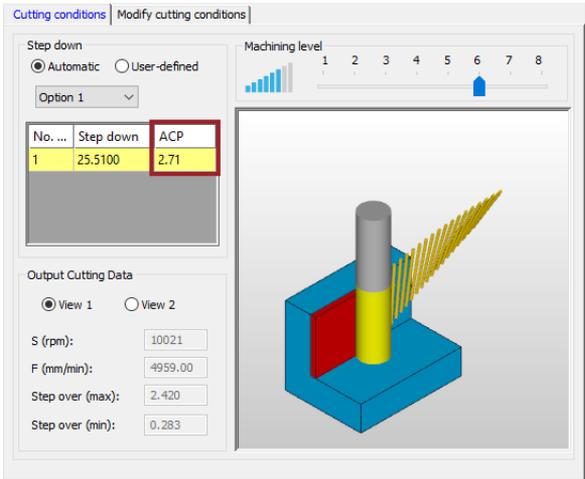


Step down

Using the proven algorithms of iMachining 2D and the Technology Wizard, roughing tool paths are generated first via deep Step down cutting passes.



For Step down, the operation ACPs are automatically calculated by the Wizard and are displayed on the Technology Wizard page.



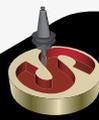
The ACP value reflects the number of Axial Contact Points (not counting the bottom) the defined tool has with the vertical wall it is producing, along a vertical line.

According to iMachining theory, the closer the ACP value is to a whole number (≥ 1.0), the less likely it is that vibrations will develop. Favorable ACPs are therefore taken into account when the Step down depths are calculated.

Since machining with favorable ACPs is not always possible, the Wizard provides ACPs with a corresponding color to indicate whether or not the current situation for stability is considered good.

Illustrated below are the color indications that can be used as a guide to assist you in avoiding vibrations.

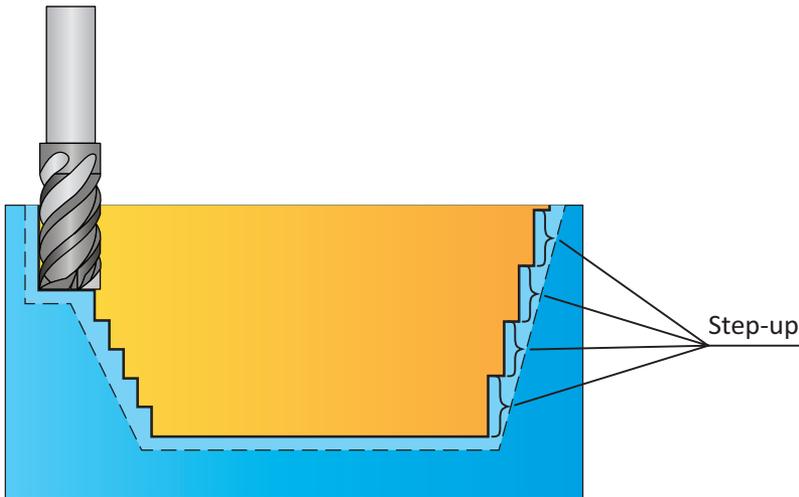
| <table border="1"> <thead> <tr> <th>No. ...</th> <th>Step down</th> <th>ACP</th> </tr> </thead> <tbody> <tr> <td>2</td> <td>15.0000</td> <td>2.01</td> </tr> </tbody> </table> | No. ... | Step down | ACP | 2 | 15.0000 | 2.01 | <table border="1"> <thead> <tr> <th>No. ...</th> <th>Step down</th> <th>ACP</th> </tr> </thead> <tbody> <tr> <td>1</td> <td>13.0000</td> <td>1.74</td> </tr> </tbody> </table> | No. ... | Step down | ACP | 1 | 13.0000 | 1.74 | <table border="1"> <thead> <tr> <th>No. ...</th> <th>Step down</th> <th>ACP</th> </tr> </thead> <tbody> <tr> <td>1</td> <td>5.0000</td> <td>0.67</td> </tr> </tbody> </table> | No. ... | Step down | ACP | 1 | 5.0000 | 0.67 |
|--|---|--|-----|---|---------|------|--|---------|-----------|-----|---|---------|------|---|---------|-----------|-----|---|--------|------|
| No. ... | Step down | ACP | | | | | | | | | | | | | | | | | | |
| 2 | 15.0000 | 2.01 | | | | | | | | | | | | | | | | | | |
| No. ... | Step down | ACP | | | | | | | | | | | | | | | | | | |
| 1 | 13.0000 | 1.74 | | | | | | | | | | | | | | | | | | |
| No. ... | Step down | ACP | | | | | | | | | | | | | | | | | | |
| 1 | 5.0000 | 0.67 | | | | | | | | | | | | | | | | | | |
| <p>Green = Good Preferred</p> | <p>Yellow = Not so good Medium likelihood of vibrations</p> | <p>Red = Bad High likelihood of vibrations</p> | | | | | | | | | | | | | | | | | | |



If the Wizard displays less than preferred ACPs, it is recommended that you take action to produce more favorable ones such as manually changing No. steps or better matching the tool to the current machining situation.

Rest Rough (Step-up)

After Step down, rest roughing tool paths are then generated in Step-up mode to remove the rest material on sloped surfaces of general shaped 3D parts or on higher horizontal surfaces of 3D prismatic parts.



During the Step-up process, the axial depth of cut gets smaller every time a new higher step is machined.

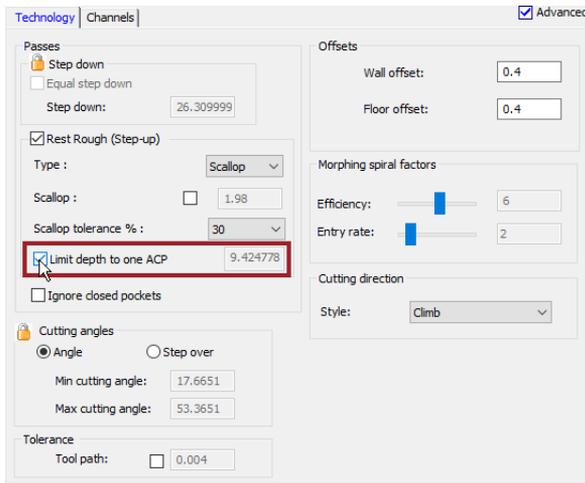
In recent iMachining 3D enhancements, the Technology Wizard was improved to automatically recalculate every Step-up depth and provide for each one the appropriate Cutting conditions.

The Cutting conditions are adjusted dynamically and cannot be displayed in the iMachining Operation dialog box. Also not shown for each Step-up depth are the corresponding ACPs.

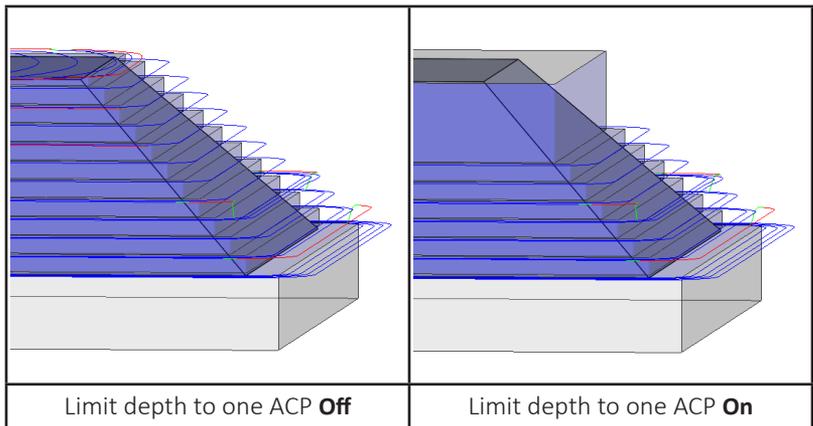
As you might expect, the ACPs become less favorable as the axial depth of cut gets smaller. When machining the highest steps at the top, where < 1.0 ACP is contacting the material, there is a higher likelihood of vibrations.



In InventorCAM 2019 is the **Limit depth to one ACP** option, appearing on the Technology page of the iMachining Operation dialog box.



When this option is enabled, the iMachining 3D Step-up cutting passes are limited to the depths having ≥ 1.0 ACP. The value in the corresponding field reflects the Step-up depth equivalent to 1.0 ACP. Any depth less than the displayed value will be avoided.



Limiting the Step-up depth enables you to match a more suitable tool with the remaining depth, thus avoiding vibrations otherwise caused by possible situations of instability.

The Complete Range of Manufacturing Applications Inside Autodesk® Inventor®

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

The Revolutionary iMachining® Module

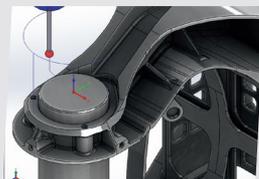
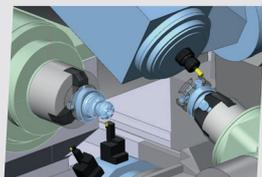
InventorCAM's iMachining module is a giant leap forward in CNC machining technology, reducing cutting times by up to 70% and increasing tool life dramatically. iMachining achieves these advantages by using a patented "Controlled Stepover" technology and managing feed rates throughout the entire tool 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 Autodesk Inventor Integration

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

InventorCAM Powers Up Your Autodesk Inventor System into the Best CAD/CAM Solution.



[youtube.com/SolidCAMProfessor](https://www.youtube.com/SolidCAMProfessor)
[youtube.com/SolidCAMiMachining](https://www.youtube.com/SolidCAMiMachining)



[facebook.com/SolidCAM](https://www.facebook.com/SolidCAM)
[facebook.com/iMachining](https://www.facebook.com/iMachining)



solidcam.com/successes/imachining-successes

