InventorCAM
2015

The unique, revolutionary Milling technology

imachining®
patent by SolidCAM

TIME SAVINGS
70%
... AND MORE!

iMachining FAQ

Frequently Asked Questions

CERTIFIED
AUTODESK INVENTOR 2015
AUTODESK

InventorCAM + Inventor
The Complete Integrated Manufacturing Solution

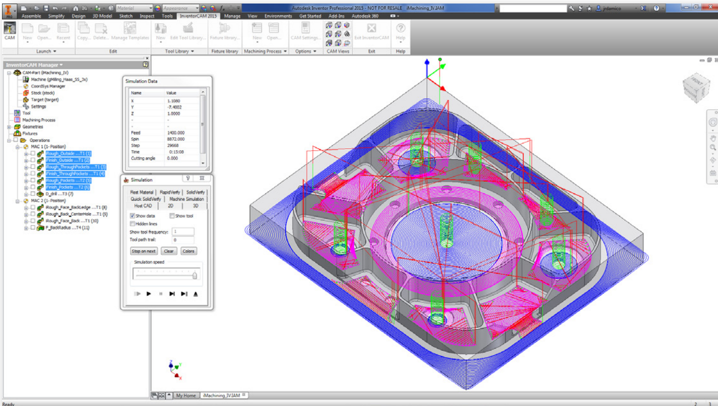
InventorCAM
iMachining – The Revolution in CAM!

Contents

Document number: IViMFAQENG3-2015

What is InventorCAM iMachining?

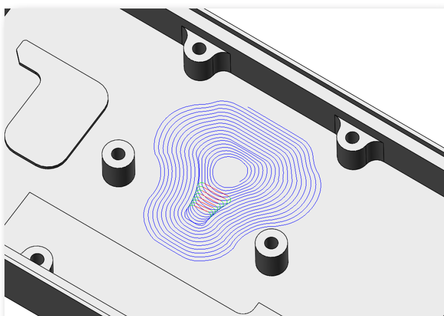
InventorCAM's iMachining™ is an intelligent high speed milling technology, designed to produce fast and safe CNC programs to machine your mechanical parts with *first part success* performance. The word *fast* meaning significantly faster than traditional machining at its best and the word *safe* meaning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

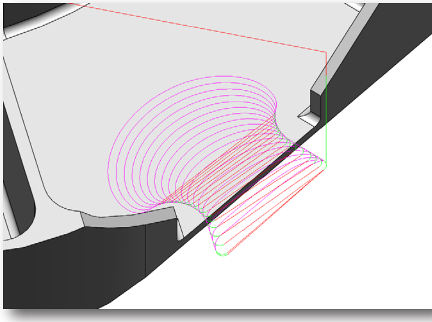


To achieve these goals, the iMachining technology uses advanced, patented algorithms to generate smooth tangent tool paths, coupled with matching conditions, that together keep the mechanical and thermal load on the tool constant, while cutting thin chips at high cutting speeds and deeper than standard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

iMachining tool paths

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



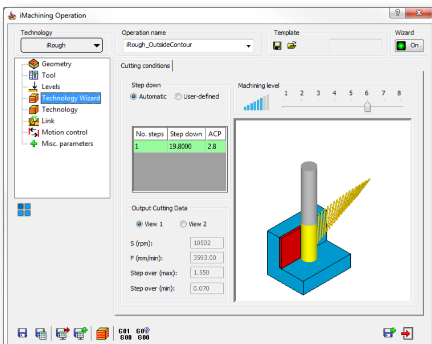


iMachining uses proprietary **constant load one-way tool paths** to machine narrow passages, separating channels and tight corners. In some open areas, where the shape is too irregular to completely remove with a single spiral, proprietary topology analysis algorithms and channels are used to subdivide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

iMachining Technology Wizard

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

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



What are the important Stock Material properties?

General

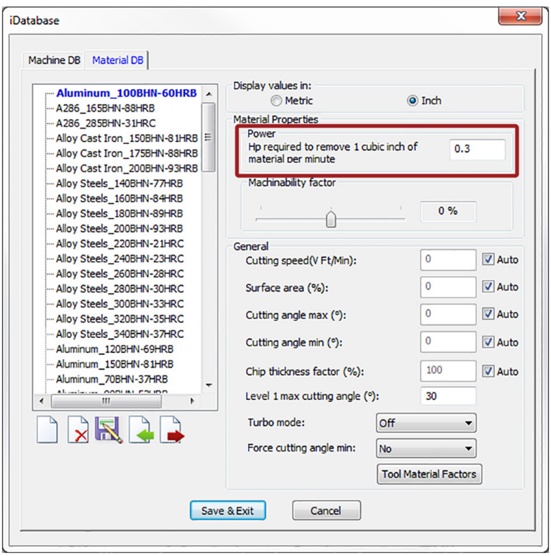
Different materials require different amounts of force to cut them. The physical property of a material that determines the force required for a particular cut is the Ultimate Tensile Strength (**UTS**), given in units of MPa (Mega Pascal) in Metric units or psi (pound per square inch) in English units.

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.

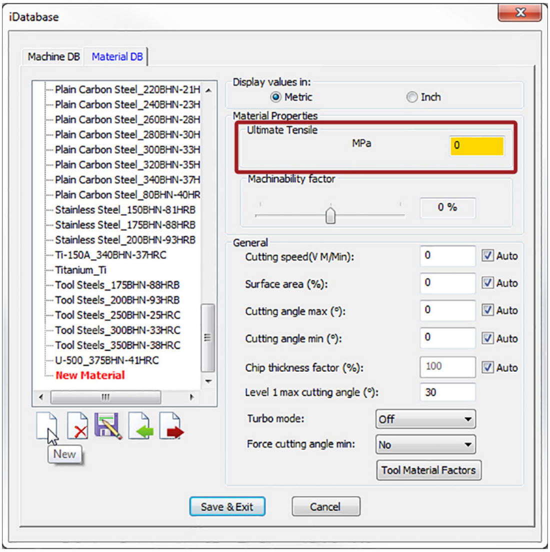
All InventorCAM versions are shipped with a basic Material Database containing more than 70 different materials.

History

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 databases 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.

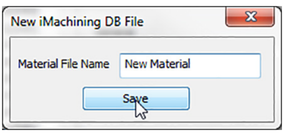
Defining new material entries in the Material Database

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.

How do I find the UTS value of a material?

Make sure you know the exact specification of your material

Case Study: An InventorCAM customer needed to cut a part made of Titanium. On www.matweb.com they searched Titanium and got a whole list of Titanium materials. They selected the first entry, Titanium Ti, which is the pure form of the metal. In the Mechanical Properties section, they found that the **UTS** was 220 MPa. Accordingly, they defined the material with the given value. Then, they selected the newly entered material from the Material Database list in the iMachining Data section of the Milling Part Data dialog box. They defined their iMachining operation, saved and calculated it, generated the GCode, and started cutting. Their tool broke after 5 seconds in the cut.

When the customer called our support center, we quickly understood that they were trying to cut an aerospace part. The material was then identified as Ti – 6Al – 4V, a very common aerospace material.

We advised the customer to search this specific material on www.matweb.com. They informed us there were six different entries of Ti – 6Al – 4V on MatWeb, ranging in **UTS** from 860 to 1170 MPa. The customer said they did not know which one was their material, and it was too late in the day to contact their supplier. We advised them to use the entry with the highest value of **UTS**, 1170 MPa.

When in doubt, use the highest value in the list. Later you can decide, based on the cutting sound and rate of tool wear, whether or not it is safe to change to a lower value in the list.

The best way, of course, is to find out the exact material specification with the help of your material supplier or your customer.

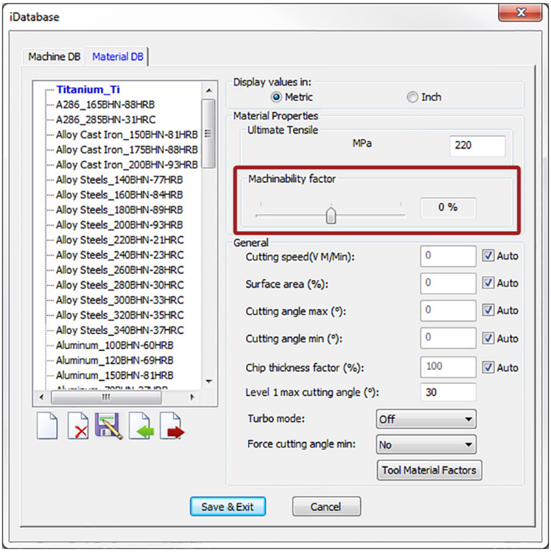
If there are many entries to choose from...

Always start with the highest **UTS** value – this is absolutely *safe*. 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.

Considering the Machinability factor of a material

After machining, you may discover that you can cut your material 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. Because the same materials are made by many different manufacturers, it should be expected that tolerances exist between your material and its given **UTS** value, making it more or less machinable.

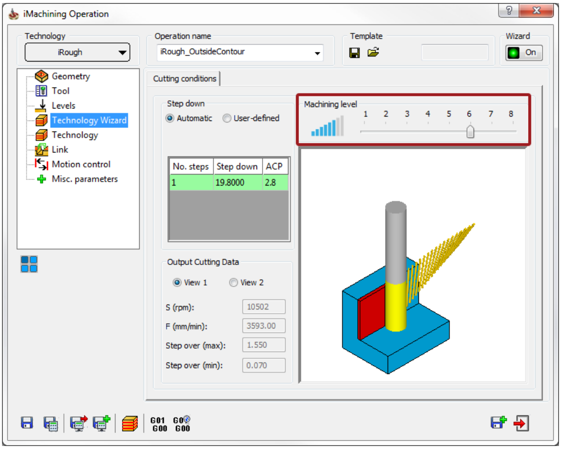
The recently introduced **Machinability factor** enables you to alter the hardness of a material without changing its given **UTS** value. This option is available on the Material DB tab of the iDatabase dialog box.



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.

What is the role of the Machining level slider?

The **Machining level** slider provides an iMachining user with the means to conveniently and intuitively control the Material Removal Rate (**MRR**) when machining their part. The Machining level selected by the user, through moving the slider, informs the Technology Wizard how aggressively to machine the part.



As every experienced machinist knows, increasing the feed by 10% without changing anything else will increase the **MRR** by 10%. (Actually, a little less due to rapid moves and time wasted on acceleration). Approximately the same increase can be achieved by increasing the side step by 10%. You may also know that these actions might have negative side effects, like stalling the spindle because you exceeded its maximum Torque, or reducing the tool life as a result of the higher chip thickness involved.

The same experienced machinist might also know that increasing *both* the feed and the spindle speed by 10% will increase the **MRR** without changing the chip thickness, although it will increase the cutting speed by 10% and increase the power output required from the spindle. If this machinist knows the higher power is available, their cooling arrangement is good enough, the tool is sharp enough and its coating still intact, they might venture to make these increases and thus reduce the cycle time. If they are a real expert, they will know there is a likelihood the tool will not last as many parts as before. They may choose to make the increases anyway, due to a tight schedule, knowing there are enough tools to complete the run.

On the other hand, if the sound of cutting indicates the onset of vibrations after making the increases, the experienced machinist will immediately go back to the original Cutting conditions realizing that the machining setup (rigidity and state of the machine and rigidity of the work and tool holding) is not rigid enough for the higher aggressiveness.

These are the kinds of decisions the Technology Wizard makes, using similar reasoning, based on sophisticated algorithms that analyze the entire set of factors, properties and limitations which characterize the machining set up (the part geometry, material properties, tool properties and machine limitations). The knowledge-based Wizard uses the known interdependence between all these factors to suggest the optimal combination of Cutting conditions for the job. Its algorithms work hand-in-hand with those of the **iMachining Intelligent High Speed Tool Path Generator** to produce the optimal, *fast* and *safe* CNC program to machine the part delivering **First Part Success** performance.

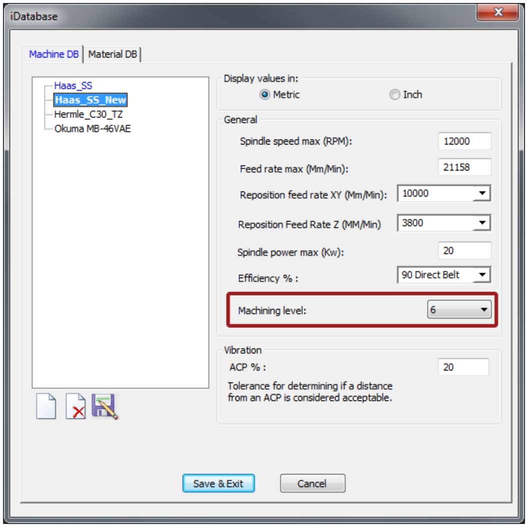
However, as we have seen above, there are factors that influence the attainable **MRR** and tool life (such as the basic rigidity of the machine, work and tool holding, and the machine's level of maintenance) as well as the desired compromise between high **MRR** and long tool life, influenced by your production schedule and cost structure that are difficult to accurately quantify. Instead, the Wizard provides the **Machining level** slider that enables you to easily and intuitively incorporate the combined effect of these factors in the Wizard's decision making process.

Machine Default Level

The correct method of using the **Machining level** slider is to assign each machine in the workshop with a **Machine Default Level**, which reflects the basic machine rigidity and its state of maintenance.

The assigned default level should not be influenced by the speed, power or acceleration capability of the machine. These parameters are known to the Wizard from the Machine Database. The **Machine Default Level** should only reflect the machine's tendency to develop vibrations. An older, ill-maintained, non-rigid machine should be assigned a very low default level: between 2 and 4. A brand-new, rigidly constructed machine should be assigned a very high default level even if it is a very slow machine: we recommend level 6 Turbo (see the *What is the Turbo Mode of the Machining levels?* section below). There will be enough time to push it up to level 7 or 8 Turbo after listening to the first cut, providing everything sounds and looks perfect. If you only need to cut one part, the difference in cycle time would not matter much anyway.

This **Machine Default Level** is defined only once and is stored in the Machine Database, together with all the other constant machine parameters (Spindle speed max, Feed rate max, etc). You only need to update this default level every 2-3 years, and after a crash or a major maintenance procedure.



Preparing the CNC program for a new setup

Before using iMachining to generate a CNC program for a new setup, you need to assess the rigidity of the work and tool holding, and measure the balance and TIR (True Indicator Reading) of the tool in its holder. If they are not good, reduce the operation **Machining level** by 1 or 2 from the initial default level of the machine.

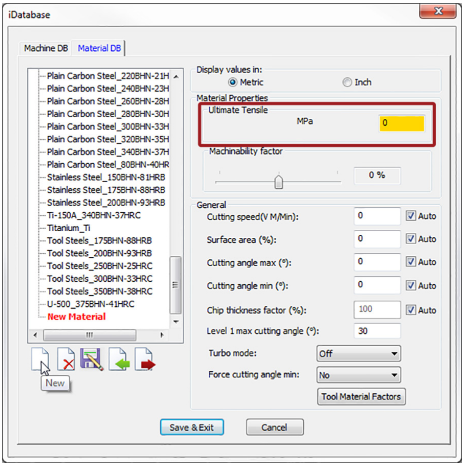
Use the resulting **Machining level** to cut the first part. Listen to the sound of cutting and assess the resultant surface quality and tool wear. If there are more parts to cut, and the previous cut was good, you may want to increase the **MRR** or decrease it to get longer tool life, depending on your schedule, tool availability and cost structure. All you need to do is to move the **Machining level** slider one position up or down, calculate a new tool path and cut another part.

The reason why it is possible to increase the level is because the Wizard, although aiming to cut as fast as is wise, always uses Cutting conditions values that are below the safe maximum by a reasonable margin, leaving enough room for taking a more risky cut.

But beware, the risk is real! The **Machine Default Level** was set according to a subjective assessment of the machine's condition. This assessment may be optimistic, and so might be the assessment of the work clamping and tool holding.

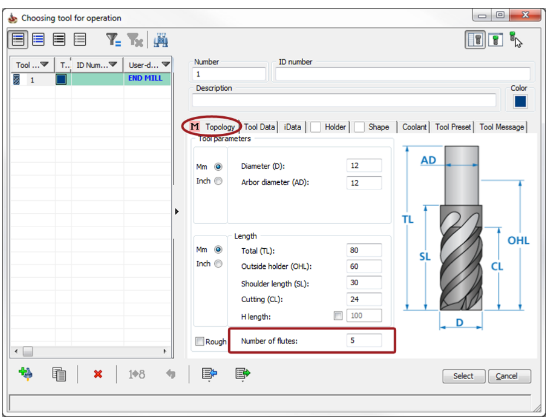
Material UTS

In the *What are the important Stock Material properties?* section, we have seen the importance of the **UTS** of a material. This is not a free parameter for the user to set a value to their liking, but it is worth mentioning to stress how dramatically it affects the Cutting conditions and therefore how critical it is to set the correct value.



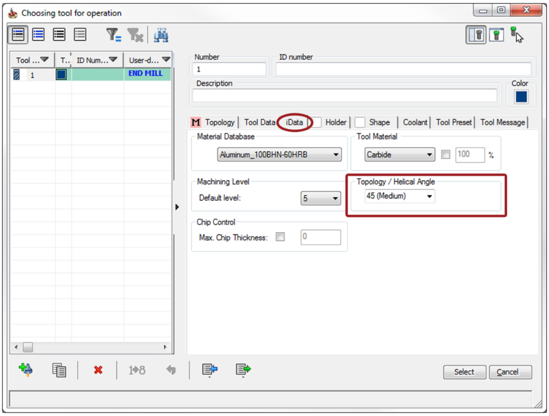
Number of Flutes

Another important parameter, which value is not free to set by the user, is the number of flutes of the end mill. Changing the **Number of flutes** will change the Cutting conditions (usually, just the feed).



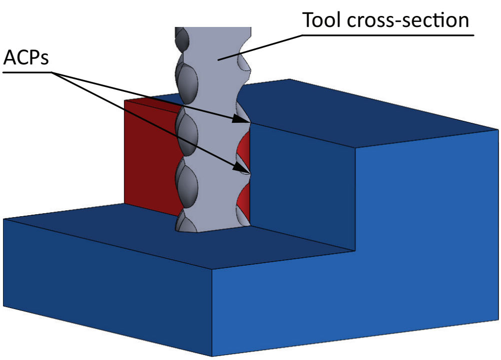
Tool Helix Angle

The **Helical Angle** of the flutes is in a class of its own. Changing the **Helical Angle** only changes the Axial Contact Points (**ACP**) indication, which by itself has currently no effect on the Cutting conditions, though it may (should) drive the user to decide to change the tool or the Step down or reduce his machining level to avoid vibrations. It should be mentioned that the helix angle has a strong effect on the downwards force on the tool; if ignored it can result in the tool being pulled out of its holder, with devastating effects.



Axial Contact Points (ACP)

This is not a user-defined parameter. 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 producing, along a vertical line.



If the axial depth of the cut is d, the tool diameter is D with N flutes, and the flute helix angle is β, we can calculate the Flute Pitch P as follows:

(Flute Pitch) $P = \pi D \times \tan(90 - \beta)$

Since the tool has N flutes, the vertical distance p between neighboring cutting edges (Fine pitch), is given by:

(Fine pitch) $p = P/N$

The *ACP* can then be calculated by asking how many Fine Pitch intervals can fit in depth D. The answer is:

$ACP = D/p$

Now the question is, "How does knowing the ACP value help us to cut better?" The answer is simple:

According to the iMachining theory, the closer the ACP value is to a whole number (≥ 1), the less likely it is that vibrations will develop.

By default, there is a 20% tolerance on any ACP over 1.

So if you get an ACP of 1.0 or 1.1 or 1.2 or 1.8 or 1.9, you are safe. Having vibrations is less likely.

The same is true, if you get 2.0 or 2.1 or 2.2 or 2.8 or 2.9.

If you get an ACP of 1.3, 1.4, 1.5, 1.6, 1.7, or 2.3, 2.4, 2.5, 2.6, 2.7 etc, you should think of a way to either change it (e.g., change the number of down steps) or change the tool, or reduce the Machining level.

The Technology Wizard will alert you whether or not the situation for stability is good based on ACPs. The output grid changes color to indicate the current situation:

- **Green** = Good- Preferred
- **Yellow** = Not so good- Medium likelihood of vibrations
- **Red** = Bad- High likelihood of vibrations

No. steps	Step down	ACP
2	15.0000	2.0

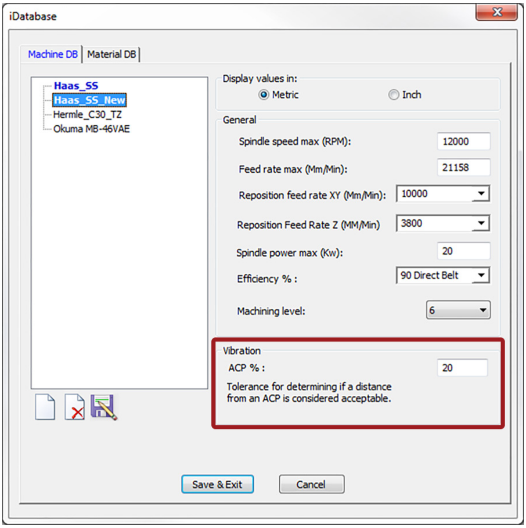
No. steps	Step down	ACP
1	13.0000	1.7

No. steps	Step down	ACP
1	5.0000	0.7

Because the reaction of cutting force is transmitted to the tool and from there to the machine, favorable ACP values are taken into account when generating the depths in an effort to avoid vibrations.

However, it's just not possible to always be machining with preferred ACPs. But by monitoring the ACP values and the cutting results, over time you may find that matching a tool to the current depth to get good ACPs is beneficial.

Now in the Machine Database, there is a parameter to define the **ACP %** tolerance specific to a given machine. This tolerance enables you to control the ACP indication and how the 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.

As mentioned, the ACP tolerance is set to 20% by default (for new machines). With this setting, the ACP indication will show that a value of 1.1 or 1.2 or 1.8 or 1.9, for example, is acceptable and the depth will be painted green for good stability.

The recommended ACP tolerance is in fact 20%. In most cases, you will not need to alter this setting.

Spiral Efficiency

iMachining generates Morphing spiral tool paths whenever it needs to clear a completely open or completely closed pocket area, which does not have the shape of a circle. This means it generates tool paths with different side steps in different directions. See **Figure 1** below: The Effect of Spiral Efficiency.

As a result, the average side step is smaller than the maximum side step. This makes the average MRR less than the maximum MRR possible. This means that a morphing spiral is potentially less efficient than a regular round spiral.

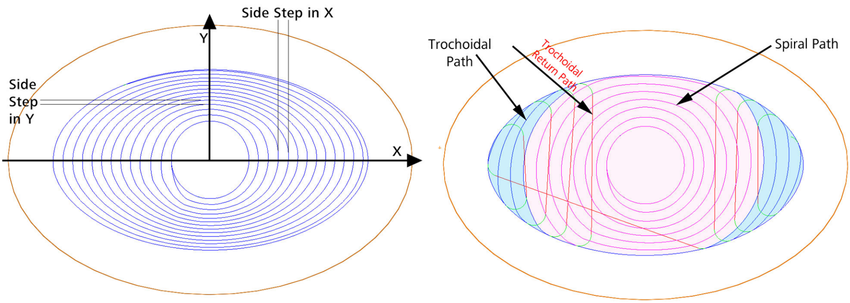


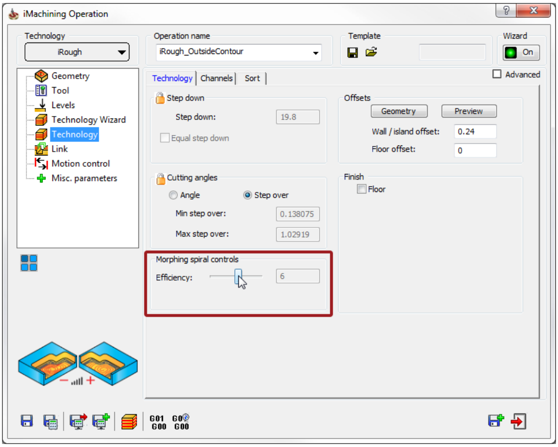
Figure 1A: Low efficiency setting:
Spiral with strong morphing clears complete area

Figure 1B: High efficiency setting:
Spiral with little morphing is rounder and clears only part of area, the rest is cleared by a Trochoidal Path

There are three reasons why we are doing this:

1. Since the Technology Wizard adjusts the feed at every point along the tool path in order to maintain a constant cutting force on the tool, the actual loss in the average MRR is, in many cases, negligibly small or even zero. This greatly depends on the maximum feed the machine can achieve. With very slow machines, the Wizard cannot fully compensate for some of the very small side steps indicated by the morphing action, because the maximum feed of the machine is not high enough. In such cases, if your first priority is high average MRR and long tool life is less of an issue, you can instruct iMachining to limit the extent of morphing of the spirals.

You can limit the morphing by selecting a higher value of Spiral Efficiency with the **Efficiency** slider. This slider can be used in the Morphing spiral controls section on the Technology page of the iMachining Operation dialog box.



2. The second reason is based on the old saying "You give a little to gain a lot." Our aim is to get higher global efficiency for the whole pocket or part, and for this we are willing to sacrifice a little in the local efficiency of a specific spiral.

   Comparing the tool paths in case (a) with that of case (b) in **Figure 1**, we notice that while the morphing spiral of (a) manages to clear the whole area of the pocket, the conventional round spiral in (b) terminates (when reaching the pocket wall) after only clearing 55% of the pocket area. The remaining area needs to be cleared with trochoidal-like tool paths, which are by definition about 36% to 50% less efficient than round spirals, depending on the maximum acceleration of the machine and the feed used for cutting.

   If we define the efficiency of the round spiral as 100%, and use a machine and a cutting feed that produce a trochoidal-like efficiency of 55%, we can calculate the total efficiency in case (b) as: 55% of the area at efficiency 100% (round spiral), plus 45% of the area at efficiency 55% (trochoidal-like), which is 55 + 24.8 = ~ 80% efficiency.

On the other hand, the efficiency of the morphing spiral in case (a) is just over 89%. It is not easy to calculate. However, you could measure it by running this exact shape pocket on your machine. Actually, you will find case (a) in iMachining has an efficiency of over 94%, because iMachining increases the feed when the side step is smaller than the maximum specified.

If we now look at the relative efficiency of (a) to (b), we get 89/80 = 1.11. This means that (a) completes the cut in 11% less time than (b). This is without adjusting the feed when the side step is smaller.

With the iMachining feed adjustment, the cycle time for (a) is (80/94 = 0.851) 15% shorter than that of (b). This, however, is only the difference in efficiency for the simple convex shape in **Figure 1**.

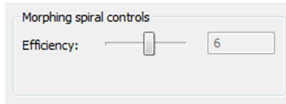
When we come to deal with more general shapes, which have concave sections in their contours, the difference in efficiency becomes much larger and the reduction in cycle time reaches beyond 30% in favor of the iMachining morphing spiral.

3. The third reason is our wish to extend the tool life to the maximum possible. It is well known that a continuous spiral cut causes less wear on the tool than repeated short cuts with their associated lead ins and lead outs from the material.

   As you can see in **Figure 1**, the morphing spiral on average reduces the portion of the total pocket area to be cleared by trochoidal-like tool paths, to less than 30%. Without iMachining's ability to generate morphing spiral tool paths, this average portion rises to over 60% of the total pocket area. This assures that with the iMachining tool paths, the tool is cutting continuously most of the time, suffering much less wear than when in the repeated interruption mode of trochoidal-like cuts.

The **Efficiency** slider enables the user to control the efficiency in the spiral tool paths.

Moving the slider to the right, increases the spiral efficiency, while moving it to the left decreases it.



Increasing the efficiency reduces the variation of the side step permitted in the spiral, making the side steps in all directions more equal and accordingly producing a rounder spiral, looking more like a circle.

Decreasing the efficiency allows iMachining to use more of the side step range specified by the Technology Wizard. This produces a spiral, which looks less like a circle and covers a greater part of the area, by managing to morph itself into the narrower parts of the area. See **Figure 2** below.

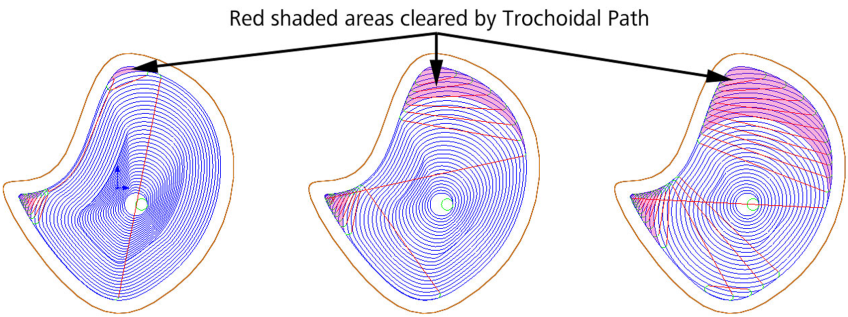


Figure 2A Efficiency level 1

Figure 2B Efficiency level 5

Figure 2C Efficiency level 10

Figure 2: Same pocket cut with 3 different spiral efficiency settings

The default setting of the **Efficiency** slider is **6**. We recommend leaving it in this position unless there is good reason to change it. However, it is a good idea to experiment with different positions, and simulate the tool paths to appreciate the effect of using the slider.

Some users, who use expensive tools regularly, use an **Efficiency** level of **3** or less to reduce the use of trochoidal-like tool paths. It depends on your priorities and cost structure (relative cost per part of machine time, tooling and labor). Using very low efficiency levels will increase the cycle time for some geometries, while increasing the tool life.

Future plans: InventorCAM plans to develop an automatic Spiral Efficiency level setting algorithm, with means for users to indicate their priorities.

The *priority* indicated by the user will be one of three:

1. **Minimum cycle time** – a short delivery deadline, or an expensive machine and a low cost tool
2. **Maximum tool life** – an expensive tool, or you are committed to deliver six parts by morning and you only have one tool in stock
3. **Minimum cost** – the algorithm will automatically find the right balance between cycle time and tool life, using your input regarding the hourly machine cost and cost of the tool

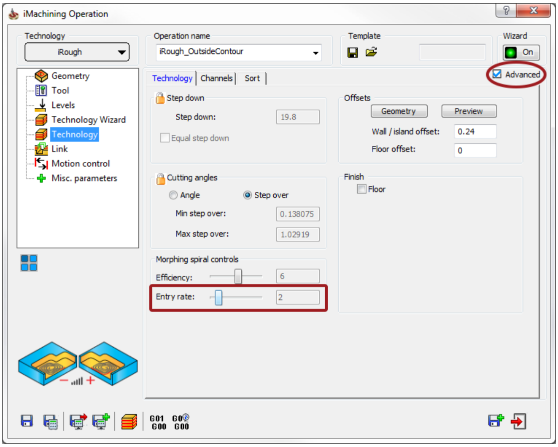
This option will be activated when a user selects the **Automatic** option for setting the Spiral Efficiency.

If a user selects the **Manual** option, they will be able to stay with the existing method of setting their preferred efficiency using the **Efficiency** slider.

Note: When using the **Automatic** option, the new algorithm will calculate the level for each spiral separately. Since even in one 2D pocket, there may be more than one spiral tool path, each spiral will be constructed with its own level calculated by the new algorithm according to the selected *priority*. However, when using the **Manual** option, the level selected by the **Efficiency** slider is global and will affect all spirals in the iMachining operation in the same way.

Entry rate slider

The **Entry rate** slider sets the rate at which a spiral tool path first enters the material. All spirals approach the material from air, whether it is from the outside of an open pocket in the case of a converging spiral, or from the inside (a pre-drill or a helical entry) in the case of a diverging spiral.



We have found for hard materials, it is better to enter the material more gradually and not directly lead in to the initial radial depth determined by the side step appropriate for the specific shape of the morphing spiral.

Figure 3: The red parts of the path show the gradual entry

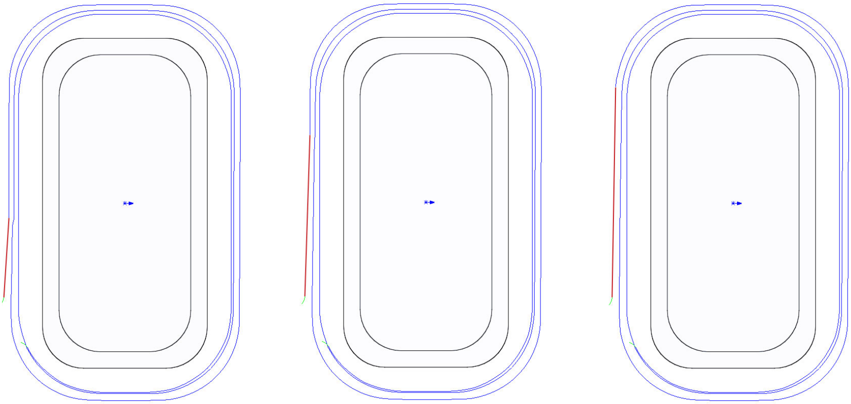


Figure 3A: Entry Rate Level 1 **Figure 3B:** Entry Rate Level 5 **Figure 3C:** Entry Rate Level 10

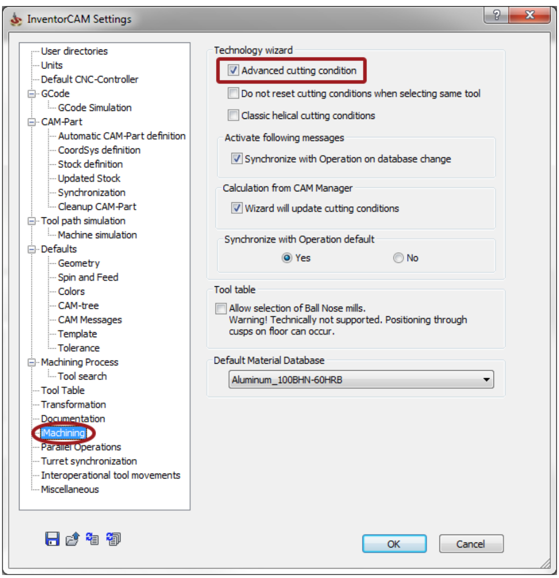
Although the **Entry rate** is automatically set by the Technology Wizard in accordance with the properties of the stock material, the developers decided for the sake of flexibility and user-friendliness, to provide users with the means to override this value. Moving the slider to the right increases the rate of entry and vice versa. The value displayed to the right of the slider only indicates the relative rate and has no fixed units.

If in doubt, change the rate by 4 or 5 notches, calculate and simulate in the Host CAD mode to observe the new **Entry rate**.

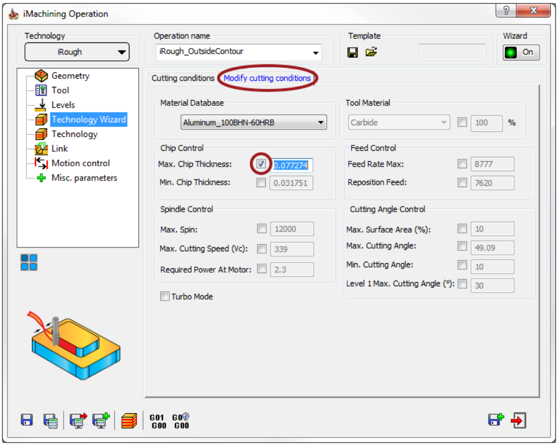
Advanced Mode

This is a special mode which provides users with additional flexibility and control. The **Advanced cutting condition** option can be activated globally (in the InventorCAM Settings) or locally per part (in the Part Settings).

1. Open the desired **Settings**, go to the **iMachining** page and select the **Advanced cutting condition** check box.



2. In the iMachining Operation dialog box, switch to the **Technology Wizard** page. In addition to the standard **Cutting conditions** tab, the Wizard will now show a new tab – **Modify cutting conditions**.

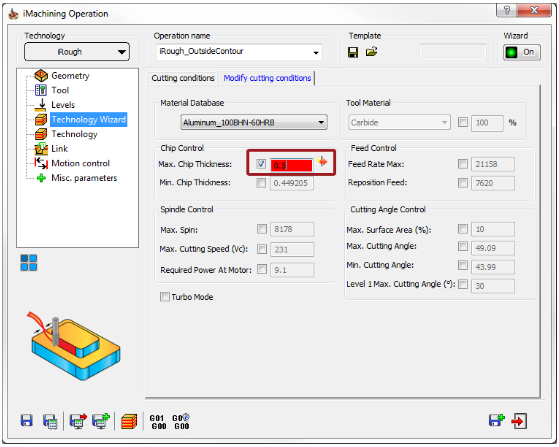


3. On the Technology Wizard page, switch to the **Modify cutting conditions** tab. You now have the option of modifying any one or more of the Cutting conditions parameters. We strongly recommend using it only if manipulation of the Machining level slider does not produce your desired result.

4. On the **Modify cutting conditions** tab, you can see that all parameter fields are initially disabled. To modify the value in any field, select the check box next to it. Before you modify any value, read the following important note.

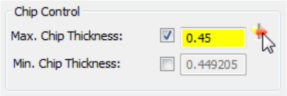
Note: The values appearing on the **Modify cutting conditions** page are always those corresponding to **Machining level 8** (Normal or Turbo, whichever is the current mode). If you have chosen a level different from 8 on the Machining level slider, you will not get the value that you entered on the **Modify cutting conditions** page. In the Output Cutting Data, you will see the newly interpolated value between the original level 1 value and the new value, which you have just set for level 8.

5. As you start modifying fields, you may find the field background color changing to red, with a border-crossing arrow appearing next to the field.



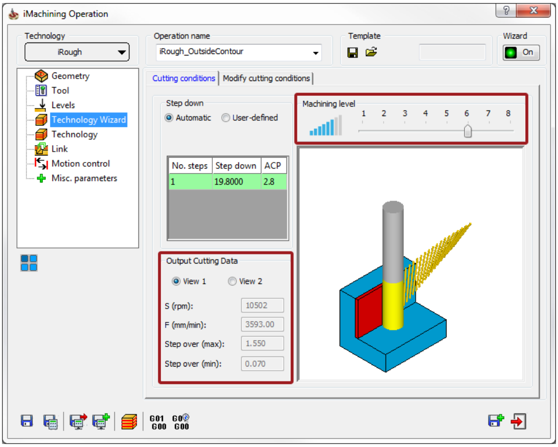
This simply signifies that the chance intermediate value in the field (e.g., resulting from one digit being deleted in the field) cannot be reconciled with the machine limitations, or with some other parameter value you may have already modified. If the red color persists after you finish modifying the field, it signifies that the final value you have set for the field is not reconcilable with the other values and constraints, and you are advised to change the situation.

6. One simple way to adjust the values is to click the icon next to the field. The Wizard will calculate the nearest reconcilable value to the one you have set, and replace your value with the calculated one, while the field background color changes to yellow. When all values are adjusted, you can then click the **Save & Calculate** button.



7. An even simpler way is to deselect the check box next to the field. This restores the original value given by the Wizard and removes all background colors.

8. Another way to reconcile the values is to continue modifying other values that are responsible, at least in part, for the mismatch, until everything is resolved. This is not an easy task.



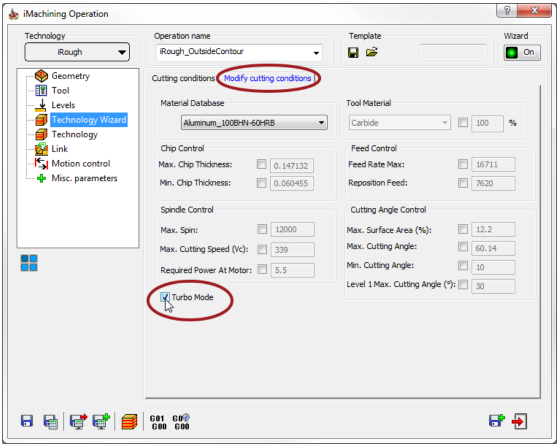
The purpose of the Machining level slider is to enable you to change all the parameters together in a synchronized manner, which gives you *easy* and *safe* control over the machining aggressiveness.

9. There is the path of least resistance, and the most risky option, of turning off the watchful eye of the Wizard. Click the green light at the upper right corner of the iMachining Operation dialog box.

The light turns red, which means the Wizard is now turned **Off**, and you are fully responsible for the consequences.

What is the Turbo Mode of the Machining levels?

The **Turbo Mode** option is available for selection on the Modify cutting conditions tab of the Technology Wizard page.

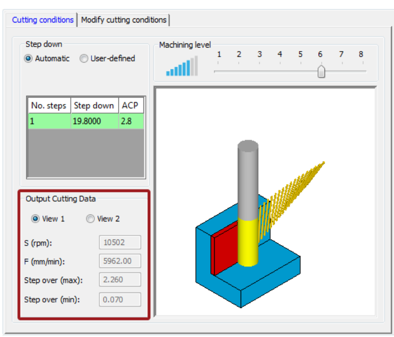


If you select this option, all the levels of the Machining level slider become more aggressive to the extent that the MRR of each level is about **25% higher** than before.

This means that the MRR of level 5 Turbo is about 25% more than the MRR of level 5 Normal, and so on. This option was added for customers who need a higher MRR than the MRR of level 8.

However, since the Cutting conditions are constrained by the machine's limitations, (e.g., the Wizard cannot set the feed or spindle speed higher than the maximum capable by the machine) it is not always possible to increase the MRR by simply increasing the feed or spindle speed (for example, that of level 7) by 25%. In such cases, the Wizard may have to go back and change other parameters (e.g., the maximum engagement angle) to be able to reach the desired 25% increase.

For these reasons, it is not always easy to understand the logic of the changes in the values you see displayed in the Output Cutting Data. Do not be concerned however, the Wizard will make sure the end result is as close to what you asked for as possible by your machine.



Why does iMachining need Channels and Moats?

Channels and Moats are unique features of iMachining. They are designed to enable the Tool path generator (Pgen) to divide the area of a pocket into sub-areas in such a way that most of the total area can be removed using iMachining's unique morphing spirals, rather than with trochoidal-like tool paths, thus reducing the cycle time and extending tool life.

Channels are cut using small trochoidal-like tool paths to produce constant width slots, along strategic routes determined by special topology analysis algorithms. Channels are open at both ends, allowing the tool free passage.

Moats are a special subset of channels and are cut around islands, whenever a spiral or trochoidal-like tool path hits an island. This unique feature of iMachining makes it possible to start a new morphing spiral, by allowing the tool free passage around the island, separating it from the remaining areas that still need removal.

Figure 4: The Effect of the Moat

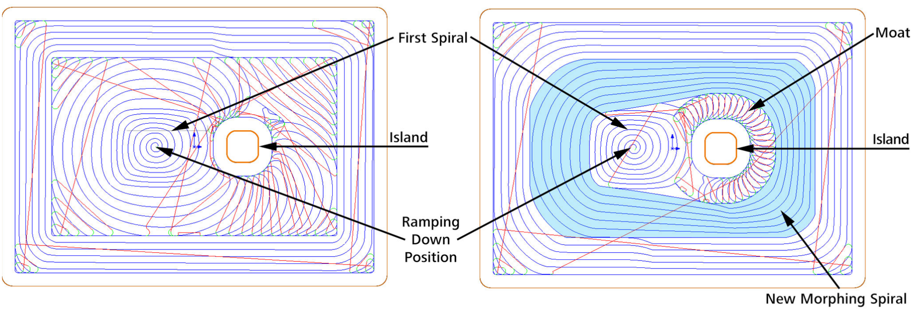


Figure 4A Time: 3:05 *(without Moat)*
The area around the first spiral and the island has to be cleared by trochoidal tool paths

Figure 4B Time: 2:52 *(with Moat)*
Cutting the moat enables iMachining to generate a new morphing spiral around the combined area of the first spiral and the island with the moat

Channels

Besides the cutting of Moats, **Channels** are used to enable the use of spirals in cases where spirals cannot ordinarily be used.

Shown below are a few examples:

Example 1

The open pocket below has an aspect ratio (ratio of length to width of the smallest rectangular box that contains the pocket) of 2:1. In the drawing, the longest dimension is 200 mm (7.875in) and the shortest is 100 mm (3.95 in).

Even if the ratio between the maximum side step and the minimum side step enables the construction of a morphing spiral that can clear the entire pocket area (see **Figure 5A** below), the cost in cycle time could be very high (e.g., if the machine's maximum feed was not high enough to compensate for the use of half the maximum side step for most of the tool path). In such cases, iMachining cuts one or more channels that cut the pocket into two or more manageable areas (**Figure 5B**), with the result that it now needs to clear two or more pockets.

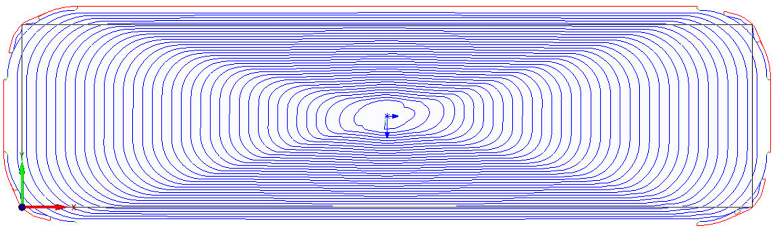


Figure 5A - Time 5:04 and tool wear at this extreme morphing is higher

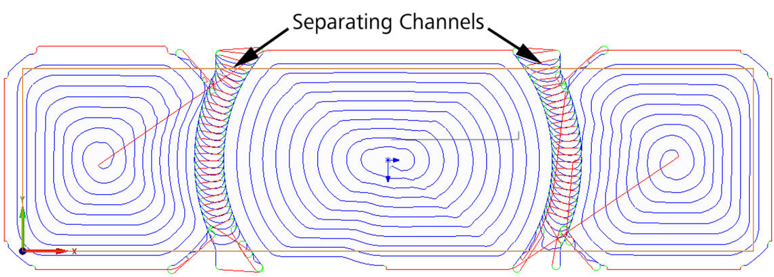


Figure 5B - Time 3:40 and tool wear is lower

These pockets can now be cleared with maximum possible **MRR**, with the only penalty being the time to cut the separating channels.

Example 2

The semi-open pocket below (**Figure 6**) cannot be cleared with a spiral. However, iMachining calculates the time it would take to separate the pocket area from the closed edge (wall) at the top, using a separating channel, and the time it would take to clear the separated area (the now open pocket) with a single spiral. iMachining then compares the sum of these times to the time it would take to clear the original pocket area using trochoidal-like tool paths. If the separation plus spiral is shorter than the trochoidal-like tool path, iMachining will separate as described (**Figure 6B**).

Figure 6: Additional Uses for Separating Channels

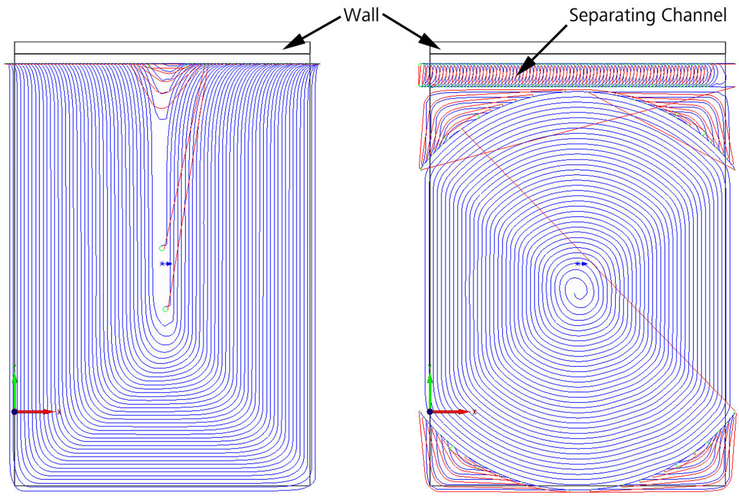
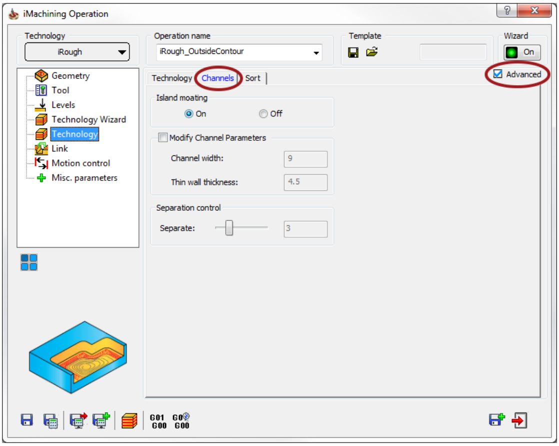


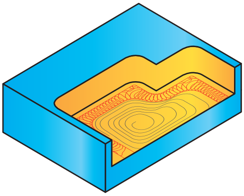
Figure 6A: Time 5:20
No separating channel - Semi open pocket completely cleared by Trochoidal Paths

Figure 6B: Time 4:32
After separating from wall it becomes an open area and is then mostly cleared by spiral paths

When the **Channels** tab is opened on the Technology page and the **Advanced** check box is enabled, you can see that there are a number of parameters that let you control the behavior of the **Channels** feature:

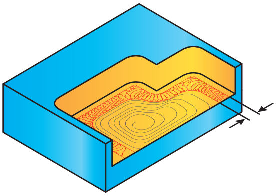


1. **Island moating** – This option is activated only if **On** is chosen.



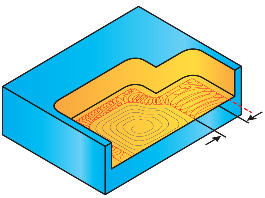
2. **Modify Channel Parameters** – When you select this check box, you can modify the values in the following fields:

   2.1 **Channel width** – This is the width of all channels cut in the current operation. It is the width between centers (of the tool). The default value is automatically set equal to the tool diameter, which results in a channel with a physical width of twice the tool diameter.



In most situations, you should not change this value unless you have special reason. As mentioned above, all separating channels are open at both ends (otherwise they will not separate), which means that towards the end of cutting them, the tool breaks out from material into air. If you increase the **Channel width** beyond the default of tool diameter, the front through which the tool breaks into air gets longer. In soft materials, this is not a problem, but in hard materials this could break the tool, as the front gets thinner and thinner. For more information, see below.

2.2 **Thin wall thickness** – Sometimes, during the machining of a pocket, temporary thin walls are left behind (created), only to be subsequently removed. These thin walls must be addressed carefully, otherwise they can cause vibrations, excessive tool wear and even tool breakage. This is especially true when cutting hard metals.



Every time your tool path breaks out from material into air, there is a transit situation of a thin wall which is subsequently removed. An example of this was shown above with separating channels.

Example 3

Another example illustrates what happens when iMachining decides to cut a moat or a channel near an open edge of a pocket.

Figure 7: Maximum Thin Wall Thickness

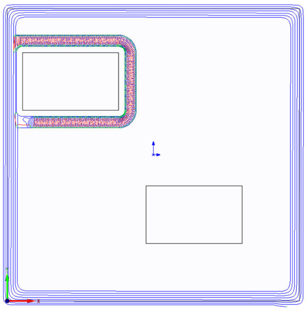


Figure 7A

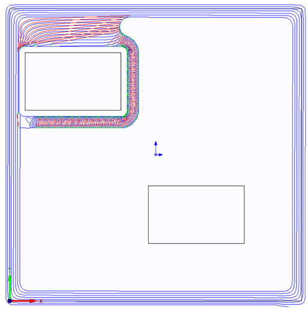


Figure 7B

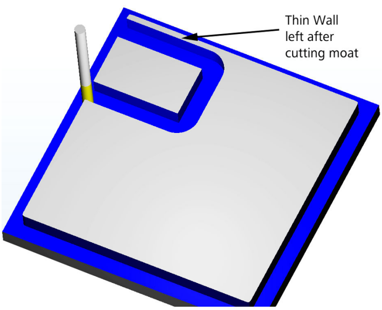


Figure 7C - Thin Wall thickness set to: 2mm

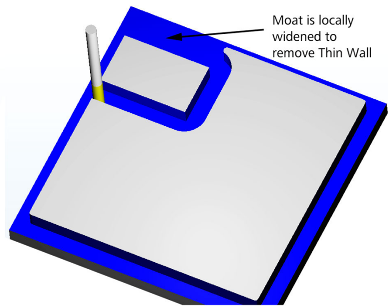


Figure 7D - Thin Wall thickness set to: 4mm

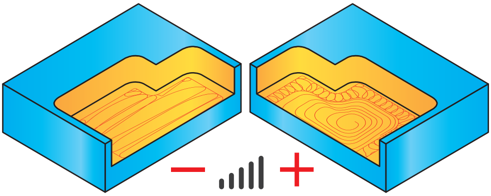
In such cases, the channel or moat will leave a thin wall that will later need to be removed.

iMachining, which always looks ahead, recognizes these cases and automatically widens the channel locally, so as to prevent the formation of a thin wall.

The system sets a default value for the maximum **Thin wall thickness** that should still be considered dangerous and which should be prevented (and likewise any wall which is thinner than that maximum) by this local widening.

If you consider the default value too small, which means that in your opinion even a slightly thicker thin wall may still be dangerous and should be prevented, by all means increase this value. However, we recommend you do not reduce the value below the default.

3. **Separation control**



As already explained above, iMachining uses channels to separate areas, which cannot be completely cleared by a spiral, and also to divide areas of large aspect ratios in two, in order to make it possible to clear them (or in the latter case, to clear them efficiently) by spirals.

The decision whether to separate or not is made on the basis of efficiency. However, to decide whether it is more efficient to separate or not, iMachining needs to be able to calculate the machining time for each alternative method and compare the times. Currently, iMachining has no knowledge of the maximum acceleration of a machine's axes. For this reason we have provided a special slider in the **Channels** page, called **Separate**, that the user can use to inform iMachining to lean more in favor of separation or less.

Moving the slider to the right (higher separation factor) will result in more separations. It works in such a way as if the higher separation factor informs iMachining that the machine can accelerate faster than an average machine, and therefore that the channel machining will take less time than on an average machine. Most users have no need to move this slider from its default value.

Only users with especially high or especially low acceleration machines may find they should move it.

The Technology Wizard produces on-the-fly Cutting conditions

The first and most basic parameter of the **Cutting conditions** is the cutting speed (**Vc**). After deciding on **Vc**, the spindle speed (**S** (measured in RPM)) can be calculated easily if you know the diameter (**D**) of the end mill ($S = Vc/\pi D$). Once you have **Vc** and **D** and **S**, the next decision is the chip thickness (**CT**), which is limited by the maximum spindle power available and the quality, strength and rigidity of the tool. Usually, the tool manufacturers publish for each tool the recommended maximum **CT** for each material type. When the **CT** has been decided, the feed can be directly calculated.

The important question is, how do you decide on **Vc**? The surprising fact is that, contrary to popular belief, *there is no such thing as the right cutting speed for a given material*. At least, not in High Speed Milling.

If you have a high quality tool with the right coating for the given material, a very rigid machine and set-up and very good cooling, you can cut the material at the maximum capability of the machine, if the tool paths are all tangential and the mechanical and thermal load on the tool is kept constant throughout.

For example, most tool manufacturers will recommend cutting Ti – 6Al – 4V at 50-60 meters per minute. With iMachining tool paths, using a good, rigid, fast machine and a suitable tool and good cooling, we succeeded in cutting Ti – 6Al – 4V at 250 meter per minute!

What does it mean? It means that with good tool paths, a good tool, machine and set-up along with good cooling, you can cut every material at very high speeds, much higher than most experienced professionals believe.

We can cut at any cutting speed, but heat and vibration create problems.

If everything is perfect for example, we can cut Titanium at any speed from 50 meter/min to 500 meter/min.

When cutting speed increases, vibration, heat and required spindle power increases. When vibration increases, tool wear increases and the tool will finally break. When heat increases, the tool will melt and break. When the required spindle power increases, it will finally exceed the maximum machine spindle power.

So what is limiting the cutting speed for a given material?

The answer is temperature and vibrations. If we have a very good tool with suitable coating for the given material and very good cooling, the temperature rise can be limited to a bearable value even at cutting speeds which are 5 to 7 times greater than the normally recommended speed. The remaining factor which will limit the cutting speed (and the chip thickness and feed) is vibrations. Vibrations cause shock loads on the cutting edge, which quickly starts to break. This means that if we want to cut very fast, we have to make sure there are no vibrations.

Limiting the temperature and vibrations is more difficult in hard-to-machine materials like Hardened Steel, Stainless Steel, Titanium, Inconel, Hastaloy, Wespaloy, etc. By definition, their resistance to cutting is higher, causing more bending of the tool which quickly brings the onset of vibrations, and also causing more heat to be produced by the friction and the plastic deformation of the chip. Also, at higher speeds, chip thickness and feeds, the resistance to cutting is higher, causing more heat and vibrations.

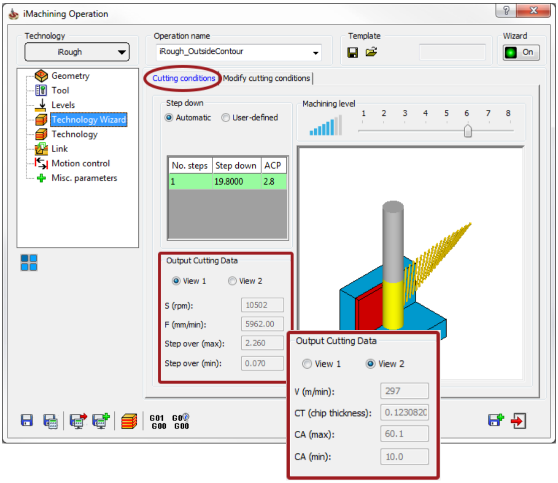
How can you make sure that you will not have vibrations?

1. If your machine is very rigid (good construction, relatively new and well-maintained), and you are careful to have a very rigid work holding arrangement, and you use a good tool holder and the tool is well balanced in the holder and is running true (central), then you have no reason for concern and can cut very fast (level 8 Turbo).
2. If you cannot provide *all* the above, you will have to use a lower Machining level, which will depend on *the state of the machine* and *the set-up*.

In iMachining, the Technology Wizard calculates 16 sets of **Cutting conditions** combinations, all of them suitable for cutting the given material with the selected tool on a perfect machine with a perfect set-up.

The 16 sets of **Cutting conditions** combinations are the Normal mode of levels 1~8 and the Turbo Mode of levels 1~8.

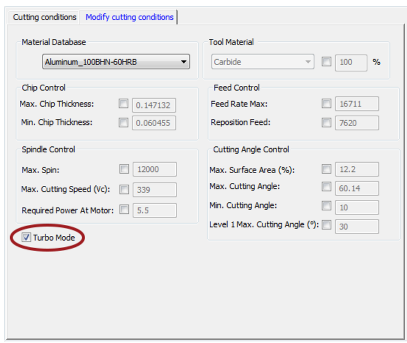
The **Cutting conditions** are a package of min and max engagement angles (CA), max feed (F) and spin (S), Step over range, chip thickness (CT) and so on.



Level 1 of the Turbo Mode is not the next level after level 8 Normal.

For every Normal level (1-8), the corresponding Turbo level has 25% more MRR than the Normal level.

If you want more MRR than you get with level 8 Normal, you should use the Turbo Mode. If everything is perfect (machine, tool, work and tool holding and cooling), it is possible to use level 8 Turbo.



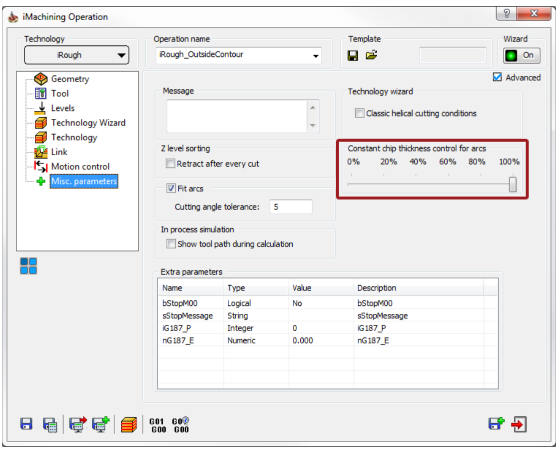
Cooling is just as important as selecting the correct Machining level. Always arrange *perfect* cooling. When level 8 Turbo is used, we can cut very fast but heat becomes a problem. And so, more cooling is necessary.

Controlling chip thickness to achieve even faster cycle times

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.

Accordingly, the developers added a slider called **Constant chip thickness control for arcs**. This 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.

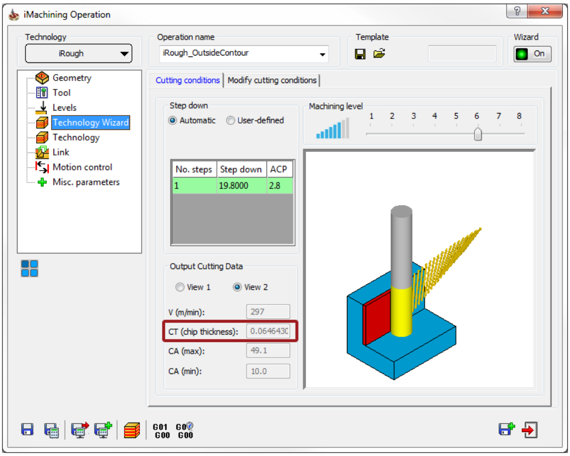


Moving the slider to **0%** informs iMachining to maintain a consistent feed rate between cutting in a straight line (G1) and cutting in a corner (G2). The result is faster cycle times; *but beware, with increased CT in corners comes increased tool load.*

What is the difference between chip thickness and chip load?

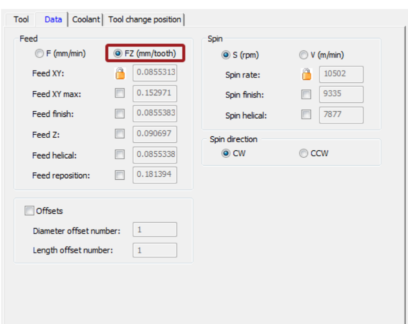
Chip thickness

In iMachining, chip thickness (CT) can be described as the thickness (at the widest point) of the undeformed chip that is sheared away from the material. On the Technology Wizard page, the CT value for the operation appears in View 2 of the Output Cutting Data.



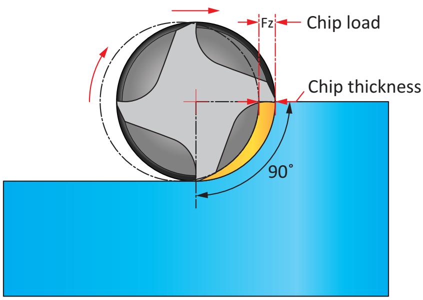
Chip load

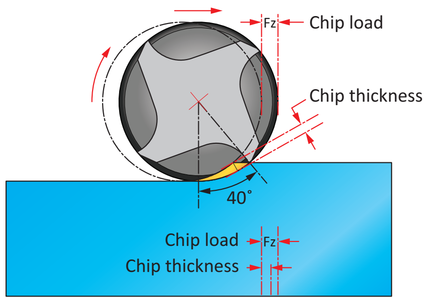
Not to be confused with chip thickness, chip load defines the feed rate of the tool, measured in units of distance per tooth (Fz (mm/tooth) in Metric units or Fz (inch/tooth) in English units). The Fz values can be viewed on the Data tab of the Tool page.



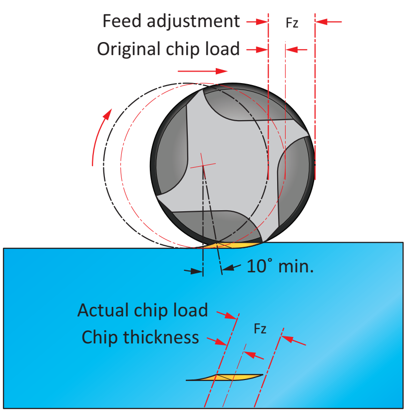
Relationship between chip thickness and chip load

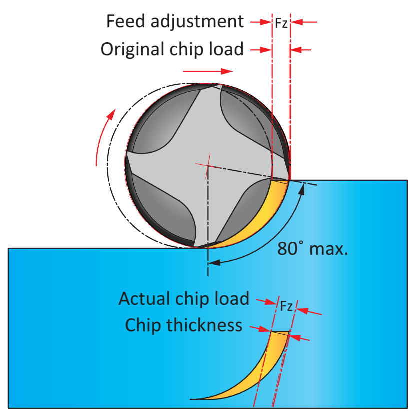
Traditionally, the only instance where you would see equal values of chip thickness and chip load is when the cutting angle is at 90º. Anything less than 90º will inherently produce a chip that is thinner than the specified chip load, resulting in uncontrolled tool load variations.





In iMachining calculations however, morphing spiral tool paths are generated according to a dynamically varying cutting angle that can range anywhere between 10º and 80º depending on the selected Machining level. To compensate for the dynamically varying cutting angle, the Technology Wizard automatically increases and adjusts the feed at every point along the tool path in order to maintain the specified chip thickness, which is matched to the original chip load as shown below.





By the automatic synchronization of these and other values, the Cutting conditions always remain optimal to achieve a constant load on the tool.

How can I judge the quality of a cut by the Sound, Look and Feel of it?

Good cutting is cutting with no vibrations. So if the sound gives you a sensation of vibrations, it means that you have vibrations.

If you hear a shrieking sound, like a train putting on its breaks, it is not good cutting. It could be that the tool and workpiece are heating up (cooling not perfect or level too high) or your tool is chipped.

If you see red hot sparks flying out of the cutting zone, it means the chips are too hot, and the level is too high or cooling is not good.

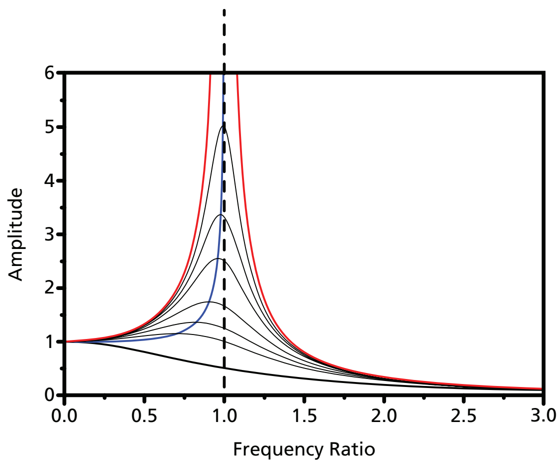
At the end of cutting the part, the part should not be hot. In good High Speed Milling, most of the heat should be absorbed by the chips and removed from the cutting zone as they fly out. So, when the Cutting conditions and cooling are good, the part is quite cool at the end of the cut.

What causes Vibrations and how does iMachining help me avoid them?

Reaction of cutting force is transmitted to the tool and from there to the machine. If the machine and set-up are not rigid enough, vibrations will start. If you keep increasing feed and spindle rotation speed, eventually the reaction becomes vibration.

Sometimes, the vibration makes resonance with the machine natural frequency, and the vibration becomes stronger.

Resonance in Milling



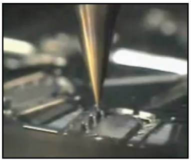
The amplitude of vibrations increases as the driving frequency approaches the resonant frequency of the machine. The driving frequency is that of the tool's flutes entering the material. An end mill with 4 flutes rotating at 600 RPM, enters the material 2400 times a minute, which translates to a driving frequency of 40 (2400/60) oscillations per second (Hz).

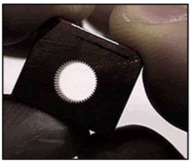
If the natural frequency of the machine is around 40 Hz, the above milling action will cause the machine to resonate, and the result will be strong vibrations.

In such a situation of resonance, it is sometimes possible to get out of the resonance frequency range by *increasing the Machining level*. *Increasing the depth of cut* may also help. Of course, *decreasing the level* is also an option.

How does iMachining perform in a micro-machining environment?

For very small tools, the Technology Wizard will generate small engagement angles (CA) and very small chip thickness (CT), so the *Cutting conditions will be suitable*.





www.makino.com

Usually with micro-machining, the most important requirement is the accuracy and surface finish, not speed or cycle time. Therefore, select a Machining level which is 2 or 3 levels below the usual.

What exactly is iMachining 3D?

iMachining 3D is the newest and most complete addition to the growing iMachining product family from InventorCAM. It is an automatic high speed milling CNC program generator for optimal roughing, rest machining and semi-finishing of molds, complex 3D parts and 3D prismatic parts.

iMachining 3D uses the 3D Model of the initial stock material and of the desired target geometry of the part, as input. It produces an optimal high speed milling CNC program that removes all the material that needs to be removed and that can be removed by the selected tool, and also produces an Updated Stock model (USM), reflecting all material removal by the operation, as output.

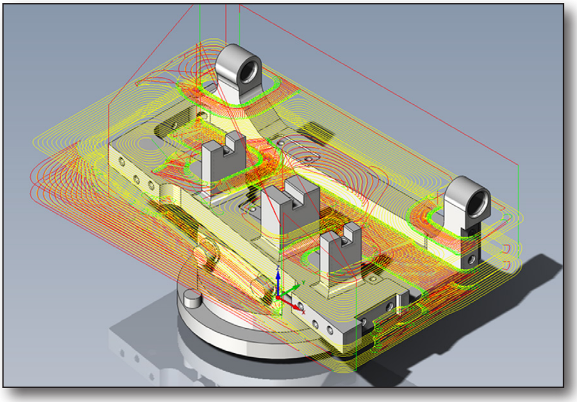
By default, iMachining 3D works in the mode of **Cut only the Rest material**, which enables it to utilize the USM from the previous operation, or the 3D Model of a casting or a forged part, as input for the starting Stock model of the current operation. During calculation of the tool path, this Stock model is dynamically updated by each cutting move, and thus reflects the exact shape of the remaining stock at every stage of the machining process.

By limiting the tool path to only the rest material, iMachining 3D guarantees that *no time is wasted on air cutting* of volumes previously removed or volumes that were empty to begin with (e.g., 3D Model of a casting or a forged part).

What makes iMachining 3D so unique?

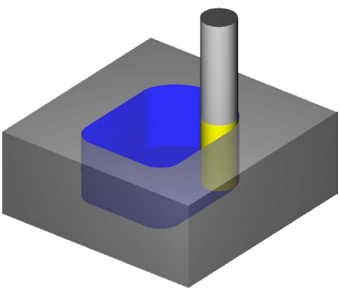
iMachining 3D is unique for several reasons:

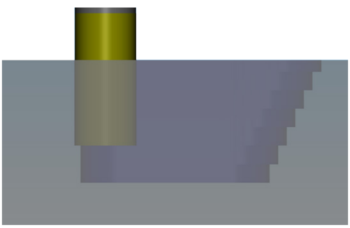
1. iMachining 3D *uses proven algorithms of iMachining 2D* to generate cutting tool paths at different Z-levels, by analyzing and determining which volume to remove next, at what Z-level. Doing so, iMachining 3D is able to achieve the shortest possible cycle time for the complete operation.



2. Using *intelligent localized machining*, roughing tool paths are generated with deep Step down passes in the first isolated milling region. After achieving the final reachable depth (by the current tool) of the current region, rest roughing tool paths are then generated in Step-up mode to remove all rest material on sloped surfaces of general shaped 3D parts or on higher horizontal surfaces of 3D prismatic parts.

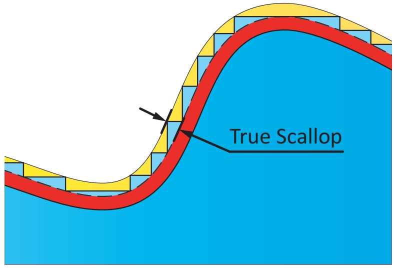
   In some cases, the isolated region may merge with a larger region requiring further deep Step down milling after rest roughing its upper layer (e.g., a wide pocket which splits into two or more, deeper smaller pockets at the bottom).



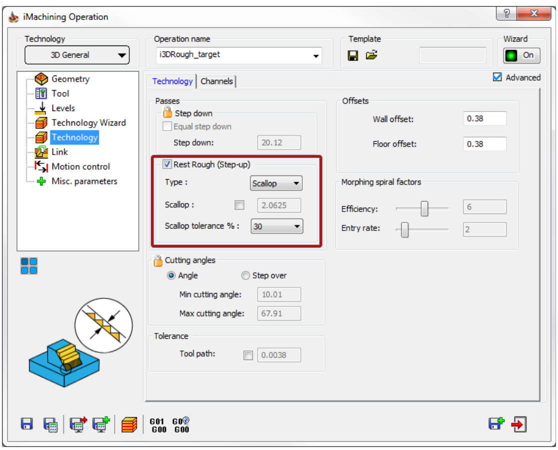


This sequence of Step down and Step-up is repeated for region after region until the last one is complete. It was later introduced that the user can also choose the option of completing **All down steps then step-up**.

By default, the height of the steps during Step-up is automatically calculated based on the current tool diameter. According to the local slope of each individual surface, the height of these steps change dynamically in order to maintain the specified Scallop vale throughout the operation. Every scallop produced is therefore a **True Scallop**.



You also have the option to override the given value of **Scallop** with your own in the Rest Rough (Step-up) section on the Technology page of the iMachining Operation dialog box.

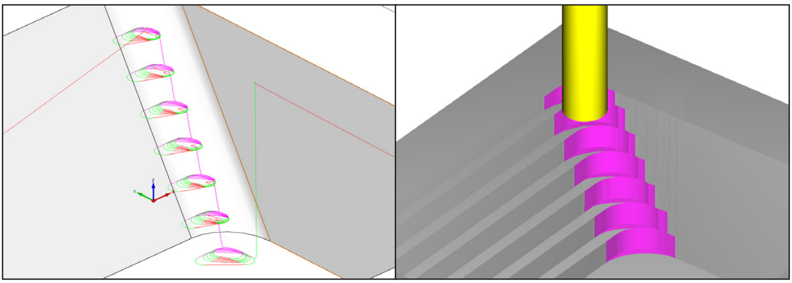


Another important parameter is the **Scallop tolerance**, which works in conjunction with the Scallop value. There are two optional values to choose from – **30%** (default selection) and **10%**. This tolerance is applied to the specified Scallop value. It enables iMachining 3D to join two steps on two adjoining slopes, which would otherwise be cut at slightly different Z-levels, and perform one long cut at the same Z-level.

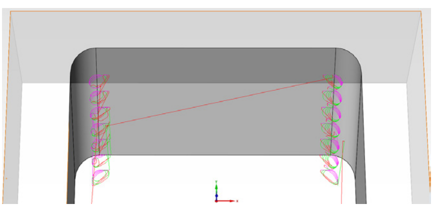
The effect of the default tolerance is to produce an actual scallop that may be up to 30% larger than the specified Scallop value. The total tool path length and cycle time will be appreciably shorter.

Using the override check box and specifying a smaller Scallop value will result in finer passes, which can be beneficial by helping to avoid semi-finishing operations. However, the calculation time and cycle time will be proportionately longer.

3. The next iMachining 3D operation will likely get the USM from the previous operation, and the same target geometry of the part, as well as be assigned a smaller diameter roughing tool to automatically remove material in areas not reachable by the previous larger tool.



Where appropriate, large down steps into pockets and bottle necks (too small for the previous tool) will be followed by smaller Step-up rest roughing tool paths. As always, knowledge of the updated shape of the remaining stock ensures that there will be no air cutting.



How is iMachining 3D different from iMachining 2D?

iMachining 3D produces a complete, ready to run CNC program with optimal Cutting conditions for roughing and rest roughing an entire 3D part, with True Scallop on all slopes, all in a single operation.

> **iMachining 2D** needs to be instructed how the milling of a part is to be broken down into separate pocket operations, and the order in which they have to be machined. In a single operation, **iMachining 2D** can only remove a single horizontal (thick or thin) prismatic slice of material.

iMachining 3D analyzes the Target model and automatically recognizes all its features and depths, without the need for chain picking or specifying feature information like Pocket depth, etc. All the volumes that need to be removed are then subdivided into milling regions. Roughing tool paths are generated in thick horizontal slices (Step down) followed by rest roughing tool paths in thin horizontal slices (Step-up) for each isolated milling region. Using sophisticated analysis, the optimal order of milling those slices is determined to achieve iMachining's unique **local machining** feature, which results in eliminating almost all retracts and long position moves.

> **iMachining 2D** needs every pocket to be defined separately by its depth and by chaining or by sketching its contours.

iMachining 3D acquires all its information about the Stock model – its extent, its shape, and information about material already removed by previous operations, from the USM.

> **iMachining 2D** requires that additional contours be sketched and defined for all stock material that lies outside the part geometry.

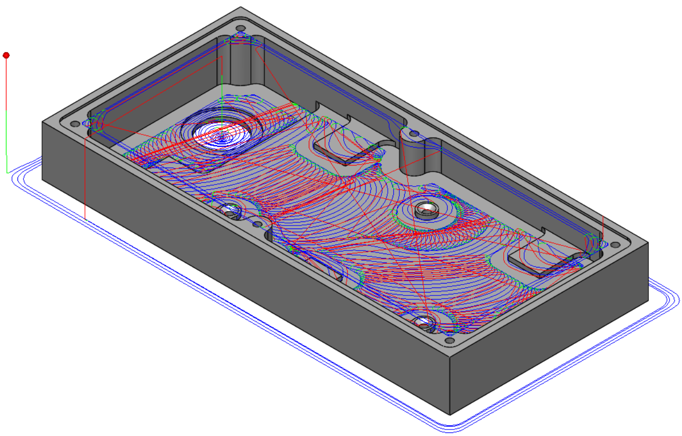
iMachining 3D automatically recognizes sloped surfaces and determines the optimal Step-up for each specific slope, for the next cut to produce the specified True Scallop.

> **iMachining 2D** does not deal with slopes; it only understands plane and vertically ruled surface geometries.

Can iMachining 3D automatically mill prismatic parts?

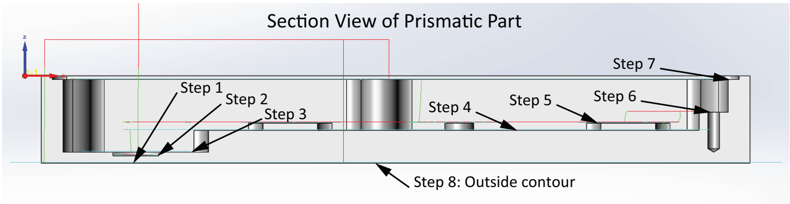
Yes, indeed! With iMachining 3D, *you can perform the roughing and rest roughing of an entire prismatic part that includes hundreds of pockets and islands in a single operation*, without chaining or sketching a single contour. All that's needed is the solid model of the target and solid model of the stock. Let the intelligence of iMachining 3D do the rest, automatically and optimally.

The example below illustrates a typical iMachining 3D Prismatic tool path.



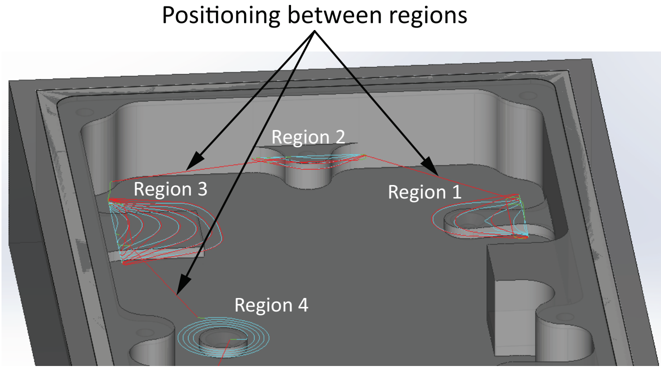
When machining 3D prismatic parts, performance and efficiency is automatically maximized to achieve the shortest possible cycle time. Using iMachining 3D over iMachining 2D provides the following four benefits:

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.

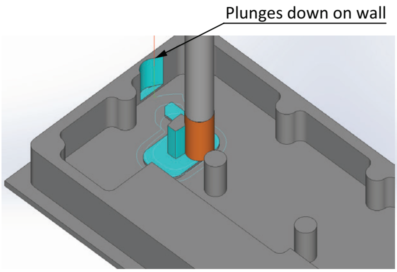


2. iMachining 3D performs intelligent sorting of 2D Z-level regions. Non-cutting moves are reduced by the 3D Z-level ordering and localized machining of 2D tool path regions.

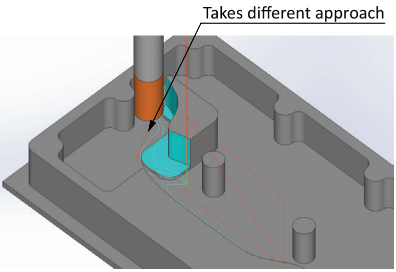
3. iMachining 3D performs smart positioning between 2D Z-level regions. Long position moves are reduced by the 3D Z-level linking and localized machining of 2D tool path regions.



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



iMachining 2D with large tool



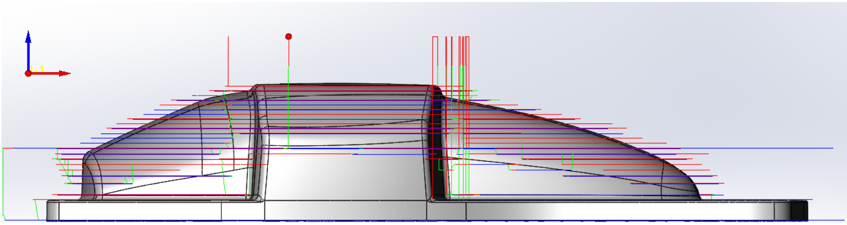
iMachining 3D with large tool

How fast is iMachining 3D relative to other 3D systems?

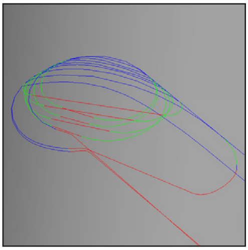
iMachining 3D is faster than other 3D high speed milling systems *by a larger factor* than iMachining 2D is compared to other 2D high speed milling systems.

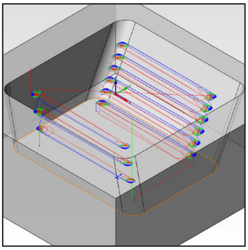
iMachining 2D creates CNC programs that complete the machining of your parts in a cycle time which is up to 70% shorter than all other CAM systems. iMachining 3D cycle times are up to 90% shorter than all other CAM systems.

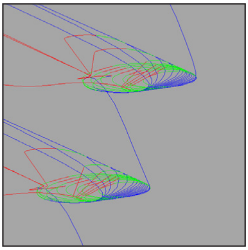
The reason being that since all iMachining 3D cutting moves are basically horizontal cutting moves generated by the algorithms of iMachining 2D and the Technology Wizard, iMachining 3D cuts material as fast as iMachining 2D.



Additionally, with its localized machining and optimal ordering of machining sequences, *iMachining 3D eliminates almost all retracts and long position moves and all air cutting*, which no other 3D high speed milling system can do.





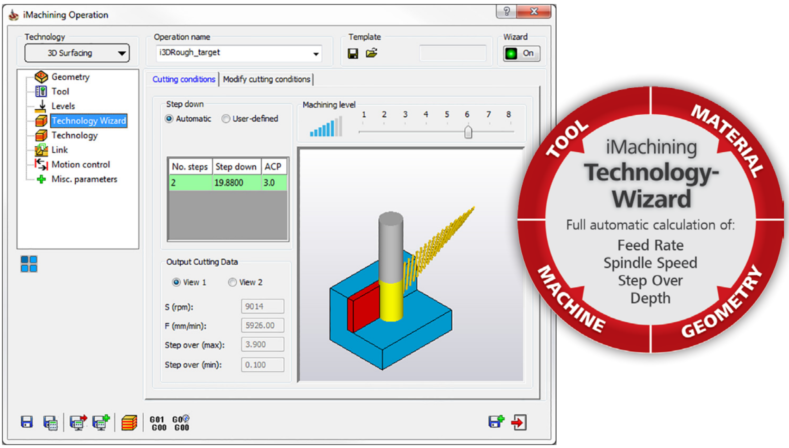


iMachining 3D, with its per slope scallop driven Step-up rest machining, also generates milling paths that remove only the minimum amount of material on slopes that is necessary to produce the specified True Scallop, without wasting time on surfaces that don't require cutting at the current Z-Level. In principle, this means that the cycle time in *iMachining 3D is almost pure minimum-cutting time*.

What are the advantages of iMachining 3D over other 3D high speed milling systems in addition to those described above?

There are several more advantages:

1. A major additional advantage is the proven Technology Wizard, which *automatically calculates the optimal Cutting conditions for each cutting path separately*, making for effortless *first part success* every time. This saves a lot of programming time and cycle time, tools and material, otherwise spent on long expensive trial and error work required with other 3D systems to achieve a reasonably efficient tool path.



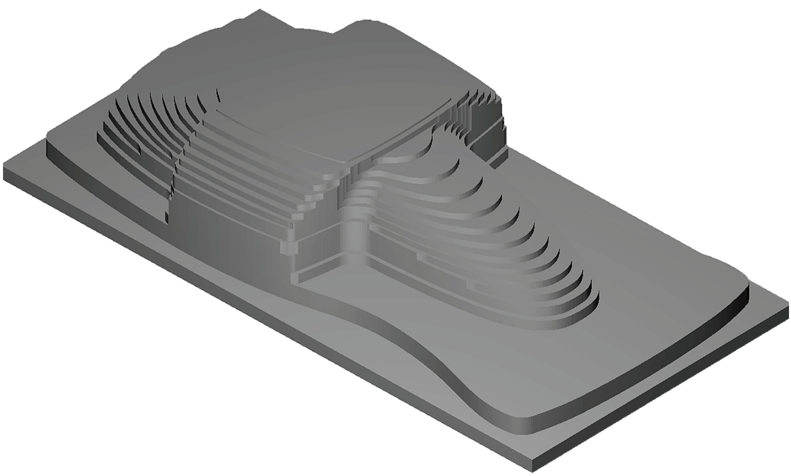
2. A very important additional money saving advantage of iMachining 3D over other 3D high speed milling systems is its **minimum machining** feature on sloped surfaces. This feature restricts the rest roughing tool paths at any Z-Level to only cut material that, if left uncut, would exceed the maximum True Scallop specified by the user.

All other 3D high speed milling systems cut steps on slopes, even in situations that do not require any cutting on those slopes, to stay within the specified Scallop value (in 3D systems that allow the user to specify a Scallop value – most 3D systems only let the user specify a constant step-up amount).

This unique **minimum machining** feature therefore results in:

- *Reduced total cutting path length during Step-up*, resulting in *reduced tool and machine wear and further reduced cycle time*.
- *A much more even amount of material left on the slopes*, making the subsequent finishing operations (using the HSM module) able to run faster with smaller tool load variations, and therefore a *further reduction in cycle time and tool and machine wear*.

3. During the Step-up procedure, the axial depth of cut gets smaller every time a new higher step is machined. iMachining, using the functionality of the Technology Wizard, increases the feed rate and engagement angle of the tool paths that machine the higher step, by the exact amount required to maintain the Wizard specified constant load on the tool, as it cuts the smaller depth. As a result, the machining time of that step is shorter than it would have been without the feed and angle increase.



How do I avoid mistakes that may shorten tool life?

iMachining produces complete (i.e., including all Cutting conditions) CNC programs for *fast* and *safe* milling of your part on your chosen machine with *first part success* and longer tool life than what you would get with traditional machining technologies. So, how can it be possible that a tool breaks or prematurely wears out?

The answer typically appears in some *mismatch* between your inputs and the realities of the machining environment. Here is a list of possible mismatches:

The dimensions or location of the stock material are not the same as those defined in the CAM-Part Definition

Case Study: This actually happened to a very experienced InventorCAM customer, who mistakenly used a piece of stock leftover from a previous batch of the same part. In this previous batch, the stock was defined and prepared with slightly larger dimensions than the new batch.

The result: The tool broke at the second detour rapid repositioning move on attempting to move through the excess material, where air should have been.

The wrong tool was allocated to the tool magazine pocket

This situation is familiar to many users.

The wrong material was selected from the Material Database or the material entry had the wrong value of UTS

Case Study: This situation happened to a potential customer who invited us to do a live test cut in his workshop. The mission was to cut a part made from Titanium. Everything was set up correctly, but the tool turned red after 2 minutes in the cut and broke. After investigation, it turned out that the material selected from the Material Database was Titanium, which is a pure unalloyed form of the metal, and has an UTS of 220 MPa. The stock material on the machine was the aerospace alloy Ti – 6Al – 4V, used globally for aircraft structural parts, and has an UTS of 1170 MPa. For more information, see *What are the important Stock Material properties?*.

The wrong Machining level was set in relation to the state of the machine, rigidity of the workpiece holding and TIR of the tool

Case Study: The test cutting of a Ti – 6Al – 4V part was performed at an InventorCAM customer's workshop. The machine was in poor condition, so a default Machining level of 5 was chosen. The TIR was 0.02 mm (0.0008 in) which is too large for the 0.06 (0.0024) chip thickness the Technology Wizard suggested. However, the work holding seemed rigid enough, so finally a Machining level of 4 was selected on the slider. The start of the cut was a helical entry into a closed pocket, roughly at the center of the workpiece, down to a depth of 24 mm (1 in) with a 16 mm (0.625 in) 4 flute premium carbide end mill. After the entry, the tool started on a diverging spiral to clear out the pocket. At first everything looked and sounded okay, but as the spiral widened, the sound got worse and worse.

We immediately stopped the machine. There was intense vibration, and had we let the cutting continue, the tool would have completely worn after one part. Upon a closer examination, we realized that the workpiece holding was very problematic. The part was bolted onto a thick base plate held between an old removable horizontal 4th axis which was clamped onto the table, and a center, mounted on a tailstock. Apparently, the gearbox of the 4th axis was slightly worn and had some play in it. As long as the cutting was near the center of the part, directly in line with the 4th axis, everything was stable enough, but as soon as the tool started cutting some distance away from center, the play in the 4th axis gave way to the cutting force and strong vibrations set in.

The result: We reduced the Machining level to 2 and resumed cutting. This time there was almost no discernible vibration, and the tool lasted 8 parts. For more information, see *What is the role of the Machining level slider?*.

The Cutting conditions provided by the Technology Wizard were modified with erroneous or over-optimistic values with the Wizard turned off

When all the elements of the machining environment (machine, work holding, tool and tool holding) are in good condition (rigid, sharp, balanced and central), it is possible to work with a Machining level of 8 Turbo.

It is possible that a very experienced user knows they could go higher with the feed rate, cutting speed, chip thickness or the combination of all three. It may be that the user is using a special tool designed for bigger chip thickness or higher cutting speed. It may be that the user simply knows from experience that their material can be cut faster on their machine.

Whatever the case, *it is highly recommended to only modify the Cutting conditions with the Wizard turned on*. Provided the user is not exceeding the maximum power or maximum feed rate or spindle speed of the machine, the Wizard will allow you to input your preferred values. This is the safest way. It helps prevent mistakes that may happen when the Wizard is turned off, and cannot oversee your actions.

Coolant issues

Good cooling is critical in iMachining, as in all forms of high speed milling. Without it, the tool’s temperature rises uncontrollably, and the tool disintegrates quickly.

All hard-to-machine metals (e.g., Titanium alloys, Nickel alloys, Stainless Steels, etc.) should be cooled with a good, high pressure, high flow of an appropriate cooling emulsion. The coolant should be directed at the cutting portion of the tool from a steep angle and from at least three directions (four is ideal), to avoid the possibility of the stream being blocked by the geometry of the partially cut part. Using cooled emulsion will enable you to increase the MRR.

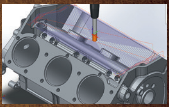
Other materials (even Hardened Mold Steels) should use air cooling, with good supply and high pressure, similarly directed. Higher MRR can be achieved using cooled air. A Venturi tube air cooler that costs less than US $500.00 will reduce the temperature of the workshop’s standard 6 atmosphere air supply by about 20 °C and enable you to increase the MRR by 20% or more.

InventorCAM 2015

iMachining **FAQ**

Frequently Asked Questions

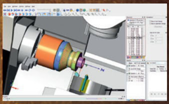
The complete range of manufacturing applications inside Autodesk Inventor



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