



The Revolutionary CNC Milling Technology – integrated in Siemens **NX**



iMachining for **NX** Frequently Asked Questions



Saves 70% and More in CNC Machining Time



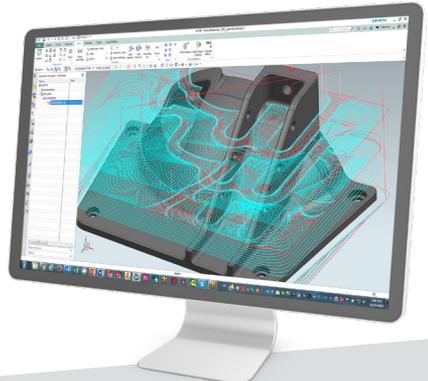
Drastically extends Cutting Tool Life



Avoids Guesswork: Optimum CNC Settings with the unique iMachining Technology Wizard



Immense Savings in Programming Time

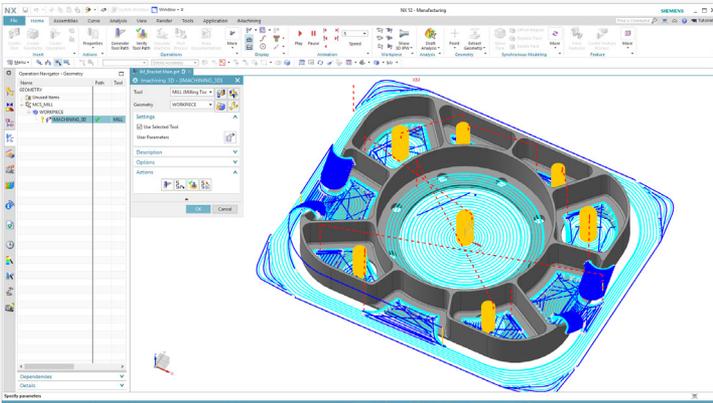


Contents

What is iMachining for NX?	1
What are the important...	3
How do I find the UTS value of a material?	7
What is the role of the Machining levels?	9
What are the main parameters in iMachining?	12
What is the Turbo Mode...	24
Why does iMachining need Channels...	25
How do I set the Cutting conditions...	32
What is the difference/relationship...	36
How can I judge the quality of a cut...	38
What causes vibrations...	39
How does iMachining perform?	40
What exactly is iMachining 3D?	41
What makes iMachining 3D so unique?	42
How is iMachining 3D different...	45
Can iMachining 3D automatically mill...	46
How fast is iMachining 3D...	48
What are the advantages of iMachining 3D...	49
How do I avoid mistakes...	51

What is iMachining for NX?

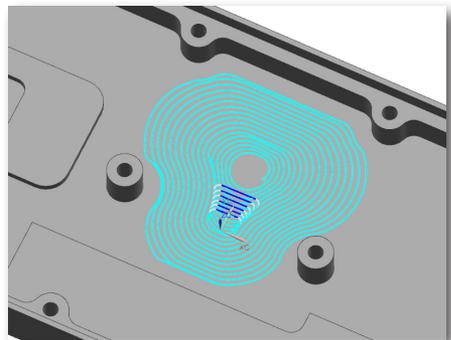
Developed by SolidCAM, iMachining™ is an intelligent high speed milling technology that is designed to produce fast and safe CNC programs to machine mechanical parts with *first part success* performance. The word *fast* meaning significantly faster than traditional machining at its best and the word *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).

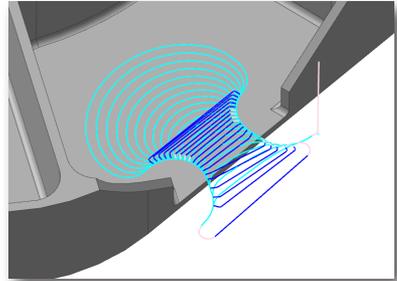
iMachining tool paths

iMachining generates morphing spiral tool paths that 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. As a result, iMachining manages to cut irregularly shaped areas with a single continuous spiral.



To machine narrow passages, separating channels and tight corners, iMachining uses proprietary constant load one-way tool paths.

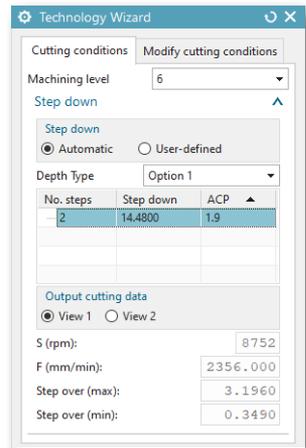
In some open areas, where the shape is too irregular to completely remove with a single spiral, proprietary topology analysis algorithms and channels are used to subdivide the area into a few large irregularly shaped sub-areas and then machines each of them by a suitable morphing spiral, achieving over 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 also performs an automatically optimized tool path when finishing 2.5D features. The finish tool path is executed in several consecutive steps with intelligent tool movements, all of which would be programming intensive and difficult to achieve using traditional machining methods. Because of its highly systematic approach to finishing and dedication to eliminate over engagement, the iMachining technology is able to further maximize tool life.

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 cutting 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 setup conditions and to their production requirements (quantity, schedule and tooling costs).



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

What are the important...

What are the important stock material properties?

General

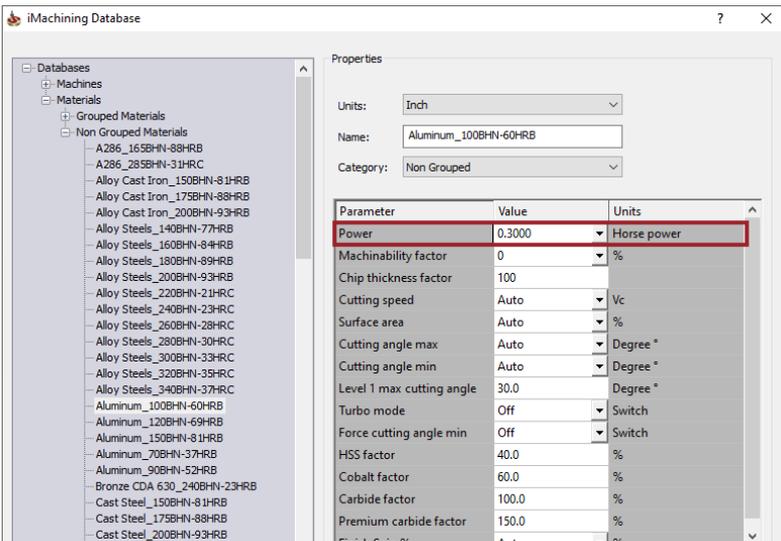
Different materials require different amounts of force to cut them. The physical property of a material that determines the force required for a particular cut is the Ultimate Tensile Strength (UTS), given in units of MPa (Mega Pascal) in Metric units or psi (pound per square inch) in English units.

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

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

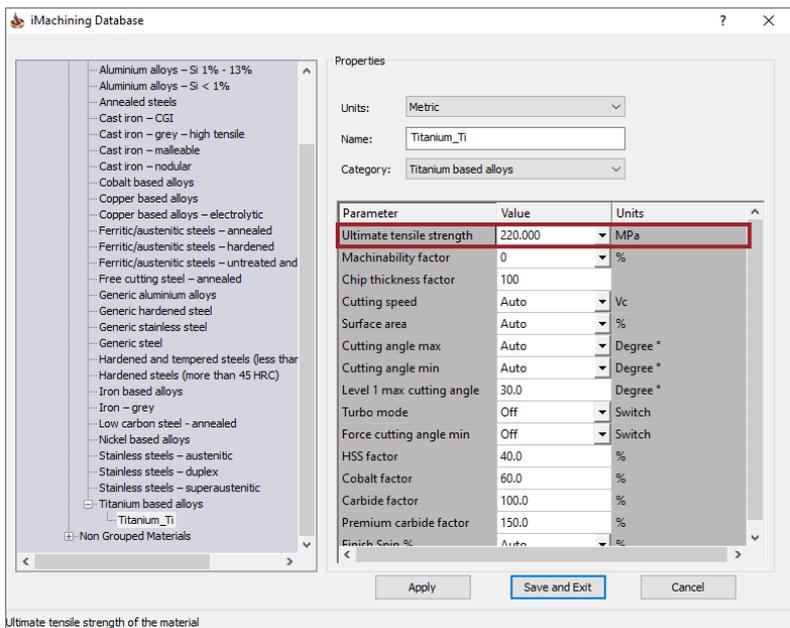
History

When the Technology Wizard was first developed by SolidCAM, it was designed to use a different material property, called Power Factor, to calculate the cutting force. This property 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 its initial development, which uses the UTS property to calculate the Cutting conditions. 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 interface for defining a new material, so that it would only accept UTS for newly entered materials.



The current situation is that the included materials are all defined in terms of their Power Factor rating; all newly added materials will be defined in terms of their UTS rating.

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

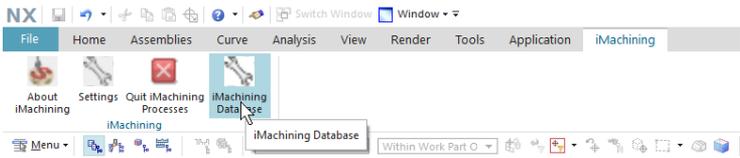
Defining new material entries in the Material Database

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.

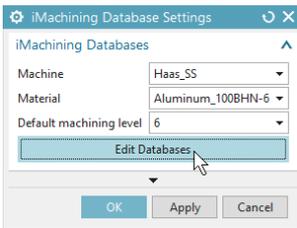
With the iMachining Database editing dialog box and the use of material UTS, it can be done quickly and easily. There are only two required inputs. The first input is the material name, which only serves to help you visually identify the specific material in the list and therefore must be unique, but 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:

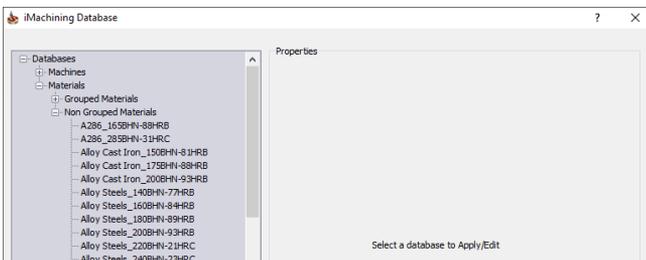
1. On the **iMachining** tab of the NX Ribbon bar, click **iMachining Database**.



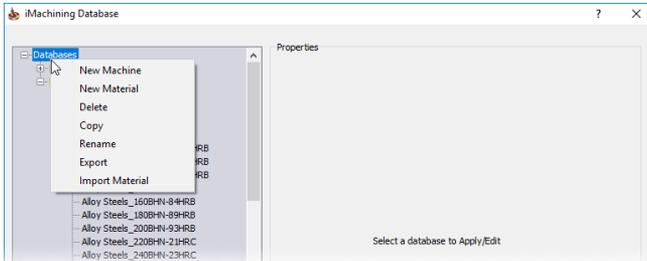
2. When the iMachining Database Settings dialog box opens, click the **Edit Databases** button.



The iMachining Database dialog box is displayed.

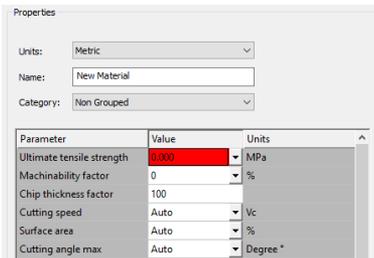


- Right-click anywhere in the Databases tree on the left.

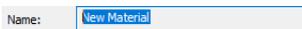


The iMachining Database managing commands appear.

- Select **New Material** to add your new material to the Materials list. The material property data appears on the right.



- In the Name field, enter the name of your new material.



- Using the Category list, select a predefined category into which you want your new material grouped (recommended but not required).



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

Parameter	Value	Units
Ultimate tensile strength	0.000	MPa

- If you have more materials to add, click **Apply** and then repeat steps 3 through 6 above. Otherwise, click **Save and Exit** to save your new material(s) to the iMachining Database and close the dialog box.

How do I find the UTS value of a material?

Make sure you know the exact specification of your material



Case Study: An iMachining 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 list within the iMachining Database Settings dialog box. They defined their iMachining operation, saved, generated and post processed it, and then started cutting.

Their tool broke after five 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 either your material supplier or your customer.

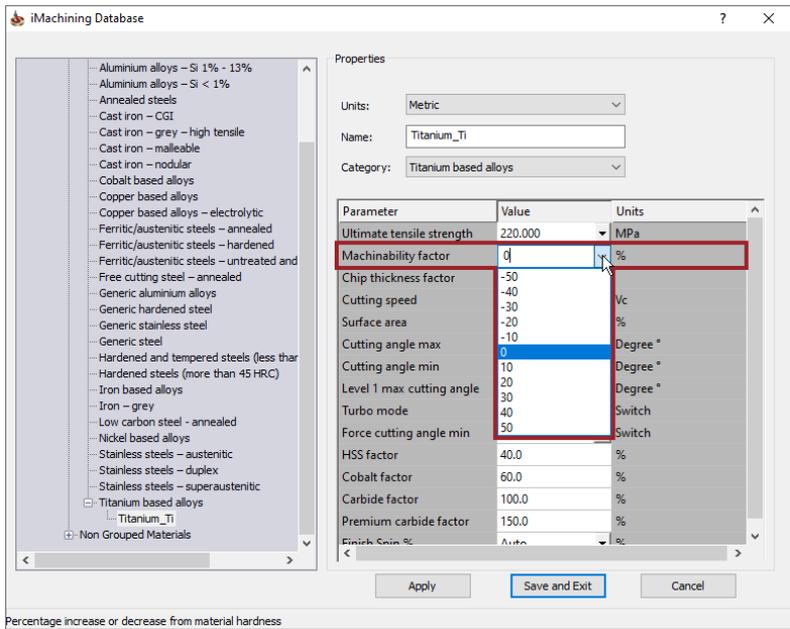
If there are many entries to choose from...

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 selection 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 selection 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.

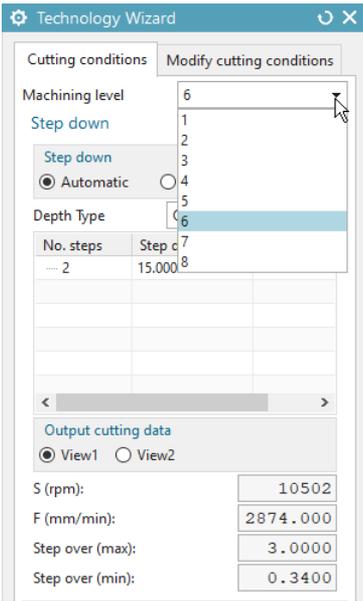
The Machinability factor enables you to alter the hardness of a material without changing its given UTS value. This option is available for all materials in the iMachining Database dialog box.



Selecting a higher positive value informs iMachining that your material is less hard than indicated by its UTS property 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 levels?

The Machining levels in iMachining provides the user with the means to conveniently and intuitively control the Material Removal Rate (MRR) when machining their part.



The Machining level selection informs the Technology Wizard how aggressively to machine the part.

As every experienced machinist knows that 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 available 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 setup (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, all while 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 levels that enable 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 levels is to first 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 Turbo level 6. There will be enough time to push it up to Turbo level 7 or 8 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.

The screenshot shows the 'iMachining Database' window. On the left, a tree view shows 'Databases' > 'Machines' > 'Defined By VMID' > 'Standard' > 'Haas_SS' > 'Haas_SS_New'. On the right, the 'Properties' section for 'Haas_SS_New' is shown. It includes a 'Units' dropdown set to 'Metric' and a 'Name' field containing 'Haas_SS_New'. Below this is a table of parameters:

Parameter	Value	Units
Spindle speed max	12000	RPM
Feed rate max	21158	MM per minute
Reposition feed rate XY	10000	MM per minute
Reposition feed rate Z	3800	MM per minute
Spindle power max	20.0	Kilowatt
Efficiency	90	%
ACP tolerance	20.00	%
Machining level	6	Integer

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 increase or decrease the Machining level by one number, calculate a new tool path and cut another part.



It is possible to increase the level 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.

What are the main parameters in iMachining?

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.

The screenshot shows the 'Properties' window for 'Titanium_Ti'. The 'Ultimate tensile strength' parameter is highlighted with a red border, showing a value of 220.000 and units of MPa. Other parameters include Machinability factor (0%), Chip thickness factor (100), Cutting speed (Auto), Surface area (Auto), Cutting angle max/min (Auto), Level 1 max cutting angle (30.0), Turbo mode (Off), Force cutting angle min (Off), HSS factor (40.0%), Cobalt factor (60.0%), Carbide factor (100.0%), and Premium carbide factor (150.0%).

Parameter	Value	Units
Ultimate tensile strength	220.000	MPa
Machinability factor	0	%
Chip thickness factor	100	
Cutting speed	Auto	Vc
Surface area	Auto	%
Cutting angle max	Auto	Degree °
Cutting angle min	Auto	Degree °
Level 1 max cutting angle	30.0	Degree °
Turbo mode	Off	Switch
Force cutting angle min	Off	Switch
HSS factor	40.0	%
Cobalt factor	60.0	%
Carbide factor	100.0	%
Premium carbide factor	150.0	%

Flutes

Another important parameter, whose value is not free to set by the user, is the number of flutes of the end mill.

The screenshot shows the 'Milling Tool-5 Parameters' dialog box. The 'Flutes' parameter is highlighted with a red border, showing a value of 4. The dialog also includes a legend with diagrams of a mill tool showing dimensions (L, FL, R1, D, A, B) and a table of dimensions.

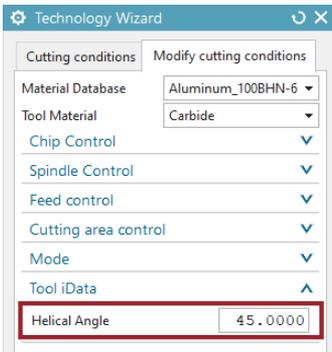
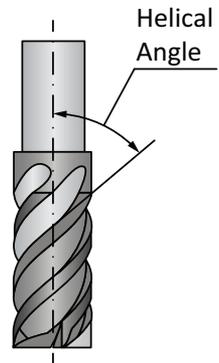
Dimensions	Value
(D) Diameter	9.5000
(R1) Lower Radius	0.0000
(B) Taper Angle	0.0000
(A) Tip Angle	0.0000
(L) Length	80.0000
(FL) Flute Length	24.0000
Flutes	4

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.

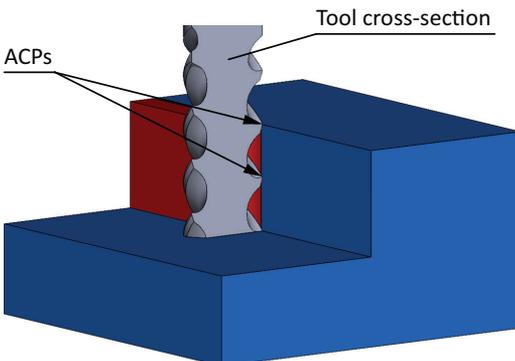
Changing Helical Angle only changes the Axial Contact Points (ACP) indication, which by itself currently has no effect on the Cutting conditions, though it may (should) drive the user to decide to change the tool or the Step down or reduce their Machining level to avoid vibrations.



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.

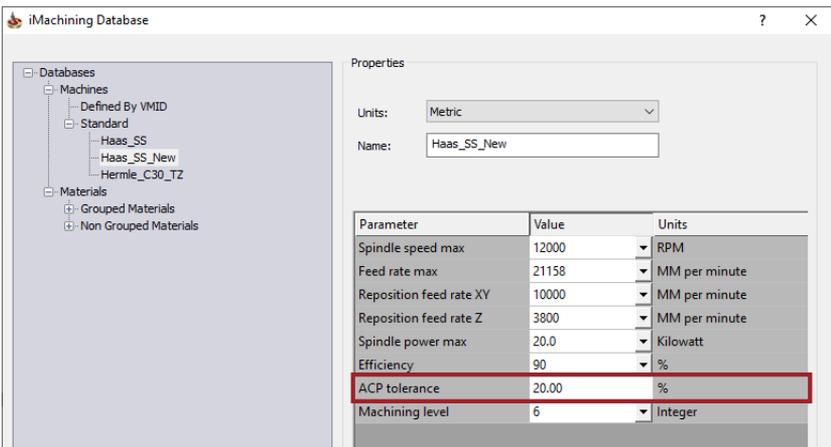
The Technology Wizard provides ACPs with a corresponding color to alert you of the current situation for stability and whether or not it is considered good. The color indication is as follows:

No. steps	Step down	ACP ▲	No. steps	Step down	ACP ▲	No. steps	Step down	ACP ▲
.... 2	15.000000	2.010378 1	13.000000	1.742328 1	5.000000	0.670126
<p>Black font Good – Preferred</p>			<p>Yellow font Not so good – Medium likelihood of vibrations</p>			<p>Red font Bad – High likelihood of vibrations</p>		

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.

In the machine properties of the iMachining Database, the ACP tolerance can be modified for any given machine. By changing this tolerance, you can control the ACP indication and how the Wizard outputs the depths.



If the tolerance was set to 0, for example, 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 is acceptable and the depth will be highlighted green for good stability.



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

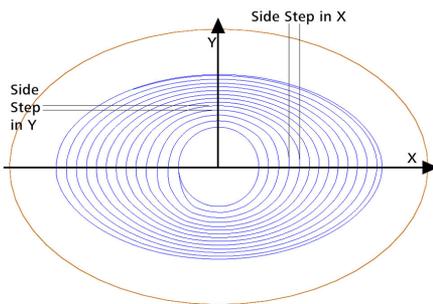


Figure 1A: Low efficiency setting:
Spiral with strong morphing clears complete area

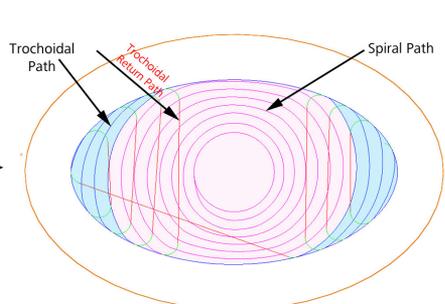


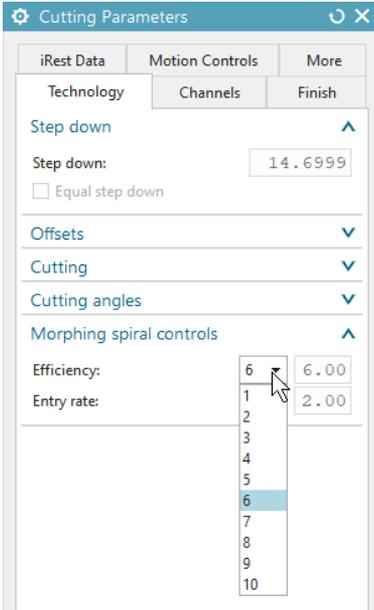
Figure 1B: High efficiency setting:
Spiral with little morphing is rounder and clears only part of area,
the rest is cleared by a Trochoidal Path

There are three reasons why iMachining does 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.

Using the Efficiency setting, you can limit the morphing by selecting a higher value of Spiral Efficiency.

The Efficiency setting appears in the Cutting Parameters dialog box > Technology tab > Morphing spiral controls group.



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 path in Figure 1A with that of Figure 1B, we notice that while the morphing spiral of case 1A manages to clear the whole area of the pocket, the conventional round spiral in case 1B 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 $55 + 24.8 = \sim 80\%$ efficiency.

On the other hand, the efficiency of the morphing spiral in case 1A 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 1A 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 cases 1A to 1B, we get $89/80 = 1.11$. This means that 1A completes the cut in 11% less time than 1B. And this is without adjusting the feed when the side step is smaller.

With the iMachining feed adjustment, the cycle time for 1A is $(80/94 = 0.851)$ 15% shorter than that of 1B. 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.

Shown 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.

Increasing the Efficiency level 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 level 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.

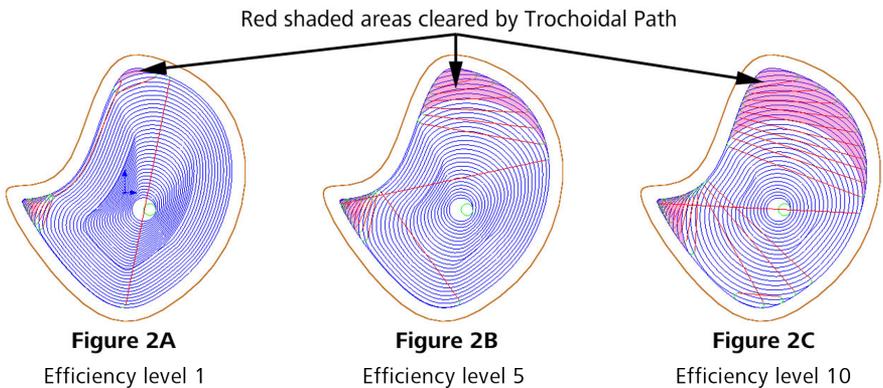


Figure 2: Same pocket cut with 3 different spiral efficiency settings

The Efficiency is set at a default level of 6. We recommend leaving it at this level unless there is good reason to modify it. However, it is a good idea to experiment with different levels, and simulate the tool paths to appreciate the effect of using the efficiency control.

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 maximize tool life but will also increase the cycle time for some geometries.

Future plans: SolidCAM plans to develop an automatic Spiral Efficiency level setting algorithm with a means for users to indicate their priorities. The *priority* indicated by the user will be one of three:

1. **Minimum cycle time** – intended for a short delivery deadline or an expensive machine and a low cost tool.
2. **Maximum tool life** – intended for an expensive tool or when you 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.

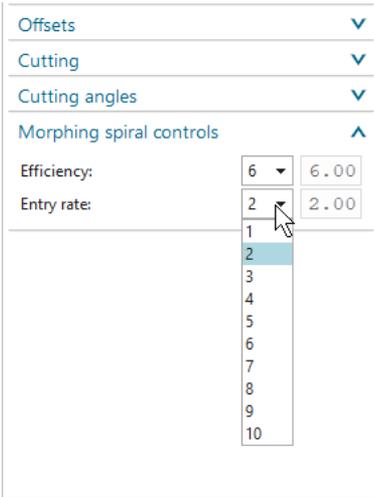


Note: When developed, the new algorithm will calculate the level for each spiral separately. Since even in one 2D pocket, there may be more than one spiral tool path, each spiral will be constructed with its own level calculated by the new algorithm according to the selected *priority*.

However, when using the existing method, the selected level will be global and will affect all spirals in the iMachining operation in the same way.

Entry rate

All spirals approach the material from air, whether from the outside of an open pocket in the case of converging spirals or from a pre-drill/helical entry on the inside of a closed pocket in the case of diverging spirals.



The Entry rate setting defines the rate at which the morphing spiral tool path first enters the material.

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

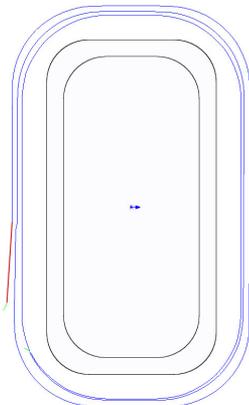


Figure 3A: Entry Rate Level 1

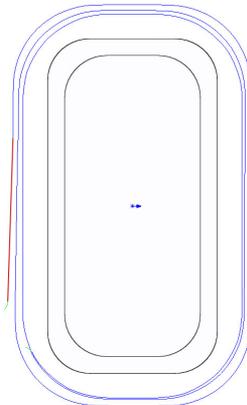


Figure 3B: Entry Rate Level 5

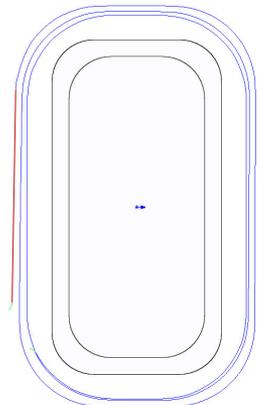


Figure 3C: Entry Rate Level 10

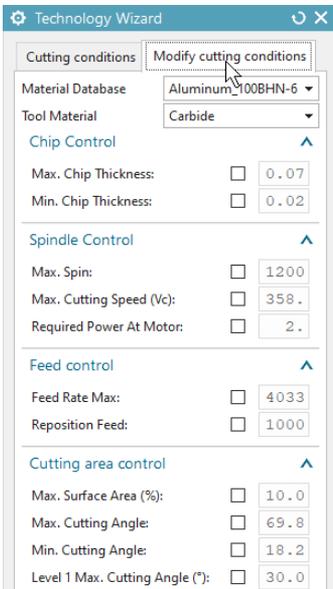
Although Entry rate is automatically set by the Technology Wizard in accordance with the properties of the stock material, the developers decided for the sake of flexibility and user-friendliness, to provide users with the means to override the default value. Selecting a higher value increases the rate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



If in doubt, change the rate by 4 or 5 levels, save and generate again the tool path to observe the new entry rate.

Modify cutting conditions

The Modify cutting conditions feature provides users with additional flexibility and control over the Cutting conditions.



You can modify any of the Cutting conditions parameters in the Technology Wizard dialog box > Modify cutting conditions tab.

It is strongly recommend that you only make modifications if your desired result cannot be produced via selection of the Machining levels.

All parameters are calculated automatically and their corresponding fields are initially disabled.

To modify a value, select its override check box to open the field for editing.



Note: The values appearing on the Modify cutting conditions tab 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, your modified value(s) will not be shown in the Output cutting data. Instead, 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.

When modifying values, be aware and consider the following:

1. After modifying a value and clicking outside the field, you may find that the value changes to a different value automatically.

This signifies that the chance intermediate value in the field cannot be reconciled with the machine limitations or with some other parameter value you may have already modified.

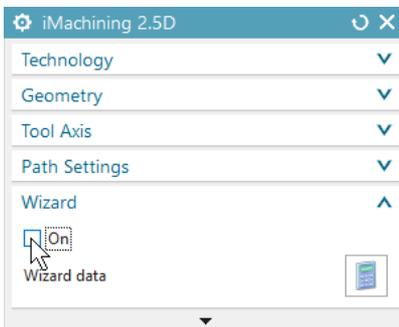
The Wizard calculates the nearest reconcilable value to the one you have entered and replaces your value with the calculated one.

2. The simplest way to reconcile the value is to deselect the check box next to the field. This restores the original value given by the Wizard.
3. Another way to reconcile the value 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 levels is to enable you to change all these parameters together in a synchronized manner, which gives you *easy* and *safe* control over the MRR and machining aggressiveness.

4. There is the path of least resistance, and the most risky option, of disabling the Wizard.

This can be done by deselecting the **On** check box in the Wizard group of the Operation dialog box.

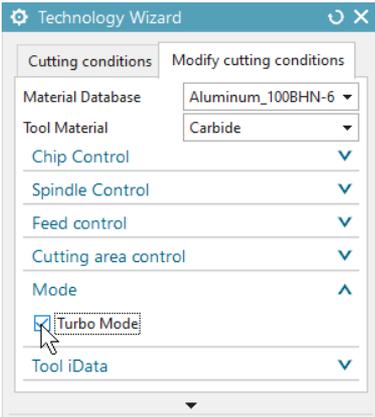


Warning!: When the Wizard is disabled, you are fully responsible for the consequences.

What is the Turbo Mode...

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 dialog box.



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

This means that the MRR of Turbo level 5 is about 25% more than the MRR of Normal 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, understanding the logic of the changes in the values you see displayed in the Output cutting data is not always easy. Do not be concerned however, the Wizard will make sure the end result is as close to your intentions as possible by your machine.

Output cutting data: View 1	Output cutting data: View 2																								
<table border="1"><thead><tr><th colspan="2">Output cutting data</th></tr><tr><td><input checked="" type="radio"/> View1</td><td><input type="radio"/> View2</td></tr></thead><tbody><tr><td>S (rpm):</td><td>10395</td></tr><tr><td>F (mm/min):</td><td>4386.000</td></tr><tr><td>Step over (max):</td><td>2.9300</td></tr><tr><td>Step over (min):</td><td>0.5200</td></tr></tbody></table>	Output cutting data		<input checked="" type="radio"/> View1	<input type="radio"/> View2	S (rpm):	10395	F (mm/min):	4386.000	Step over (max):	2.9300	Step over (min):	0.5200	<table border="1"><thead><tr><th colspan="2">Output cutting data</th></tr><tr><td><input type="radio"/> View1</td><td><input checked="" type="radio"/> View2</td></tr></thead><tbody><tr><td>V (m/min):</td><td>310</td></tr><tr><td>CT (chip thickness):</td><td>0.0974</td></tr><tr><td>CA (max):</td><td>67.5000</td></tr><tr><td>CA (min):</td><td>27.1000</td></tr></tbody></table>	Output cutting data		<input type="radio"/> View1	<input checked="" type="radio"/> View2	V (m/min):	310	CT (chip thickness):	0.0974	CA (max):	67.5000	CA (min):	27.1000
Output cutting data																									
<input checked="" type="radio"/> View1	<input type="radio"/> View2																								
S (rpm):	10395																								
F (mm/min):	4386.000																								
Step over (max):	2.9300																								
Step over (min):	0.5200																								
Output cutting data																									
<input type="radio"/> View1	<input checked="" type="radio"/> View2																								
V (m/min):	310																								
CT (chip thickness):	0.0974																								
CA (max):	67.5000																								
CA (min):	27.1000																								

Why does iMachining need Channels...

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 maximizing 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

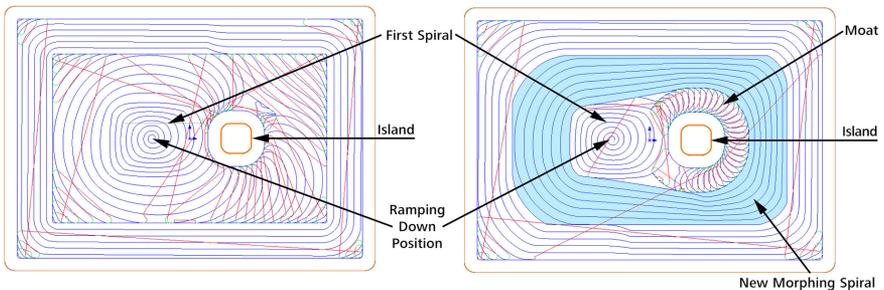


Figure 4A Time: 3:05 (without Moat)
The area around the first spiral and the island has to be cleared by trochoidal tool paths

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

Besides the cutting of moats, iMachining cuts channels to enable the use of spirals in cases where spirals cannot ordinarily be used.

On the following pages, the use of channels are illustrated in a few examples and the parameters that let you control the behavior of the Channels features are described in detail.

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 (7.875 in) 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 5: Uses for Separating Channels

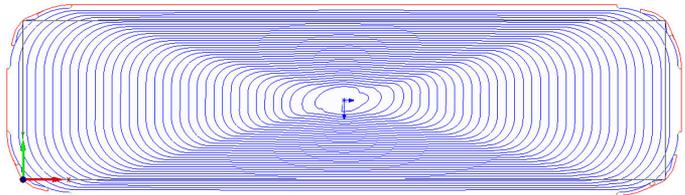


Figure 5A - Time 5:04 and tool wear at this extreme morphing is higher

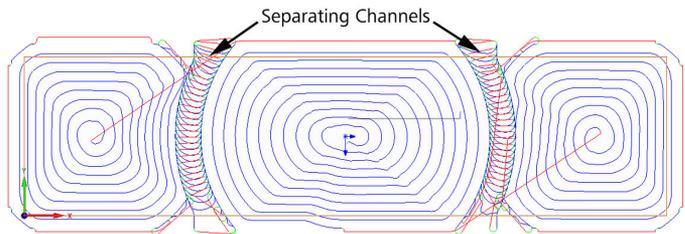


Figure 5B - Time 3:40 and tool wear is lower

These pockets can now be cleared by the maximum possible MRR with the only penalty being the time it takes 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

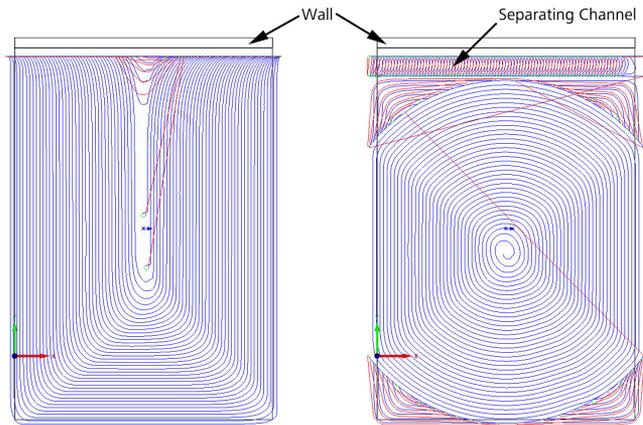
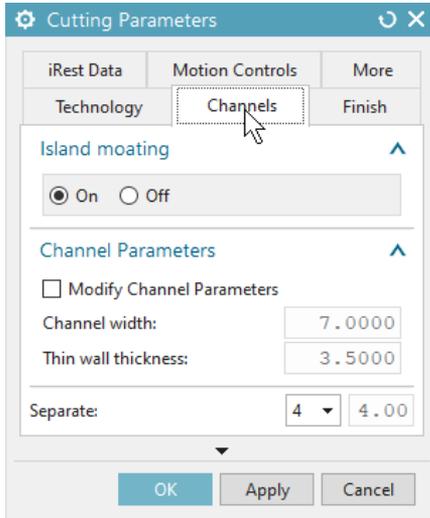


Figure 6A: Time 5:20
No separating channel - Semi open pocket completely cleared by Trochoidal Paths

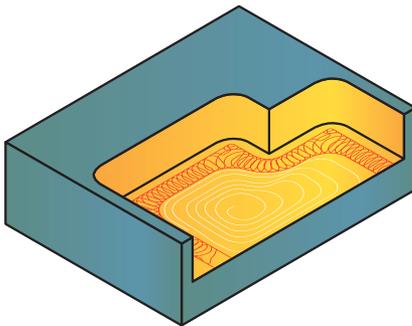
Figure 6B: Time 4:32
After separating from wall it becomes an open area and is then mostly cleared by spiral paths

In the Cutting Parameters dialog box > Channels tab, there are a number of parameters that allow you control the behavior of the Channels features.



Island moating

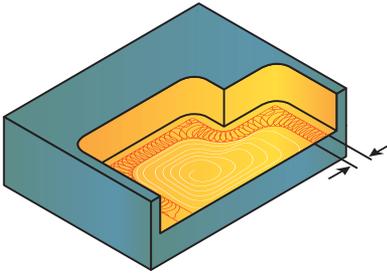
On by default, iMachining performs the cutting of moats automatically.



Modify Channel Parameters

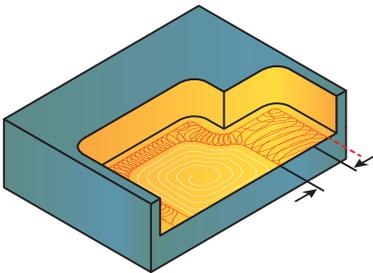
When you select this check box, you can modify the default values of the Channel width and Thin wall thickness parameters.

Channel width – This is the width of all channels cut in the active 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.

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

This example illustrates the effect of cutting a moat or channel near the open edge of a pocket.

Figure 7: Maximum Thin Wall Thickness

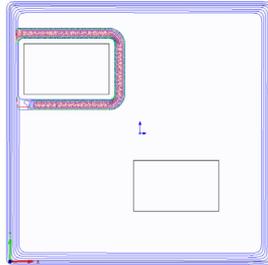


Figure 7A

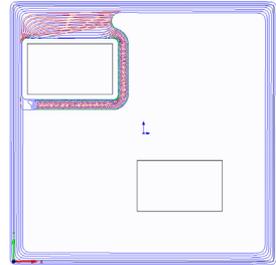


Figure 7B

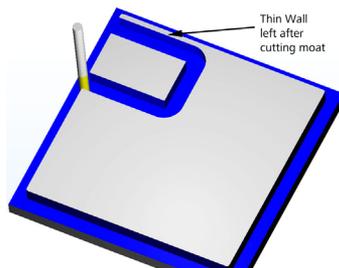


Figure 7C - Thin Wall thickness set to: 2mm

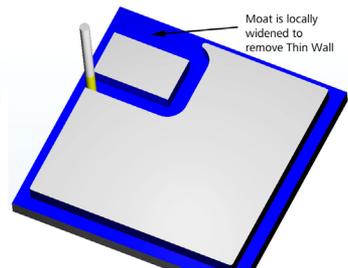


Figure 7D - Thin Wall thickness set to: 4mm

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 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.

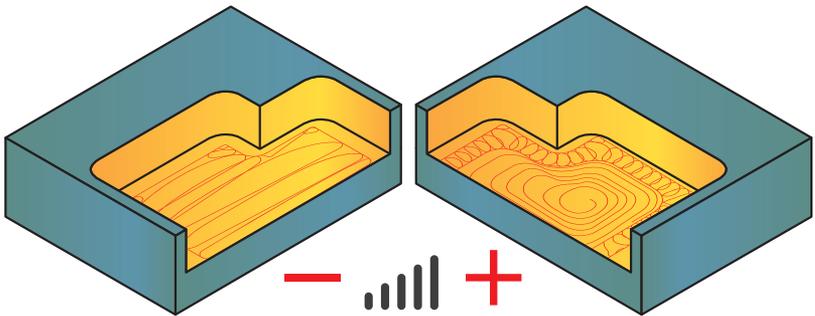


If you consider the default value too small, which means that in your opinion even a slightly thicker thin wall may still be dangerous and should be prevented, by all means increase this value. However, we recommend you do not reduce the value below the default.

Separation control

As mentioned, 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, the Separate setting on the Channels tab is provided so that the user can inform iMachining to lean more or less in favor of separation.



Increasing the value permits a higher separation factor, which results 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 the channel machining will take less time than on an average machine.

The Separation is set at a default level of 4 and most users will have no need to change it. Only users with especially high or especially low acceleration machines may find they need to change it.

How do I set the Cutting conditions...

How do I set the Cutting conditions in iMachining?

The first and most basic parameter of the Cutting conditions is the cutting speed (V_c). After deciding on V_c , the spindle speed (S (measured in RPM)) can be calculated easily if you know the diameter (D) of the end mill ($S = V_c/\pi D$). Once you have V_c 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 V_c ? 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 setup 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 loads on the tool are 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 setup 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.

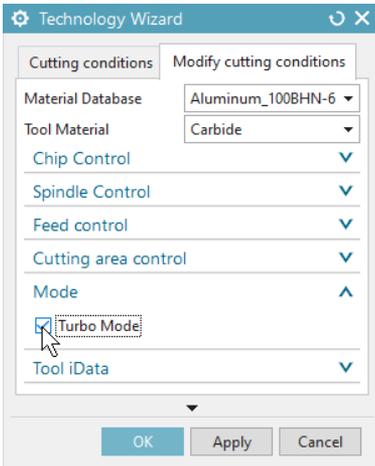
How can you make sure that you will not have vibrations?

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 (Turbo level 8).
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 setup*.

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 setup.

The 16 sets of Cutting conditions combinations are the Normal Mode levels 1~8 and the corresponding Turbo Mode 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 of the Normal Mode. For every Normal level (1-8), the corresponding Turbo level has 25% more MRR than the Normal level.

Output cutting data: View 1 with Turbo Mode enabled	Output cutting data: View 2 with Turbo Mode enabled																								
<table border="1"> <tr> <td colspan="2">Output cutting data</td> </tr> <tr> <td><input checked="" type="radio"/> View1</td> <td><input type="radio"/> View2</td> </tr> <tr> <td>S (rpm):</td> <td>10395</td> </tr> <tr> <td>F (mm/min):</td> <td>4386.000</td> </tr> <tr> <td>Step over (max):</td> <td>2.9300</td> </tr> <tr> <td>Step over (min):</td> <td>0.5200</td> </tr> </table>	Output cutting data		<input checked="" type="radio"/> View1	<input type="radio"/> View2	S (rpm):	10395	F (mm/min):	4386.000	Step over (max):	2.9300	Step over (min):	0.5200	<table border="1"> <tr> <td colspan="2">Output cutting data</td> </tr> <tr> <td><input type="radio"/> View1</td> <td><input checked="" type="radio"/> View2</td> </tr> <tr> <td>V (m/min):</td> <td>310</td> </tr> <tr> <td>CT (chip thickness):</td> <td>0.0974</td> </tr> <tr> <td>CA (max):</td> <td>67.5000</td> </tr> <tr> <td>CA (min):</td> <td>27.1000</td> </tr> </table>	Output cutting data		<input type="radio"/> View1	<input checked="" type="radio"/> View2	V (m/min):	310	CT (chip thickness):	0.0974	CA (max):	67.5000	CA (min):	27.1000
Output cutting data																									
<input checked="" type="radio"/> View1	<input type="radio"/> View2																								
S (rpm):	10395																								
F (mm/min):	4386.000																								
Step over (max):	2.9300																								
Step over (min):	0.5200																								
Output cutting data																									
<input type="radio"/> View1	<input checked="" type="radio"/> View2																								
V (m/min):	310																								
CT (chip thickness):	0.0974																								
CA (max):	67.5000																								
CA (min):	27.1000																								

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

Cooling is just as important as selecting the correct Machining level. Always arrange *perfect* cooling. When Turbo level 8 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

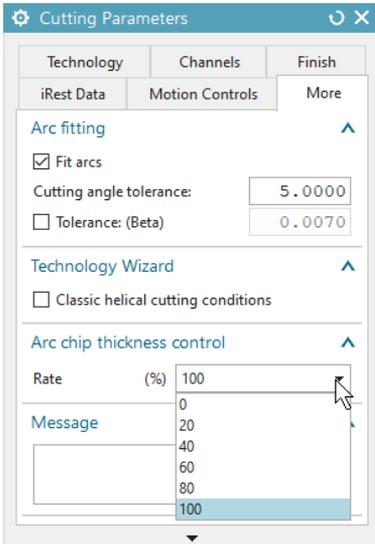
By default, the iMachining technology maintains a constant chip thickness (CT) when cutting in corners. To do so, the feed rate is reduced automatically.



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

With the automatic feed correction for arcs comes increased cycle times, and you may determine that a faster cycle time is more desirable than maintaining a constant CT.

In the Cutting Parameters dialog box > More tab > Arc chip thickness control group, you can specify the rate of chip thickness deviation when cutting in corners. The Rate is set to 100% by default; If kept at the default rate, iMachining is informed to maintain a constant CT.



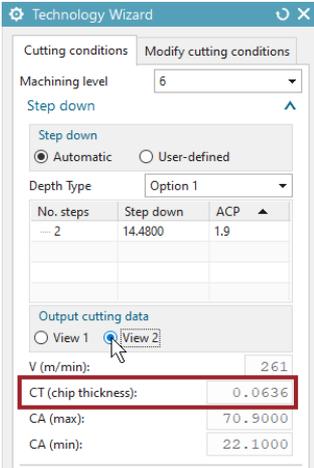
If you set the Rate to 0%, iMachining is informed, when cutting in a corner (G2), to use a feed rate consistent with cutting in a straight line (G1). The result is faster cycle times; *but beware, with increased CT in corners comes increased tool load.*

What is the difference/relationship...

What is the difference/relationship between chip thickness and chip load?

Chip thickness

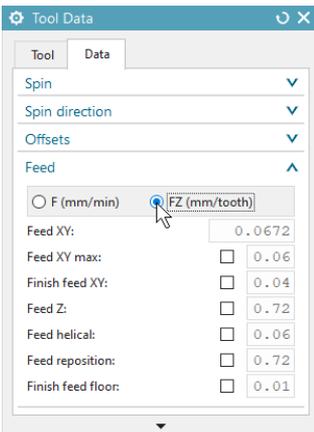
In iMachining, chip thickness (CT) can be described as the thickness (at the widest point) of the undeformed chip that is sheared away from the material.



The CT value for the operation appears in the Technology Wizard dialog box > Cutting conditions tab > Output cutting data: View 2.

Chip load

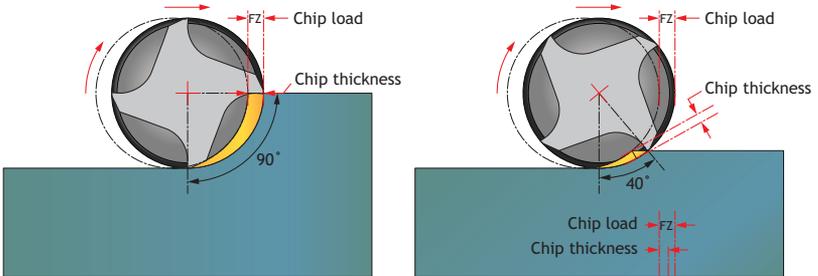
Not to be confused with chip thickness, chip load defines the feed rate of the tool, measured in units of distance per tooth (FZ (mm/tooth) in Metric units or FZ (inch/tooth) in English units).



The chip load (FZ) values can be viewed in the Tool Data dialog box > Data tab > Feed group.

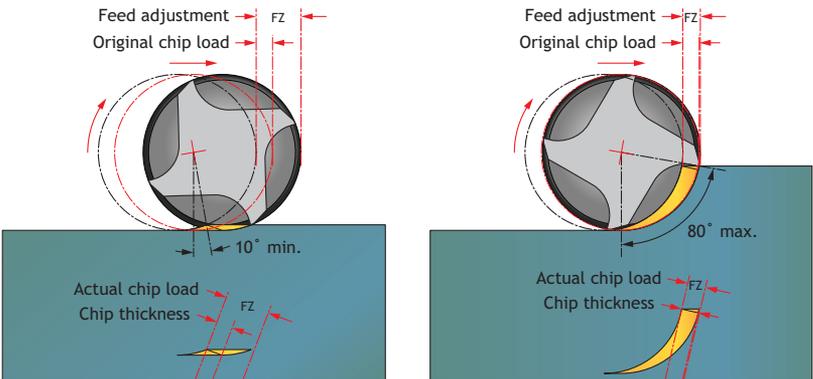
Relationship between chip thickness and chip load

Traditionally, the only instance where you would see equal values of chip thickness and chip load is when the cutting angle is at 90° . Anything less than 90° 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 10° and 80° depending on the selected Machining level.

To compensate for the dynamically varying cutting angle, the Wizard automatically adjusts the feed rate 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, it is possible for the Wizard to achieve a constant load on the tool and Cutting conditions that are always optimal.

How can I judge the quality of a cut...

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 Machining 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 Machining 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.

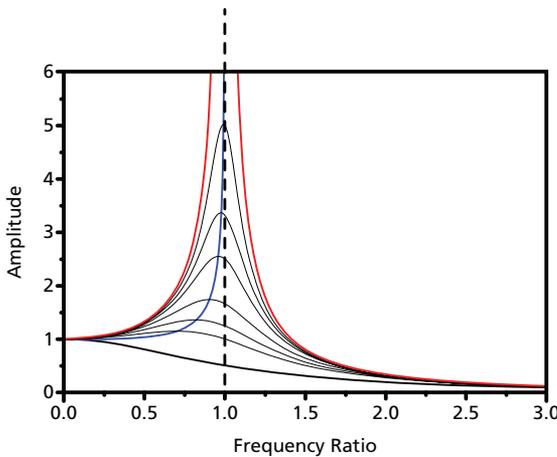
What causes vibrations...

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 the setup are not rigid enough, vibrations will start. If you keep increasing the feed and spindle rotation speed, the resulting reaction eventually becomes vibration.

Sometimes, the vibration makes resonance with the machine's 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).

When 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 situations of resonance, it is 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 Machining 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.

On older, low power machines

Since early 2012, iMachining has supported the use of low power machines – 2 Hp (1.49 Kw) or less. Accordingly, special Cutting conditions exist for machines defined with limited power capabilities. In short, iMachining is capable of reducing load on the machine tool, while still being able to keep the cutting efficient.



Rest assured, you can dust off those older, low power machines and put them back to work with iMachining.

What exactly is iMachining 3D?

SolidCAM's iMachining 3D 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 provides a complete machining solution for all your 3D parts when combined with other NX CAM operations for finishing. iMachining 3D can even be used to rough your 3D parts entirely followed by using iMachining 2D to finish prismatic parts or any planar surfaces of general shaped parts.



iMachining 3D uses both the Blank geometry representing the initial stock and the Part geometry representing the desired target, 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. Produced as output and used by subsequent operations is the *In Process Workpiece* (IPW) of NX CAM, reflecting all material removal by the operation.

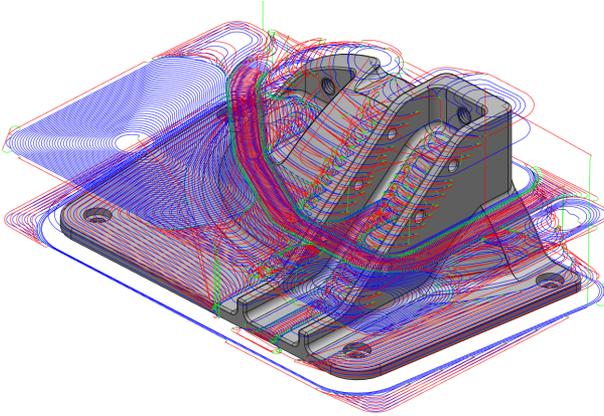
By default, iMachining 3D works in a mode that cuts only the rest material, which enables it to utilize the IPW from previous operations, or the Blank geometry of a casting or a forged part, as input for the starting stock of the active operation. During calculation of the tool path, the IPW is dynamically updated by each cutting move, and thus reflects the exact shape of the remaining stock material 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., Blank geometry of a casting or a forged part).

What makes iMachining 3D so unique?

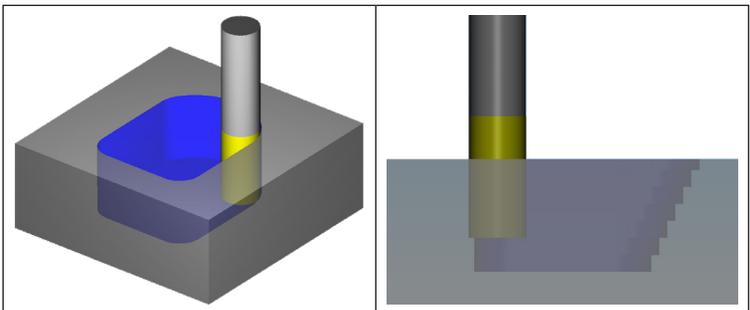
iMachining 3D is unique for several reasons:

1. iMachining 3D uses *proven algorithms of iMachining 2D* to generate cutting tool paths at different planar levels, by analyzing and determining which volumes to remove at what planar levels. Doing so, iMachining 3D achieves the shortest possible cycle time for the complete operation.



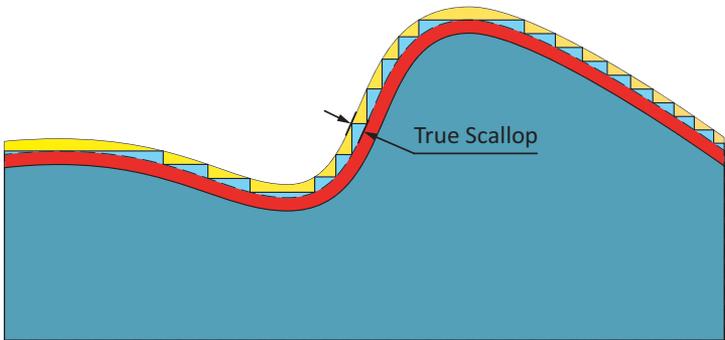
2. *Intelligent localized machining* – roughing tool paths are generated with deep Step down passes in the first isolated milling region. After achieving the final reachable depth (by the current tool) of the current region, rest roughing tool paths are then generated in Step-up mode to remove all rest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



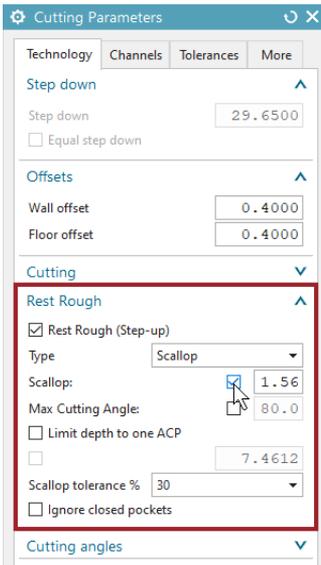
This default sequence of Step down and Step-up is repeated for region after region until the last one is complete.

The height of the steps during Step-up is automatically calculated based on the diameter of the selected tool. According to the local contour of each individual surface, the height of these steps change dynamically in order to maintain the specified Scallop value throughout the operation.



Every scallop produced is therefore a True Scallop.

If preferred, you also have the option to override the given value of Scallop with your own.



In the Cutting Parameters dialog box > Technology tab > Rest Rough group, selecting the override check box will open the field for editing.

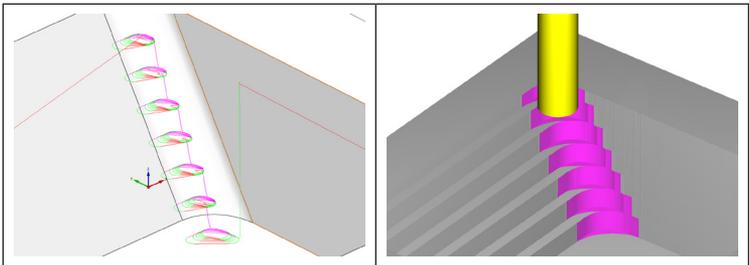
Another important parameter is the Scallop tolerance, which works in conjunction with the Scallop value. There are two optional values to choose from – 30% (default selection) and 10%. This tolerance is applied to the specified Scallop value. It enables iMachining 3D to join two steps on two adjoining contoured surfaces, which would otherwise be cut at slightly different planar levels, and perform one long cut at the same planar 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 to specify a smaller Scallop value will result in finer passes, which can be beneficial by helping to avoid semi-finishing operations. However, the calculation and cycle times will be proportionately longer.

3. The next iMachining 3D operation will get the IPW from the previous operation and the same Part geometry, and will likely be assigned a smaller diameter roughing tool to automatically remove material in areas unreachable 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 stock shape ensures that there will be no air cutting.



How is iMachining 3D different...

How is iMachining 3D different from iMachining 2D?

iMachining 3D produces a complete, ready to run CNC program with optimal Cutting conditions for both the roughing and rest roughing your 3D part entirely, with a True Scallop on all contoured surfaces, all in a single operation.

iMachining 2D needs to be instructed on how you want to machine the part, separated into appropriately ordered pocket operations. In a single iMachining 2D operation, only horizontal (thick or thin) planar slices of material can be removed.

iMachining 3D analyzes the Part geometry 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 subdivided into milling regions. Roughing tool paths are generated in thick planar slices (Step down) followed by rest roughing tool paths in thin planar slices (Step-up) for each isolated milling region. Using sophisticated analysis algorithms, the optimal order of milling those slices is determined to achieve iMachining's unique **local machining** feature, which results in eliminating nearly all retracts and long position moves.

iMachining 2D also analyzes the Target model (with its Feature Recognition technology). The Geometry definition process is simple when using the Feature Recognition Modes, but it requires minimal selections (e.g., faces or chains) to define and recognize the machinable features and their depths. You also have the option of using NX CAM's part boundaries.

iMachining 3D acquires all its information about the Blank geometry – its extent, its shape, and information about material already removed by previous operations, from the IPW of NX CAM.

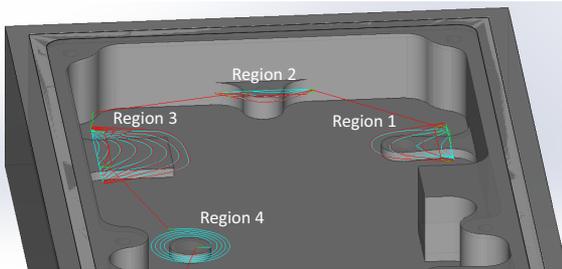
iMachining 2D also acquires information about the Blank/IPW when using the Feature Recognition Modes. Only when using NX CAM's part boundaries must you sketch and define additional contours for any stock material that lies outside the Part geometry.

iMachining 3D automatically recognizes contoured surfaces and determines the optimal Step-up for the local contour of each surface, for the next cut to produce the specified True Scallop.

iMachining 2D is not designed to handle contoured surfaces, only planar and vertically ruled surface geometries.

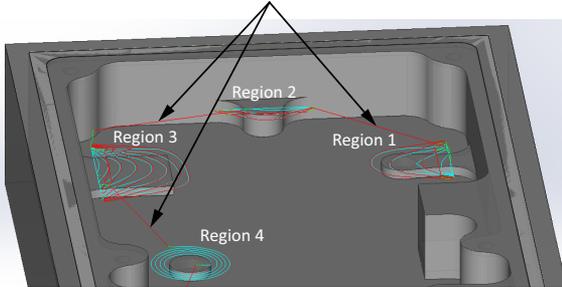
- iMachining 3D performs intelligent sorting of 2D planar regions. Non-cutting moves are reduced by the 3D planar ordering and localized machining of 2D tool path regions.

Sorting of regions

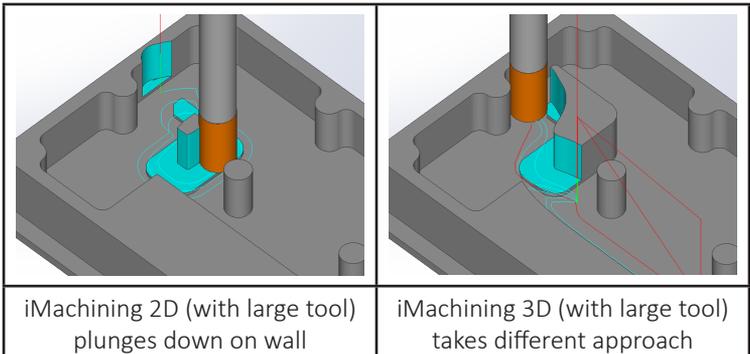


- iMachining 3D performs smart positioning between 2D planar regions. Long position moves are reduced by the 3D planar linking and localized machining of 2D tool path regions.

Positioning between regions



- iMachining 3D provides automatic protection of the Part geometry. Large tools can safely be used in confined spaces.



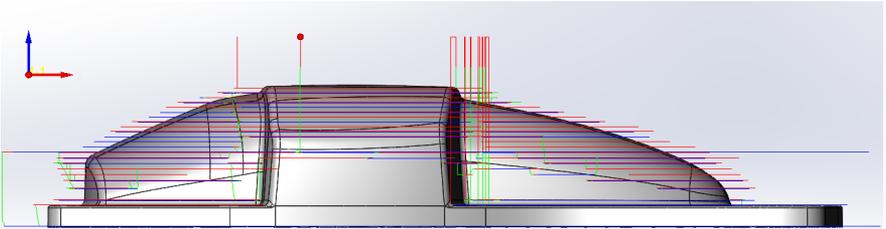
How fast is iMachining 3D...

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

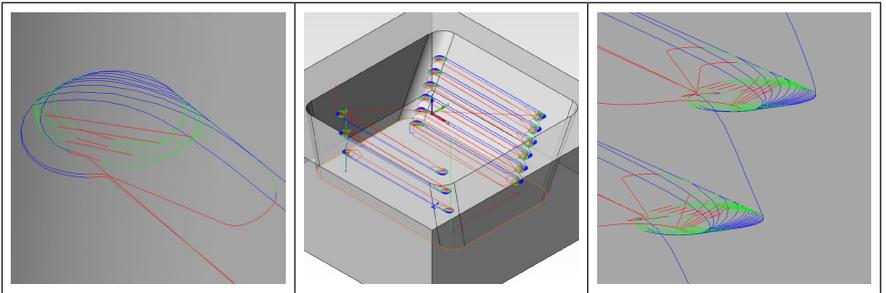
iMachining 3D is faster than other 3D high speed milling strategies *by a larger factor* than iMachining 2D is compared to other 2D high speed milling strategies.

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 strategies. Notably, iMachining 3D cycle times are up to 90% shorter than all other CAM strategies.

The reason being that since all iMachining 3D cutting moves are basically planar cutting moves generated by the algorithms of iMachining 2D and the 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 strategy can do.



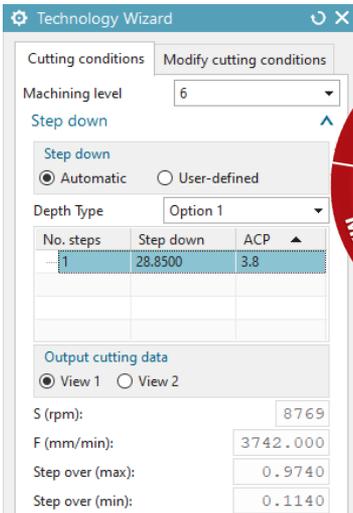
iMachining 3D, with its scallop driven Step-up rest machining, also generates cutting paths that remove only the minimum amount of material on contoured surfaces necessary to produce the specified True Scallop, without wasting time on areas that don't require cutting at the current planar level. In principle,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

What are the advantages of iMachining 3D...

What are the advantages of iMachining 3D over all other 3D strategies in addition to those described already?

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 that would otherwise be spent on long expensive trial and error work that is typically required with other 3D strategies to achieve a reasonably efficient tool path.



2. A very important additional money saving advantage of iMachining 3D over other 3D strategies is its minimum machining feature on contoured surfaces. This feature restricts the rest roughing tool paths at any planar level to only cut material that, if left uncut, would exceed the maximum True Scallop specified by the user.

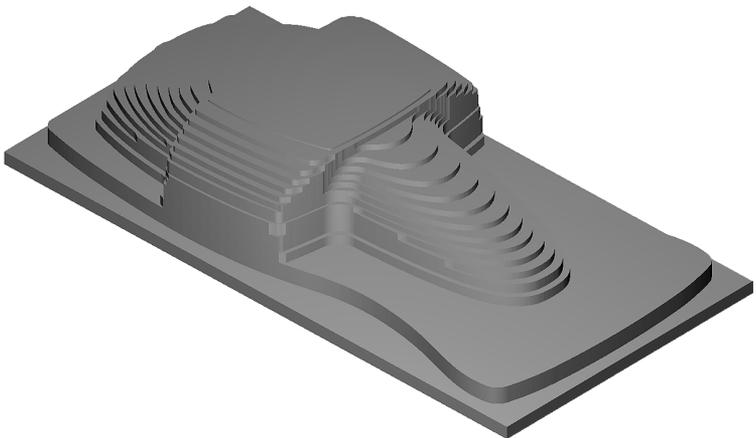


All other 3D high speed milling strategies cut steps on contoured surfaces, even in situations that do not require any cutting on those surfaces, to stay within the specified Scallop value (in 3D strategies that allow the user to specify a Scallop value – most 3D strategies only let the user specify 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 surfaces, making subsequent finishing operations 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 3D, 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...

How do I avoid mistakes that may shorten tool life?

iMachining produces complete (i.e., including all Cutting conditions) CNC programs for both *fast* and *safe* milling of your part on your chosen machine with not only *first part success* but also 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.

Described below are some possible mismatches.

The dimensions or location of the stock material are not the same as those defined for the Blank geometry



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

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 a customer’s workshop. The machine was in poor condition, so a default Machining level of 5 was selected. 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 chosen. 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 with the 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 developed.

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.

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 Turbo level 8.

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 that is 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 may be, *it is strongly recommended to only modify the Cutting conditions with the Wizard enabled*. 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 disabled, and cannot oversee your actions and intervene when necessary.

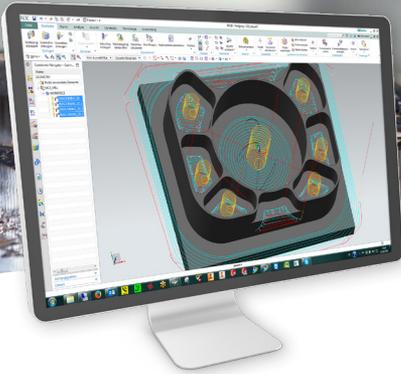
Coolant issues

Good cooling is critical in iMachining, as in all forms of high speed milling. Without it, the tool's temperature rises uncontrollably and causes the tool to disintegrate 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 a 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 USD \$500.00 can reduce the temperature of the workshop's standard 6 atmosphere air supply by about 68°F (20°C) and enable you to increase the MRR by 20% or more.

The Power of iMachining right
inside NX – a great Synergy!



The Revolutionary Milling Technology integrated in **NX**

imachining®

patent by SolidCAM

- ▶ iMachining – with its unique patented “morphing spirals”, intelligent island separation and moating, achieves the shortest cycle times in any industry - in any material, on any machine.
- ▶ The cutting angle constantly adjusts between a minimum and a maximum value, while the feed is dynamically adjusted to ensure constant mechanical and thermal load on the tool - providing you with the longest tool life at the highest Material Removal Rate (MRR) possible.
- ▶ Since spirals have higher MRR than Trochoidal tool paths, iMachining utilizes various, patented strategies that enable it to use spirals much more often than Trochoidals.
- ▶ iMachining automatically reduces vibrations to prevent excessive tool wear and also decreases the likelihood of forming temporary thin walls that often lead to costly tool damage.
- ▶ Using advanced rest material knowledge, iMachining maintains the highest level of cutting tool engagement by avoiding “air-cuts” and repositioning moves.
- ▶ The iMachining Technology Wizard provides automatic, optimal feed and speed values for different materials and CNC machines to ensure “first-cut” success.



“We have freed up a lot of production capacity, have significantly increased our productivity, and reduced tool costs by more than half. The investment has more than paid for itself. For us, iMachining is definitely a quantum leap.”

M. Torghele, Liebherr, Nenzing, Austria

“We use iMachining with nearly all our products – from 2.5D-, 3D- up to indexial 5-Axis Machining. From today’s point of view, it is a no-go to work without it.”

A. Winkler, Zrinski AG, Germany

“Every day we don’t use iMachining we are losing money!”

Rotary Airlock, USA



Visit our website for more iMachining success stories, videos and customer quotes

www.imachining4nx.com