InventorCAM 2015

The unique, revolutionary Milling technology

iMachining®

patent by SolidCAM

TIME SAVINGS

70%

... AND MORE!

Miscellaneous **FEATURES**

iMachining 2D | iMachining 3D

CERTIFIED AUTODESK INVENTOR 2015 AUTODESK

InventorCAM + Inventor

The Complete Integrated Manufacturing Solution

InventorCAM

iMachining – The Revolution in CAM!

Contents

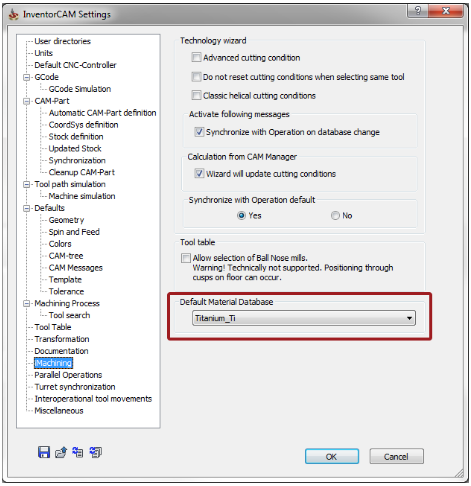
Document number: IViMMFENG1-2015

Notable Features in iMachining

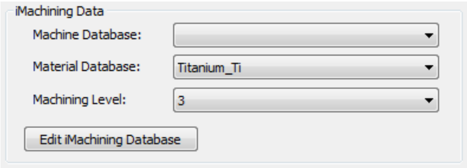
1. Default settings for the iMachining Data selections

Every time a new CAM-Part is created, you have to define the iMachining Data selections if you want to use the iMachining technology. If not in the CAM-Part Definition, the machine and work material selections must be made upon adding the first iMachining operation to your CAM-Part.

Just added, the **Default Material Database** setting enables you to choose a default work material for all newly created CAM-Parts.



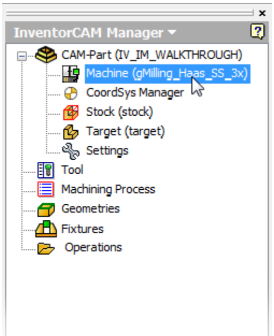
When defining the CAM-Part, the default selection will appear automatically in the iMachining Data area of the Milling Part Data dialog box.



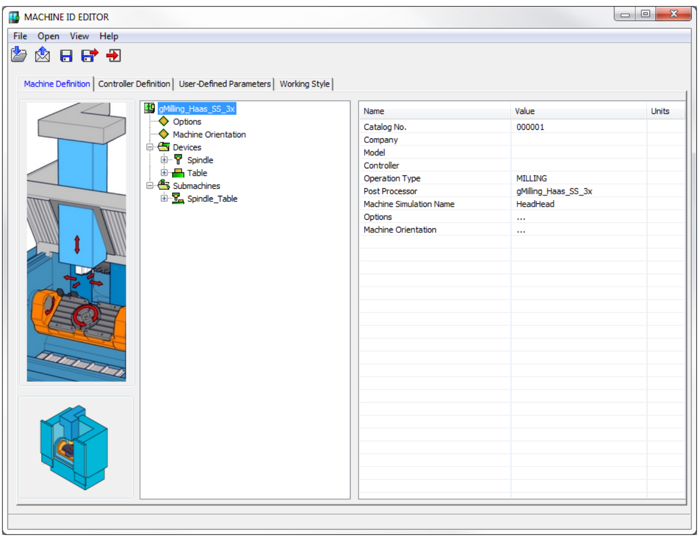
Also just added, the InventorCAM **Machine ID Editor** can be used to define a default machine and work material that is associated to your post-processor. You have to edit the accompanying *.vmid (Virtual Machine ID) file of the CNC-Machine Controller.

To define the default iMachining Data selections in the Virtual Machine ID file from the CAM Manager, the working order is as follows:

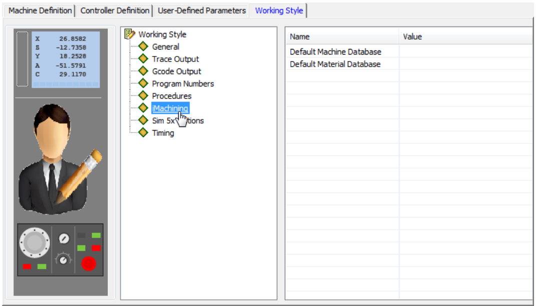
1. In the InventorCAM Manager, double-click the **Machine** subheader.



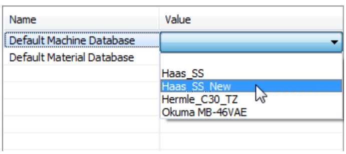
The MACHINE ID EDITOR dialog box is displayed.



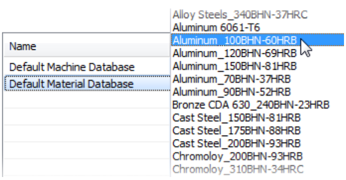
2. Switch to the **Working Style** page and click **iMachining**. The default settings for the iMachining Data selections appear in the Name column.



3. Click the empty **Value** field next to Default Machine Database. The relevant iMachining Database menu is activated, which enables you to define a default machine that is associated to your post-processor.

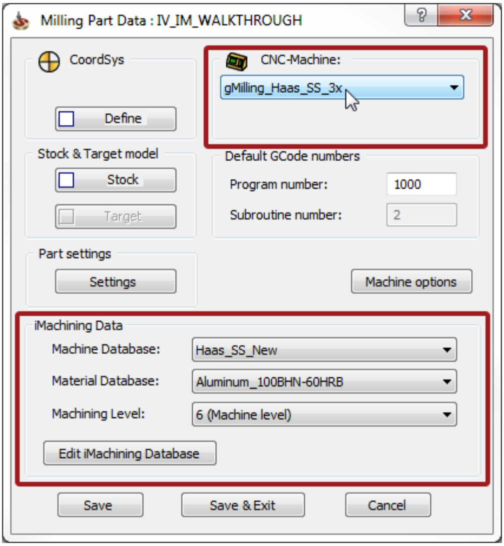


4. Click the empty **Value** field next to Default Material Database. The relevant iMachining Database menu is activated, which enables you to define a default work material that is associated to your post-processor.

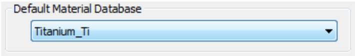


5. In the main menu, click the  button to save the changes to the *.vmid file and exit the Machine ID Editor.

After choosing the CNC-Machine Controller in the CAM-Part Definition of newly created CAM-Parts only, the default selections will appear automatically in the iMachining Data area of the Milling Part Data dialog box.



If there is no **Default Material Database** selection in the Virtual Machine ID file, that which is chosen in the InventorCAM Settings will be used.

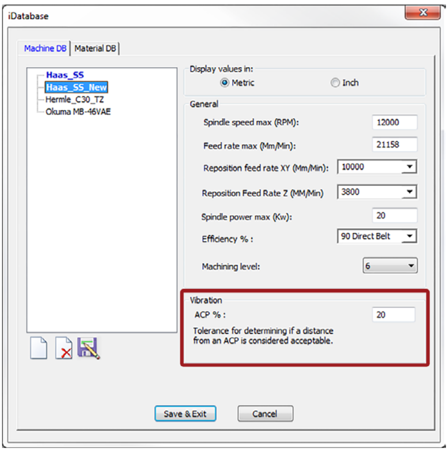


2. ACP tolerance in the Machine Database

The iMachining Technology Wizard calculates and displays the ACP value, which reflects the number of Axial Contact Points the defined tool has with the vertical wall it is cutting. The reaction of this cutting force is transmitted to the tool and from there to the machine.

According to iMachining theory, the closer the ACP value is to a whole number (≥ 1), the less likely it is that vibrations will develop. Favorable ACP values are therefore taken into account when the depths are generated. However, it's just not possible to always be machining with perfect ACPs.

Just added, the Machine Database now includes an **ACP %** parameter. This tolerance enables you to control the ACP indication and how the iMachining Technology Wizard outputs the depths.



If the tolerance was set to 0, the Wizard would output an increased No. steps with a shallower Step down. With a higher tolerance, on the other hand, the Wizard will output a reduced No. steps with a deeper Step down.

For new machines, the ACP tolerance is set to 20% by default. With this setting, the ACP indication will show that the situation for stability is good if you get a value of 1.1, 1.2, 1.8 or 1.9 for example.

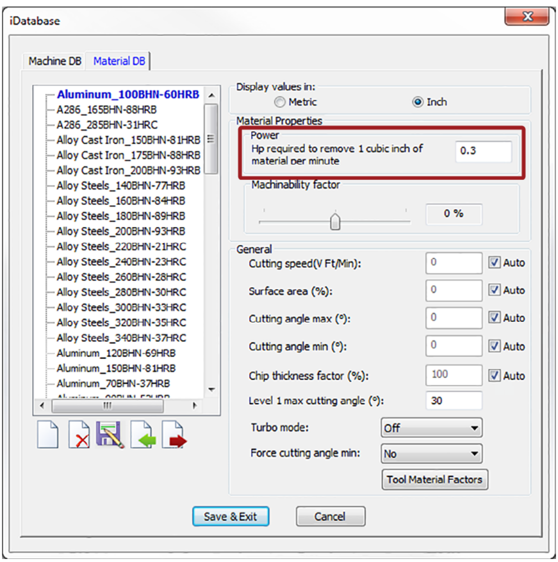
3. Material Properties – UTS versus Power Factor

Different materials require different amounts of force to cut them. The physical property of a material that determines the force required for a particular cut is the Ultimate Tensile Strength (**UTS**), given in units of MPa (Mega Pascal) in Metric units or psi (pound per square inch) in English units.

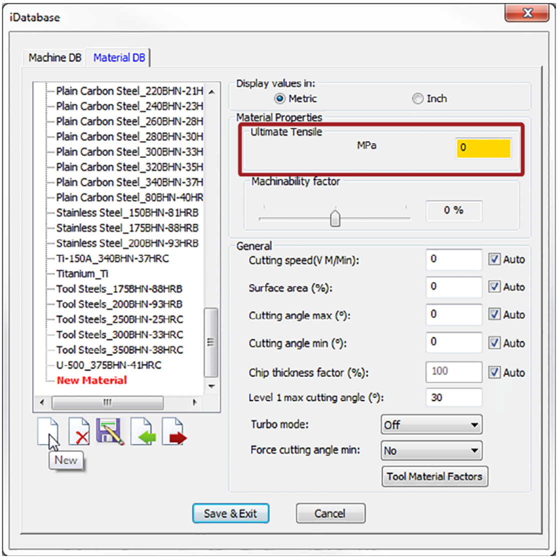
The iMachining Technology Wizard totally depends on the correct **UTS** value to produce good Cutting conditions, which is why it is critical to ensure that any material you decide to cut has the accurate **UTS** value assigned to it in the Material Database.

All InventorCAM versions are shipped with a basic Material Database containing more than 70 different materials.

> When the Technology Wizard was first developed, it was designed to use a different material property to calculate the cutting force. This property is called the **Power Factor** of the material, which specifies the power required to cut 1 cubic centimeter of material per minute (in Metric units of Kw), or 1 cubic inch of material per minute (in English units of Hp – Horse Power). This is an engineering property of the material, which is based on its physical properties, but is not so readily available in standard material property data resources such as www.matweb.com.



For this reason, the developers decided to build a parallel algorithm in the Technology Wizard after the initial release, which calculates the Cutting conditions using the **UTS** property. Since customers already had material tables based on **Power Factor**, the developers decided to leave the original algorithm in the system and allow the Wizard to use either property, depending on the property stored in each material record. The developers also decided to dynamically change the dialog box for defining a new material, so that it would only accept **UTS** for newly entered materials.



The current situation is that materials defined before 2012 are all defined in terms of their **Power Factor** rating; all materials defined since then have been and will be defined in terms of their **UTS**.

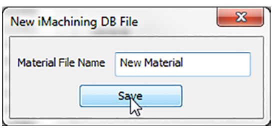
It should be clear that both methods of definition are equivalent and the Wizard produces the same efficient Cutting conditions with either method.

It is apparent that the 70+ materials supplied with the system cannot cover the needs of every customer for all their parts. Remember that there are over 5,000 different materials used in the industry. This means that users often need to add new materials to the Material Database.

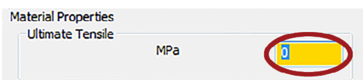
With the new iDatabase editing dialog box and the use of material **UTS**, it can be done quickly and easily. There are only two required inputs. The first input is the material `name`, which only serves to help you visually identify the specific material in the list and therefore must be unique, but need not be identical to its standard name. The second input is the material **UTS** rating, which can be easily found on www.matweb.com.

To define a new material for iMachining, follow these simple steps:

1. On the Autodesk Inventor Ribbon, click **InventorCAM 2015** -> **Options** -> **iMachining Database**.
2. When the iDatabase dialog box appears, switch to the **Material DB** tab.
3. Click the button at the bottom of the list.
4. Enter the `name` in the Material File Name field of the New iMachining DB File dialog box.



5. Find and input the **UTS** value for your newly added material.



4. Machinability factor in the Material Database

It is known that the same materials are often made by many different manufacturers. Therefore, it should be expected that tolerances exist between your material and its given UTS value. In turn, such tolerances make the material more or less machinable based on the subsequent changes in its physical properties.

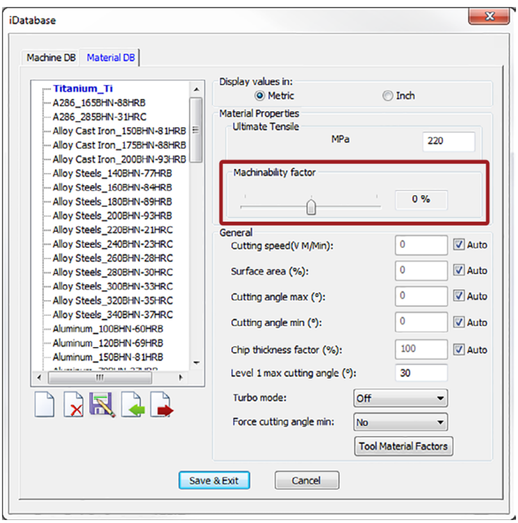
After machining your newly supplied material for the first time, you may discover that it can be cut faster than the Machining level slider or Turbo Mode permits. In most cases, this means that your material is less hard than specified by your property data resource.

When defining new material entries in the Material Database, *it has been and is still recommended to identify the exact material specification* (e.g., Titanium Ti – 6Al – 4V) with the help of your material supplier.

If there are many entries to choose from when searching www.matweb.com, *it has been and is still recommended to always start with the highest UTS value* – this is absolutely *safe*. The higher the UTS value, the harder the material. It may result in gentler cutting than is possible, which you can subsequently correct using the Machining level slider or make an effort to find the exact specs of the material and its UTS, but at least you can start cutting.

In some instances, even when the exact material specification was identified and the correct UTS value defined, customers discovered that they could cut their material faster than the Machining level slider or Turbo Mode permitted. At that point in time, it was recommended to change the UTS value in the Material Database accordingly.

Now, the recently introduced **Machinability factor** enables you to alter the hardness of a material without changing its given UTS value. This option is available on the Material DB tab of the iDatabase dialog box.

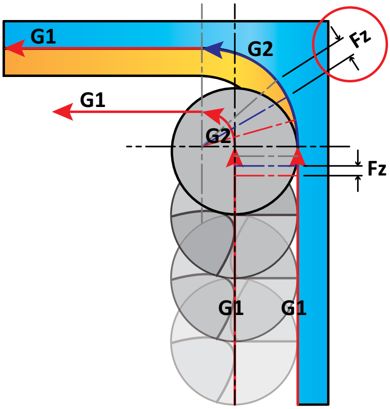


Moving the slider in the positive direction informs iMachining that your material is less hard than indicated by its UTS value and is more machinable by the specified percentage. Accordingly, the Technology Wizard will output more aggressive Cutting conditions by default.

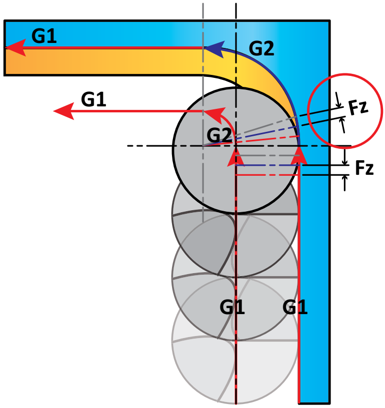
5. Feed correction for arcs

When cutting in a straight line (G1), the feed at the center of the tool is identical to the feed at the wall of the workpiece (periphery feed). However, when cutting in a corner (G2), the periphery feed is much higher. As a result, tool wear increases due to undesired chip thickness (CT).

When iMachining was developed, it was designed to produce not only *fast* but also *safe* CNC programs. It was discovered early on that in order to maintain a constant CT, it is important for iMachining to dynamically reduce the feed at the center of the tool when periphery feed increases on arcs.



Without iMachining
Feed Correction



With iMachining
Feed Correction

Currently, iMachining is able to automatically adjust the feed at the center of the tool in order to maintain a constant CT on arcs. As a result, tool load remains constant; tool life is not only increased but dangerous Cutting conditions are also decreased.

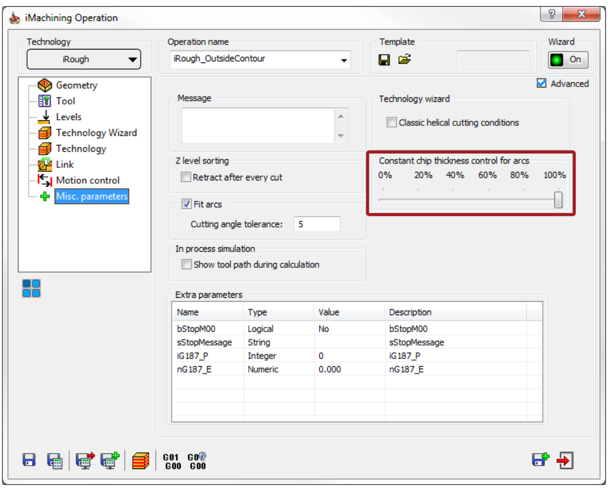
When milling aggressively, like with iMachining, feed correction for arcs is believed to be critical.

6. Constant chip thickness control for arcs

In early 2013, the iMachining technology established a feed correction for arcs, which was implemented to maintain a constant chip thickness (CT) when cutting in corners. A constant CT in corners is achieved by automatically reducing the feed rate. Some customers found that this feature increased their cycle times, and they determined that faster cycle times is more desirable than maintaining a constant CT.

When milling aggressively, like with iMachining, it is believed that feed correction for arcs is essential. By maintaining a constant CT, it is proven that tool load is kept constant and tool life is increased. In addition, the likelihood that dangerous Cutting conditions will develop is decreased.

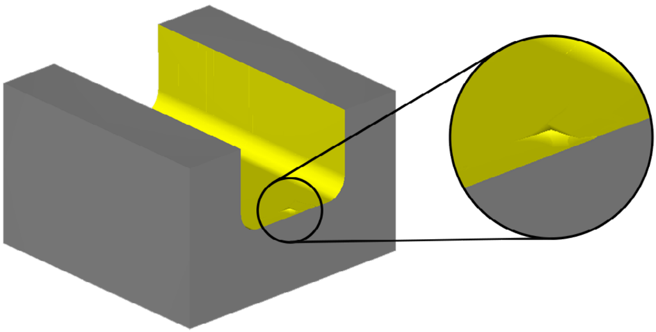
Now, the recently introduced **Constant chip thickness control for arcs** option enables you to control the feed correction for arcs. It can be found on the Misc. parameters page of the iMachining Operation dialog box, and the position of the slider is set to **100%** by default. If kept at **100%**, iMachining is informed to maintain a constant CT when cutting in corners.

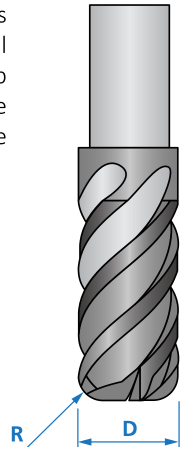


Moving the slider to **0%** informs iMachining to maintain a consistent feed rate between cutting in a straight line (G1) and cutting in a corner (G2). The result is faster cycle times; *but beware, with increased CT in corners comes increased tool load.*

7. Bull nose support

Prior to 2013, when using a bull nose mill in iMachining, tool path was generated based on the outside dimension, not considering the radial void between the bottom flat and full diameter. As a result, a cusp (material) would remain on the floor, as shown in the example below. The potential for a crash increased as the tool would high feed through the cusp, also causing damage to the tool.



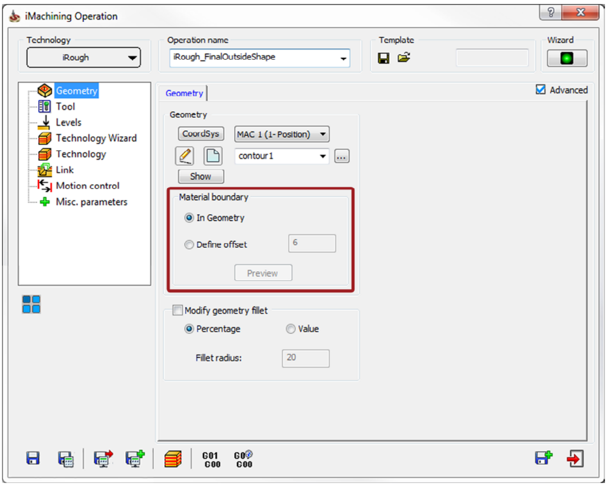


Currently, iMachining generates tool path taking into account the radial void between the bottom flat and full diameter of a bull nose mill. Thus, there is never a remaining cusp nor a potential for crashes.

With the support of bull nose mills, tool life and successful workpiece turnovers increase.

8. Material boundary option for Geometry definitions

Introduced in early 2013, the **Material boundary** feature enables you to automatically generate geometry chains by defining an offset. The offset is generated to one side of an existing closed or open geometry chain. By eliminating the need to sketch a working area in Autodesk Inventor, this method of Geometry definition can help reduce programming time.



The **Material boundary** feature is only suitable for certain circumstances, where the starting stock is marginally larger than the target and the initial shape is the same as the final shape.

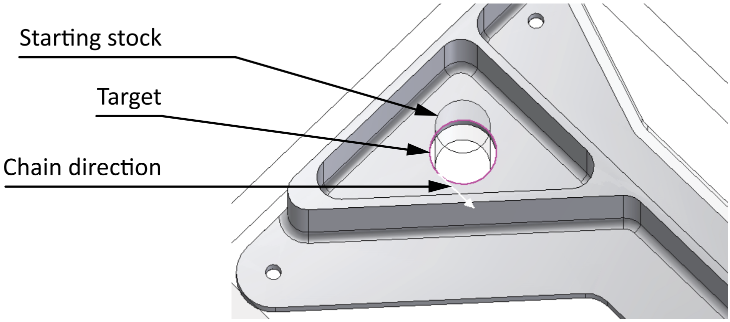
When selecting chains, it is important to note that the offset is generated to the left side of the chain direction; therefore, when using this method of Geometry definition, chains must always be picked to indicate climb cutting.

Provided that the size and shape meet the above requirements, a Material boundary can be used to define the following three geometry types:

Closed pocket with entry geometry

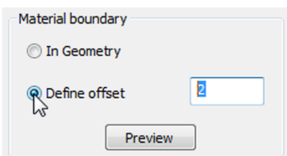
Any shape can be used for entry geometry, but it must be the same as the final shape.

In this example, a hole that is considered a precut area is used for entry.



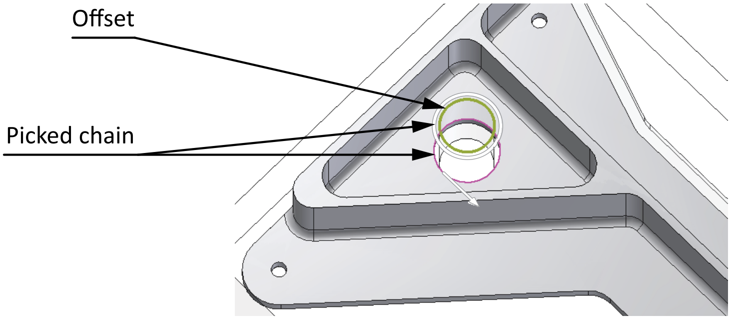
To define this type of geometry using the **Material boundary** feature, the working order is as follows:

1. To start the Geometry definition, click the button on the Geometry page of the iMachining Operation dialog box.
2. Pick the chain entities that define the pocket contour. When closed, the selected chain is displayed in the Chain List of the Geometry Edit dialog box. To ensure the entry chain is generated on the inside of the selected closed chain, the chain direction must indicate climb cutting. Click **Finish** to confirm the chain selection and exit the Geometry Edit dialog box.
3. In the Material boundary section, choose the **Define offset** option and enter the appropriate offset `value`. To accurately define the offset, you have to consider the size of the tool in addition to the starting stock.



As defined by the offset `value`, the iMachining technology automatically generates the internal entry chain to complete the Geometry definition.

Clicking the **Preview** button enables you to verify that the auto-generated geometry chain is correct.



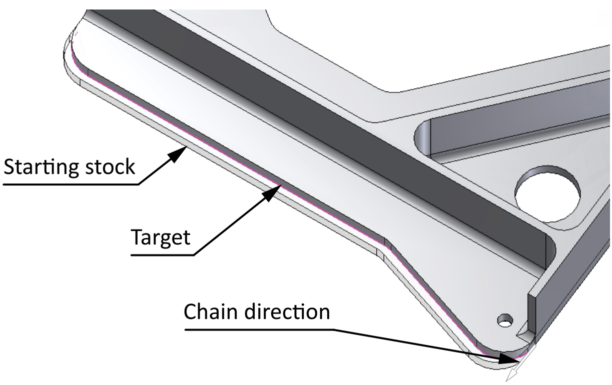
The geometry chain projected in white represents the picked chain, while the one projected in green represents the auto-generated chain.

Open pocket with island

A boss feature already machined and then later reduced in size is a typical example of this type of geometry. The working order is similar to the above with only one difference. The chain on island contour is selected; and as defined by the offset `value`, the iMachining technology automatically generates the external chain to complete the Geometry definition.

Semi-open pocket

In this example, the outer shape was modified from the original specifications with only a marginal reduction of one side.



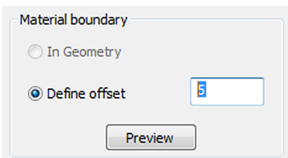
To define this type of geometry using the **Material boundary** feature, the working order is as follows:

1. To start the Geometry definition, click the button on the Geometry page of the iMachining Operation dialog box.
2. Pick the chain entities that define the semi-open pocket contour. In the Edit chain section of the Geometry Edit dialog box, click the **Accept Chain** button to define the chain as open. The selected chain is displayed in the Chain List.

   Ensure that the chain direction indicates climb cutting. Click **Finish** to confirm the chain selection and exit the Geometry Edit dialog box.

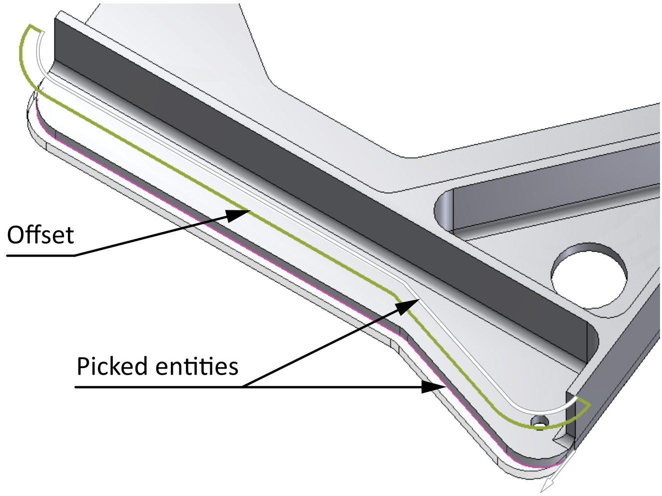
   Since the geometry chain is defined as open, the **In Geometry** option is deactivated and the **Define offset** option is made the default selection.

3. In the Material boundary section, enter the appropriate offset `value` in the **Define offset** text field. To accurately define the offset, you have to consider the size of the tool in addition to the starting stock.

As defined by the offset `value`, the iMachining technology automatically generates the remaining chain entities to complete the Geometry definition.

Clicking the **Preview** button enables you to verify that the auto-generated chain entities are correct.

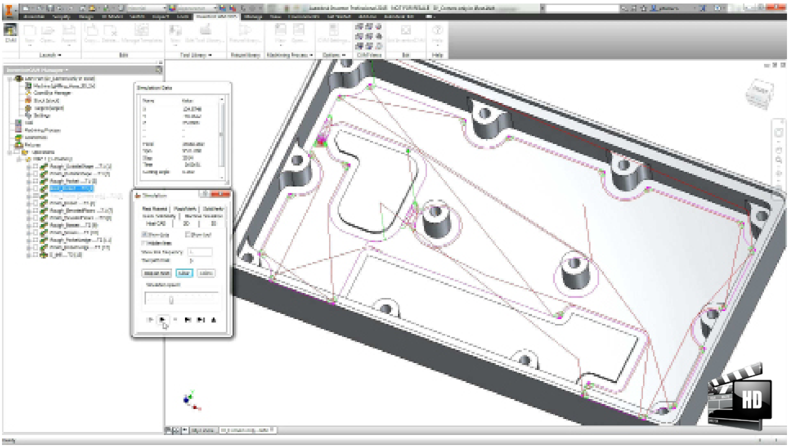


The chain entities projected in white represent the picked entities, while those projected in green represent the auto-generated entities.

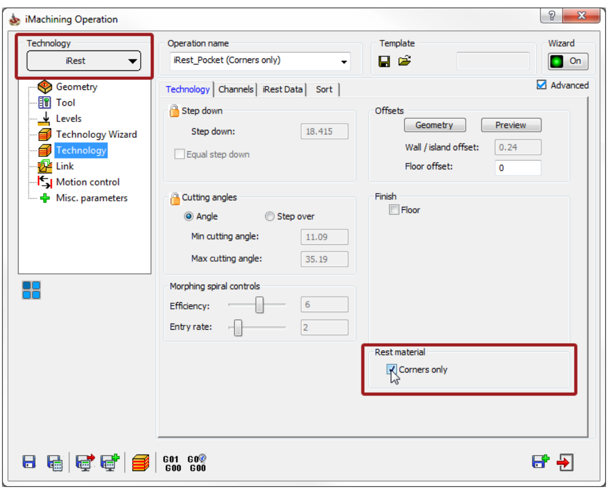
Regardless of the geometry type, the auto-generated chains and chain entities are always defined as open, meaning that the tool will approach the material and start machining from those chains.

9. Corners only in iRest

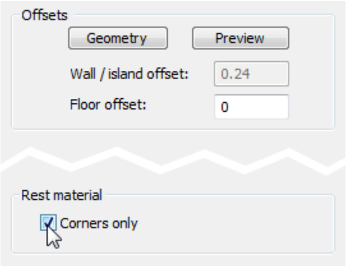
Prior to 2013, the iRest tool path was only able to perform rest machining in the corners and along the walls of an entire pocket, as shown in the Host CAD simulation video below. **Click to activate**, and then right-click to enable **Full Screen Multimedia**. Pressing the **Esc** key will **End Full Screen Multimedia**.



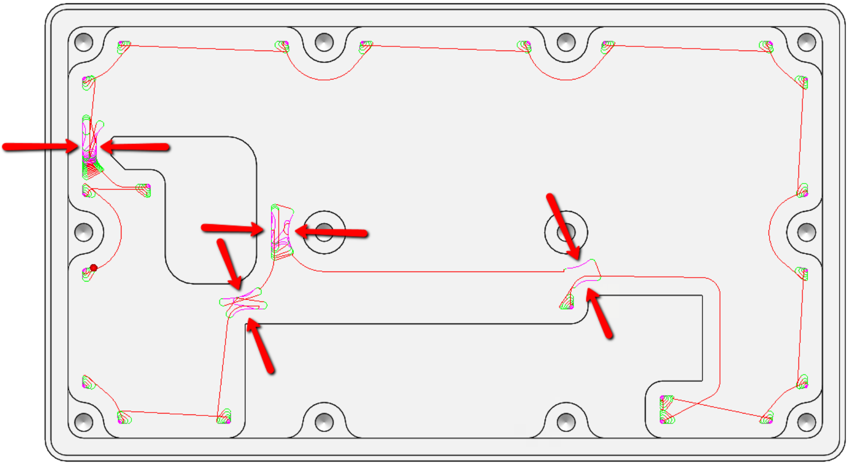
In early 2013, the option to rest machine **Corners only** in iRest was introduced. This option can be enabled in the Rest material section on the Technology page.



Corners only limits the machining of rest material to only the corners (and not the walls) of the pocket. When enabled, the Wall/island offset parameter is inherited from the previous roughing operation and the text field is locked from editing.



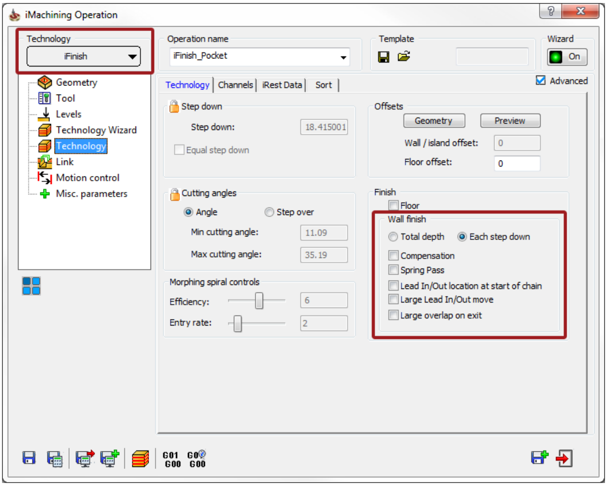
The example below illustrates an iRest tool path with the **Corners only** option enabled. The corners are cleared as well as the remaining stock in the tight areas unreachable by the previous roughing tool.



This option not only reduces cycle time, but it can also further improve efficiency when a tight Wall/island offset is specified in the previous roughing operation.

10. Tool path optimization strategies for iFinish

After iMachining was first released, customers began requesting that some tool path options be added for iFinish. So in early 2013, new optimization strategies started to become available as shown below on the Technology page of the iMachining Operation dialog box.



When defining an iFinish operation, any one or more of the following tool path options can be selected in the Wall finish section:

Compensation

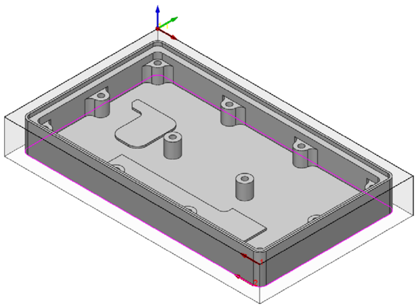
This option allows for small adjustments of the finish pass on the CNC-Machine. The adjustment is only relative to the walls (and not the corners) of the pocket.

Spring Pass

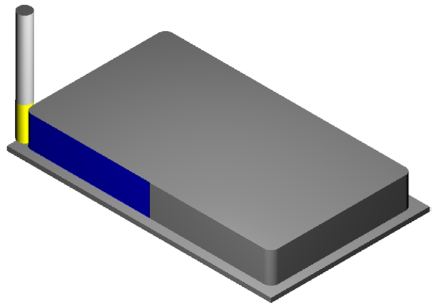
This option provides a secondary pass in addition to the primary finish pass. It aids in offsetting tool deflection to ensure that parts are finished more accurately and dimensionally correct. The tool does not disengage or retract away from the material between the two passes, so no time is wasted on positioning moves.

Lead In/Out location at start of chain

Prior to 2013, the tool would automatically lead in and lead out of the cut in the middle of the longest chain length chosen in the **Geometry Selection**. In the example below, the geometry is selected on the lower contour of the Target model as shown. In the **SolidVerify Simulation**, you can see where the tool leads in.



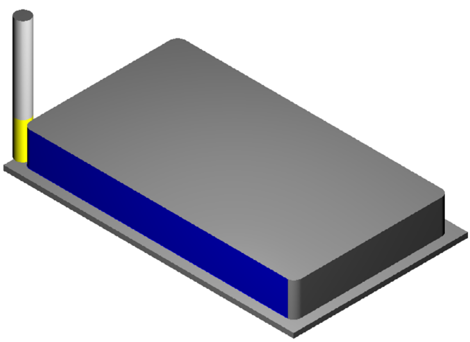
Geometry Selection



SolidVerify Simulation

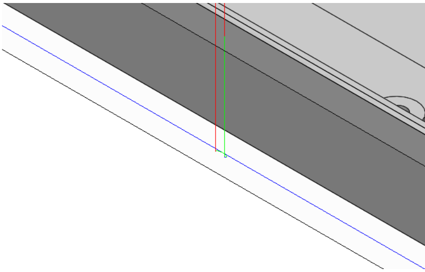
Currently, the option of **Lead In/Out location at start of chain** enables you to define the start location of the lead in and lead out for the finish pass according to the first picked chain entity (regardless of its length).

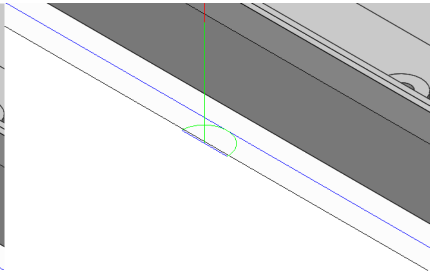
With the same **Geometry Selection** as above, the **SolidVerify Simulation** below shows how the tool leads into the cut at the start of the first picked chain entity when using the option of **Lead In/Out location at start of chain**.



Large Lead In/Out move

This option enables you to define a large lead in and lead out for the finish pass. Customers prefer using a **Large Lead In/Out move** when their older machines require the compensation line to be greater than the tool diameter.



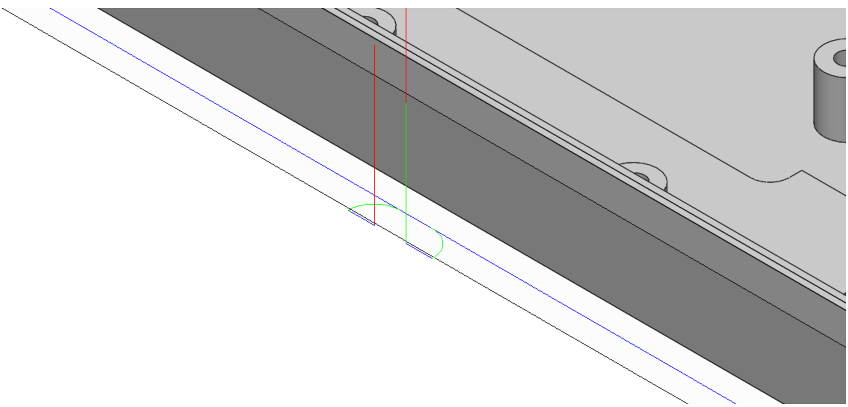


Typical arcs　　　　Large arcs

In the comparison above, note that the lead in/out arc is much larger than a typical lead in/out arc when using the option of **Large Lead In/Out move**. The tool used for this example has a diameter of 9.5 mm (0.375 in).

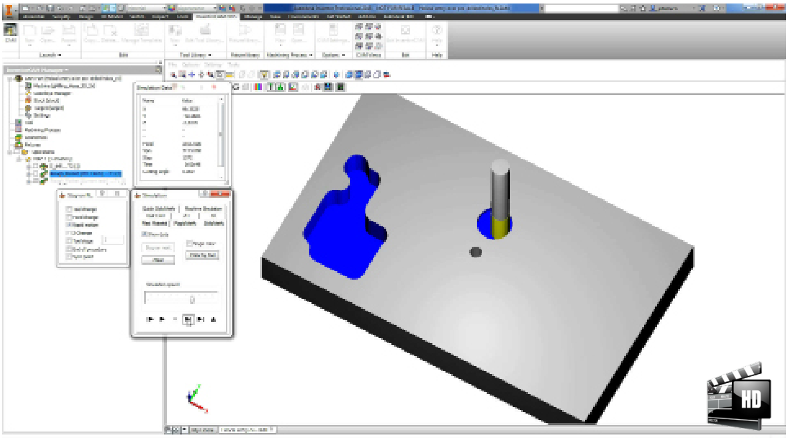
Large overlap on exit

Recently added to the tool path optimization strategies for iFinish is the option to perform a **Large overlap on exit**. As shown in the example below, this option enables you to extend the movement of the tool (collinear to the entry point) prior to making the arc lead out.

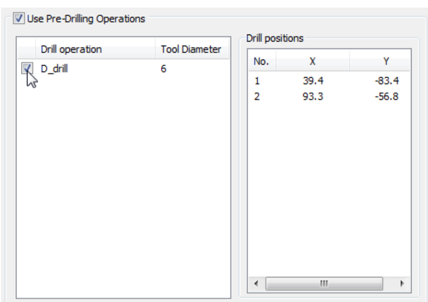


11. Automatic helical entry over small pre-drilled holes

Prior to 2013, an iRough operation ignored pre-drilled holes (if smaller than the roughing tool) and performed a helical entry into the pocket wherever iMachining calculated best. In the SolidVerify simulation video below, notice how the tool ignores the pre-drilled holes when entering the material. **Click to activate**, and then right-click to enable **Full Screen Multimedia**. Pressing the **Esc** key will **End Full Screen Multimedia**.

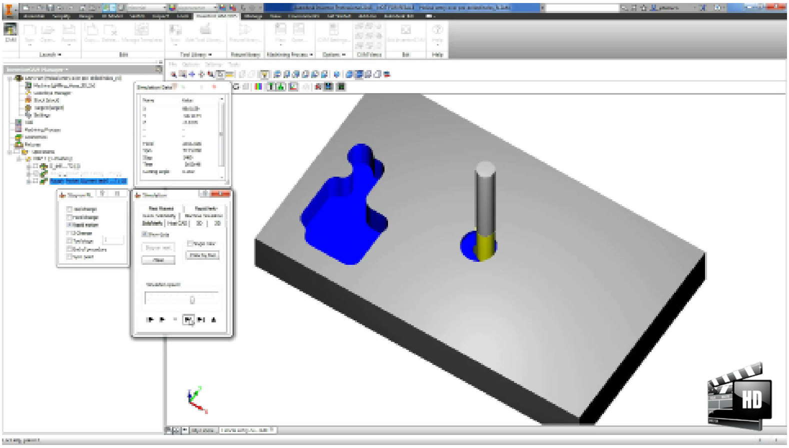


Currently, an iRough operation will perform automatic helical entry over small pre-drilled holes when the option to **Use Pre-Drilling Operations** is selected. The iMachining technology extracts the data from previously applied Drilling operations and stores it under the Pre-Drilling tab on the Link page.



When the desired Drill operation is chosen, the X- and Y-coordinates appear in the Drill positions list; the data is used for entry and if the Tool Diameter is smaller than the current roughing tool, iMachining performs a helical entry directly over the small pre-drilled hole(s).

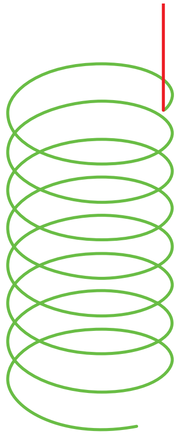
In the SolidVerify simulation video below, you'll see that the tool now performs an automatic helical entry over each of the small pre-drilled holes. **Click to activate**, and then right-click to enable **Full Screen Multimedia**. Pressing the **Esc** key will **End Full Screen Multimedia**.

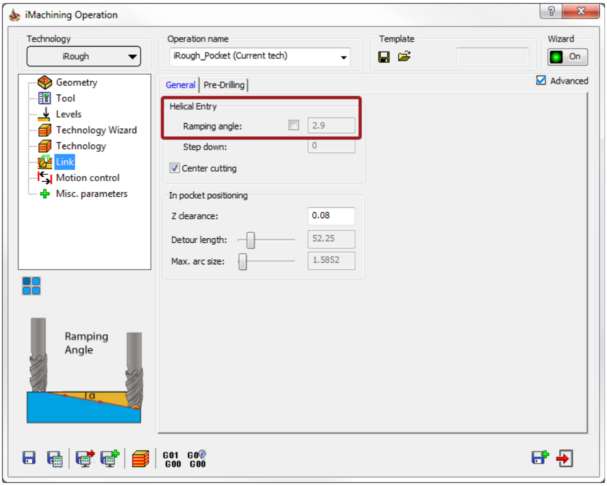


Note that using a small pre-drilled hole as a helical entry point for an iRough operation can dramatically reduce stress on the tool; and as a result, tool life is extended even further.

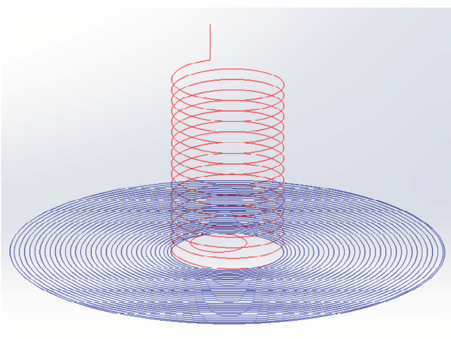
12. Helical entry cutting conditions

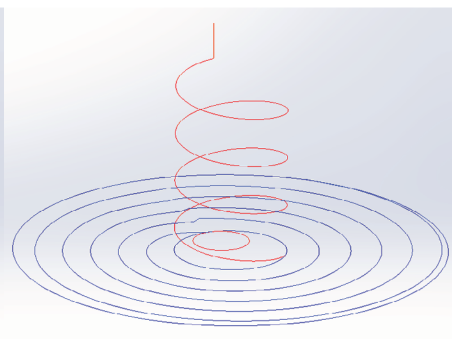
When the Geometry is defined as a closed pocket in iMachining, the tool enters the material in a spiral movement according to the parameters defined in the Helical Entry section on the Link page.





The **Ramping angle** parameter defines the aggressiveness of the descent angle by which the tool enters the material. Introduced in early 2013, the iMachining Technology Wizard is now designed to automatically calculate the helical cutting conditions based on material hardness and aggressiveness of the Machining level slider.



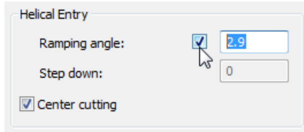


Ramping angle at Machining level 1 | Ramping angle at Machining level 8

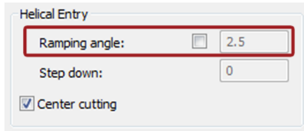
The example above illustrates the effect of the Machining level slider on the Helical Entry.

Warning: When using more aggressive values, cooling can become a concern. Larger values will generate more heat and proper cooling should be applied when necessary.

An override check box is provided so the **Ramping angle** value can be set manually, in the instance you want the tool to perform a Helical Entry at an aggressiveness that is not relative to the position of the Machining level slider.



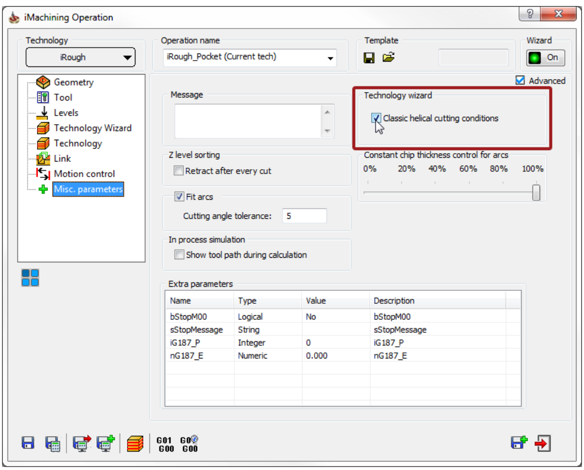
Prior to 2013, the iMachining technology output the value of `2.5` degrees for all operations by default, which is believed to be the absolute safest maximum descent angle by which the tool enters the material.



If the value of `2.5` degrees is preferred for all your operations, iMachining enables you to select an option called **Classic helical cutting conditions**.

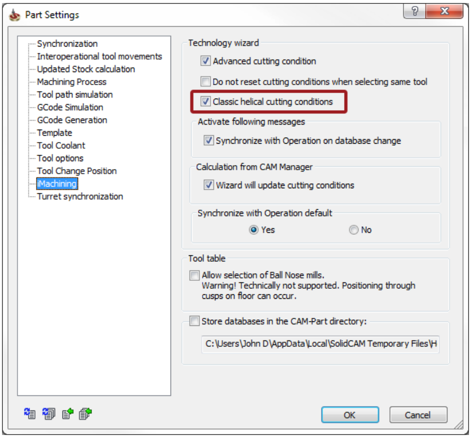
There are three ways to enable the **Classic helical cutting conditions** option:

1. **On a per operation basis** – in the Technology wizard section on the Misc. parameters page of the iMachining Operation dialog box.

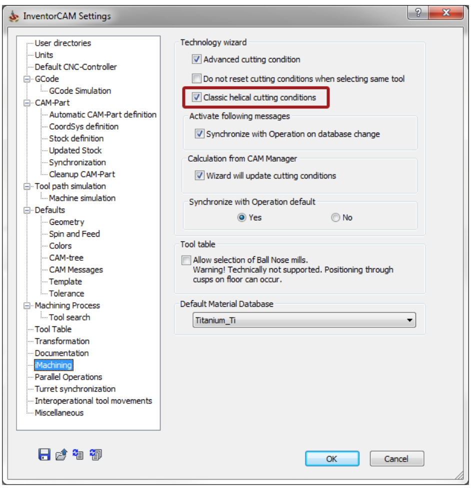


Classic helical cutting conditions is only used by the current operation; a **Ramping angle** override must not be used.

2. **Per project** – in the Technology wizard section on the iMachining page of the Part Settings dialog box.



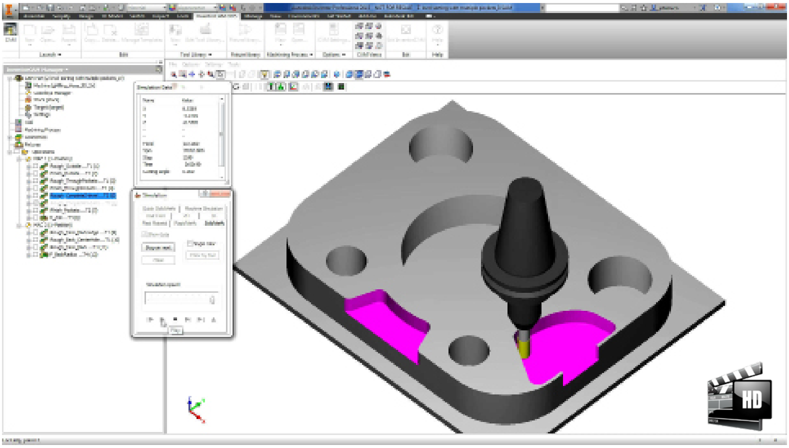
3. **Globally** – in the Technology wizard section on the iMachining page of the InventorCAM Settings dialog box.



It is important to note that the **Classic helical cutting conditions** are kept for all operations when converting 2012 CAM Projects to newer versions.

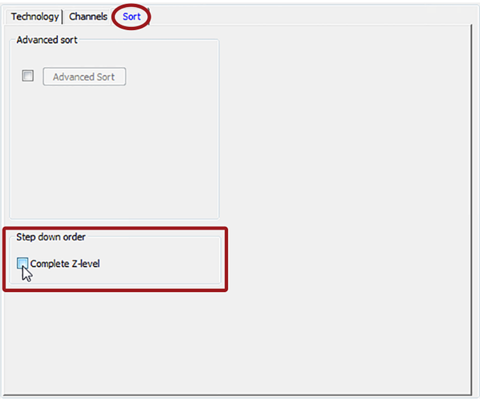
13. User option for Z-level sorting with multiple pockets

Prior to 2013, when multiple pockets were defined in one operation and more than one Step down was specified, the first Z-level was machined for every pocket before moving down to the subsequent Z-levels, as shown in the SolidVerify simulation video below. **Click to activate**, and then right-click to enable **Full Screen Multimedia**. Pressing the **Esc** key will **End Full Screen Multimedia**.

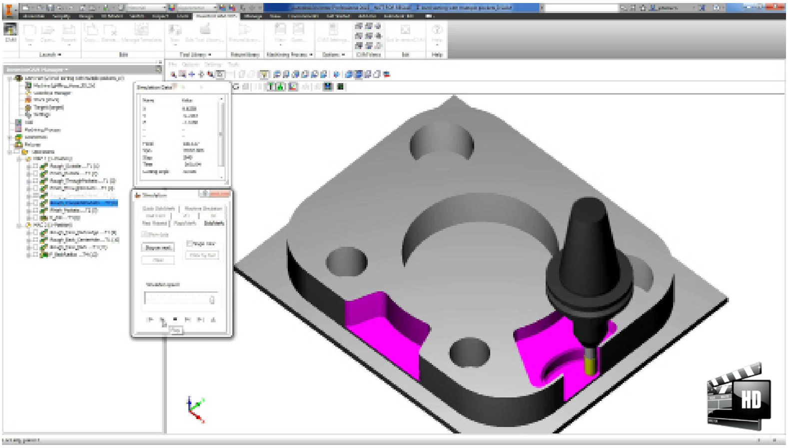


In early 2013, a user option for Z-level sorting was added. The **Complete Z-level** option appears on the Sort tab of the Technology page and is enabled by default.

When this option is disabled, the iMachining technology is informed to complete all Z-levels of each pocket before moving on to the next series of pockets.



In the SolidVerify simulation video below, you'll see that each pocket is machined completely as a result of the **Complete Z-level** option being disabled. **Click to activate**, and then right-click to enable **Full Screen Multimedia**. Pressing the **Esc** key will **End Full Screen Multimedia**.



Depending on the arrangement of pockets, it may be more beneficial to disable the **Complete Z-level** option. Doing so can help improve the cycle time by reducing retracts and long position moves.

14. Same tool selection (new defaults with user setting)

Prior to 2013, the Cutting conditions that were produced as a result of using any Technology Wizard overrides were kept in an iMachining operation when the same tool was selected from the Part Tool Table.

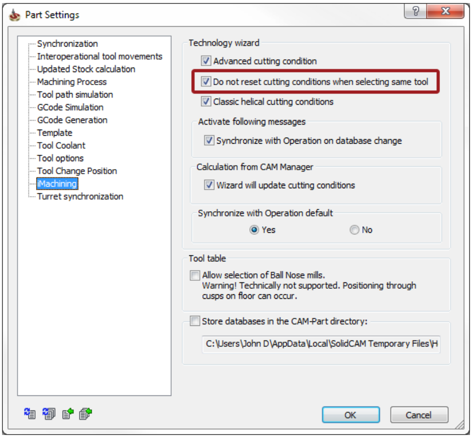
Since early 2013, any overrides are now cleared and the default Cutting conditions are calculated upon clicking the **Select** button to close the Choosing tool for operation dialog box (even if the same tool is selected).

Although it is recommended to leave the setting disabled, you do have the option to keep your user-defined Cutting conditions by enabling **Do not reset cutting conditions when selecting same tool** in the Settings.

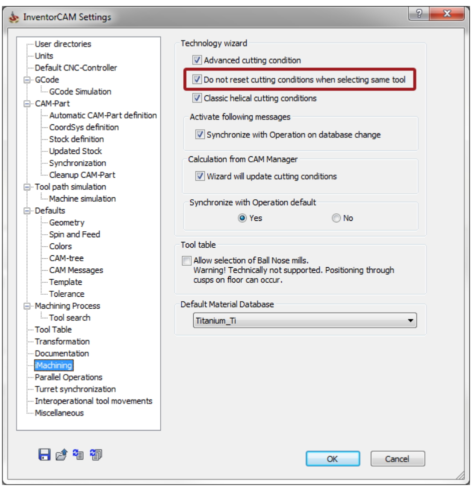
When using this option, it is critical for you to be aware that the Cutting conditions are kept and that they may no longer be optimal or *safe*.

There are two ways to enable the **Do not reset cutting conditions when selecting same tool** setting:

1. **Per project** – in the Technology wizard section on the iMachining page of the Part Settings dialog box.



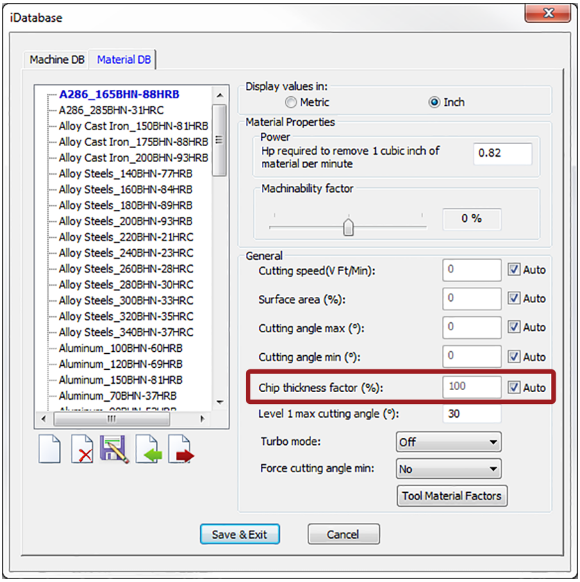
2. **Globally** – in the Technology wizard section on the iMachining page of the InventorCAM Settings dialog box.



15. Chip thickness factor

Prior to 2013, it was not possible for users to adjust the chip thickness (CT) for all tools cutting a specific material at one time.

Since early 2013, the iMachining technology enables you to adjust CT for all tools at one time by setting the **Chip thickness factor** of the material.



This parameter appears in the General section on the Material DB tab of the iDatabase dialog box. The default setting is **Auto** at 100%. Using the override check box, you can manually set the **Chip thickness factor** between 50 and 200%.

Note that manually adjusting the **Chip thickness factor** for a specific material can save time, especially when programming a part that uses many tools of different sizes.

There are three ways in which you can access the materials listed in the iMachining Database to adjust the **Chip thickness factor**:

1. On the Autodesk Inventor Ribbon, click **InventorCAM 2015** -> **Options** -> **iMachining Database**. When the iDatabase dialog box appears, switch to the **Material DB** tab.

2. In the CAM-Part Definition, click the **Edit iMachining Database** button in the iMachining Data area. When the iDatabase dialog box appears, switch to the **Material DB** tab.

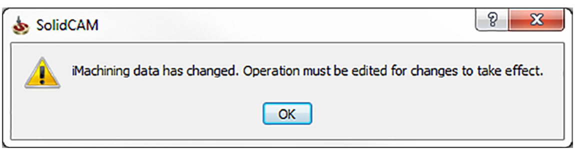
3. If the iMachining Database selections are not chosen in the CAM-Part Definition, you will be prompted to define the machine and material when the first iMachining operation is added to the CAM-Part. At which point, you can adjust the **Chip thickness factor** for your chosen material.

If you find that increasing or decreasing the feed rate or spindle speed cuts better, you are most likely modifying CT. If you increase the feed at your machine to 120% for example, you are increasing CT by 20%. Thus, increasing the **Chip thickness factor** to `120`% will eliminate the need for adjusting the feed.

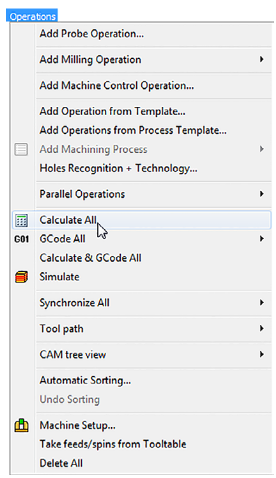
If you find that certain materials cut better at specific feeds or speeds, you can "lock in" the appropriate **Chip thickness factor** for that material. It is up to you to determine what `value` is best.

16. Wizard can update cutting conditions from the CAM Manager

Prior to 2014, if data was modified externally from an iMachining operation (i.e., changes in the CAM-Part Definition or in the Part Tool Table), you were required to manually edit and recalculate each operation individually in order for the Technology Wizard to update the Cutting conditions.

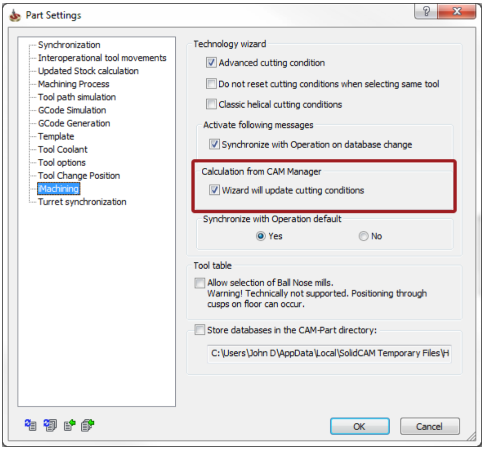


With the recently added **Wizard can update cutting conditions** setting, the calculation can now be performed from the CAM Manager, allowing you to recalculate several iMachining operations at once using the **Calculate All** command.

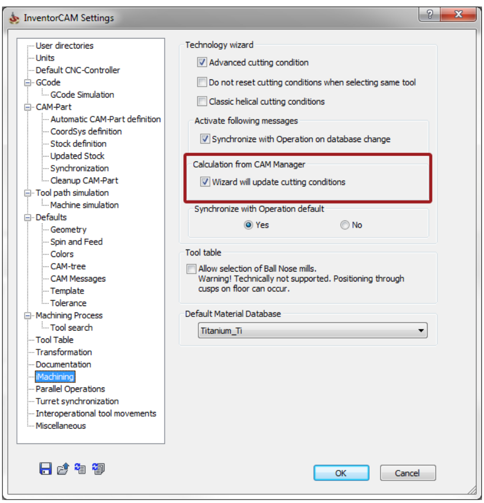


The **Wizard can update cutting conditions** setting can be enabled in the following two ways:

1. **Per project** – in the Calculation from CAM Manager section on the iMachining page of the Part Settings dialog box.



2. **Globally** – in the Calculation from CAM Manager section on the iMachining page of the InventorCAM Settings dialog box.



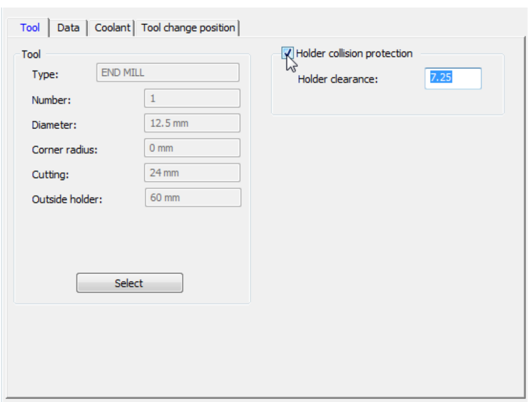
17. Low power machine support – 2 Hp (1.49 Kw) or less

In early 2013, iMachining implemented the support of low power machines. Accordingly, special Cutting conditions exist for machines defined with limited power capabilities. In short, iMachining is capable of reducing load on the machine tool, while still able to keep the cutting efficient.

It's time to dust off those older, low power machines and put them back to work with iMachining.

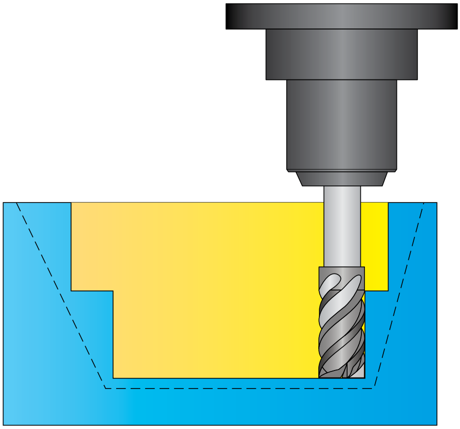
18. Holder collision protection in iMachining 3D

Since early 2014, you can select the option of **Holder collision protection** when using iMachining 3D. With this option enabled on the Tool page, the calculated tool path is automatically adjusted to avoid contact between the defined tool holder and the Updated Stock model at every stage of the machining.

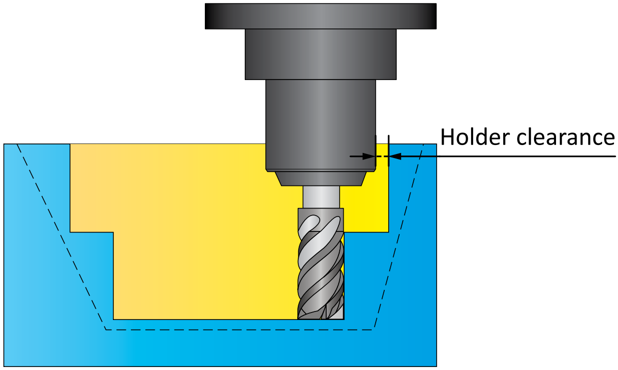


Additionally, the **Holder clearance** parameter enables you to define how close the holder can approach the material during the machining. By default, this `value` is automatically calculated based on the current tool diameter.

Without holder consideration, the extension of the tool from the holder needs to be long in order to machine deep pockets with steep walls.



With holder consideration, the extension of the tool from the holder can be short and strong, enabling the tool to run faster and more aggressive. However, because the tool path is constrained, it may be possible that not all material that can be removed by the operation is machined.

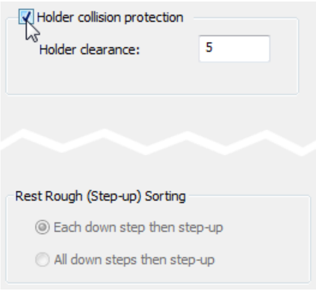


19. User-defined sorting options in iMachining 3D

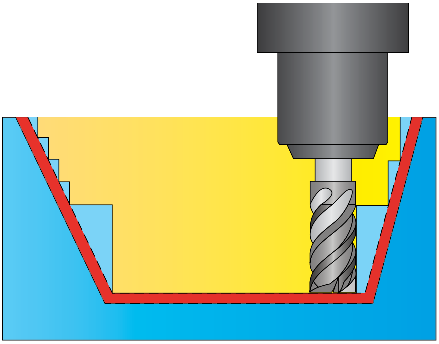
Now appearing on the Link page of the iMachining Operation dialog box, the following two methods were added in early 2014 for choosing the order of roughing and rest roughing tool path passes in iMachining 3D:

1. **Each down step then step-up** (default selection) – this option successively performs the Step-up rest roughing after each Step down roughing pass is achieved.
2. **All down steps then step-up** – this option performs the Step-up rest roughing after the final Step down roughing pass is achieved.

When **Holder collision protection** is enabled on the Tool page, **Each down step then step-up** remains the default selection and cannot be changed.

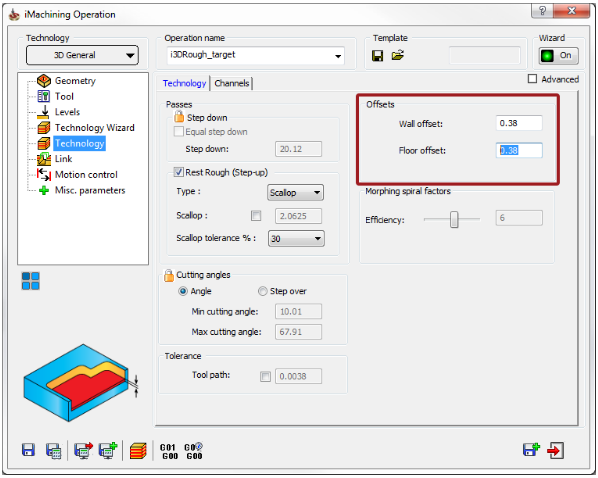


Each down step then step-up provides maximum clearance for the tool holder, which allows the extension of the tool from the holder to be short and strong. In most cases, all material that needs to be removed and that can be removed by the operation is machined.



20. Floor offset parameter in iMachining 3D

Recently added, iMachining 3D enables you to define a **Floor offset** that is separate from the **Wall offset**. The updated parameters appear on the Technology page of the iMachining Operation dialog box.

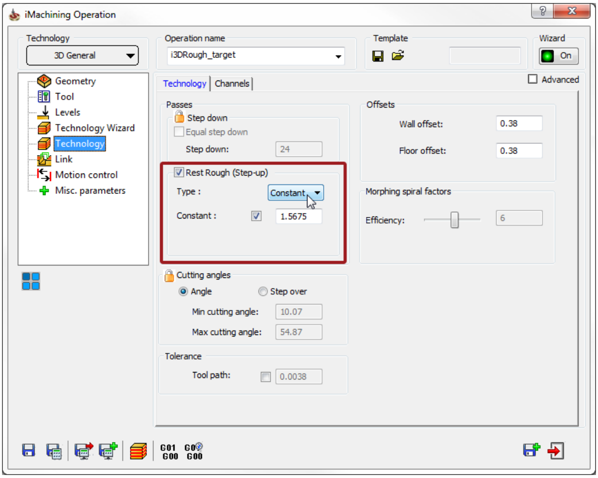


When converting existing CAM-Parts (e.g., from InventorCAM 2014), if the **Finish floors** option was enabled in the Offsets section of an operation, then a **Floor offset** of 0 is automatically specified for that operation in InventorCAM 2015 and later.

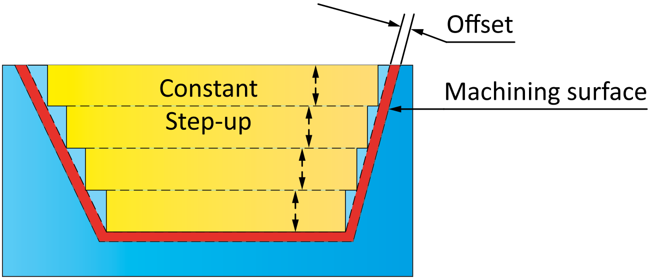
If the **Finish floors** check box was not enabled in an earlier version, then the **Floor offset** is made equal to the **Wall offset** as shown in the iMachining Operation dialog box above. For new CAM Projects, the offsets are also made equal by default.

21. Constant Step up in iMachining 3D

Just added, iMachining 3D enables you to choose how the rest roughing tool path passes are performed.



When **Constant** is selected, the height of the steps during Step-up are made constant. The value in the text field defines the vertical distance of all steps, regardless of the local slope of each individual surface (steep or shallow).

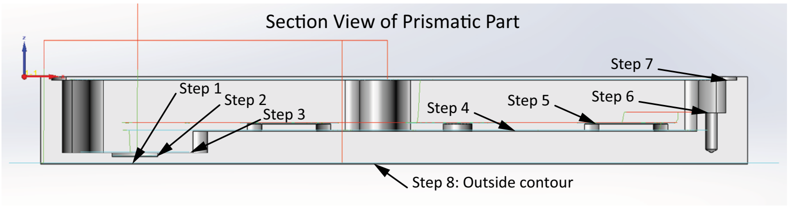


22. iMachining 3D for prismatic parts

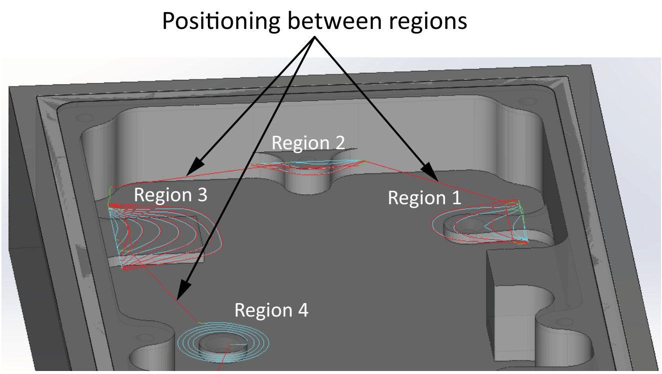
With the newly introduced **3D Prismatic** technology in InventorCAM 2015 SP3, it is currently recommended to use iMachining 3D for the roughing, rest machining and semi-finishing of 3D prismatic parts.

When machining 3D prismatic parts, performance and efficiency is automatically maximized to achieve the shortest possible cycle time. Using iMachining 3D over iMachining 2D provides the following four benefits:

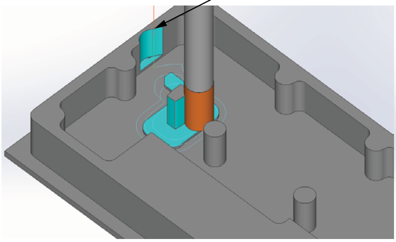
1. iMachining 3D performs the deepest step downs first to remove the most amount material, resulting in optimized depths of cut. Material Removal Rate (MRR) and tool life are maximized and the need for full retracts is eliminated.



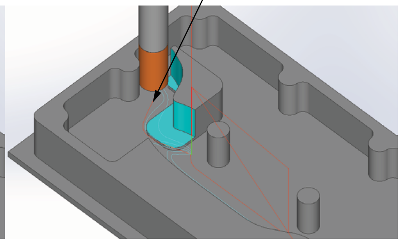
2. iMachining 3D performs intelligent sorting of 2D Z-level regions. Non-cutting moves are reduced by the 3D Z-level ordering and localized machining of 2D tool path regions.
3. iMachining 3D performs smart positioning between 2D Z-level regions. Long position moves are reduced by the 3D Z-level linking and localized machining of 2D tool path regions.



4. iMachining 3D provides automatic protection of the Target model. Large tools can safely be used in confined spaces.



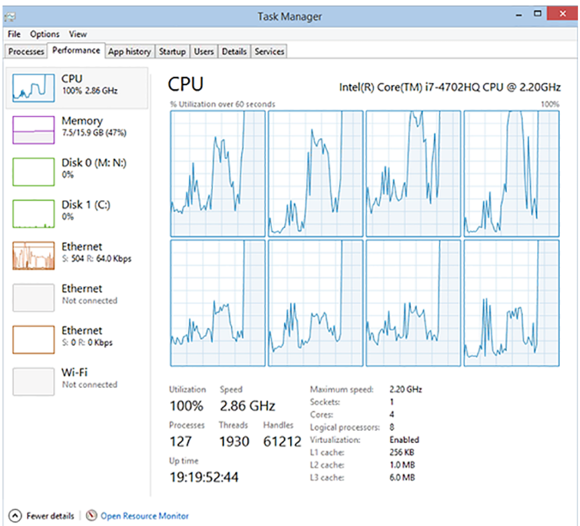
iMachining 2D with large tool



iMachining 3D with large tool

23. Parallel calculation in iMachining

With the implementation of multi-core and multithreading support, the iMachining technology is now able to achieve even faster calculation times. Such techniques, which in this case are complementary, enable your computer to perform more tasks with greater overall CPU performance.



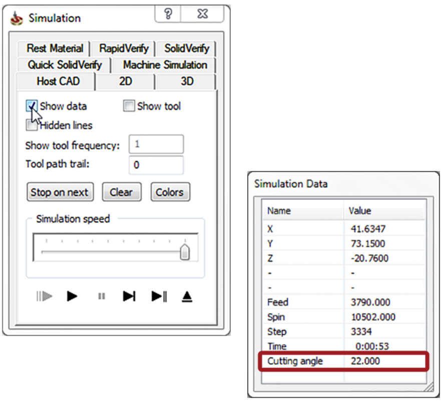
This means that the calculation of an iMachining operation can be distributed and solved simultaneously across multiple central processing units (cores). Multithreading is then performed by each core to manage and complete the task more efficiently.

When using iMachining 3D specifically, where tool path calculation times can be long, you will notice the highest impact since multiple Z-levels can now be calculated simultaneously.

24. Cutting angle feedback during simulation

When simulating an iMachining operation, you can now view real-time changes in the cutting angle of the tool path.

The cutting angle is displayed in the Simulation Data dialog box along with other tool path information such as coordinates of the current point, time, feed and spin. To open the Simulation Data window, enable the **Show data** option in the Simulation control panel.



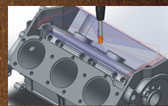
The **Cutting angle** field displays the tool's engagement in the material at every step of the simulation.

InventorCAM 2015

Miscellaneous FEATURES

iMachining 2D | iMachining 3D

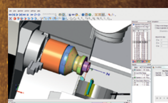
The complete range of manufacturing applications inside Autodesk Inventor



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

The InventorCAM 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 and 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 of 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 automatic thin wall avoidance, iMachining™ brings efficiency to a new level for CAM users.

Highest level of Autodesk Inventor integration

InventorCAM provides the highest level of CAD integration, with seamless, single-window integration and full associativity to Autodesk Inventor. The integration ensures the automatic update of tool paths for CAD revisions.

InventorCAM powers up the user's Autodesk Inventor system into the best CAD/CAM solution.

YouTube Broadcast Yourself™

www.youtube.com/SolidCAMProfessor
www.youtube.com/SolidCAMiMachining

www.facebook.com/SolidCAM
www.facebook.com/iMachining

iMachining Successes

www.inventorcam.com/us/imachining/imachining-successes/

www.inventorcam.com