**High Speed Machining** has so far disappointed many people in the machining industry by failing to live up to the expectations raised by early publications and marketing hype.

The reasons for this failure are many, but in most cases includes a combination of the following problems:

1. Very little or no attention to the user’s need for assistance in deciding on suitable cutting conditions.
2. Very little or no attention to the dependence of cutting condition parameters on the state of the machine tool.
3. Very little or no attention to the dependence of good changing cutting conditions on the local geometry of the tool path.
4. Very little or no effort to try and maximize the area cut by spiral tool paths, and reducing the area cut by trochoidal tool paths.
5. Very little or no attention to vibrations and chatter, and possible ways to reduce them.

SolidCAM, the well established leader in integrated CAM software, decided in 2006 to develop the first product that will truly meet the expectations of High Speed Machining and provide an optimal solution to the problems stated above. After four years of intensive research and development, SolidCAM is proud to announce the newest addition to its family of CAM software modules, **iMachining™**.

**iMachining™** was designed from scratch to automatically generate CNC programs that will adhere to the principles of HSM, enabling much shorter machining time, longer tool life and a significant reduction in machine-tool wear, compared to conventional milling methods and to existing CAM systems purporting to deliver working HSM tool paths.

An extremely important additional benefit of using **iMachining™** is the great reduction in mechanical and thermal stress suffered by the workpiece compared to conventional milling. This is achieved thanks to the unique feature of iMachining which actively ensures exact cutting conditions for each
point along the tool path, and to iMachining’s success in achieving the highest average percentage in the industry of area cut with spiral tool paths. These benefits are of major importance to a number of industries, including the **Aerospace Industry** and the **Mold Industry** (see later in this document).

**Machining Time** can be shortened by increasing the average Material Removal Rate (MRR):

1. MRR can be increased by reducing the frequency of non-cutting moves, i.e. repositioning moves and retracts. This means keeping the tool cutting most of the time and cutting with spiral tool paths most of the time.

2. Retracts can be kept to a minimum by carefully analyzing the geometry and topology of the remaining material, finding the shortest, relatively smooth repositioning detour path to the destination, at the current z level of the tool, comparing it to the total length of the alternative retract path, and selecting the shorter of the two.

3. MRR can be further increased by increasing the chip volume. The chip volume is the product of the axial depth of cut, the chip thickness and the chip length. The chip length is determined by the tool radius, the tool engagement angle, the feed and the spindle speed. The higher the feed the longer the chip. The lower the spindle speed, the longer the chip.

4. A large chip volume does not by itself ensure high MRR. We also need to produce many chips every minute. The number of chips cut per minute is determined by the product of the spindle speed and the number of flutes of the end mill.

   **Note:** Since iMachining generally cuts at high surface speeds and small chip thickness, under the right conditions and with good air cooling, chips tend to evacuate quickly and well out of the cutting zone without clogging the flutes of the cutter. This makes it possible to use tools with more (and therefore smaller) flutes than in conventional milling. This is another way iMachining can increase MRR.
**Tool life** can be increased by ensuring a steady mechanical and thermal load on the tool:

1. The major factor influencing tool wear is vibration. Using a suitable set of values for the cutting conditions parameters at each point along the tool path goes a long way towards minimizing harmful vibrations.

2. To increase tool life, care should be taken to avoid sudden changes in chip thickness and feed, to keep the tool cutting constantly most of the time, and keeping to a minimum the frequency of repositioning.

3. Tools wear out and break when excessively heated, and when vibrations set in. With the high cutting speeds of iMachining, experiments show that a significant portion of the heat produced by the cut is removed from the cutting zone by the chips flying out. Tool heating may be further reduced by using forced air cooling, preferably from 2-4 different directions. Even if suitable cutting conditions are used, vibrations may set in, if the workpiece is not firmly held in place, or if the tool is not held firmly enough.

4. The rigidity of all motion related elements of the machine tool (bearings, guideways, ballscrews etc.) is critical in the efforts to reduce vibrations and chatter. When using a less rigid machine, selecting a less aggressive level of cutting conditions will help to reduce vibrations. The **Technology Wizard** user interface has a special slider to select the desired **Milling Aggressiveness Level**. Changing the level of aggressiveness by simply moving the slider, results in the calculation of a new set of matching values for the cutting conditions parameters.

**Machine-tool wear** is reduced by the same conditions that reduce tool wear:

1. Avoiding vibrations.

2. Keeping mechanical load as steady as possible.
**HSM principles:**

1. Use surface speeds (= spindle speed X tool diameter X pi) and feeds 4 – 15 times higher than used in conventional machining.*

2. Reduce chip thickness to bring required power to within machine’s limits. Beware: there is a minimum thickness, below which the tool is rubbing, not cutting.

3. Avoid excessive vibrations and chatter by improving work holding and tool clamping, and by selecting matching values of the cutting condition parameters.

4. Make sure chips are free to fly out of cutting area. Use forced air to assist chip evacuation (through spindle air, if available).

5. Keep mechanical load on the tool steady by continuously cutting with constant chip thickness. Ideally, always cut in a spiral tool path.

*It has been discovered that increasing cutting speed and feed beyond conventionally held limits, tool loads (mechanical and thermal) start to go down, probably due to changes in plasticity of the material at the cutting point, and also to changes in heat transfer regimes. Apparently, under suitable conditions, chips flying out carry a lot of the heat with them.

**HSM benefits**

- Shorter machining time 1.5 – 4 times faster
- Longer tool life 4 - 30 times longer
- Longer Fatigue life of product. Reduced inner stresses.
- Reduced machine – tool wear.
- Practical machining of hard and hardened metals. Molds in 64 RC steel.
The iMachining 2D Pocket Operation

iMachining 2.5D  is a new addition to the list of operations offered by SolidCAM’s operations menu.

It is a 2D pocket milling operation. Any number of pockets may be defined for one operation. A pocket can, as usual, be an open, closed or semi open pocket, with or without islands. When more than one pocket is defined in a single operation, all the parameter values specified (except pre-drills), apply equally to all. iMachining 2.5D offers the choice of:

1. iMachining 2.5D Rough cut
2. iMachining 2.5D semi finish (Rest Material milling)
3. iMachining 2.5D Finish cut

For one part geometry, there can be any number of semi finish operations, as long as the tool diameter specified is smaller than that of the previous operation.

iMachining 2.5D Rough cut:

This rough cut operation will automatically generate a high speed milling tool path that will rough out all the superfluous material that is reachable by the specified tool, leaving the specified wall offset on islands and walls, within the specified accuracy.

Entry to a closed pocket can be via a pre-drilled hole. If none was specified, one will be automatically milled using a helical path, in the most strategic position for maximizing spiral tool paths.

iMachining 2.5D Semi Finish: -

The end mill selected for this operation must be smaller in diameter than that of the tool used for roughing the pocket(s).

This Semi Finish cut operation will automatically generate a high speed milling tool path that will rough out all the superfluous material left uncut by the previous operation, that is reachable by the specified tool, leaving the specified wall offset on islands and walls, within the specified accuracy.
The wall offset specified for the semi finish operation must be larger than the minimum stepover and not larger than the maximum stepover, specified by the Technology Wizard, for the Finish cut that will follow.

**iMachining 2.5D Finish cut: -**

The order of machining of the finish operation in iMachining 2.5D is:

- Machining of all excess material in all internal corners and narrow passages.
- Profile machining around all walls and islands in a single pass.
How SolidCAM iMachining achieves its goals

1. iMachining has two main parts:
   - The Technology Wizard
   - The Automatic Tool Path Generator

![Fig 1. the Technology Wizard Screen](image)

The **Technology Wizard** incorporates a sophisticated mathematical model of the behavior of all the machining variables and their interdependence. When presented with a machining task (which includes target and stock geometries, tools, material and machine data), the Wizard finds the optimum cutting conditions for sustainable high speed machining.

It should be noted that the Technology Wizard is not a set of pre-calculated look up tables. It carries out complex iterative optimizing calculations aimed at producing values for the cutting parameters that will produce fast, chatter free, constant load
machining, taking into account the limitations of the machine, the target geometry, the material and the tool.

In addition, since it is generally impossible to keep a constant feed throughout the milling process (acceleration/deceleration limitation of machine vs small arcs etc.), and since iMachining intentionally provides a range of usable cutting angles, which it utilizes to generate its’ morphing spiral tool paths, the Technology Wizard provides the tool path generator algorithm with matching formulae, to enable it to produce an exact match of the speeds and tool engagement for each point along the tool path.

2. Using the cutting conditions produced by the Wizard, the Automatic Tool Path Generator calculates all the tool paths necessary to efficiently produce the target geometry from the given stock geometry with the required wall offset and surface finish.

![Fig 2. Tool Cutting Angle -TCA](image)

The **rough cut** algorithm uses two basic types of tool paths:

- Morphing Spiral Tool Path
- Modified D Type Tool Path
Ideally, one would want the whole area that needs to be cut, to be milled with one continuous spiral tool path. Generally this is not possible. The iMachining Rough Cut algorithm does the next best thing. It subdivides the area into the optimal number of sub-areas, each of which can be cut with a morphing spiral path, such that the total machining time for the whole area is minimal. It uses Modified D Type Tool Paths to cut slots to subdivide the area.

**Fig 2. TOOL PATH VIEW**

Key:

Morphing Spiral Tool paths .........................1 ; 8 ; 10

Modified D-Type Tool paths ....................... 2 ; 9

Sausage widening of moats & slots .......... 3 ; 4 ; 5

Moats with trapped concave areas .......... 6 ; 7
Fig 3. Shows how “sausage widenning” of moat, eliminates the thin wall that would have been left, between the moat and the material edge.

Fig 4. Shows how, after a moat has been milled around 3 islands, the software recognizes that the area that has been separated, is too irregular to be cut with a spiral, and automatically decides to cut that area into two with a separating slot.

Another unique major advantage of the iMachining Rough Cut algorithm, is the fact that, on each point along the tool paths created, the values of the chip thickness, the feed and the spindle speed are always matched up according to the optimal rules laid down by the Technology Wizard. This exact matching of the cutting conditions, coupled with the optimal subdivision of the area, produce first cut parts safely, with total machining time savings ranging from 40% to 85%, on regular, mid-range machining centers.

3. The Morphing Spiral Tool Path is so called because unlike the spiral tool paths in most other CAM systems, it gradually adapts to the shape of the area it has to clear, rather than symmetrically expanding in a circular fashion. This ability reduces machining time considerably.
Fig 5 below shows the irregularly shaped area being divided into two more regularly shaped areas (A&B), by the separating slot C.
CONTROLLING THE CHARACTER OF THE TOOL PATHS IN iMachining

Although iMachining calculates all tool paths automatically, the user may wish to influence the decisions iMachining takes, in situations where even the best machining technology allows some degree of freedom of choice. The following examples illustrate what and how the user may influence using the iMachining User Interface.

1. The Morphing Spiral
   1. Introduction
   A regular spiral is a path that moves out from a center point, (or moves in from a circle in a circular fashion, gradually increasing (or decreasing) the radius at an even rate. Such a spiral can only clear areas which are circular or nearly circular in shape. Apart from its circular shape, a regular spiral has a constant stepover everywhere. The iMachining morphing spiral, has different stepover values in different directions looking out from the center. As a result, the morphing spiral can have many shapes. It can look like a rectangle, an ellipse, a triangle and many other irregular shapes. This quality, enables iMachining to clear irregular shapes with a continuous spiral tool path, thus needing to use the D-Type tool path much less than CAM systems limited to regular spirals. And since continuous (spiral) cutting wastes no time on repositioning, iMachining, with its morphing spiral tool path, is the most efficient method of machining. However, if the shape of the area to be cleared is very irregular, like a long narrow rectangle, the stepover across the narrow dimension of the rectangle will be much smaller than the stepover in the direction of the long dimension. In such a case, (see fig 6), most of the spiral tool path has a relatively small step over. Of course, such a spiral is not very efficient, in other words, the average rate of area clearing (or material removal rate) is relatively low.
2. **Spiral efficiency** is formally defined as *a measure of the percentage of the length of the spiral tool path that has a stepover larger than 80% of the maximum stepover selected by the Technology Wizard for the current operation.*

3. It is easy to select any level of efficiency that seems appropriate by simply moving the Efficiency Slider. When we increase the efficiency, a greater portion of the spiral tool path will have a larger step over. This will make the shape of the spiral more regular (more like a circle). In fig 7, we can see the effect of increasing the efficiency from 6 to 10.

**Fig 7 Controlling the Spiral Efficiency**

a) Spiral efficiency = 6
Here we see that with efficiency = 10 we have a spiral with less loops and a more rounded shape, bigger stepover all around, and of course, the spiral cannot cut the whole area. In such a case, iMachining automatically cuts the 2 semi-open areas (left and right) using a modified D type tool path before starting the spiral.

4. When the area to be cleared is open, and the best spiral with the requested efficiency cannot clear the entire area, iMachining needs to clear the small areas that would not be cleared by the spiral, using one of two methods:

a. Using a D –Type tool path (to clear the semi open small area, see fig 8 a).

b. Cut a separation channel and then with a small spiral, clear the separated area, (see fig 8 b), unless the channel cleared the whole small area (see fig 8 c).
**Area Factor**

The decision of whether to separate or not (i.e. whether to use method a or b above), depends on the value of the **area factor**. This is a measure of the small area in relation to the cross section area of the end mill. The smaller the **area factor**, the more likely will the area be separated.

**2. Separation Channels**

This is a very useful tool iMachining uses to separate one area from another, using the D – Type tool path to cut a narrow passage between two parts of the joint area.

Separation Channels are used (unless disabled by the user – Channels Off), when the shape of an open area to be cleared, is too irregular to be cleared by a spiral tool path, (white area on right, see fig 9 a) and iMachining can see a way to separate it into two areas, each of which will be regular enough to be cleared by a separate spiral. See fig 9 b & c.

Separation Channels are usually of a constant width. The default width is about 1.5 X D, where D is the tool diameter. The user can change this width – see Channel Width in the wizard’s dialogue.

Fig 9

9a
3. Moats

Separation Channels are also used to cut Moats around islands (i.e. to cut separation channels around islands, thus separating them from the remaining area to be cut), whenever they are encountered by a tool path. (Unless disabled by the user – Channels Off).

It is strongly recommended not to disable the separating channels. In most situations they help reduce machining time by making it possible to machine a greater portion of the total area by spiral tool paths.

In fig 10, we see an open pocket with 5 islands being cut by spiraling in from the outside. After two or three passes of the spiral, an island is encountered. It is immediately moated. This moating separates the island from the rest of the area, which can be viewed as a new open pocket, which iMachining can then cut with a new spiral, until the next island is encountered. And so on. When the island being moated is near enough to another island, iMachining will join the moat of the first island with the moat of the second island.

Fig 10
4. Thin walls

Whenever a thin wall is left by a cut, there is a danger of damage being caused to the tool or the work piece when that thin wall is removed by a subsequent cut.

iMachining analyses the result of each planned separating channel before it is confirmed. If it detects that a thin wall will be left by the channel, it widens the channel locally to remove the “would be thin wall” while it is still attached to the rest of the material.

The thickness, below which a wall will be defined as thin, is called “thin wall thickness”. It is by default 2 X Maximum Step Over, but can be changed by the user. To do that, the user needs to input his preferred “thin wall thickness” value in the appropriate field in the User Interface (UI).

The local widening of the channel is called Sausage.

5. Channels: on/off

As already mentioned, iMachining can be commanded not to cut any Moats or separating channels, by simply using the off radio button under Channels in the UI.

We strongly recommend not to do that unless there is a very good technological reason to do so. When the Channels are disabled, the whole area will be cut using the D – Type tool path, without any spirals. In over 90% of cases, this will increase machining time quite considerably.
**SolidCam’s iMachining™ highlights**

Designed by Machinists for Machinists.

- iMachining’s Knowledge Based Technology Wizard, automatically computes optimal Cutting Conditions.
- Spiral cutting even of irregular shape areas.
- Controlled Tool load at each point along tool path.
- Islands are Moated to enable continuous spiral cuts, even with multiple islands.
- Automatic Thin Wall avoidance.

**Is iMachining really Revolutionary?**

**YES.**

It is the only CAM module that produces:

- Practical high speed cutting conditions that actually work, **first time**.
- Mostly spiral tool paths that actually cut your machining time in half or better.
- Fantastic savings in tooling and machine maintenance costs.
iMachining 3D and the mold industry

The Plastic Mold industry is and will be the number one growth market for iMachining 3D. Here are the reasons why:

- Cutting a mold cavity directly in a block of hardened tool steel, including a high quality finish cut, has 6 main advantages for the mold maker:
  
  1. Much shorter manufacturing time.
  2. Unattended machining.
  3. No thermal deformation, and no inner stresses left in the mold. Much lighter cuts.
  4. Much less manual finishing and polishing. The finish cut can be much closer to the final dimensions and shape
  5. Better final dimensional accuracy.

- iMachining 3D produces much faster machining tool paths for High Speed Machining, and longer tool life than competing products. This means iMachining 3D is the best system to use for cutting molds directly in hardened tool steel.

- There is a marked trend in the global market for electronic products, gadgets, appliances and even toys: bigger and bigger demand for a great variety of models. Everyone wants a different model from that of their friend/colleague/neighbor. As a result, manufacturers have to produce many more molds than they used to. This trend is growing fast, which means the rate of mold manufacturing has to grow fast too.

- The rate at which new thermoplastic polymers with ever more fantastic characteristics come out of the labs is growing exponentially. As a result, the rate at which parts that were once made of metal or wood are re-engineered to be made of plastic is also growing exponentially.

- We can therefore reasonably conclude that:

  1. The global market for plastic injection molds is growing and will continue to grow for some time.
  2. iMachining 3D will offer the mold manufacturer the best alternative.
**iMachining 3D and the Aerospace industry**

"High-speed machining (of) aluminum is becoming almost standard today," says Rudy Canchola, application manager for Mazak Corp.’s Western Regional Headquarters and Aerospace Technology Center located in Gardena, CA. For him, the greater machining challenge today is in the high-temperature alloys, like stainless 15-5 or titanium 5553 or 6Al4V. These are materials finding more and more use in aerospace. His recent projects using a variety of tools on Mazak equipment included titanium tests on the Mazak VCN-510 C VMC. "We have demonstrated machining titanium with solid carbide end mills up to 500–600 fpm. We think that is impressive," says Canchola.*

Traditionally, a lot of the small and medium sized airframe structural parts used to be made up of several thin walled parts, joined together by suitable means. The reason for that practice was the high stresses introduced into the parts when attempting to machine them with conventional milling methods in reasonable time. A recent trend, aimed at reducing cost and weight, and increasing accuracy and fatigue life, is to machine these parts using High Speed milling technology. A lot of these parts are made of Aluminum or titanium, and have very thin (0.6 mm) high walls. iMachining 3D, with its fully controlled tool load, and morphing spiral tool paths is the best CAM software for such parts.

Another trend in the Aerospace industry is replacing complex critical structural Aluminum parts with simpler, stronger, and lighter Titanium parts. Here again iMachining 3D, is the best tool for the job.

---

*Manufacturing Engineering March 2008 Vol. 140 No. 3 (Excerpts from High-Speed Machining for Aerospace)