InventorCAM 2018

The unique, revolutionary Milling technology

iMachining®

patent by SolidCAM

TIME SAVINGS **70%** ... AND MORE!

What's **NEW**

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

InventorCAM + Inventor

The Complete, Integrated Manufacturing Solution

InventorCAM

iMachining – The Revolution in CAM!

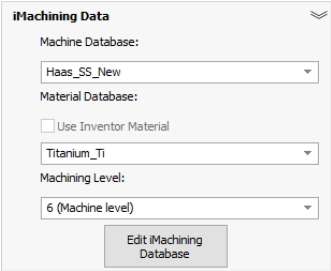
Contents

New Features and Improvements

Document number: IViMNEWENG2-2018

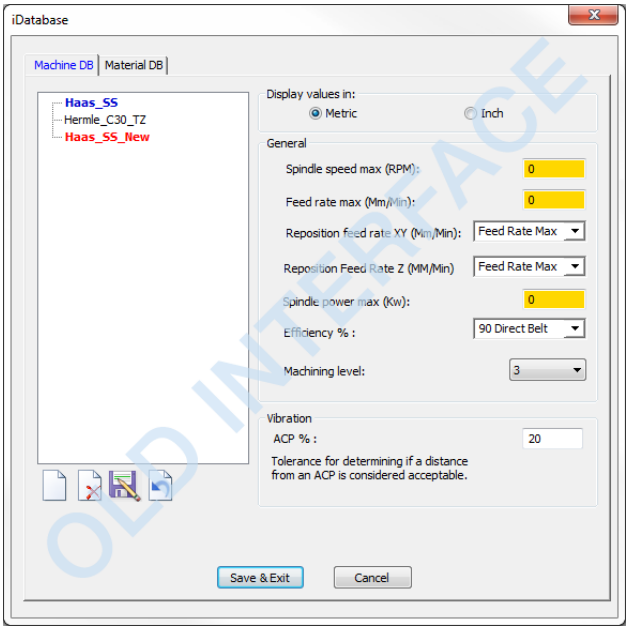
1. Improved Material Database UI with Material Groups

To use the iMachining technology, you have to define and/or select the machine and work material you want to use for the CAM-Part.



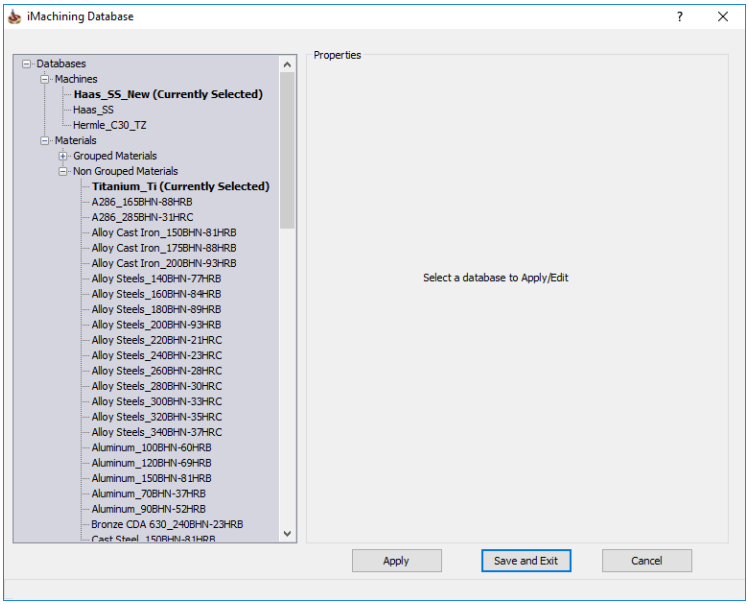
Before making the necessary selections in the CAM-Part Definition, you may have to first add your machine and work material to the iMachining Database. This is especially true if you are a new user, the materials supplied with the system do not cover all your needs or if you find that you are often cutting new materials.

In InventorCAM 2017 and earlier, new and existing machine and material definitions were managed using the iDatabase dialog box.

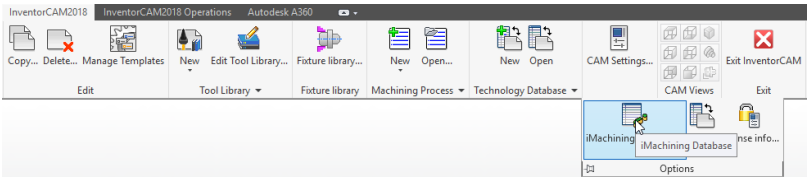


NEW

In InventorCAM 2018, the interface for managing machines and materials in the iMachining Database is completely redesigned.



This new dialog box can be accessed in the same way as previous versions of InventorCAM, either in the CAM-Part Definition using the Edit iMachining Database button or as shown from the CAM menu.



With an emphasis on materials, the new design offers predefined material categories. By grouping materials into specific categories, InventorCAM provides two major advantages:

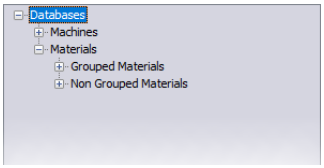
1. The iMachining technology receives enhanced material property data, enabling the Technology Wizard to produce Cutting conditions that are even more accurate than before.

2. It enables you to better manage the material files contained in the iMachining Database, improving its overall organization.

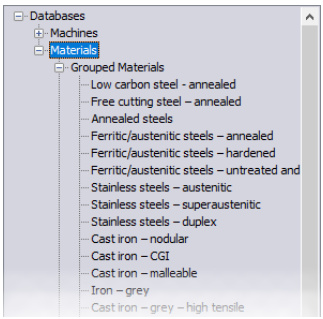
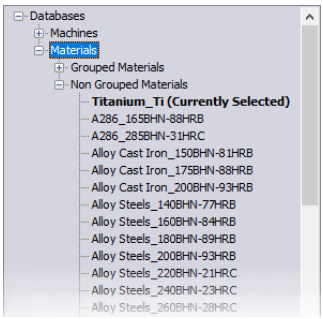
iMachining Database interface

The main features of the new iMachining Database interface are similar to the old iDatabase interface and the overall functionality remains the same. Described below are some notable differences.

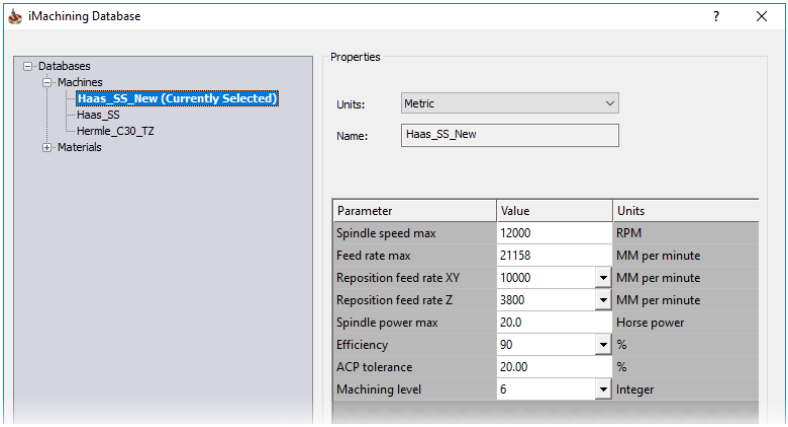
Instead of separating the machines and materials by tabs, the tree on the left now contains the Databases header at the top level. Under it are the Machines and Materials subheaders and their respective lists of available machines and materials.



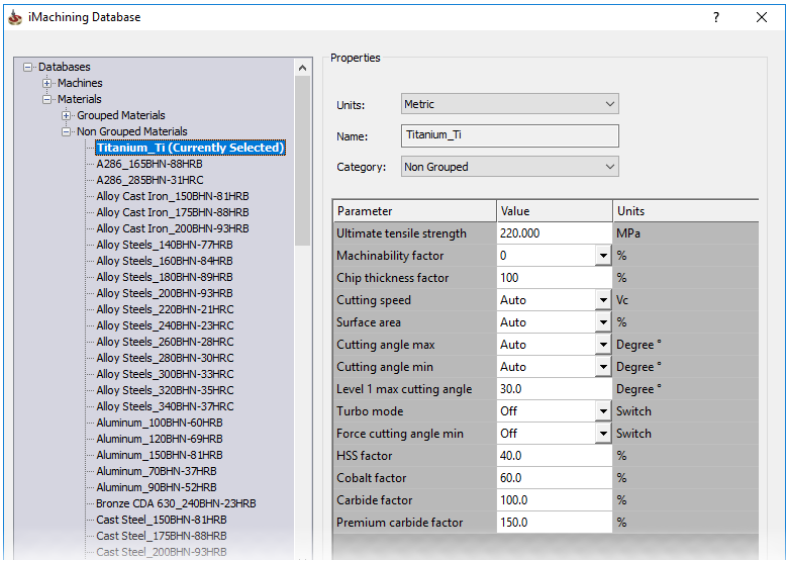
The Materials subheader contains two lower level lists, Grouped Materials and Non Grouped Materials. By default, all the materials originally supplied with the system as well as those that you have already added are organized into the Non Grouped category.

Grouped Materials	Non Grouped Materials
	

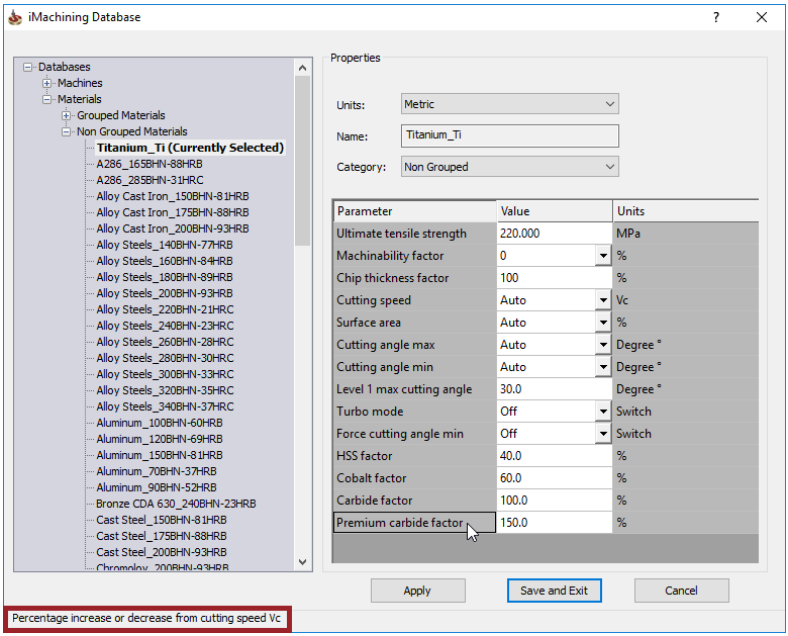
The right side of the dialog box displays the properties by which the selected machine or material is defined. Nearly all parameters and options have not changed, and you can edit the values of existing machines or materials in the same way as previous versions of InventorCAM.



Added to the property data for materials is the **Category** drop-down list, which enables you to group each material into one of the predefined material categories. Grouped materials will appear in the tree under its specified category.



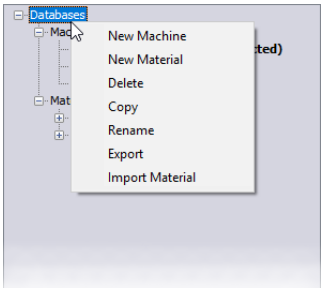
The new iMachining Database interface also displays tooltips in the bottom left corner of the dialog box. Tooltip messages are activated upon clicking any of the parameter names in the Parameter column.



Note that the tooltips are only simplified definitions. For more information about each parameter, refer to the InventorCAM Milling Help documentation.

iMachining Database managing commands

Another notable difference is the iMachining Database managing commands, which are displayed in the right-click menu of the Databases tree on the left.



The following commands enable you to:

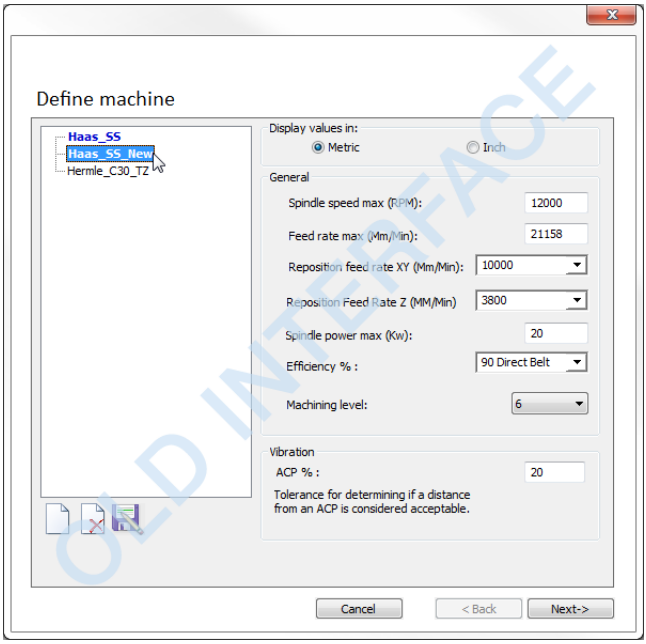
- **New Machine** – add a new machine.
- **New Material** – add a new material.
- **Delete** – delete the selected machine/material.
- **Copy** – copy the selected machine/material.
- **Rename** – rename the selected machine/material.
- **Export** – export the selected machine/material to a specified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

After making changes to the iMachining Database, you can click:

- Apply to save your changes without closing the dialog box.
- Save and Exit to save your changes and close the dialog box.
- Cancel to cancel your changes and close the dialog box. If you used Apply at any point when making changes, Cancel will revert the iMachining Database to your last applied changes.

2. iMachining Data definition requirements

In InventorCAM 2017 and earlier, it was not required to define the iMachining Data in the CAM-Part Definition. If you later wanted to use the iMachining technology, an interface similar to the old iDatabase dialog box appeared upon adding the first iMachining operation to the CAM-Part.



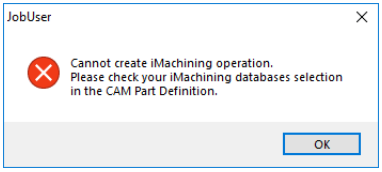
As a supplemental process, this dialog box prompted you to define and/or select the machine and work material that should be used by the Technology Wizard in its Cutting conditions calculations.

In such cases, the iMachining Data selections only saved to the CAM-Part Definition if you saved the operation. If you prematurely exited the operation without saving, the selections were lost and the process would have to be repeated upon adding again your first iMachining operation.

NEW

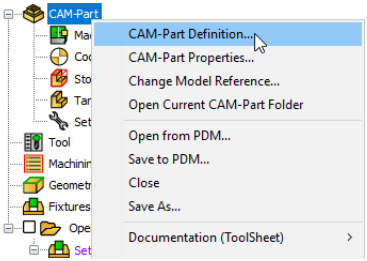
In InventorCAM 2018, the method of later defining the iMachining Data is eliminated by the developers in their continuous efforts to optimize the processing and management of data.

The iMachining Data must now be defined in the CAM-Part Definition prior to adding iMachining operations. If not, the following message appears:

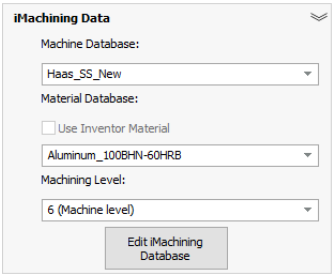


If you encounter this message, accept it by clicking **OK** and then follow these simple steps:

1. Edit the CAM-Part Definition...

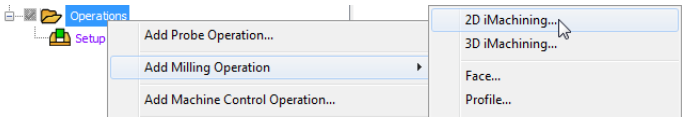


In the iMachining Data section of the Milling Part Data dialog box, define your machine and work material selections.



2. Click ✔ to confirm the CAM-Part Definition changes.

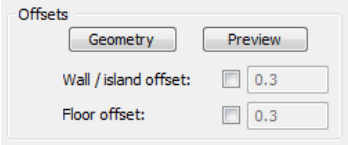
3. Now you can add iMachining operations to your CAM-Part.



3. Enhanced Modify Geometry in iMachining 2D

In iMachining 2D, the geometry is defined as a pocket that can be closed, open or semi-open (containing open edges). The Geometry definition can consist of one or more chains. Chain geometries can be defined in a number of ways and can be later modified using the Modify Geometry feature of InventorCAM.

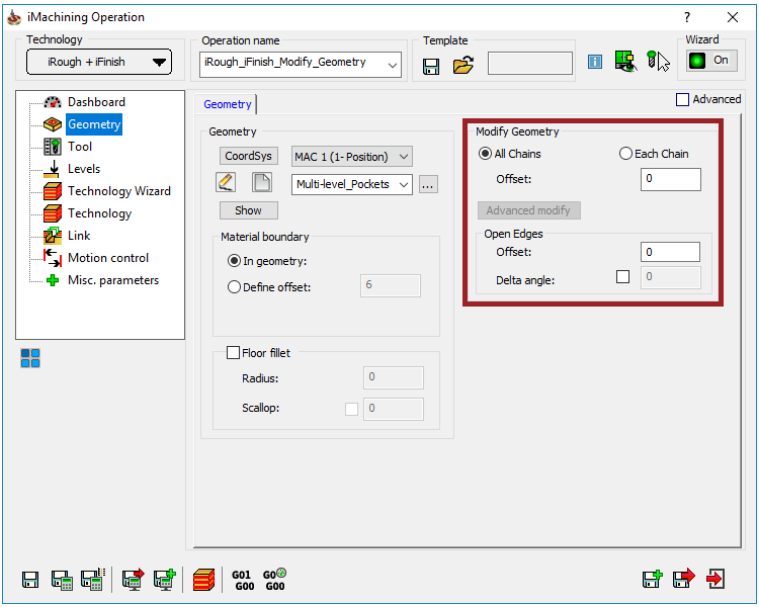
In InventorCAM 2017 and earlier, the buttons to initiate the Modify Geometry feature appeared on the Technology page of the iMachining Operation dialog box.



Clicking the **Geometry** button opened the Modify Geometry dialog box, enabling you to offset any one or more chains in the defined geometry.

Clicking the **Preview** button displayed a projection of the machining geometry (modified or not) directly on the solid model in the Autodesk Inventor graphics window.

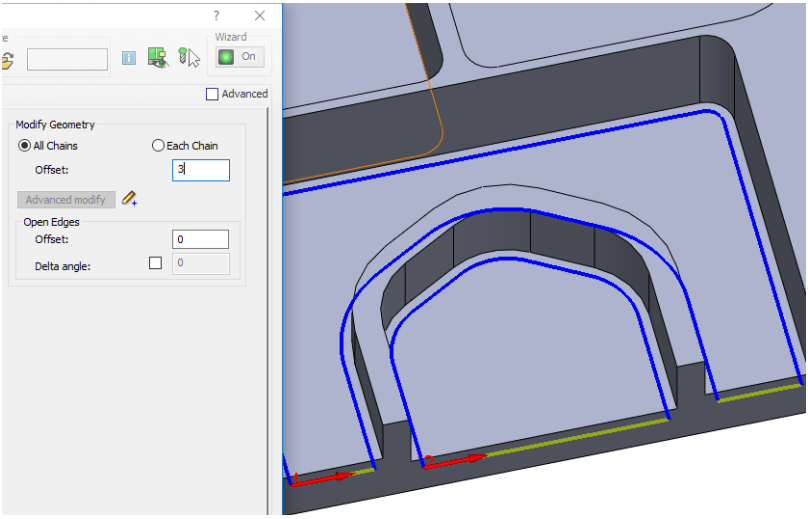
NEW In InventorCAM 2018, the Modify Geometry options and Offset parameters now appear on the Geometry page.



As described below, the Modify Geometry section enables you to specify several geometry-related modifications.

Offset all geometry chains

When the geometry consists of one or more chains and you want to offset all chains by the same value, use the **All Chains** option and enter your desired value in the Offset field.



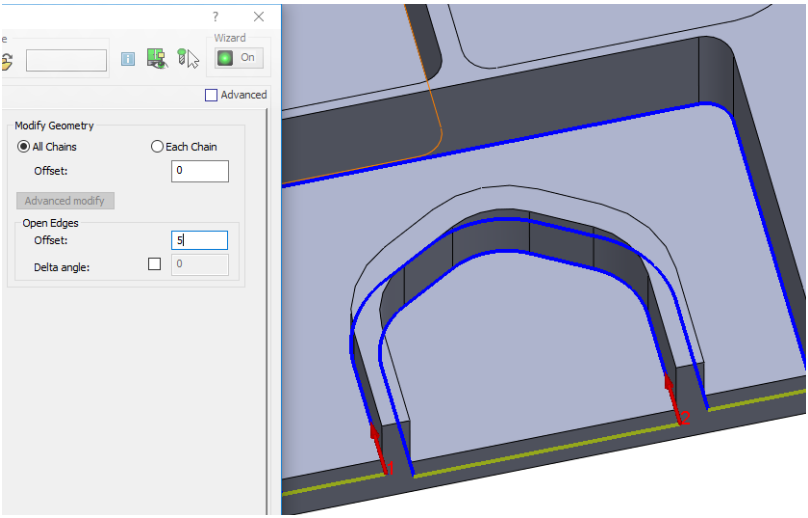
All geometry chains (closed, open and semi-open) are modified accordingly.

As a safeguard, InventorCAM does not apply negative offset values to the open edge(s) of a semi-open chain. In such cases, open edges are offset outward into air by the opposing positive value (e.g., +2 versus -2).

In cases of geometry configurations consisting of open chains (open pockets, open pockets with islands, etc.) and a negative offset value is defined, InventorCAM determines the appropriate offsetting direction. In addition, InventorCAM properly trims any intersecting chains that result from the modification.

Offset open edges only

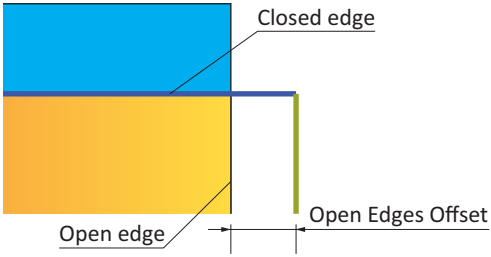
When the geometry consists of open and/or semi-open chains and you want to offset only the open edges of those chains by the same value, use the **All Chains** option and enter your desired value in the Open Edges Offset field.



Open edge(s) offsetting further enhances the Modify Geometry feature by eliminating the need to sketch geometry chains in Autodesk Inventor. Listed below are several instances where you might find this added functionality useful:

- Extra material remains on the outside of stock
- Extra material remains on an open island
- To extend open edges of a semi-open pocket that opens to the outside of a part (and the outside has not been roughed)
- To extend open edges of a semi-open pocket by the wall offset of previous operation, where a small amount of material remains (size = wall offset) on the open edges
- To extend open edges of an open island by the wall offset of previous operation, where a small amount of material remains (size = wall offset) on the open edges in the case of bottom-up cutting with multiple iMachining operations

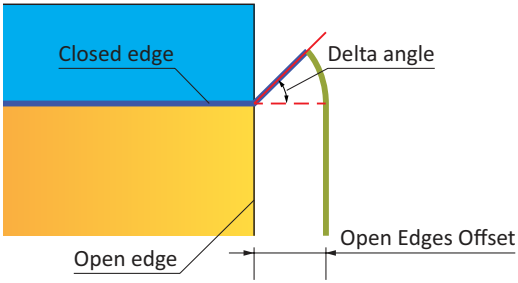
When InventorCAM offsets the open edge(s) of a semi-open chain, its closed edges are extended tangentially by default.



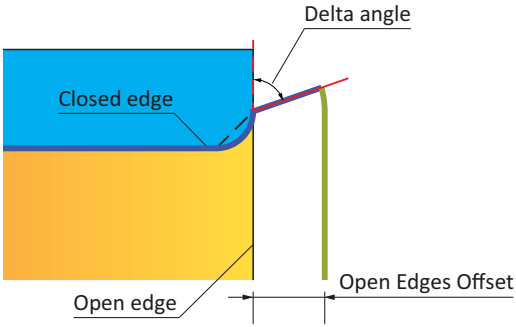
Alternatively, the **Delta angle** option can be used to apply an angle to the extended segments of closed edges. InventorCAM determines the direction to which the angle is measured according to the geometric relationship of closed to open edges.

Take, for instance, the two examples described below.

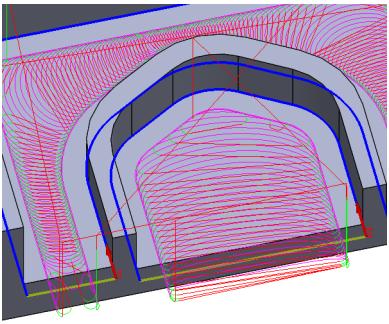
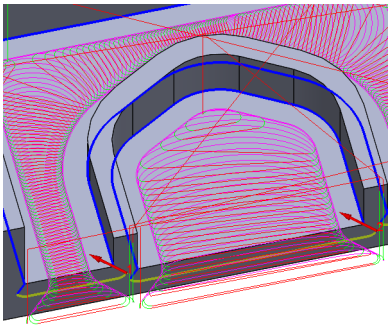
If the closed edge intersects the open edge perpendicularly, the Delta angle is measured away from the open pocket area.



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

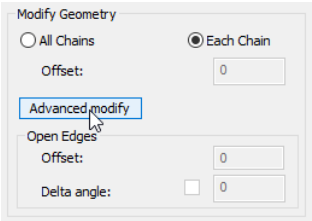


The Delta angle changes the chain boundary shape in open pocket areas, thus changing the shape of the resulting tool path.

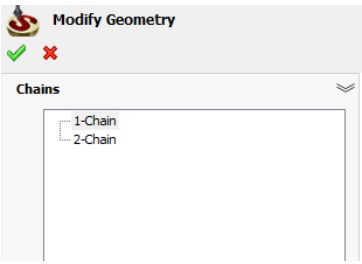
	
5 mm (0.2 in) Open Edges Offset w/ No Delta angle	**5 mm (0.2 in) Open Edges Offset w/ 45° Delta angle**

Offset each geometry chain

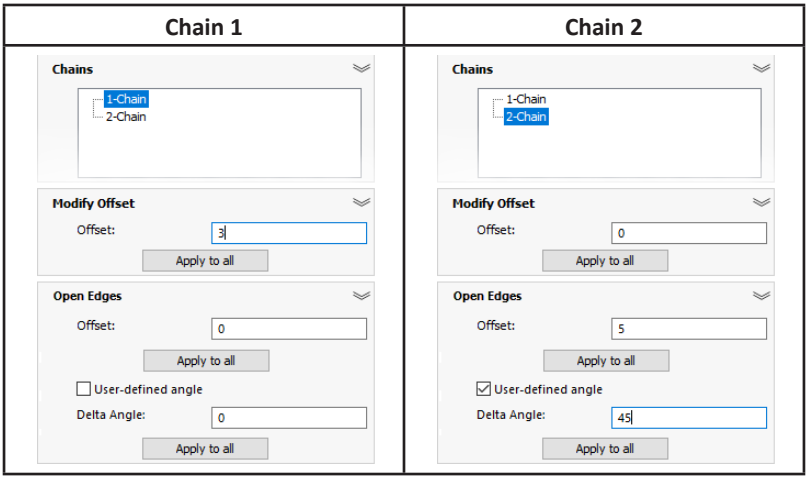
When the geometry consists of more than one chain and you want to offset each chain (closed, open and semi-open) by a different value, use the **Each Chain** option.



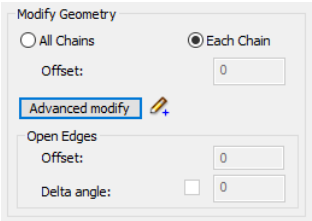
When selected, the **Advanced modify** button is activated. Clicking it displays the Modify Geometry dialog box.



As shown in the example below, the Chains list enables you to define the Offset parameters specific to the selected chain.



After confirming the Modify Geometry dialog box, an icon will appear next to the Advanced modify button.



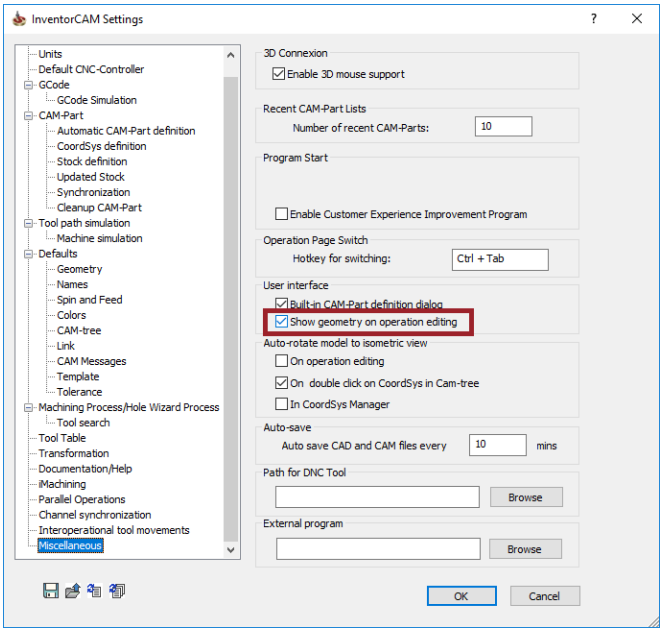
One of three icons provides you with the following feedback:

- indicates that offset modifications have been made with positive values only.
- indicates that offset modifications have been made with negative values only.
- indicates that offset modifications have been made with both positive and negative values.

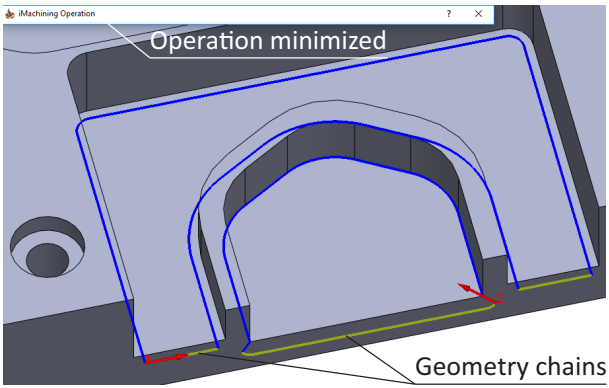
Hover your mouse pointer over the offset modification icon to see a screen tip of up to the first ten modified chains and their specified offsets.

New option for previewing geometry chains

The Preview button in previous versions of InventorCAM is eliminated and is replaced by the new option of **Show geometry on operation editing** in the InventorCAM Settings.



This setting is on by default, enabling you to always see a preview of the machining geometry (modified or not) while actively working in the operation. When using the Modify Geometry feature, changes to the geometry chains are displayed in real-time directly on the solid model.



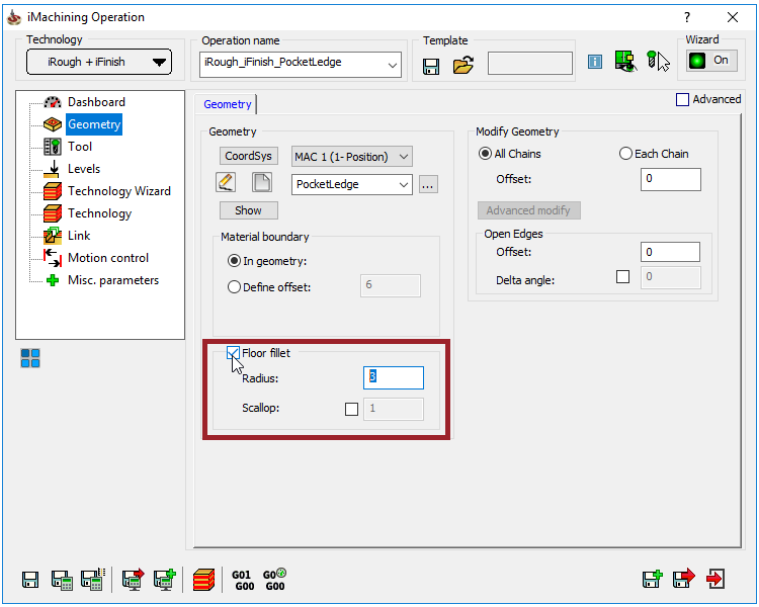
4. Floor fillet machining in iMachining 2D

Generally, the machining of floor fillets has its limitations. The proper machining of such features can be accomplished, but doing so is primarily up to the user and often requires different types of workarounds.

NEW

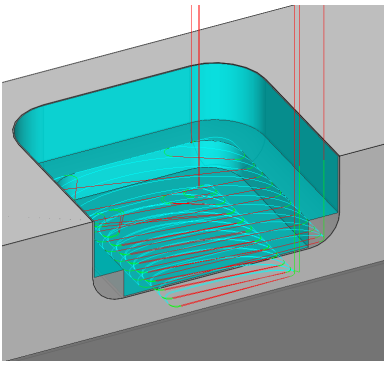
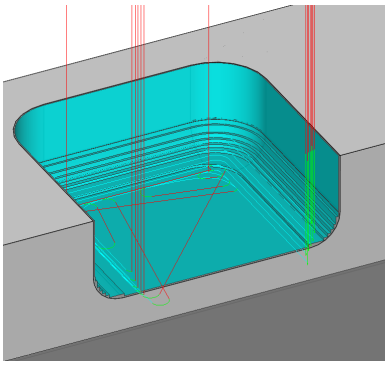
In InventorCAM 2018, the iMachining 2D option of **Floor fillet** can be used to easily machine pockets with floor fillet features. The iMachining tool path is automatically optimized, consisting of adjusted Step down (roughing) passes as well as added Step-up (rest roughing) passes.

When you enable the **Floor fillet** option, appearing on the Geometry page, its corresponding parameters are opened for editing.

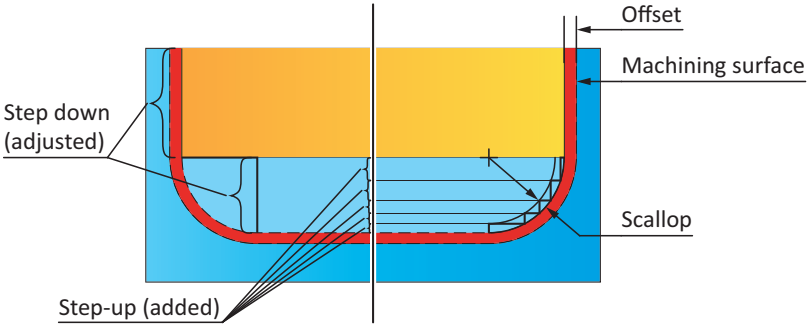


The Radius parameter is the only required input, defining the size of the fillet. Its value informs the iMachining technology of the floor fillet geometry and is used to calculate the tool path adjustments for roughing the pocket.

The Scallop value, which is suggested according to the fillet size, is used to calculate the additional tool path for rest roughing the floor fillet.

Step down tool path	Step-up tool path
	

Following the Step down passes, the Step-up passes are performed creating a staircase-like effect on the floor fillet. The Scallop parameter, which creates the effect, specifies the distance to the peak of all steps on the fillet that are measured radially from the machining surface.



Based on the curvature parameters of the fillet, the Step-up height changes dynamically in order to maintain the specified Scallop value. Similarly to iMachining 3D, every scallop produced is therefore a True Scallop.

You can select the override check box and specify a smaller value, for example, to produce finer passes. Note that smaller values will result in proportionately longer calculation and cycle times.

After the roughing/rest roughing is performed, an even amount of material remains on the fillet surface, making it more suitable for finishing. You can now also finish with ball nose mills or bull nose mills having a Corner Radius value that is smaller than the Radius value of the fillet feature.

5. Variable levels in iMachining 2D

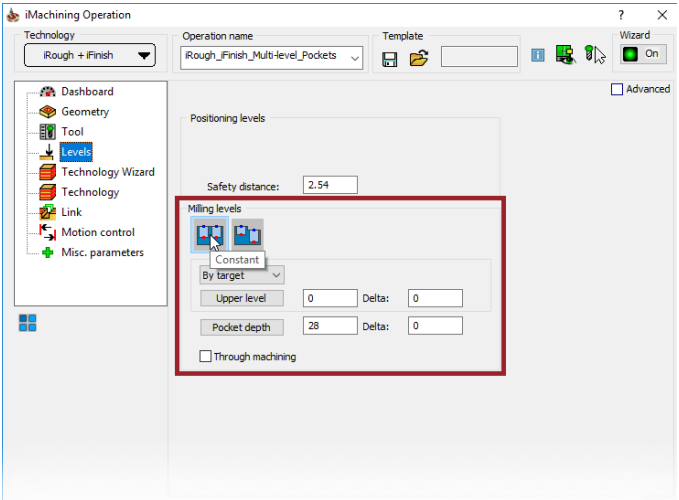
Prior to InventorCAM 2018, a single iMachining 2D operation could be defined to machine more than one chain geometry, but it was required that all geometries of the operation have the same machining levels. For geometries of different depths, it was necessary to define multiple operations.

NEW

iMachining 2D now supports the machining of geometries with different depths in a single operation. On the Levels page, the Milling levels section enables you to select the machining levels using one of two options.

Constant levels (default selection)

When **Constant** is chosen, the Milling levels parameters appear as they did in previous versions of InventorCAM.



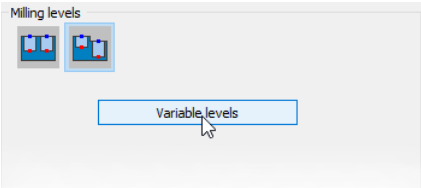
The iMachining technology uses the same Upper level, Pocket depth and Delta definitions for all chains in the geometry.

Variable levels

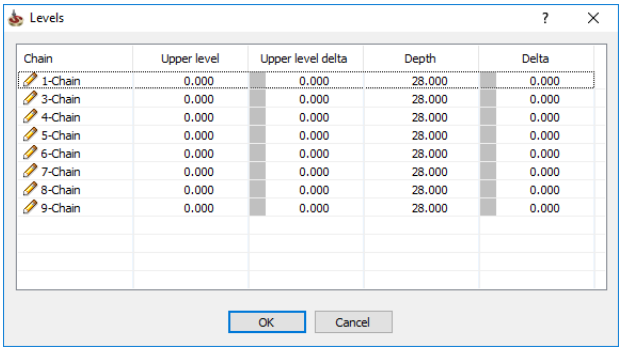
When you want to define for each chain different Upper level, Pocket depth and Delta values, use the **Variable** option.

This option can help simplify the task of programming, especially when you want to use the same tool to perform the machining of many geometries having different depths. Such a task can now be achieved in just one operation.

Upon choosing  in the Milling levels section, the parameters are hidden and the **Variable levels** button appears.

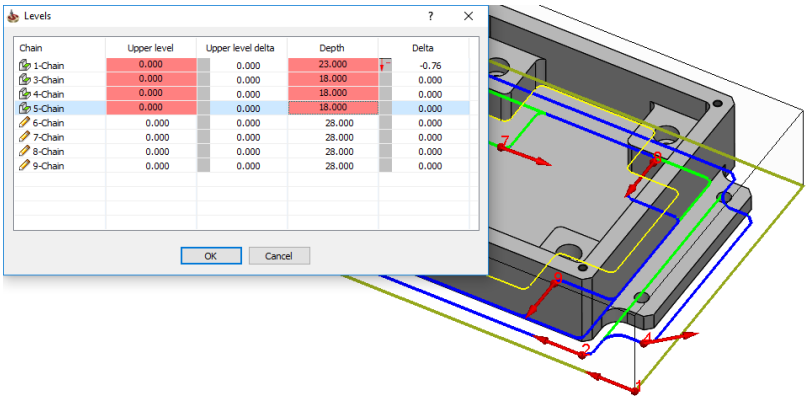


Clicking it displays the Levels dialog box.



This dialog box enables you to define separate Upper level, Pocket depth and Delta values for each chain in the geometry.

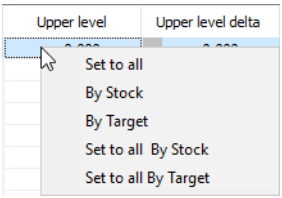
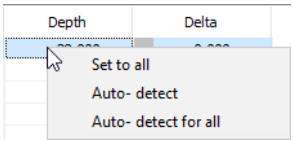
The **Chain** column contains a list of all chains defined for the operation. When a chain is selected, its geometry is highlighted in the Autodesk Inventor graphics window.



For each chain, you can select the following parameters field and then pick/edit its level directly on the solid model:

- **Upper level** specifies the Z-level at which the cutting starts for the selected chain.
- **Upper level delta** specifies the Z-level offset for the Upper level that can be changed with its associativity preserved.
- **Depth (Lower level)** specifies the Z-level at which the cutting ends for the selected chain.
- **Delta** specifies the Z-level offset for the Depth (Lower level) that can be changed with its associativity preserved.

Values can also be entered manually by second clicking a field or if you prefer, InventorCAM can automate the Levels definition by using the right-click commands of the Upper level and Depth menus.

Upper level menu	Depth menu
	
Set to all specifies for all chains the selected Upper level value.	**Set to all** specifies for all chains the selected Depth value.
By Stock specifies the value for the selected field according to the Upper level of the Stock model.	**Auto-detect** specifies the value for the selected field according to the Z-level depth of the chain.
By Target specifies the value for the selected field according to the Upper level of the Target model.	**Auto-detect for all** specifies the value for all chains according to their Z-level depth.
Set to all By Stock specifies the value for all chains according to the Upper level of the Stock model.	
Set to all By Target specifies the value for all chains according to the Upper level of the Target model.	

The following icons indicate for each chain the state of its Depth definition:

- – the Depth is not yet defined or otherwise defined by some arbitrary number and should be edited.
- – the Depth is defined, but its value is not associative to the solid model.
- – the Depth is defined and its value is associative to the solid model.

When defining Upper level delta and/or Delta depths, the direction of its measurement is represented by the arrow next to the text field:

- – the Delta depth is defined with a positive offset value (in the positive direction of the Z-Axis).
- – the Delta depth is defined with a negative offset value (in the negative direction of the Z-Axis).

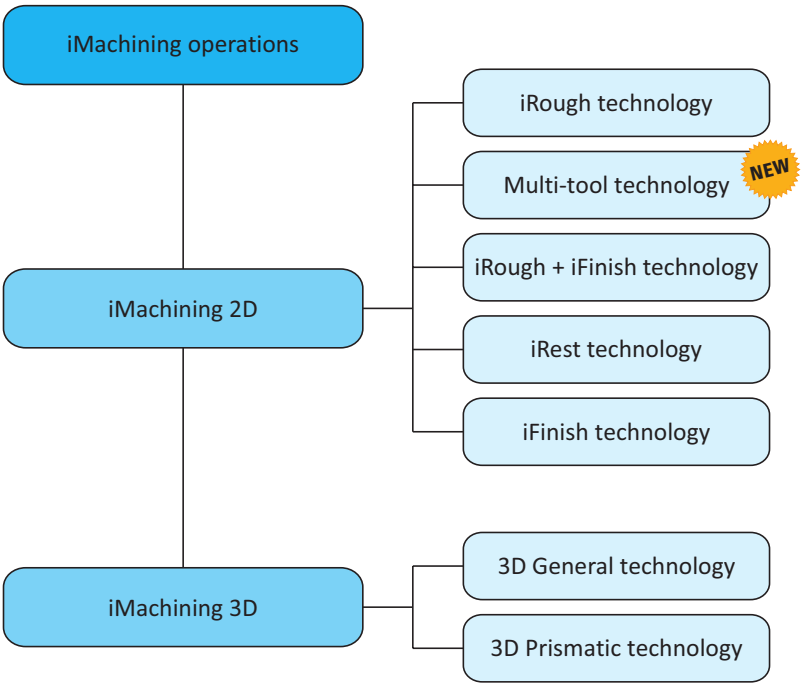
Note: The Technology Wizard displays only the Cutting conditions for the geometry having the largest total depth. If you want to see or if you want to modify the Cutting conditions for geometries of smaller total depths, each geometry must be defined in a separate operation.

6. Multi-tool functionality in iMachining 2D & 3D

The iMachining technology is constantly progressing with new features and improvements. Aside from its revolutionary tool path, combining ease of use with unique functionality and the automatic management of complex data has always been at the forefront of development, all while continuing to provide unprecedented results.

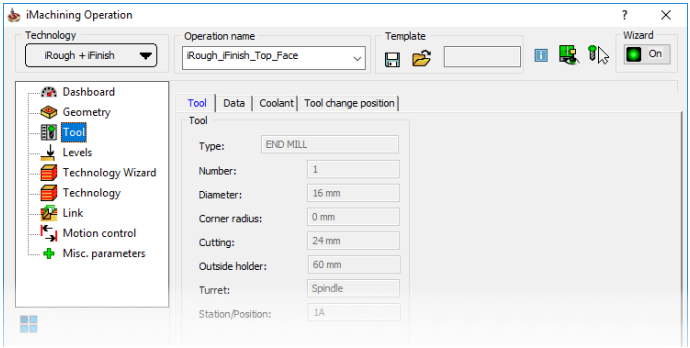
Take, for example, the Operation definition process for iMachining, which requires only three inputs: the Geometry, Tool and Levels selections. With this information, in conjunction with the iMachining Data (machine and work material) defined for the CAM-Part, the iMachining system by default calculates all other parameters and the Technology Wizard likewise provides optimal sets of Cutting conditions that are specific to each operation and to your machining case.

Whether you use iMachining 2D or iMachining 3D, each operation enables you to perform a machining task that is specific to the chosen Technology type.



The specified type determines the functionality of the operation, the availability of certain parameters and options as well as the resulting tool path.

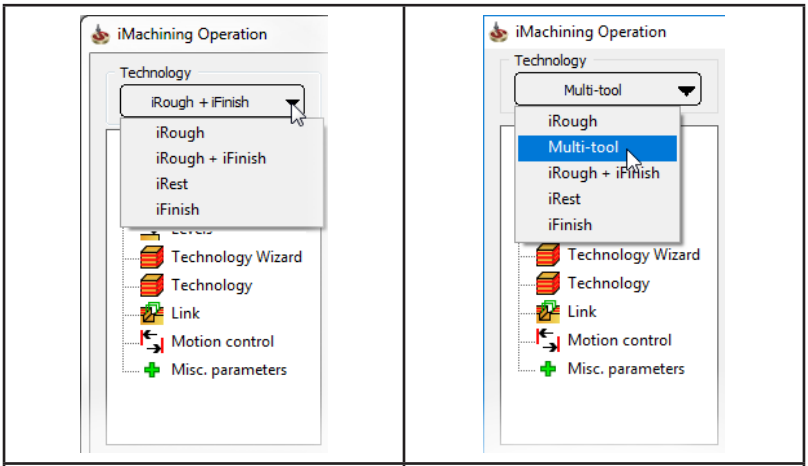
Prior to InventorCAM 2018, the Tool definition in an iMachining operation, regardless of the Technology type, could contain only one cutting tool.



Though you could use the Save & Copy feature to define a sequence of operations (e.g., roughing and finishing with multiple tools in the case of iMachining 2D), each operation had to be edited separately for each tool.

New Multi-tool Technology type in iMachining 2D

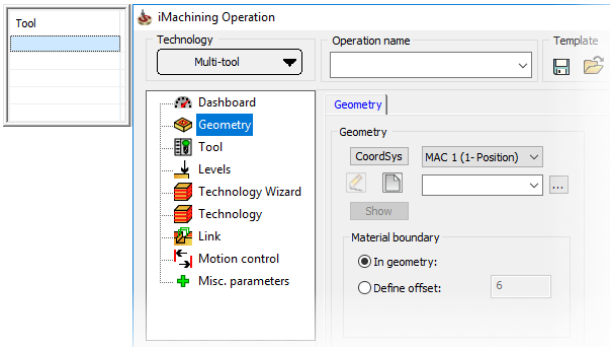
In InventorCAM 2018, iMachining 2D supports a new Technology type.



InventorCAM 2017	InventorCAM 2018

The **Multi-tool** technology was developed as a means to further simplify the task of programming. It not only offers the combined functionality of roughing and finishing but also enables you to define and edit related operations that use multiple tools, all from within a single interface.

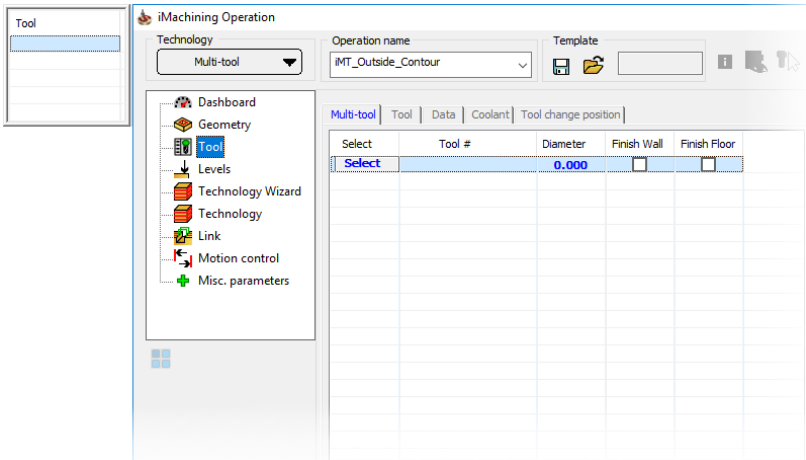
Upon choosing **Multi-tool** from the Technology drop-down list, the Tool grid window appears fixed to the top left corner of the iMachining Operation dialog box.



This is the first of two main interface features of the Multi-tool operation, which will later act as a hub after multiple tools have been defined.

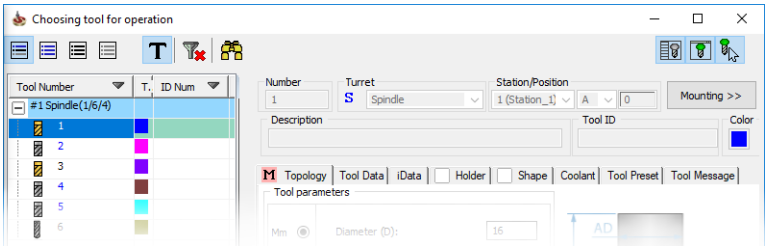
The operation can be defined as usual by making the necessary Geometry, Tool and Levels selections.

For the Tool definition, the Multi-tool tab appears on the Tool page.

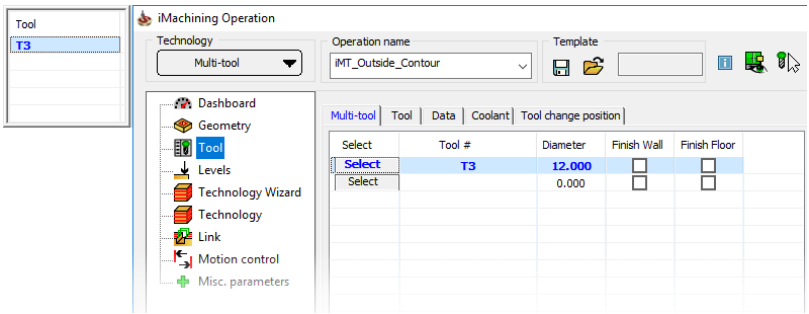


This is the second main interface feature that enables you to define and manage all the tools you want to use in the Multi-tool operation. For each tool, you can quickly assign to it the wall and/or floor finishing using the respective check boxes.

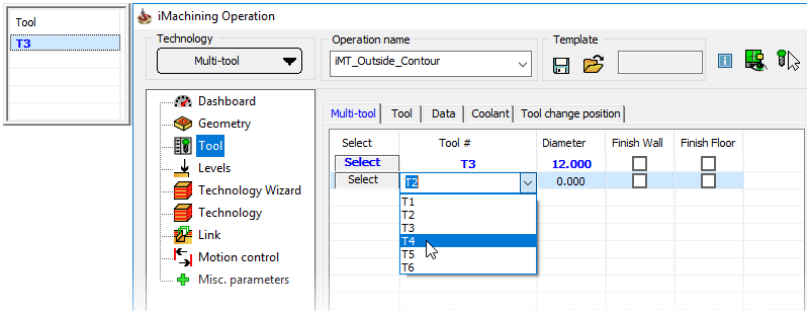
Clicking the **Select** button displays the familiar Choosing tool for operation dialog box with Part Tool Table, which can be used exactly as you would for any other operation in InventorCAM.



For iMachining Multi-tool operations, when the tool is selected, it is added to the Multi-tool table and an additional empty row appears, enabling you to select another tool.

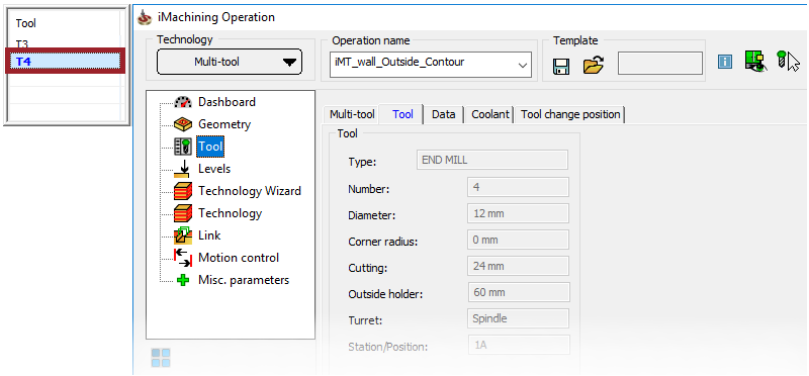


In cases where you may have a predefined Tool Library, you can bypass the Choosing tool for operation dialog box. Clicking the **Tool #** field displays a list of all the tools contained in the Part Tool Table.

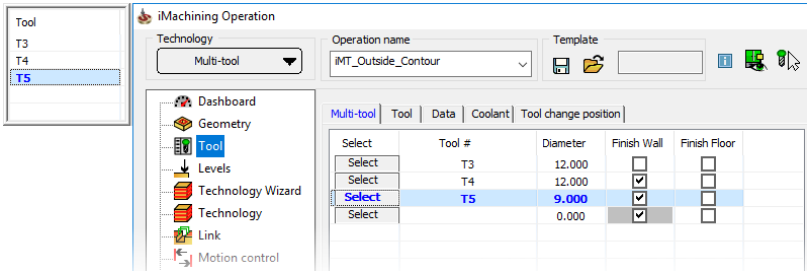


Selecting your tools in this manner can improve programming performance and efficiency by minimizing number of clicks.

Displayed in the Multi-tool table is basic information about each tool (i.e., Tool Number and Diameter). More detailed information, corresponding to the currently selected tool in the Tool grid, can be seen on the Tool tab.



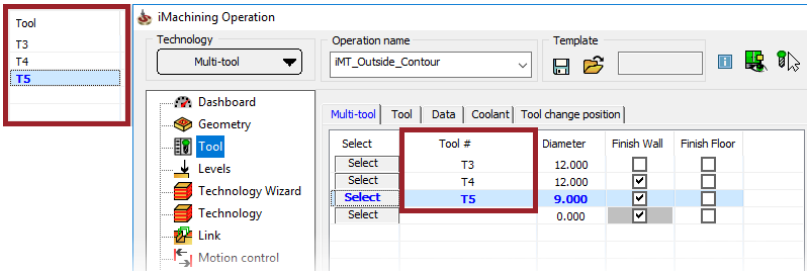
In the example below, the Tool definition contains three tools to perform the rough and finish machining of the outside contour of a bracket.



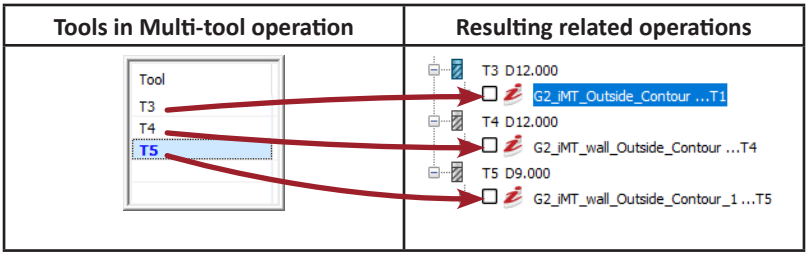
In this case, only the walls require finishing.

In iMachining Multi-tool operations, every tool performs the roughing or rest roughing plus the selected finishing option(s).

All the defined tools are reflected in the Tool grid window.

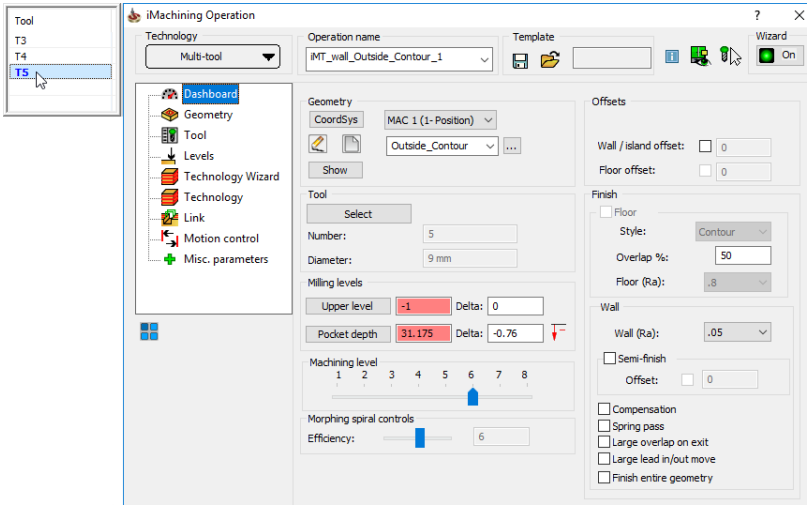


The tools in the Tool grid represent a group of sequential machining tasks (or related operations) that together make up the Multi-tool operation.



When later saving/saving and calculating, the related operations will appear separately in the CAM tree.

Using the Tool grid, the data for each of the related operations can be viewed and edited.



The data shown in the iMachining Operation interface corresponds to the currently selected tool in the Tool grid.

Data management and associativity

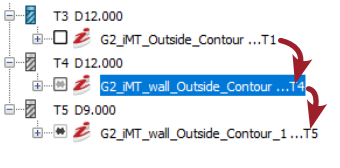
The Multi-tool technology shares and manages common data (i.e., Geometry and Levels selections) between the related operations. When common data is modified in one operation, it is automatically synchronized across all other operations.

When specifying Delta depths, like in the case of machining outside contours and you want the Multi-tool operation to cut deeper than the part bottom edge, it is recommended to make the Levels selections and enter the Delta depth value(s) prior to defining the tools. If not, the Delta depth entries will have to be repeated for each of the related operations.

All other technological data is linked between the related operations, providing three major advantages:

1. If any settings are modified in a preceding operation, subsequent operations are updated automatically.

   Upon recalculation, all operations are recalculated in sequence.

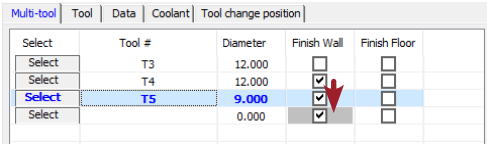


2. Offsets are inherited from preceding operations and the iRest Data of subsequent operations are managed automatically.

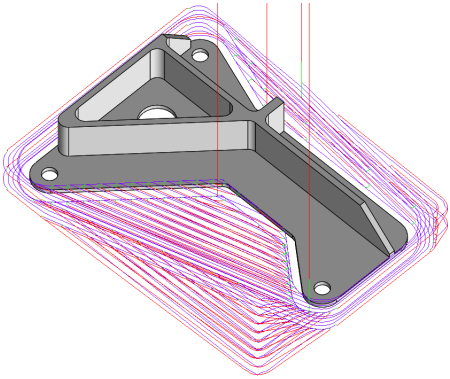
1st tool	Offsets Wall / island offset: ☐ 0.3 Floor offset: ☑ 0
2nd tool	Technology | Channels | iRest Data | Sort iRest parameters Parent operation: iMT_Outside_Contour Previous tool diameter: 12 Previous wall offset: 0.3 Previous floor offset: 0 Previous fillet radius: 1.2
3rd tool	Technology | Channels | iRest Data | Sort iRest parameters Parent operation: iMT_wall_Outside_Co Previous tool diameter: 12 Previous wall offset: 0 Previous floor offset: 0 Previous fillet radius: 0.06

3. All air cutting is eliminated by automatic detection and allocation of cutting technologies from operation to operation.

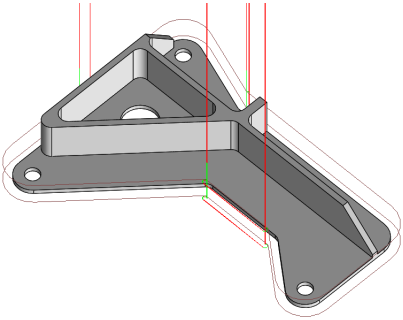
Example 1:



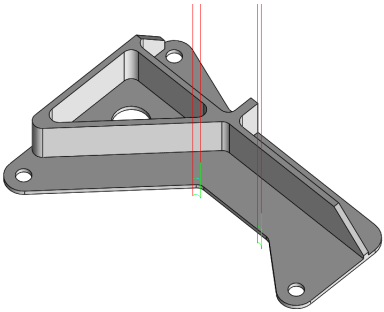
1st tool



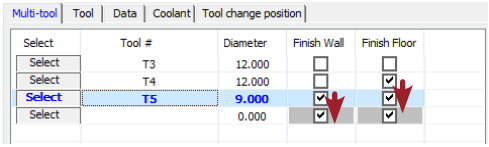
2nd tool



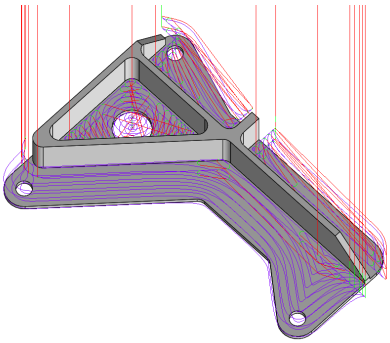
3rd tool



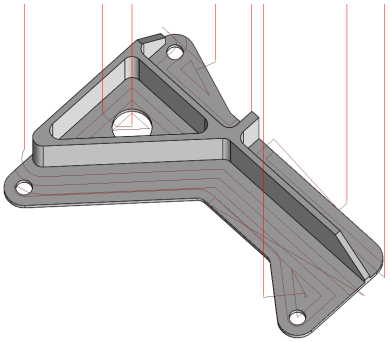
Example 2:



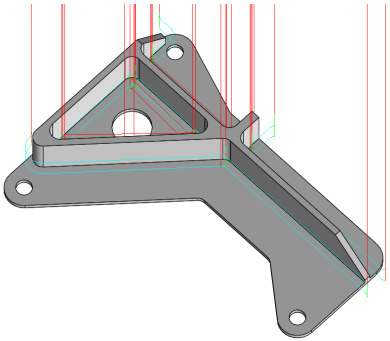
1st tool



2nd tool



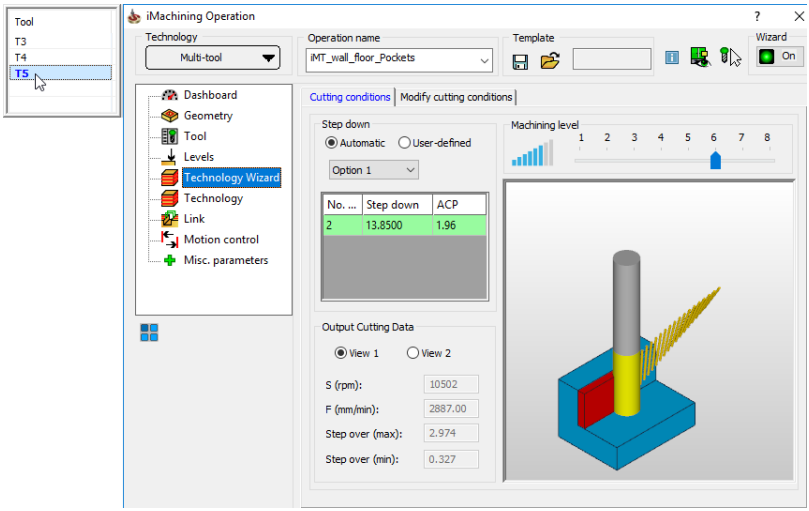
3rd tool



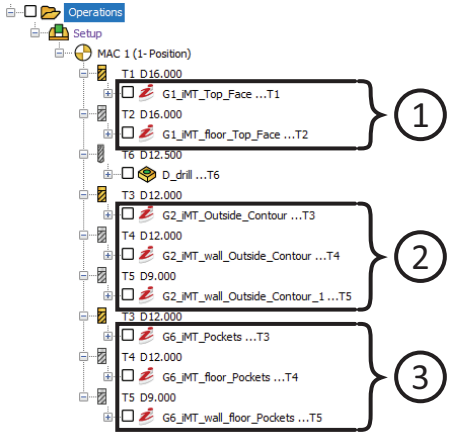
In both examples above, the tool path is automatically optimized and no time is wasted on areas already machined.

As with any other iMachining operation (iRough, iRough + iFinish, etc.) in InventorCAM, the requirements for defining a Multi-tool operation are minimal. Once the Geometry, Tool and Levels definitions are completed, you can simply use the default values calculated for all other parameters.

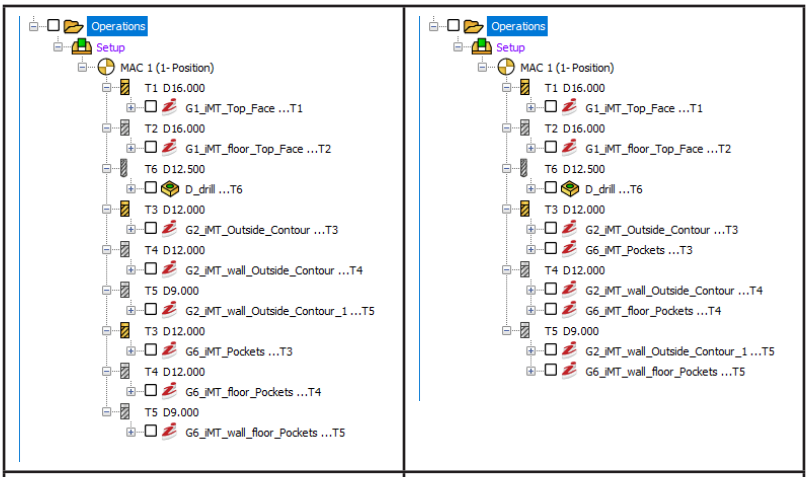
The Technology Wizard also provides for each tool, as each tool represents a separate but related operation, Cutting conditions that are optimal. By switching to the Technology Wizard page and toggling through the tools in the Tool grid, you can view the Cutting conditions specific to each tool.



The example below is a CAM tree view of the related operations resulting from just three Multi-tool operations defined, saved and calculated.



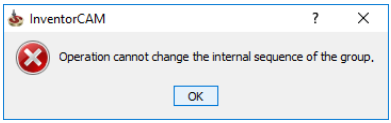
To further increase productivity, you can even change/rearrange operations in the CAM tree according to tool sequence. Doing so can help save time that would otherwise be spent on unnecessary tool changes.



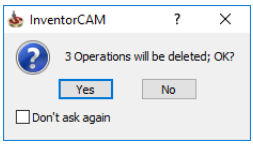
Before	After

Note: A Multi-tool operation contains internal sequences and data dependencies that must be preserved.

As a result, InventorCAM does not permit from the CAM tree the reordering of related operations in a way that will break the internal sequence.



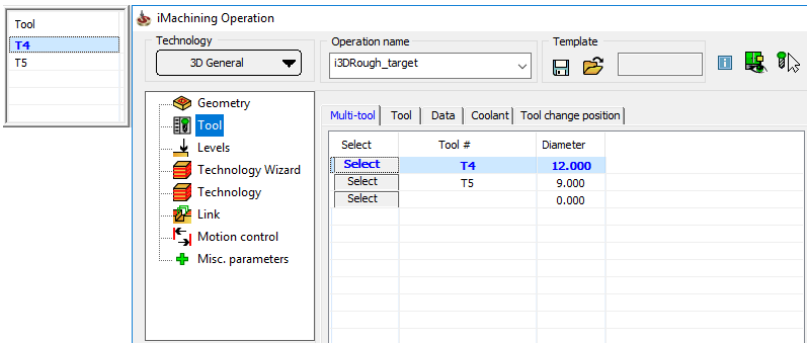
Nor can you delete any one related operation that breaks the dependency of data to another; upon deleting one, InventorCAM will prompt you to delete all.



Support of Multi-tool operations in iMachining 3D

iMachining 3D generates tool paths according to the proven algorithms of iMachining 2D and the Technology Wizard. For that reason, most of the parameters and options appearing in the iMachining Operation dialog box for iMachining 2D are shared with iMachining 3D.

For the same reason, the developers made the Multi-tool functionality available in iMachining 3D operations. Whether you are using 3D General or 3D Prismatic, both technologies enable you to define multiple tools.



The Multi-tool interface features are the same with only one difference. The Finish Wall and Finish Floor options are not available for selection because finishing is currently not supported in iMachining 3D operations.

While the Multi-tool functionality is always active and its interface features are always visible in the iMachining Operation dialog box, you are not required to use it. You can choose to:

1. Define a single machining task using just one tool.

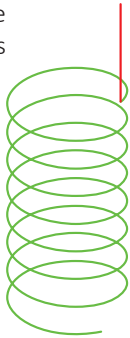
   Or from within a single interface:

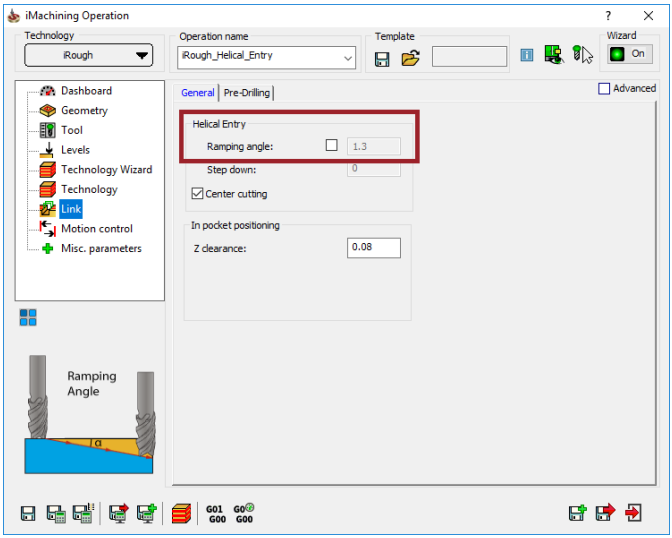
2. Using the 3D General technology (for general shaped 3D parts), define a sequence of related roughing, rest machining and semi-finishing tasks that use multiple tools.

3. Using the 3D Prismatic technology (for 3D prismatic parts), define a sequence of related roughing and rest machining tasks that use multiple tools.

7. User-defined Minimum Helical Ramping Radius

For closed geometries in iMachining operations, the tool enters the material in a spiral movement according to the Helical Entry parameters on the Link page.





Like all technological parameters of the operation, the iMachining technology automatically calculates the helical cutting conditions based on built-in formulas and algorithms. As described below, the helical cutting conditions consist of two main attributes.

Ramping angle

This parameter defines the aggressiveness of the descent angle by which the tool enters the material.

The default value is calculated according to the material hardness and the Machining level aggressiveness. The override check box enables you to modify the provided value, in the instance you want the tool to perform a helical entry at an aggressiveness that is unrelated to the position of the Machining level slider.

Ramping radius

This parameter, which is not visible within the operation, is used to calculate the size of the helical diameter of the tool path as well as the diameter of the entry hole after the helical ramping is performed.

Initially, 90% of the tool radius is used for the helical diameter calculation. If the tool radius is *Rad*, the helical diameter (*hd*) can be calculated according to the formula $hd = (0.9 \times Rad) + Rad$ (equivalent to 190% of the tool radius). The percentage is hard-coded to provide a descending helix of the maximal size, while also maintaining enough overlap through the center.

The diameter (*d*) of the entry hole can then be calculated according to the formula $d = 2 \times hd$.

Upon calculating the iMachining tool path, if the helical entry cannot fit in the designated area(s), 170% of the tool radius is instead used for the Ramping radius calculations.

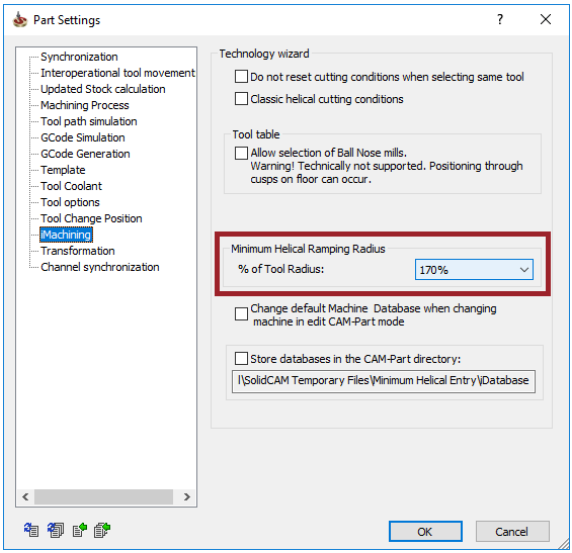
Prior to InventorCAM 2018, 170% was the minimum percentage allowed by the system with absolutely no means to modify it. If the helical entry still could not fit in the designated area(s), they were simply ignored. Such areas would require the use of smaller tools and additional operations.

NEW

The iMachining technology now provides optional minimum percentages that can be used for the Ramping radius calculations.

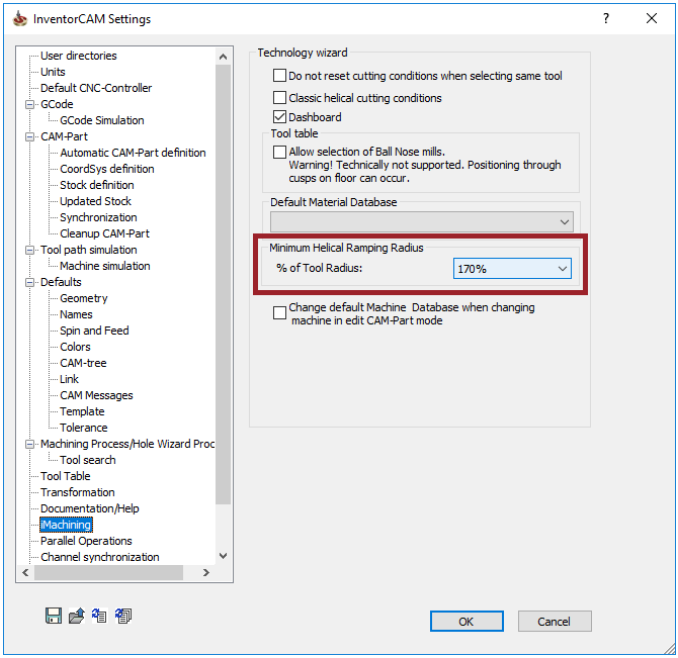
The Minimum Helical Ramping Radius is a setting that can be modified in the following two ways:

1. Per project in the Part Settings dialog box > **iMachining** page



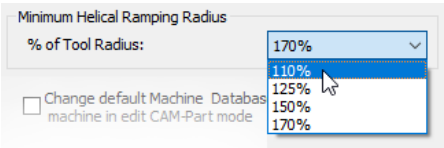
Modifying the percentage in the Part Settings affects newly added and recalculated iMachining operations of the current CAM-Part.

2. Globally in the InventorCAM Settings dialog box > **iMachining** page



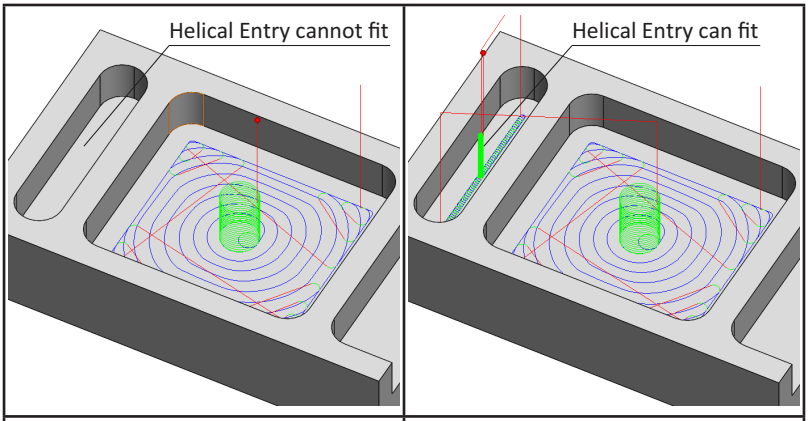
Modifying the percentage here affects all iMachining operations of newly created CAM-Parts.

As a safeguard for successful entry, 170% remains the minimum percentage that will be used by default. In the **% of Tool Radius** drop-down list however, there are three other percentages available for selection:



- **150%**
- **125%**
- **110%**

Choosing a smaller minimum percentage will allow larger tools to enter into smaller areas.

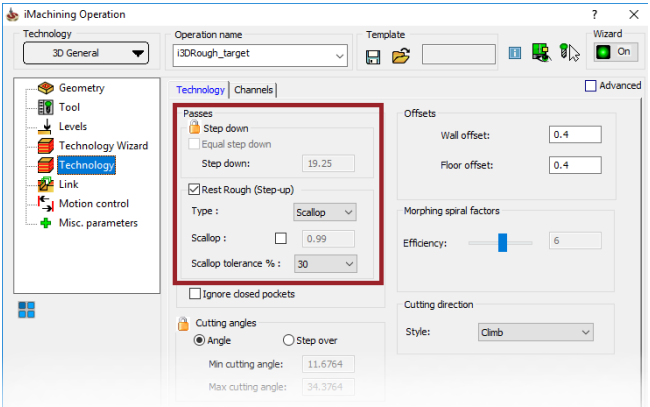


170% of Tool Radius	110% of Tool Radius

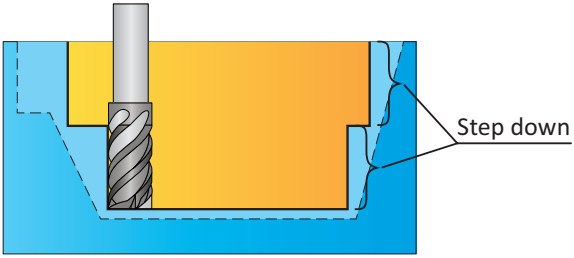
This option is generally suited for cutting soft materials and/or using high quality tools with through spindle coolant or air blast. In such cases, this option can also be used as an alternative to pre-drilling.

8. Adaptive Engagement Angle in iMachining 3D

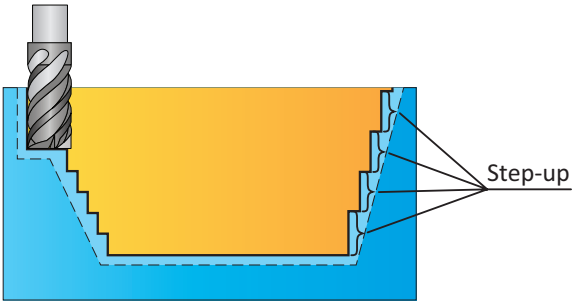
An iMachining 3D operation uses the Step down and Rest Rough (Step-up) parameters for calculation of the tool path, which consists of both roughing and rest roughing passes.



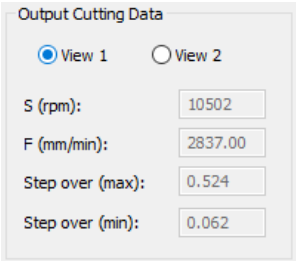
Using the proven algorithms of iMachining 2D, roughing tool paths are generated first via deep Step down cutting passes.



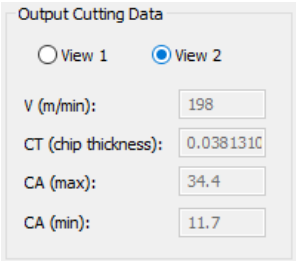
After achieving the final reachable depth, rest roughing tool paths are then generated in Step-up mode to remove the rest material on sloped surfaces of general shaped 3D parts or on higher horizontal surfaces of 3D prismatic parts.



During the Step-up process, the axial depth of cut gets smaller and usually varies every time a new higher step is machined. Although the depth of cut changes, previous versions of InventorCAM used the selected set of Cutting conditions for not only the Step down passes but also the Step-up passes.



Output Cutting Data View 1



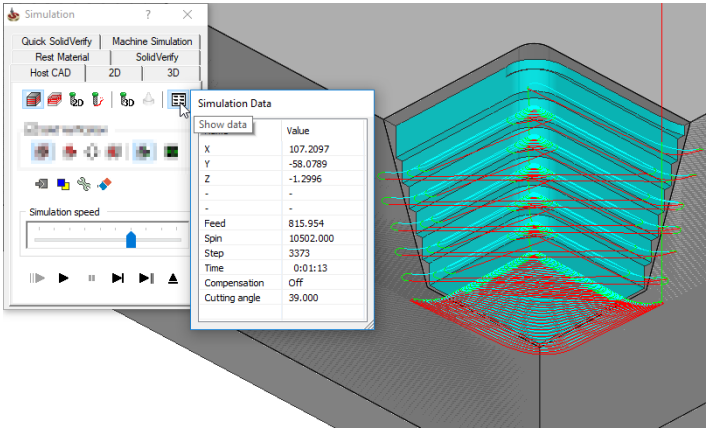
Output Cutting Data View 2

NEW

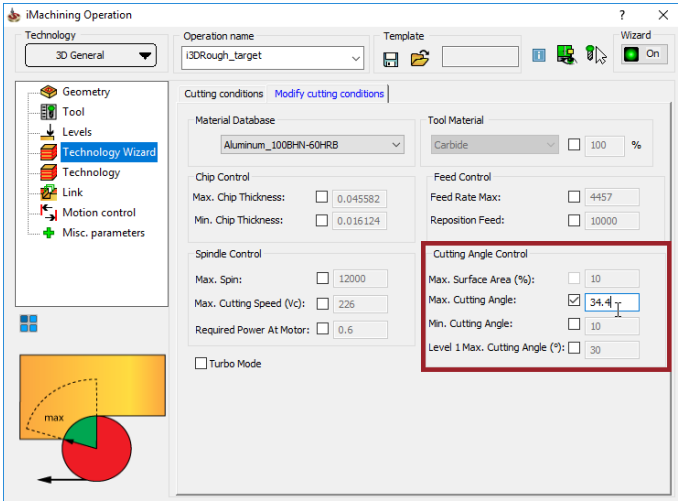
In InventorCAM 2018, iMachining 3D automatically optimizes the tool path by recalculating every depth of cut and providing for each one the appropriate engagement angle changes.

Although not shown in the Output Cutting Data, the Technology Wizard adjusts both the feed rate and cutting angle for each depth of cut. The smaller depths receive increased values by the exact amount required to maintain the specified constant load on the tool while, at the same time, resulting in shorter machining times for the higher steps.

The adjustments can be seen in the Simulation Data window in real-time when simulating the iMachining 3D tool path.



If you have tools unable to manage the cutting angle increases, it is possible to limit the cutting angle to a specified value for the operation and, in turn, for every depth of cut. To do so, use the Max. Cutting Angle override on the Modify cutting conditions tab of the Technology Wizard page.



Refer to the InventorCAM Milling Help documentation for more information about the Modify cutting conditions parameters.

9. iMachining 3D option to Ignore closed pockets

iMachining 3D analyzes the Target model and is able to recognize all its features and depths automatically. A single iMachining 3D operation will remove all the volumes of material that can be removed using the selected tool. The volumes can consist of all geometry types including semi-open, open and closed pockets.

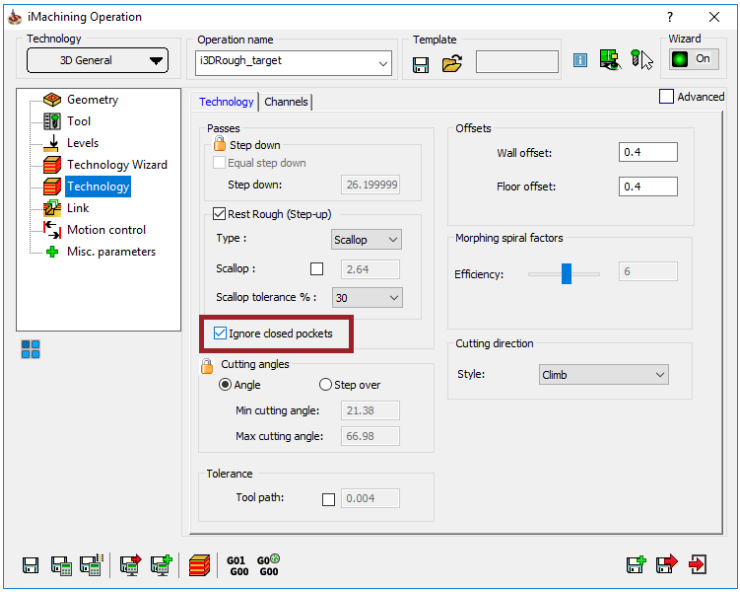
In recent years, cutting tool manufacturers have been making high performance end mills that are designed for use with iMachining tool paths. Such tools are composed of optimized cutter geometry that is best-suited for step over style strategies. It is recommended by most manufacturers, and may even include a disclaimer, that these specialized tools cannot be used to perform helical ramping into the material.

In previous versions of InventorCAM, it was not possible to use a high performance end mill in an iMachining 3D operation containing any closed pocket geometries without the risk of damaging the tool while entering the material.

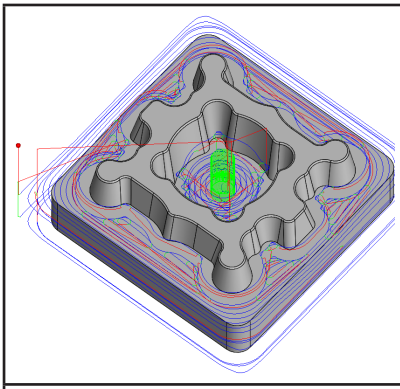
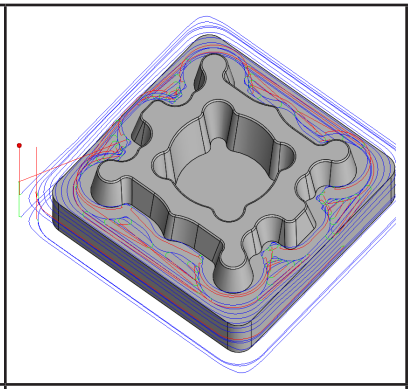
NEW

In InventorCAM 2018, the machining of closed pocket geometries can now be ignored with just one click.

The option to **Ignore closed pockets** appears on the Technology page of the iMachining operation dialog box for iMachining 3D only.



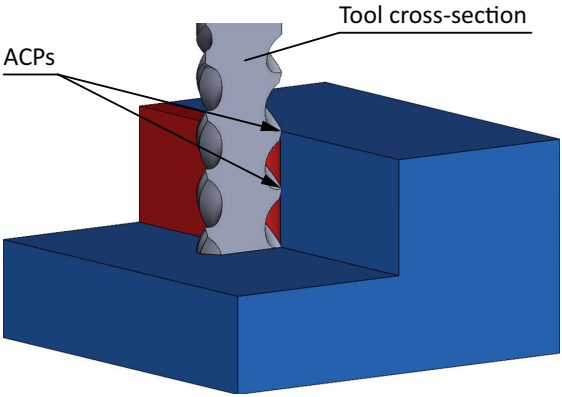
When enabled, iMachining 3D eliminates all tool path passes that would otherwise be contained in closed pocket areas (e.g., cavity features of a mold core), leaving only those volumes unmachined.

	
Ignore closed pockets not enabled (default)	**Ignore closed pockets enabled**

This option enables you to machine all semi-open and open pocket areas with one iMachining 3D operation using your high performance end mill. For the closed pocket areas that require a helical entry, you can define a separate operation using a standard end mill.

10. Option to merge ACPs in cases of 2 or more step downs

The iMachining Technology Wizard calculates and displays the operation ACPs, which reflect the number of Axial Contact Points 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), the less likely it is that vibrations will develop.

ACP is not a user-defined parameter. Instead, the Wizard provides ACPs with a corresponding color to indicate whether or not the current situation for stability is considered good. The color indication is as follows:

No. ...	Step down	ACP
2	15.0000	2.01

Green = Good
Preferred

No. ...	Step down	ACP
1	13.0000	1.74

Yellow = Not so good
Medium likelihood of vibrations

No. ...	Step down	ACP
1	5.0000	0.67

Red = Bad
High likelihood of vibrations

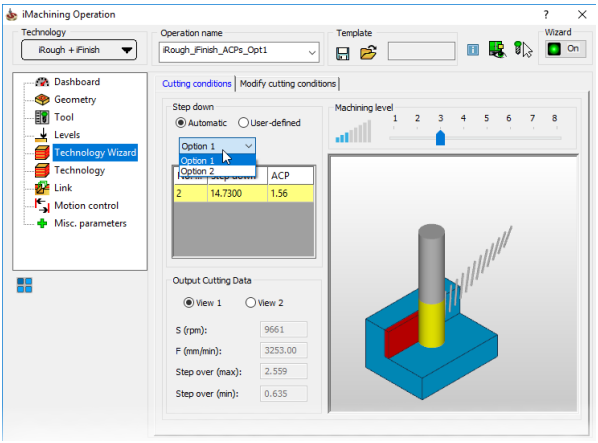
Although it is just not possible to always be machining with preferred ACPs, the color indication should be used as a guide to assist you in avoiding vibrations.

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

NEW

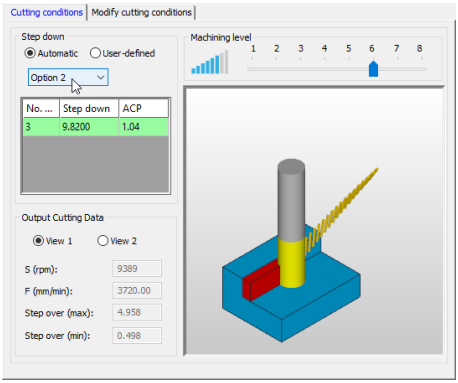
In InventorCAM 2018, the Wizard can now merge ACPs as a quick option for possibly producing more favorable ones and correspondingly better overall cutting results.

The option to merge ACPs appears on the Technology Wizard page when the Wizard is using **Automatic** (default mode) to calculate Step down.



In cases of 2 or more step downs:

- **Option 1** provides the typical Step down calculations as shown in the image above (same as previous versions of InventorCAM).
- **Option 2** provides alternative Step down calculations as shown in the image below (based on the merging of ACPs).

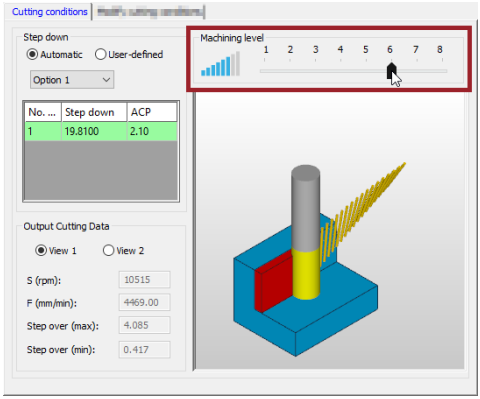


For more information about ACPs and avoiding vibrations, refer to the InventorCAM Milling Help documentation.

11. Modify cutting conditions feature is always active

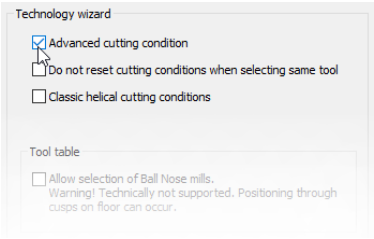
A significant part of the iMachining system is devoted to automatically calculating synchronized values of feed rate, spindle speed, axial depth of cut, cutting angles and (undeformed) chip thickness based on the mechanical properties of the workpiece and the cutting tool, while also keeping within the boundaries of your machine capabilities (maximum feeds and spindle speed, power and rigidity).

The Technology Wizard, which is responsible for these calculations, manages the complexities of all the machining factors and provides you with optimal sets of Cutting conditions combinations. Using the Machining level slider, which controls Material Removal Rate (MRR) and machining aggressiveness, you can view and select a set of Cutting conditions.

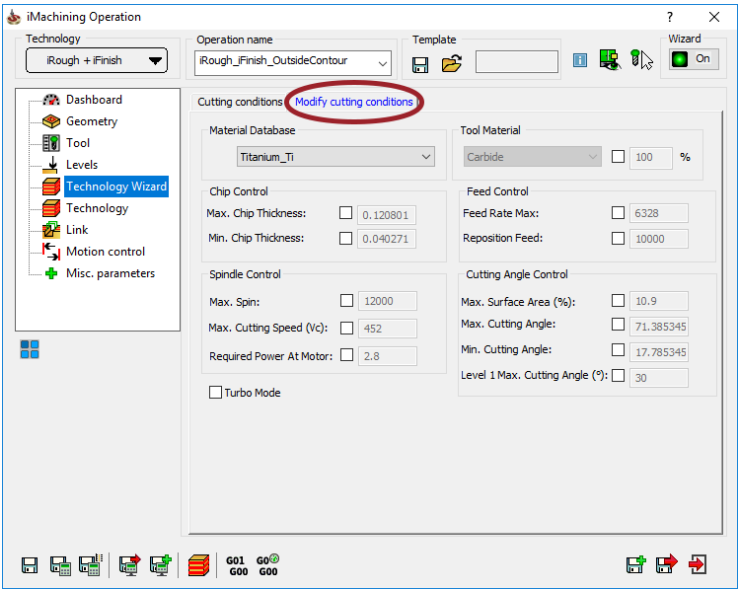


Although not recommended unless the Machining level slider does not produce your desired result, it is possible to override any one or more of the given values by using the Modify cutting conditions feature of iMachining.

In InventorCAM 2017 and earlier, the Modify cutting conditions tab was hidden from the Technology Wizard page by default and appeared only after enabling the **Advanced cutting condition** option in the SolidCAM Settings (for newly created CAM-Parts) or in the Part Settings (for the current CAM-Part/existing CAM-Parts).



In InventorCAM 2018, with overall proficiency in iMachining on the rise, users are no longer required to turn on the Advanced cutting condition setting, thus making the Modify cutting conditions tab always active.



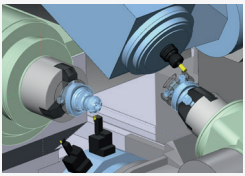
Refer to the InventorCAM Milling Help documentation for more information about using the Modify cutting conditions feature of iMachining.

InventorCAM 2018

What's **NEW**

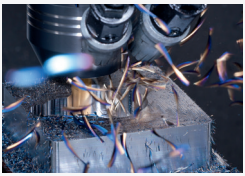
iMachining **2D** | iMachining **3D**

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.

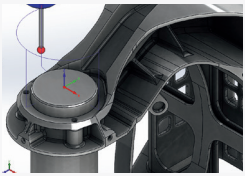
The Revolutionary iMachining® Module



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
youtube.com/SolidCAMiMachining

facebook.com/SolidCAM
facebook.com/iMachining

iMachining Successes
solidcam.com/successes/imachining-successes

www.solidcam.com

InventorCAM
iMachining – The Revolution in CAM!