The new, revolutionary Milling technology

iMachining®
by SolidCAM

SOLIDWORKS Certified Gold Product

SolidCAM + SolidWorks

The complete integrated Manufacturing Solution

SolidCAM & iMachining F.A.Q.

by SolidCAM

Frequently Asked Questions

SolidCAM

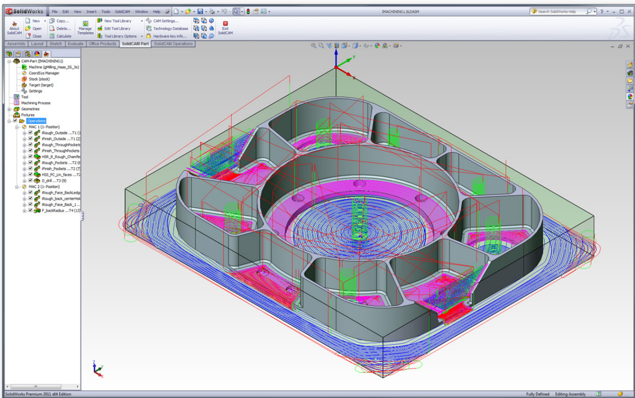
The Leaders in Integrated CAM

Contents

Document Version: 1.4

What is SolidCAM iMachining?

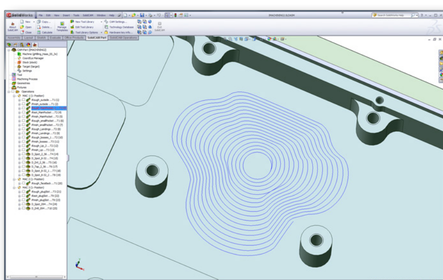
SolidCAM iMachining™ is an intelligent High Speed Machining CAM software, designed to produce fast and safe CNC programs to machine mechanical parts. The word **fast** here means significantly faster than traditional machining at its best. The word **safe** here means without the risk of breaking tools or subjecting the machine to excessive wear, whilst increasing tool life.

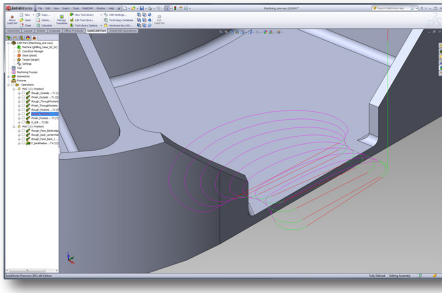


To achieve these goals, iMachining uses advanced, patent pending, algorithms to generate smooth tangent tool paths, coupled with matching conditions, that together keep the mechanical and thermal load on the tool constant, whilst cutting thin chips at high cutting speeds and deeper than standard cuts (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.



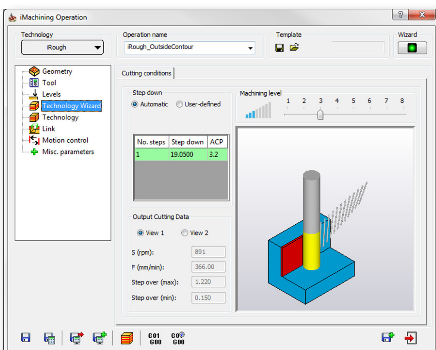


iMachining uses proprietary **Constant Load One-Way tool paths** to machine narrow passages, separating channels and tight corners. In some open areas, where the shape is too irregular to completely remove with a single spiral, it uses proprietary topology analysis algorithms and channels to subdivide 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.

The iMachining Technology Wizard

A significant part of the iMachining system is devoted to calculate matching values of Feed, Spindle Speed, Axial Depth of cut, Cutting Angle and (Undeformed) Chip Thickness, based on the mechanical properties of the workpiece and tool whilst keeping within the boundaries of the machine capabilities (Spindle Speed, Power, Rigidity and Maximum Feeds). 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 **Wizard** is dynamically adjusting the Feed to compensate for the dynamically varying cutting angle – a bi-product of the morphing spiral, thus achieving constant tool load, which increases tool life.



General

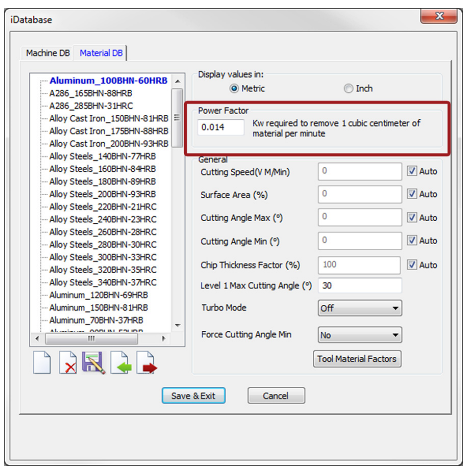
Different materials require different amounts of force to cut them. The physical property of a material that determines the force required for a particular cut is the Ultimate Tensile Strength (UTS), given in units of MPa (Mega Pascal) in metric units or psi (pound per square inch) in English units.

The iMachining Technology Wizard totally depends on the correct UTS value to produce good cutting conditions. That is why it is imperative to ensure that any material you decide to cut has the accurate UTS value assigned to it in the Materials Database.

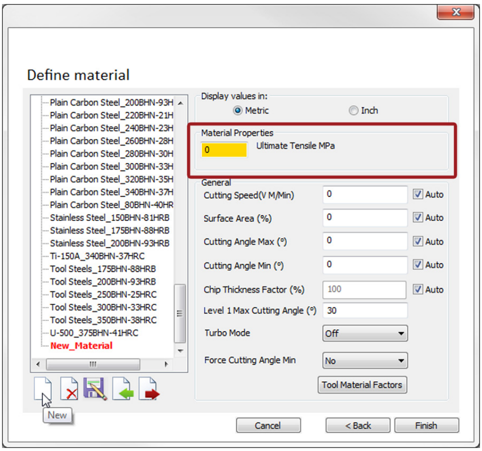
All SolidCAM versions are shipped with a basic Materials Database that contains around 70 different materials.

History

When the 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 one CC (Cubic Centimeter) of material per minute (in metric units of KW), or one 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 materials 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 materials tables based on Power Factors, 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 change the dialog box for defining a new material, so that it would only accept UTS for newly entered materials.

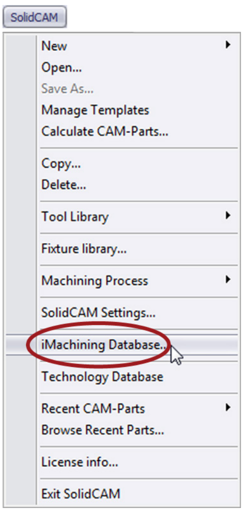


The current situation is that materials defined before 2011 are all defined in terms of their Power Factor rating, whereas all materials defined since then have been and will be defined in terms of their UTS.

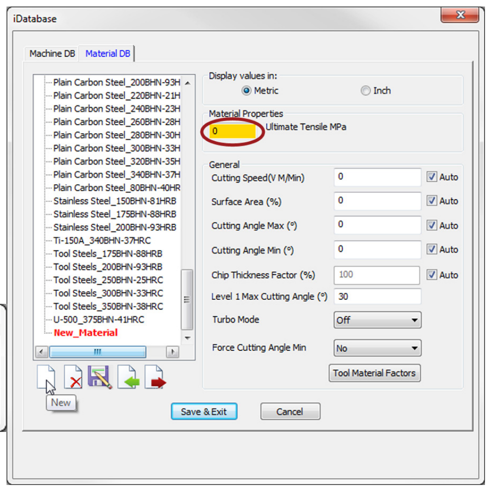
It should be clear that both methods of definition are equivalent and the Wizard produces the same efficient cutting conditions with either method.

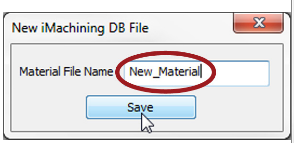
Defining new materials entries in the materials database

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 their database.



With the new Material Database editing dialog box and the use of UTS, it can be done quickly and easily. There are only two required inputs. The first 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.





How do I find the UTS value of a material?

1. Make sure you know the exact specification of your material

Case Study: A SolidCAM customer needed to cut a part out 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. They entered the value in the UTS field in the material editing dialog box and added this new material to the database. Then, they selected the newly entered material from the **Material Database** list in the **iMachining Data** section of the CAM-Part definition dialog box. They defined their iMachining operation, clicked **Save & Calculate**, generated the GCode, and started cutting. Their tool broke after 5 seconds in "the cut."

When they 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.

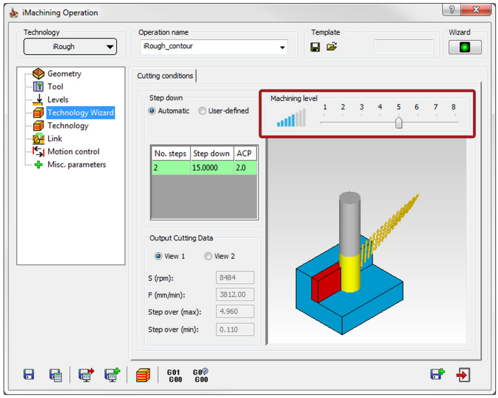
We advised the customer to search this specific material on MatWeb.com. They informed us there were six different entries of Ti – 6Al – 4V on MatWeb, ranging in UTS from 860 to 1170 MPa. The customer said they did not know which one was their material, and it was too late in the day to contact their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

2. If there are many entries to choose from, always start with the highest value of UTS

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

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.

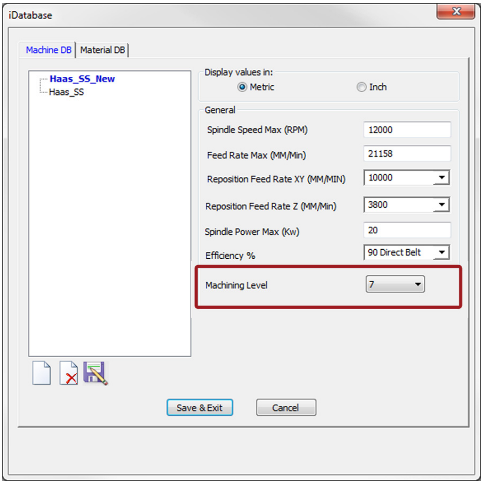
However, as we have seen above, there are factors that influence the attainable MRR and tool life (such as the basic rigidity of the machine, work and tool holding, and the machine's level of maintenance) as well as the desired compromise between high MRR and long tool life, influenced by your production schedule and cost structure that are difficult to accurately quantify. Instead, the Wizard provides the **Machining Level** slider, enabling 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 **Default Machining 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 **Default Machining Level** should only reflect the machine 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 **Default Machining Level** is defined only once and is stored in the machine database, together with all the other constant machine parameters (Maximum Feed Rate, Maximum Spindle Speed, 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 for generating 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.

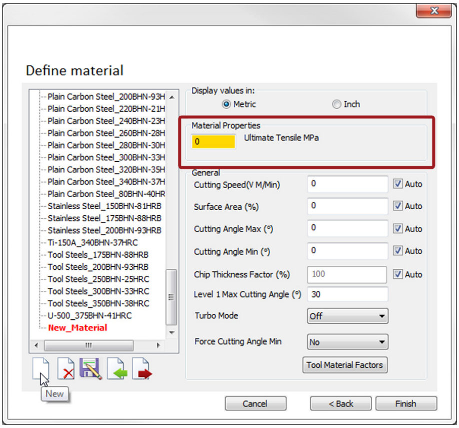
Use the resulting machining level to cut the first part. Listen to the sound of cutting and assess the resultant surface quality and tool wear. If there are more parts to cut, and the previous cut was good, you may want to increase the MRR or decrease it to get longer tool life, depending on your schedule, 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.

Remember:

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

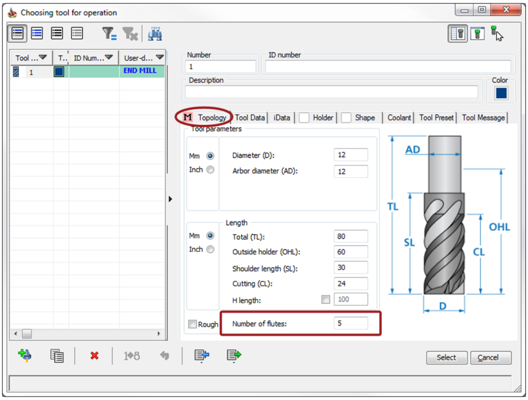
But beware, **the risk is real**. The **Default Machining Level** for the machine was set according to a **subjective assessment of its condition.** This assessment may be optimistic, and so might be the assessment of the work clamping and tool holding.

Material UTS



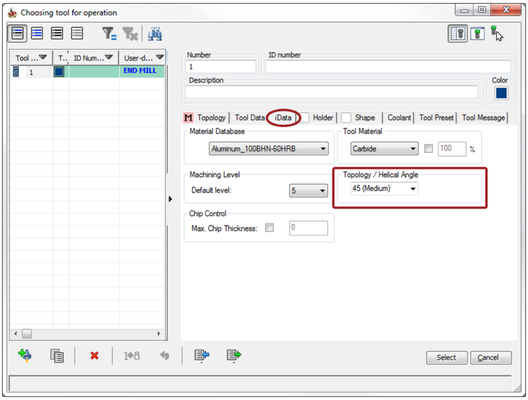
In the *What are the important Stock Material properties?* section, we have seen the importance of the UTS of a material. This is not a free parameter for the user to set a value to their liking, but it is worth mentioning to stress how dramatically it affects the cutting conditions and therefore how 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 helix angle of the flutes is in a class of its own. Changing the helix 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, which if ignored 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. It is a value calculated and displayed by the Wizard, reflecting the number of contact points the tool has with the vertical wall it is producing, along a vertical line.

If the depth of the cut is d, and the tool Diameter is D, and it has N flutes, and the flute helix angle is β, we can calculate the Pitch of the flute P as follows:

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

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

Fine pitch $p = P/N$

The ACP can now 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 help us to cut better?" The answer is simple:

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

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 the user whether or not the situation for stability is good based on ACPs. The output grid changes color to show the current situation: Red = Bad - High likelihood of vibrations; Yellow = Not so good - Medium likelihood of vibrations; Green = Good.

No. steps	Step down	ACP
30	1.0000	0.1

No. steps	Step down	ACP
2	15.0000	1.6

No. steps	Step down	ACP
2	15.0000	2.0

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

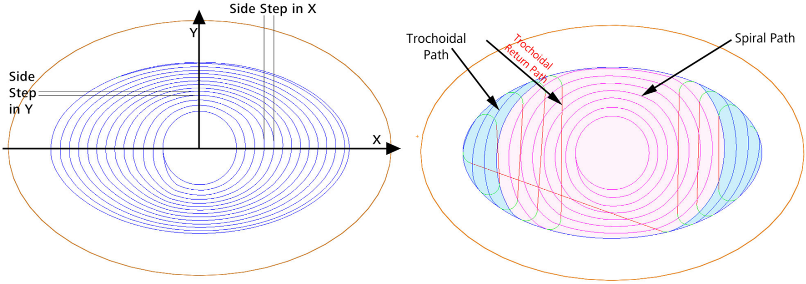


Figure 1A: Low efficiency setting:
Spiral with strong morphing clears complete area

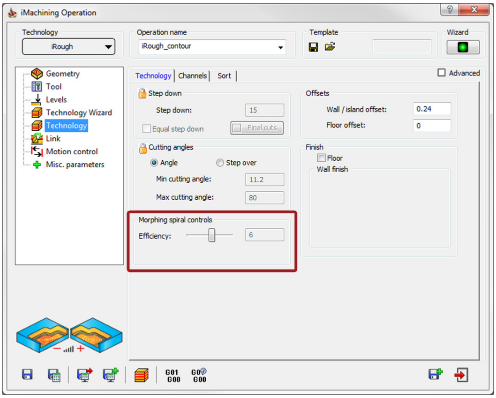
Figure 1B: High efficiency setting:
Spiral with little morphing is rounder and clears only part of area, the rest is cleared by a Trochoidal Path

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.

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 exists on the **Technology** page of the **iMachining Operation** dialog box, under the **Morphing spiral controls** section.



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) on the left with that of case (b) on the right of Figure 1, we notice that while the morphing spiral in (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.

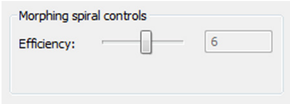
With the iMachining Feed adjustment, the cycle time for (a) is (80/94 = 0.851) 15% shorter than that of (b). This, however, is only the difference in efficiency for the simple convex shape in Figure 1.

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 we have seen above, 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.

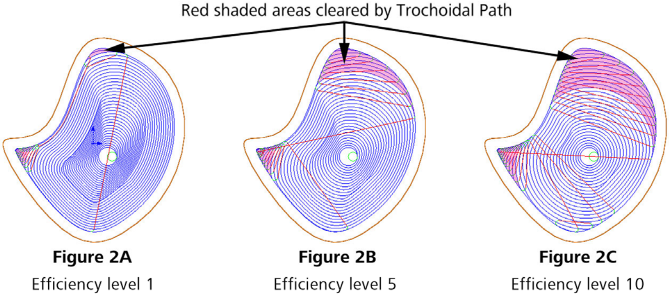


Figure 2: Same pocket cut with 3 different spiral efficiency settings

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 the 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: SolidCAM 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:

- Minimum cycle time (a short delivery deadline, or an expensive machine and a low cost tool)
- Maximum tool life (an expensive tool, or you are committed to deliver six parts by morning and you only have one tool in stock)
- 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)

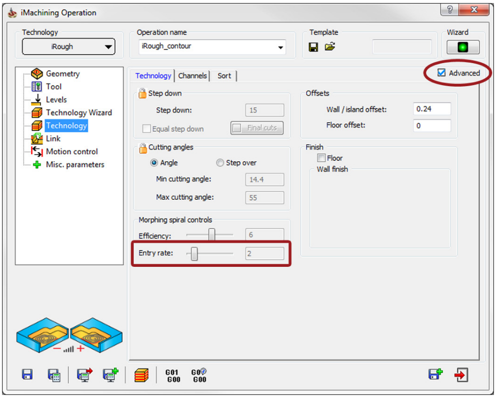
This option will be activated when a user selects the **Automatic** option for setting the **Spiral Efficiency**.

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

Note: When using the **Automatic** option of setting the spiral efficiency, the new algorithm calculates an efficiency 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 efficiency level calculated by the new algorithm according to the selected *priority*. However, in the **Manual** mode, the **Efficiency** level selected by the 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

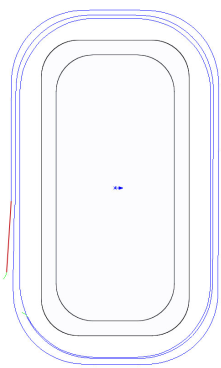


Figure 3A: Entry Rate Level 10

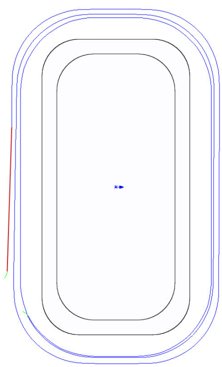


Figure 3B: Entry Rate Level 5

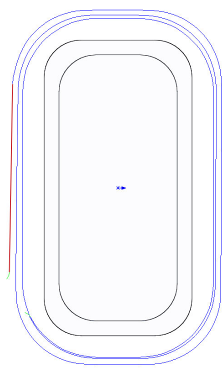


Figure 3C: Entry Rate Level 1

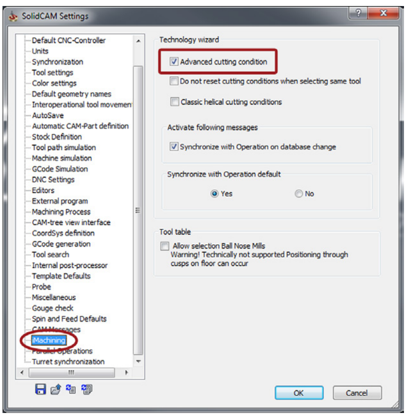
Although this **Entry rate** is automatically set by the **Technology Wizard** in accordance with the properties of the stock material, the developers decided for the sake of flexibility and user-friendliness, to provide users with the means to override this value. Moving the slider to the right increases the rate 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 - 5 units, calculate and simulate in the Host CAD mode to observe the new entry rate.

Advanced Mode

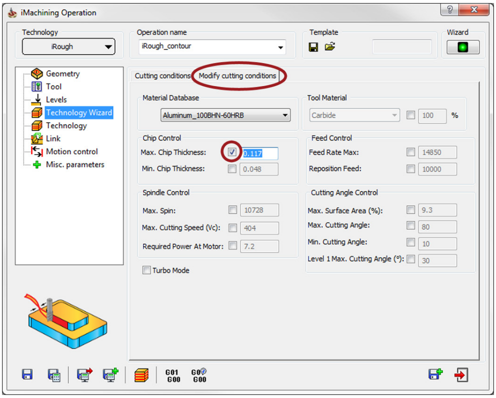
This is a special mode which provides users with additional flexibility and control options.

1. Open the **SolidCAM Settings -> iMachining** 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.**

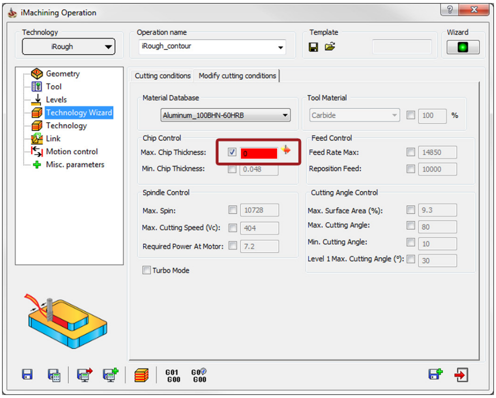
3. Switch to the **Modify cutting conditions** tab on the **Technology Wizard** page. You now have the option of modifying any one or more of the cutting conditions parameters. We strongly recommend using it only if manipulation of the **Machining level** slider does not produce your desired result.



4. On the **Modify cutting conditions** page, 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.

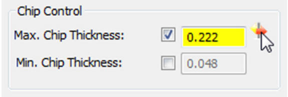
Note: The values appearing in the **Modify cutting conditions** page are always those corresponding to **Machining level** 8 (**Normal** or **Turbo**, whichever is the current mode). If you have chosen a level different from 8 on the **Machining level** slider, you will not get the value that you entered in the **Modify cutting conditions** page. In your chosen level, 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.

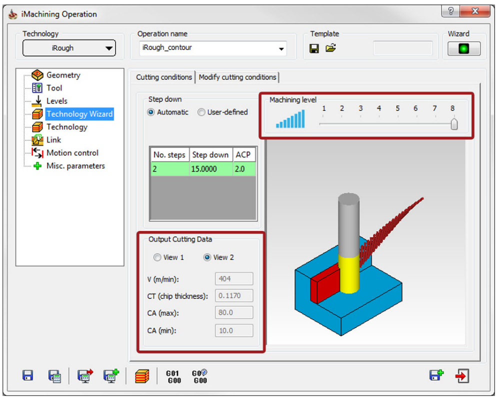
6. One simple way to adjust the values is to click the icon next to the field. The **Wizard** will calculate the nearest reconcilable value to the one you have set, and replace your value with the calculated one, while the field background color changes to yellow. When all values are adjusted, you can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



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.

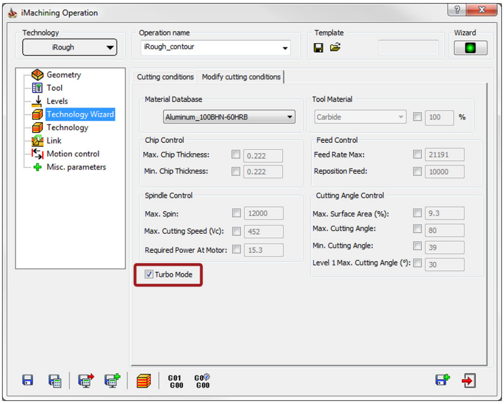


9. There is the path of least resistance, and the most risky option, of turning off the watchful eye of the **Wizard**. Click the green light at the upper right corner of the **iMachining Operation** dialog box.

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?

When the **Advanced** mode is active, a new **Turbo Mode** option appears under the **Modify cutting conditions** tab on the **Technology Wizard** page.

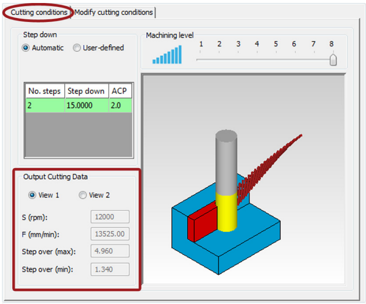


If you select this option, all the levels of the **Machining level** slider become more aggressive to the extent that the MRR of each level is about 25% higher than before.

This means that the MRR of level 5 turbo is about 25% more than the MRR of level 5, 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 on the **Cutting conditions** page. 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.



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 the tool life.

Channels are cut using small trochoidal-like 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

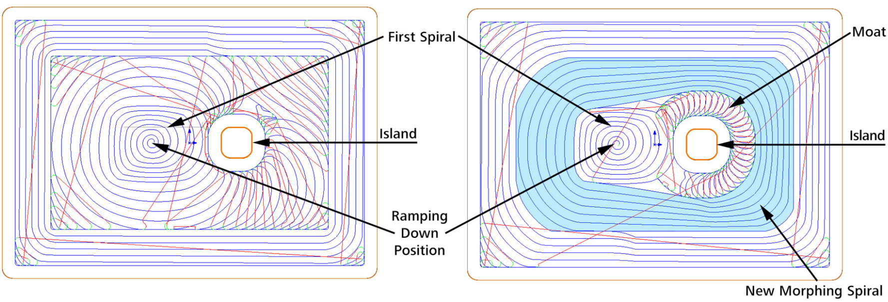


Figure 4A Time: 3:05 *(without Moat)*
The area around the first spiral and the island has to be cleared by trochoidal tool paths

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

Let us look at 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 and the shortest is 100 mm.

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.

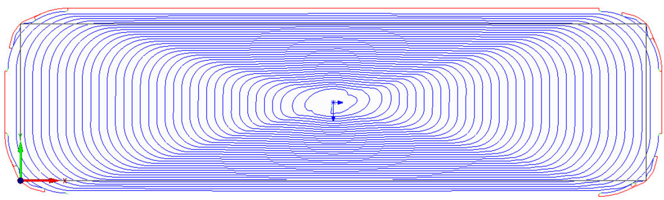


Figure 5A - Time 5:04 and tool wear at this extreme morphing is higher

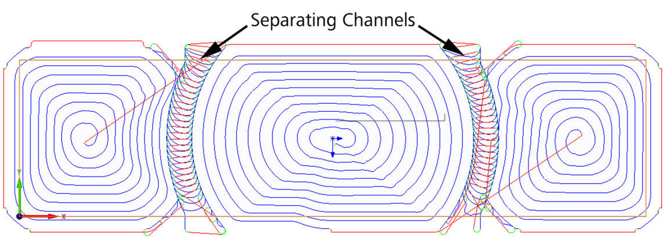


Figure 5B - Time 3:40 and tool wear is lower

These pockets can now be cleared with Maximum 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 paths. If the separation + spiral is shorter than the trochoidal-like path, iMachining will separate as described (Figure 6B).

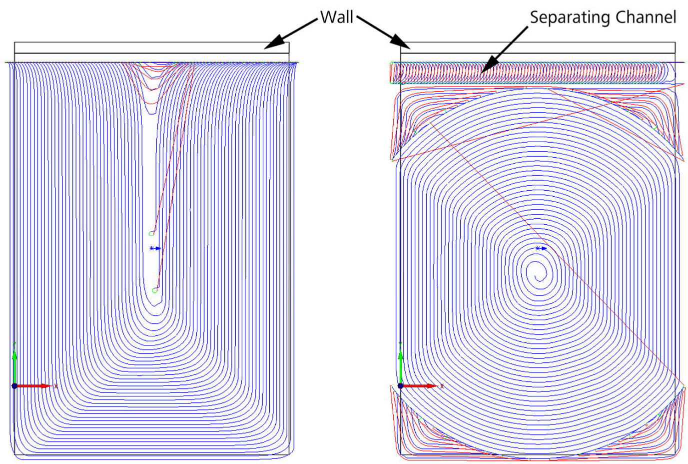
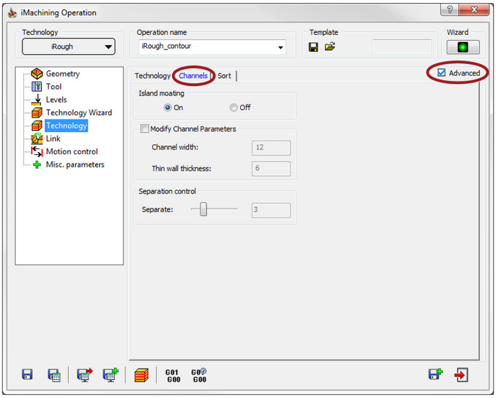


Figure 6A: Time 5:20
No separating channel - Semi open pocket completely cleared by Trochoidal Paths

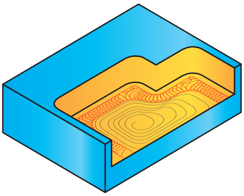
Figure 6B: Time 4:32
After separating from wall it becomes an open area and is then mostly cleared by spiral paths

Figure 6: Additional uses of separating channels

When the **Channels** page is opened and the **Advanced** check box is enabled, you can see a number of fields and controls that let you control the behavior of the Channels feature:

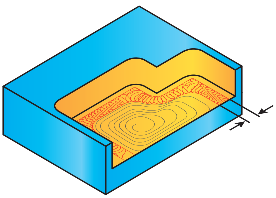


1. **Island moating** - This option is activated only if **On** is chosen.



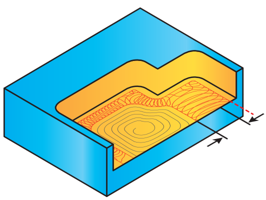
2. **Modify Channel Parameters** - When you select this check box, you can modify the values in the following fields:

   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 Thin Wall thickness 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. We saw an example of this above in the 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.

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

Figure 7: Maximum Thin Wall Thickness

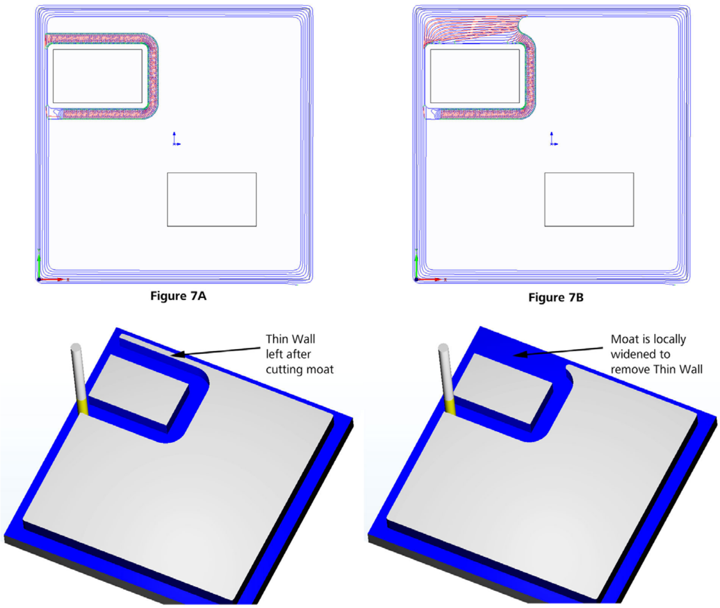


Figure 7C - Thin Wall thickness set to: 2mm

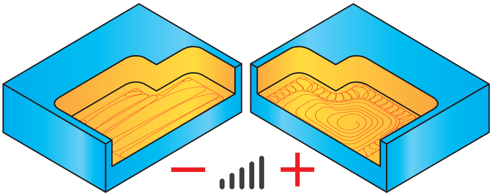
Figure 7D - Thin Wall thickness set to: 4mm

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



3.1. 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 first and most basic parameter of the cutting conditions is the cutting speed (Vc). After deciding on Vc, the Spindle speed RPM (S) 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, 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 chip thickness for each material type. When the chip thickness 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.

For example, when everything is perfect, 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 steels, 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 we make sure that we will not have vibrations?

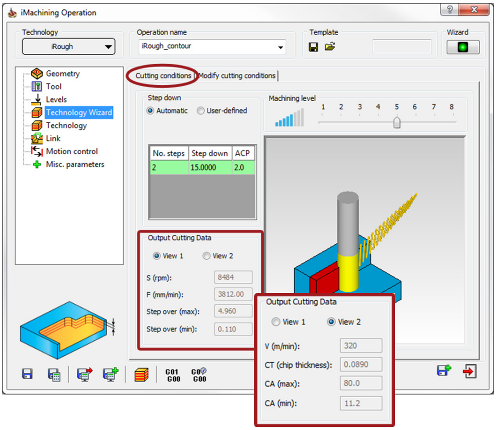
1. If our machine is very rigid (good construction, relatively new and well-maintained), and we are careful to have a very rigid work holding arrangement, and we use a good tool holder and the tool is well balanced in the holder and is running true (central), then we have no reason for concern and we can cut very fast (level 8 Turbo).

2. If we cannot provide **all** the above, we 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 condition 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 condition combinations are the Normal mode with levels 1~8 and the Turbo mode with levels 1~8.

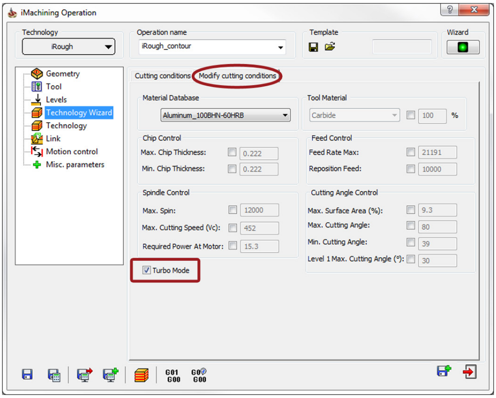
Cutting condition is a package of Min and Max engagement angles, Max feed, Spin, Min and Max chip thickness, step over and so on.



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

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 level of machining. Always arrange perfect cooling. When level 8 Turbo is used, we can cut very fast, but heat becomes a problem. More cooling is necessary when cutting with level 8 Turbo.

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

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

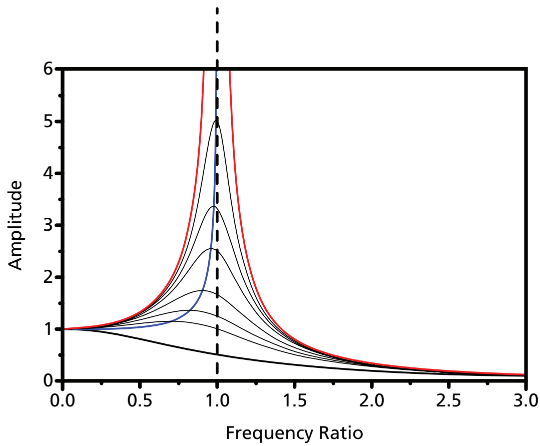
Pictures and videos with audio coming soon...

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).

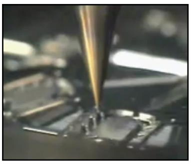
If the natural frequency of the machine is around 40 Hz, the above milling action will cause the machine to resonate, and the result will be strong vibrations.

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 wizard will generate small engagement angles and very small chip thickness, so the cutting conditions will be suitable.

- 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.



www.makino.com

What exactly is iMachining 3D?

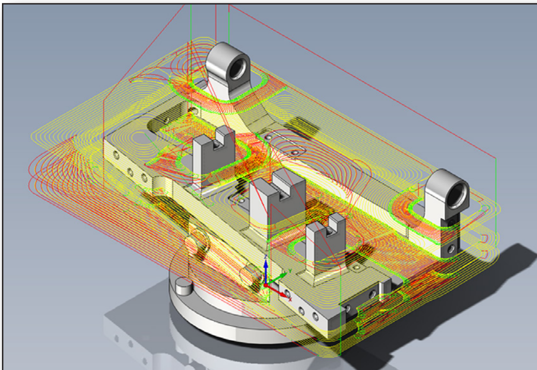
iMachining 3D is an Automatic High speed milling CNC program generator for Roughing, Rest Machining and Semi-finishing of both **prismatic** and **surfaced**, **general shaped 3D parts.**

iMachining 3D is the newest and most complete addition to the growing iMachining product family from SolidCAM, which now includes iMachining 2D & iMachining 3D.

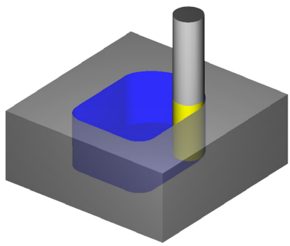
iMachining 3D uses **3D solid models** of the initial stock material and of the desired target geometry of the part, as input. It produces an optimal high speed CNC program that removes all the material that needs to be removed and that can be removed by the selected tool, and also produces an updated stock model, reflecting all material removal by the operation, as output.

By default, iMachining 3D works in the mode of "**Cut only Rest Material**," which enables it to use the Updated Stock Model from the previous operation, or the 3D model of a casting or a forged part, as the starting stock model for the next operation. In addition, during the calculation of the tool path, this initial 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 so **no time is wasted on "Air Cutting"** of volumes previously removed, or volumes which were empty to begin with (e.g. precast or forged stock).

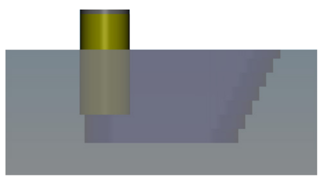
1. iMachining 3D **uses proven** **algorithms of iMachining 2D** to generate cutting tool paths at different Z levels, by analyzing and determining which volume to remove next, at what Z level. Doing so, we are able to achieve the shortest possible cycle time for the complete operation.



2. The first iMachining 3D operation for producing a part gets the initial stock solid model and the target geometry solid model as input with the first roughing tool. It then generates **roughing tool paths in large down steps** in the first isolated milling region; then *after achieving the final reachable depth (by the current tool) of the current region*, it starts the **Rest Roughing in step up mode** to remove all rest material on slopes of the region.

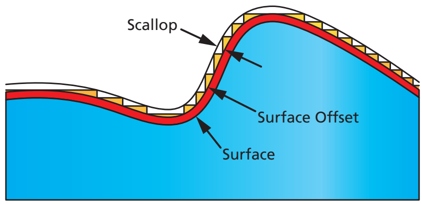


As we climb up during the step up machining, the isolated region may merge with a larger region, which may require further deep step down milling after rest roughing its upper layer (e.g. a wide pocket which splits into, two or more, deeper smaller pockets at the bottom).

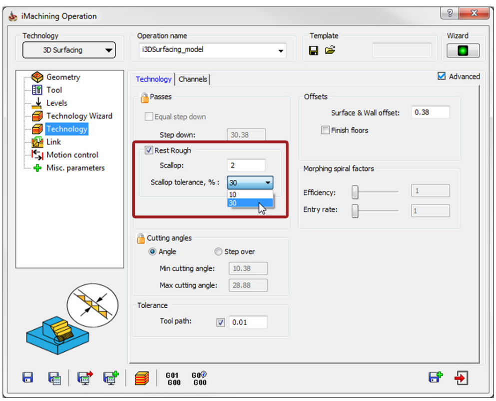


This complete sequence of step down and step up is **repeated** for region after region until the last one is complete.

The height of the steps during step up is calculated according to the local slope of each individual surface, so that the scallop height produced on each surface is of the value specified by the user. Thus, we get the name **True Scallop**.



On the Technology page of the iMachining operation dialog, you can specify the **Scallop** value after enabling **Rest Rough**.

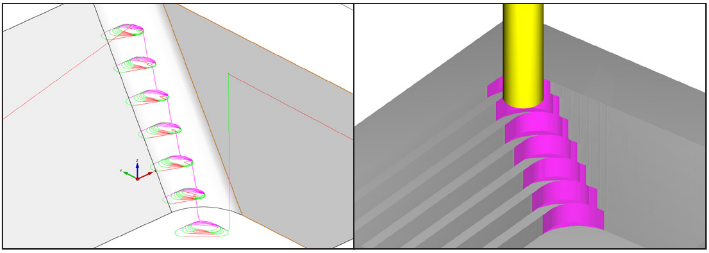


With **Rest Rough** selected, there is another important parameter field called **Scallop tolerance**. There are two optional values to choose from - 30% and 10%. This is the tolerance on your selected value of **True Scallop**. The **Scallop tolerance** enables iMachining to join two steps on two adjoining slopes, which would otherwise be cut at slightly different Z-Levels, and cut them in one longer cut at the same Z-Level, if the resultant local scallop on the two adjoining slopes (which of course is not of the preselected value) is still within the prescribed tolerance.

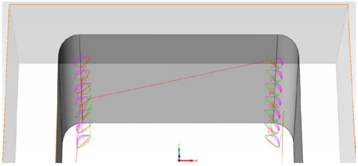
The effect of selecting the larger tolerance is to produce an actual scallop which may be up to 30% higher than the prescribed one, but the total tool path length and the cycle time will be appreciably shorter.

Note: Specifying a smaller value for True Scallop will result in a proportionately longer calculation time. By default, the True Scallop value set by iMachining 3D is proportional to the tool diameter.

3. The next iMachining 3D operation will likely get the updated stock model from the previous operation, and the same target geometry model 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 paths. As always, knowledge of the updated shape of the remaining stock ensures there will be no air cutting.



How is iMachining 3D different from iMachining 2D?

1. **iMachining 3D** produces a complete, ready to run CNC program with optimal cutting conditions, to rough and rest rough a complete 3D part, with **True Scallop** on all slopes, all in a single operation.

   iMachining 2D needs to be instructed how 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.

2. **iMachining 3D** analyzes its target solid model and automatically recognizes all pockets with their depths and gets all their geometries from the target solid model, without any need for chaining pocket contours or specific feature information like pocket depth, etc. It then subdivides all the volumes that need to be removed into thick horizontal slices (for roughing) and thin slices (for step up rest machining); and using sophisticated analysis, determines the optimal order of milling those slices to achieve iMachining's unique **Local Machining** feature, which results in eliminating almost all retracts and long positioning moves.

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

3. **iMachining 3D** acquires all its information about the stock, its extent, its shape, and information about material already removed by previous operations, from the solid model of the updated stock.

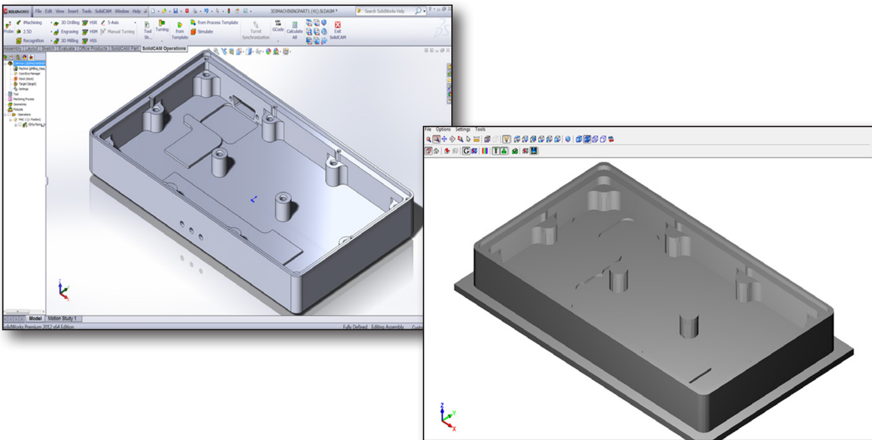
   iMachining 2D requires additional contours be sketched and defined for all stock material that lies outside the part geometry.

4. **iMachining 3D** automatically recognizes sloping surfaces and determines the optimal step up for each specific slope, for the next cut to produce the specified **True Scallop** size.

   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 rough and rest rough 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.

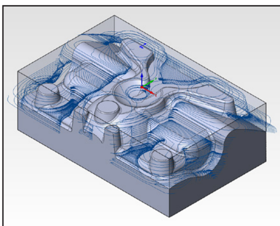


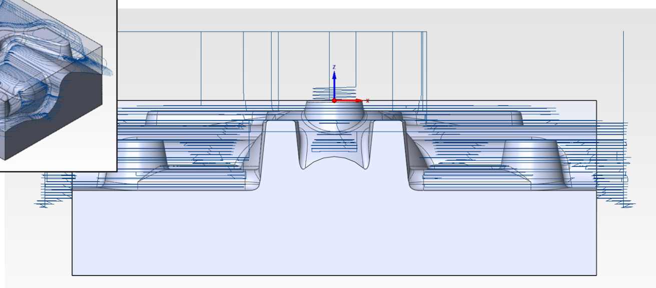
How fast is iMachining 3D relative to other 3D systems?

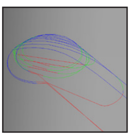
iMachining 3D is faster than other 3D High Speed Milling systems by a larger factor than iMachining 2D is compared to other 2D High Speed Milling systems.

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.

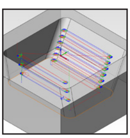
The reason being since all iMachining 3D cutting moves are basically horizontal cutting moves generated by the algorithms of iMachining 2D and its **Technology Wizard**, iMachining 3D cuts material as fast as iMachining 2D.

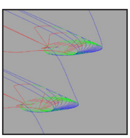






Additionally, with its localized machining and optimal ordering of machining sequences, iMachining 3D eliminates almost all retracts and long positioning moves and all air cutting, which no other 3D High Speed Milling system can do.

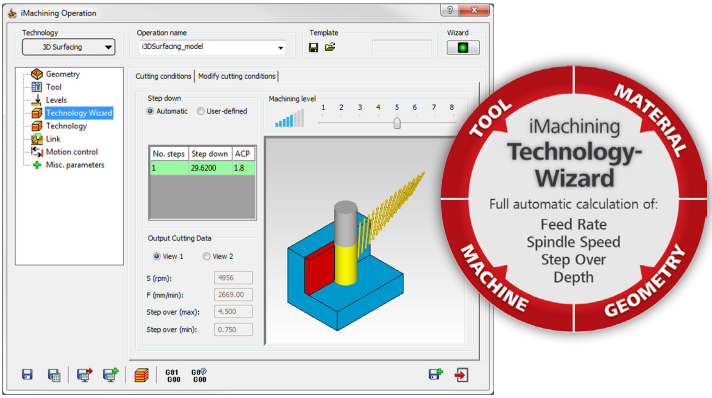




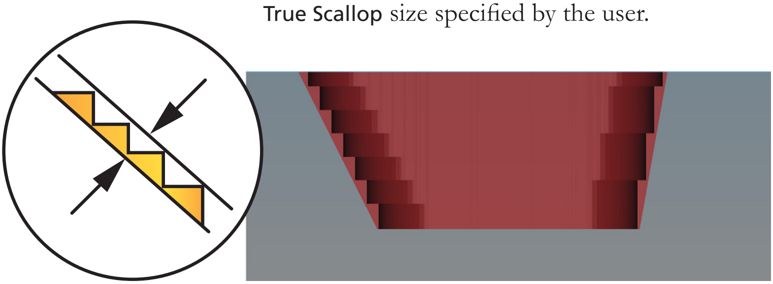
iMachining 3D, with its ***per slope scallop driven step up rest machining***, also generates milling paths that remove only the minimum amount of material on slopes that is necessary to produce the specified **True Scallop** size, without wasting time on surfaces that don't require cutting at the current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

1. A major additional advantage is the **proven <u>Technology Wizard</u>**, which automatically calculates the optimal <u>cutting conditions for each cutting path separately</u>, 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.

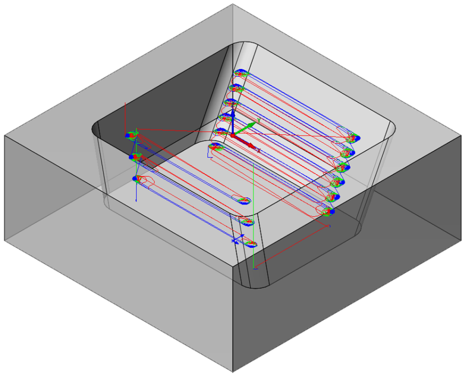


2. A very important additional money saving advantage of iMachining 3D over other 3D High Speed Milling systems is the ***Minimum Machining* feature of its step up rest machining on sloping surfaces.** This feature restricts the step up machining tool paths at any Z-Level to only cut material that, if left uncut, would violate the Maximum **True Scallop** size specified by the user.

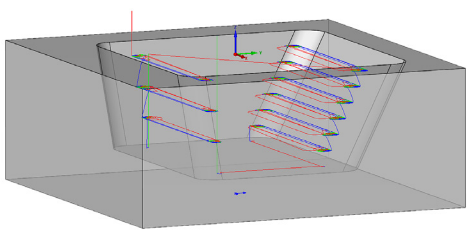


All other 3D High Speed Milling systems cut steps on slopes even in situations that do not require any cutting on those slopes at that height, to stay within the specified Scallop size (in those that allow the user to specify a Scallop size - many of the 3D HSM systems only let the user specify a constant step up amount). This unique feature therefore results in:

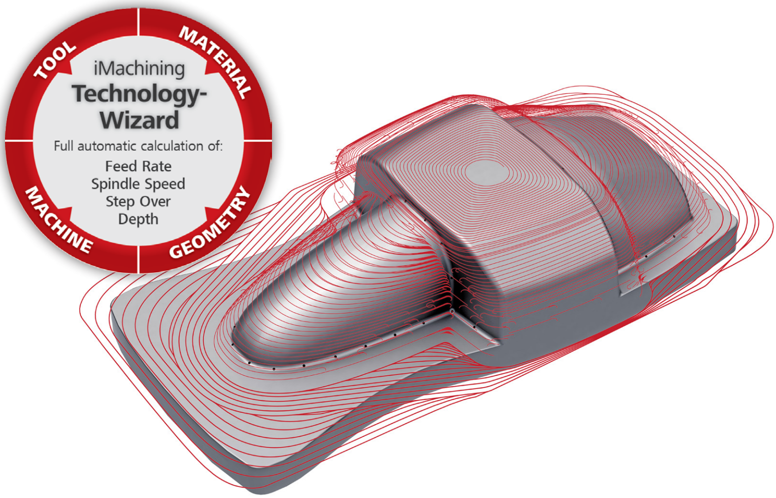
- Reduced total cutting path length during step up, resulting in reduced tool and machine wear and further reduced cycle time.



- 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 further reduction in cycle time and machine and tool 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 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 safe and fast milling of your part on your chosen machine. "**Safe**" means 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 always lies in some ***mismatch*** between the user's inputs and the realities of the machining environment. Here is a list of possible mismatches:

1. 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 SolidCAM 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.

2. The wrong tool was allocated to the tool magazine pocket

This situation is familiar to many users.

3. The wrong material was selected from the materials table, or the material entry in the table had the wrong value of Ultimate Tensile Strength (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 in 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 table was Titanium, which is a pure unalloyed form of the metal, and has an Ultimate Tensile Strength 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 Ultimate Tensile Strength of 1170 MPa. For more information, see *What are the important Stock Material properties?*.

4. The wrong Machining level was set in relation to the state of the machine, rigidity of the workpiece holding and TIR of the tool (for more information, see the *What is the role of the Machining Level slider?* section)

Case Study: The test cutting of a Ti – 6Al – 4V part was performed at a SolidCAM customer's workshop. The machine was in a poor condition, so a default machining level of 5 was chosen. The TIR was 0.02 mm which is too large for the 0.06 chip thickness the Wizard suggested, whereas 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 with a 16 mm 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 fourth 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.

We reduced the machining level to 2 and resumed cutting. This time there was almost no discernible vibration, and the tool lasted 8 parts.

5. 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 conditions (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, 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 this material can be cut faster on their machine.

Whatever the case, we highly recommend to only modify the cutting conditions **with the Wizard turned on**. As long as the user is not exceeding the maximum power or maximum feed or spindle speed of the machine, the Wizard will allow you 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.

6. Coolant issues

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

All hard-to-machine metals (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 raising the Material Removal Rate (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 will reduce the temperature of the workshop's standard 6 atmosphere air supply by about 20 °C and enable raising the MRR by 20% or more.

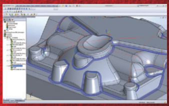
The new, revolutionary Milling technology

iMachining®
by SolidCAM

SOLIDWORKS Certified Gold Product

SolidCAM + SolidWorks
The complete integrated Manufacturing Solution

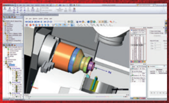
The complete range of manufacturing applications inside SolidWorks



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

The Revolutionary iMachining module

The new SolidCAM 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 Patent Pending, "Controlled Stepover" technology and managing feed rates throughout the entire toolpath, 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 morphed spiral toolpath, 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 SolidWorks integration

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

SolidCAM powers up the user's SolidWorks system into the best integrated CAD/CAM solution.

YouTube Broadcast Yourself™

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

iMachining Successes

www.solidcam.com/en/imachining/imachining-successes/

www.solidcam.com

SolidCAM
The Leaders in Integrated CAM