



TIME SAVINGS
70%
... AND MORE!



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



ed m-aerotec GmbH

iMachining for **NX** Reference Guide



Saves 70% and More in CNC Machining Time



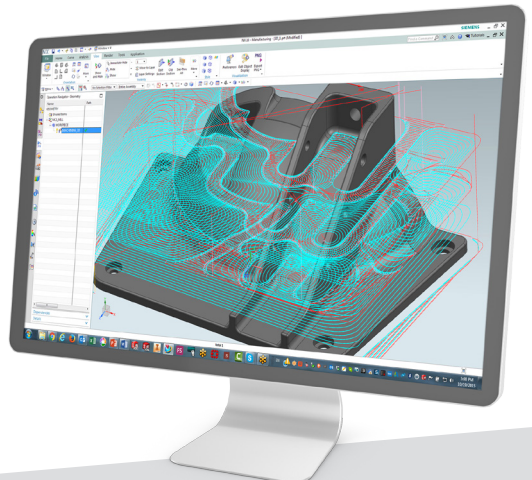
Drastically extends Cutting Tool Life



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



Immense Savings in Programming Time



www.imachining4nx.com



iMachining for NX

Reference Guide

©1995-2017 SolidCAM

All Rights Reserved

Contents

Introduction	1
Installation	1
About this document	4
About iMachining.....	4
Software versions...	6
Conventions used.....	6
 User Interface	 7
Operation Types	7
iMachining 2D	8
iMachining 3D	9
Tool Definition.....	9
Geometry Definition.....	11
User Parameters.....	12
Technology types.....	13
Wizard.....	15
Path Settings.....	16
Cut Levels.....	17
Cutting Parameters	18
Non Cutting Moves	18
Feeds and Speeds	20

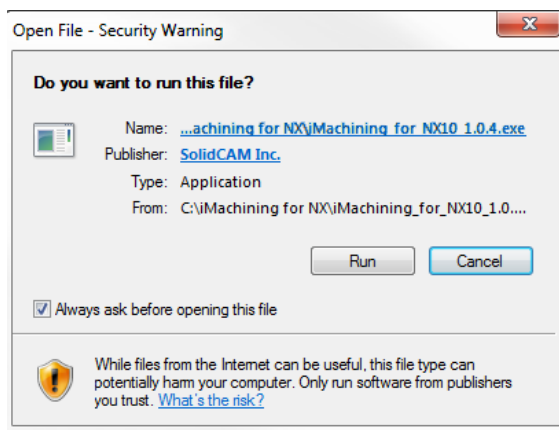
Introduction

Installation

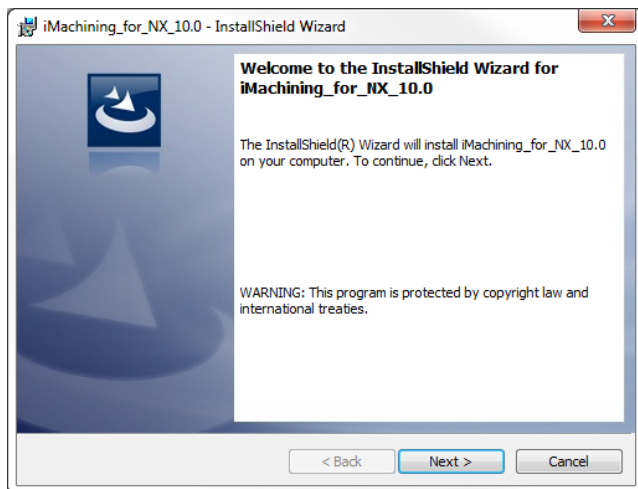
Currently, iMachining for Siemens NX CAM is available for download from a provided link. The version of iMachining for NX you will be provided is relevant to the version of Siemens NX you are currently running such as NX 9, NX 10 or NX 11.

Click the provided link to save the *.exe file, containing your new software, to your computer.

Upon double-clicking the executable file, the Installation Setup Launcher starts and prompts you to either run or cancel the initial setup that prepares the installation for your computer.



After clicking **Run**, the InstallShield Wizard appears to guide you through the installation process.



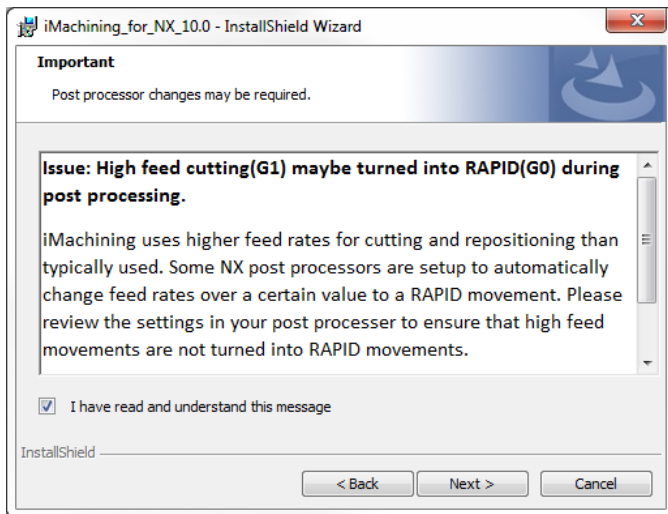
Click **Next** to continue.

In order to use the software, you must agree to the licensing terms and conditions set forth in the iMachining™ END-USER SOFTWARE LICENSE AGREEMENT.



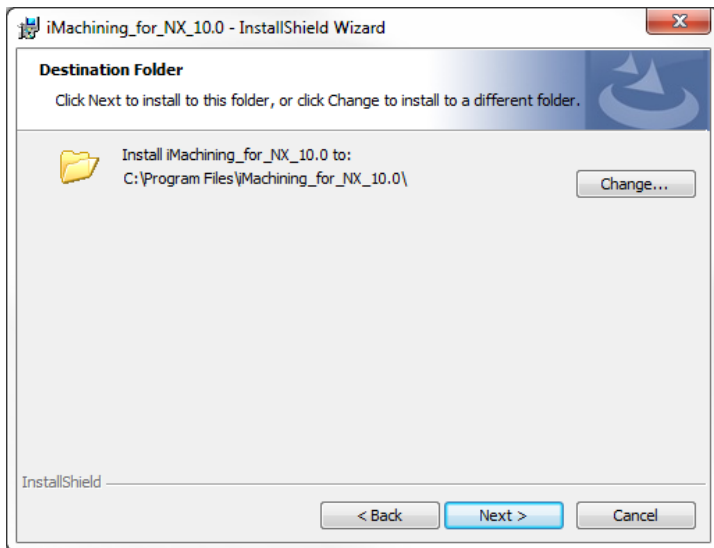
After carefully reading the license agreement, select **I accept the terms in the license agreement** to confirm that you accept the terms and then click **Next** to continue.

The iMachining technology uses higher feed rates for cutting and repositioning than typically used, which may cause issues with some NX CAM post processors. It is important that you read and understand the message displayed, and you should contact your reseller or NX CAM support if you have any questions or concerns.



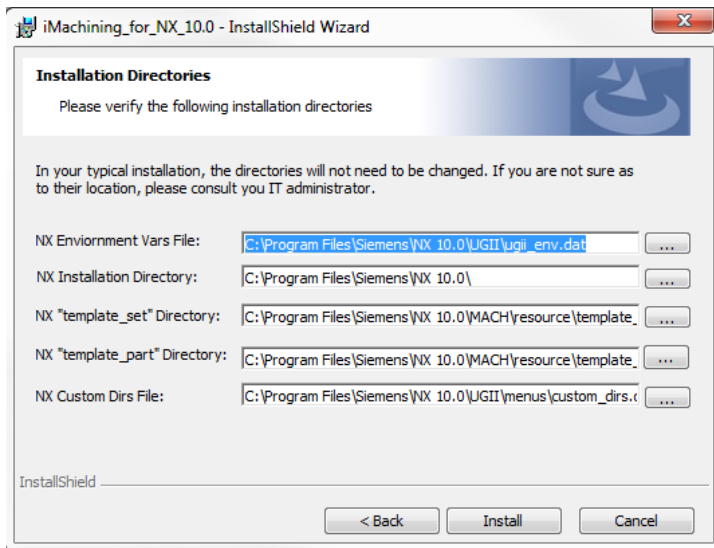
To continue with the installation, you must first confirm that you understand the contents of the message by clicking **I have read and understand this message**.

After clicking **Next**, you can choose the folder to which the software is installed. By default, the InstallShield Wizard will create the appropriate folder on the hard drive of your computer.



Click **Next** to install the software to the default folder.

Finally, you have to verify the locations of the installation directories. For iMachining to function properly, it is important that the paths for the installation and templates are correct. Typically, you should not have to change the directories. If you are unsure, it is recommended to consult with your IT administrator.



At this time, you can click **Install** to install your new software.

About this document

This Reference Guide is intended for Siemens NX CAM users and is designed to provide a basic understanding of the iMachining technology and the information needed to get started with iMachining for NX.

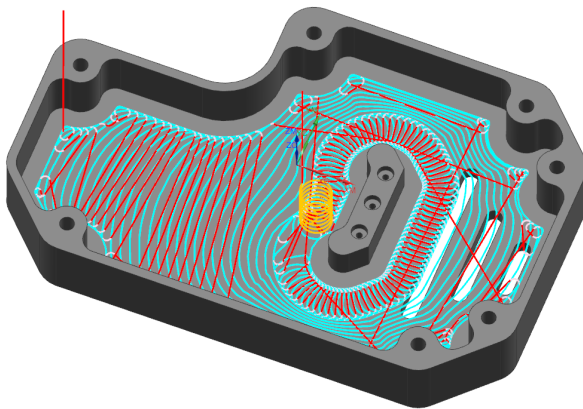
About iMachining

Developed by SolidCAM, iMachining for NX provides an integrated machining solution for Siemens NX CAM. iMachining is an intelligent high speed milling technology, designed to produce fast and safe CNC programs to machine your mechanical parts with **first part success** performance. The word **fast** meaning significantly faster than traditional machining at its best and the word **safe** meaning without the risk of breaking tools or subjecting the machine to excessive wear, all while maximizing tool life.

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

iMachining tool paths

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



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

In some open areas, where the shape is too irregular to completely remove with a single spiral, proprietary topology analysis algorithms and channels are used to subdivide the area

into a few large irregularly shaped sub-areas and then machines each of them by a suitable morphing spiral, achieving over 80% of the volume being machined by spiral tool paths. Since spiral tool paths have between 50% and 100% higher **Material Removal Rate (MRR)** than one-way tool paths, and since iMachining has the only tool path in the industry that maintains a constant load on the tool, it achieves the highest MRR in the industry.

iMachining Technology Wizard

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

Technology Wizard

Cutting conditions Modify cutting conditions

Machining level 6

Step down

Step down

☒ Automatic ☐ User-defined

No. steps	Step down	ACP
1	15.875000	2.127650

Output cutting data

☒ View1 ☐ View2

S (rpm): 10502

F (mm/min): 2957.000

Step over (max): 2.7100

Step over (min): 0.3100

OK Apply Cancel

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

Software versions used for this Reference Guide

The screenshots in this Reference Guide were made using **iMachining for NX 10.0 Version 1.0.4** integrated with **Siemens NX 10.0.2.6** running on Windows 7. If you are running on a different version of Windows, you may notice differences in the appearance of the menus and windows. These differences do not affect the performance of the software.

Conventions used in this Reference Guide

This Reference Guide uses the following typographic conventions:

Bold	This style is used to emphasize iMachining for NX and NX CAM options, commands or basic concepts. For example, clicking the User Parameters button displays the groups and parameters...
<i>Italics</i>	This style is used to emphasize NX CAM related names. For example, the <i>In Process Workpiece</i> (IPW) of NX CAM is a geometric shape...

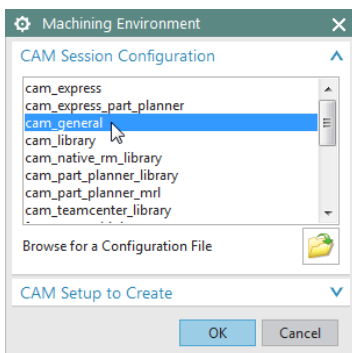
User Interface

SolidCAM is now a Siemens Software and Solution Technology Partner. Using their provided APIs, an iMachining interface has been developed that is fully integrated in Siemens NX, making it familiar to those already using NX CAM. The following user interface topics explain how to get started with iMachining for NX and what parameters are used by its exclusive, patented Technology Wizard.

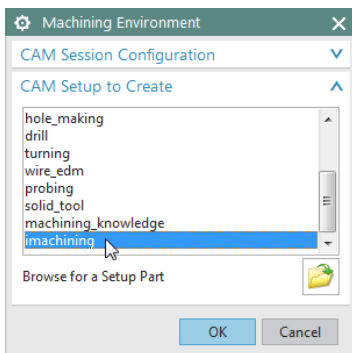
It is assumed that the overall functionality of NX CAM and the commonly used parameters found within its Manufacturing application are known by the user.

Operation Types

After installation and upon starting the Manufacturing application, the Machining Environment dialog box is displayed. In the CAM Session Configuration list, **cam_general** must be selected in order to use the iMachining technology.

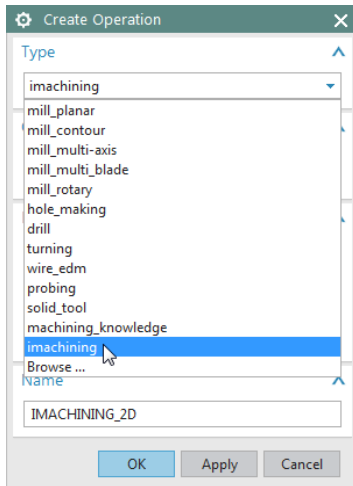


In the Setup list, iMachining appears as an available option to create the initial setup.



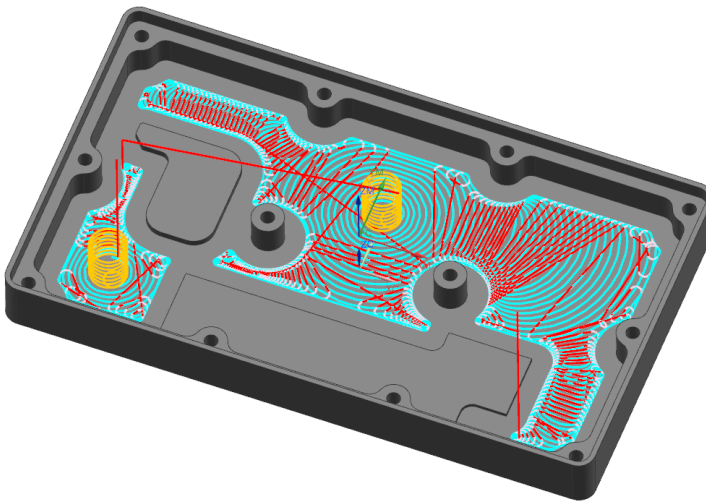
If **imachining** is selected, it will be made the default setup template of the Program parent, and operations that are created will inherit the selection as the initial operation type.

Whether it is selected to create the initial setup or not, iMachining will appear as an available Type option when creating an operation.



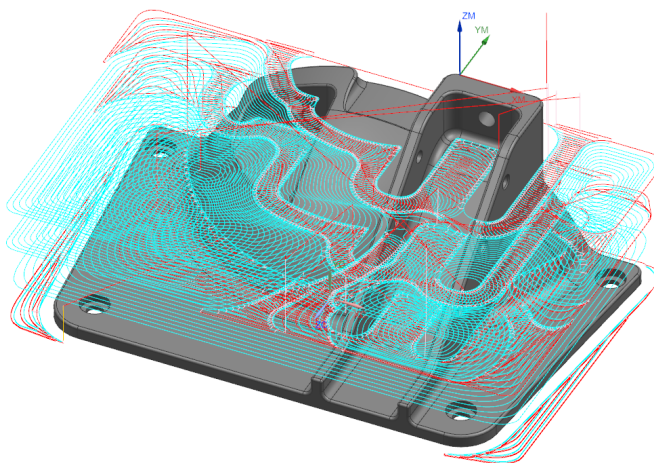
When selected, the Operation Subtype group displays the following operation templates from which to create the new operation:

- **iMachining 2D** – this milling operation subtype efficiently performs the rough, rest and finish machining of prismatic parts and features. It can be used to machine similar geometry types that are machined with the native NX *Volume Based 2.5D Milling* and *Planar Milling* operations.



Besides a reduction in programming time with its knowledge-based Technology Wizard, iMachining 2D can save up to 70% in machining time.

- **iMachining 3D** – this milling operation subtype efficiently performs the roughing, rest machining and semi-finishing of molds, complex 3D parts and 3D prismatic parts. It can be used to machine similar geometry types that are machined with the native NX *Cavity Milling* and *Z-Level Milling* operations.



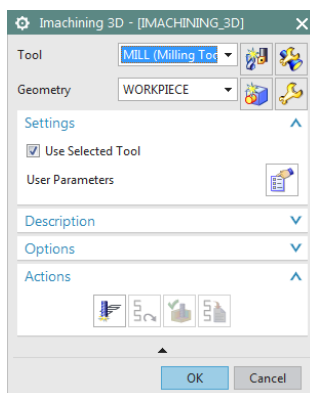
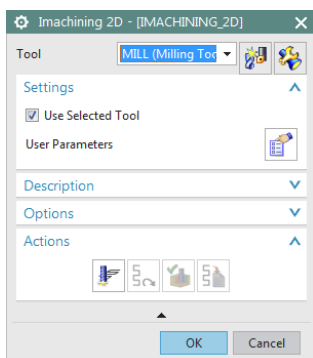
Besides a reduction in programming time with the knowledge-based Technology Wizard of iMachining 2D, iMachining 3D can save up to 90% in machining time.

iMachining is considered a custom NX Open program, so both operation subtypes use the *Mill User-Defined* processor to generate its sophisticated tool paths.

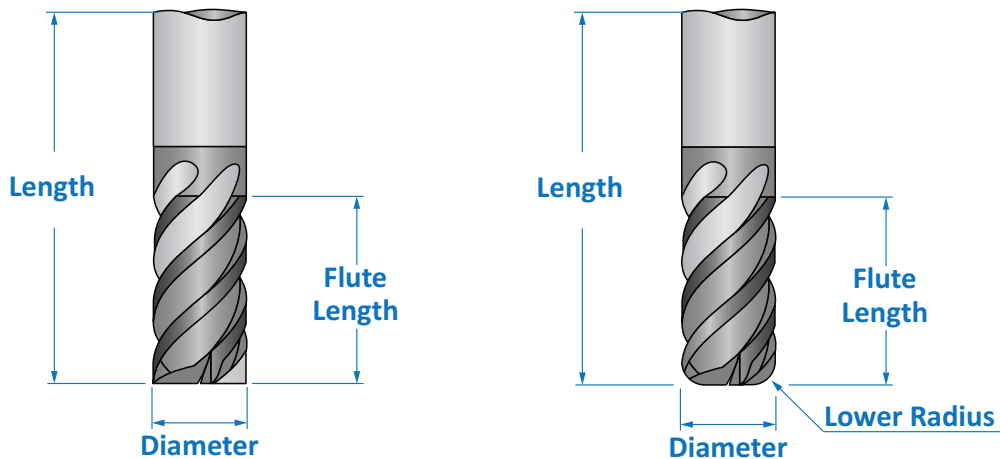
Since iMachining 3D tool paths are generated according to the algorithms of iMachining 2D and the Technology Wizard, most iMachining specific parameters are common between both operation subtypes.

Tool Definition

The Tool group, common to most NX CAM operations, is currently not available within iMachining operations. All tools must be created and inherited.



It is important to note that iMachining supports only the MILL tool subtype. Milling tools considered appropriate are characterized by flat end mills and bull nose mills (whereas the lower radius is smaller than the radius of the tool).



When defining tools for use in iMachining operations, the Technology Wizard uses the data defined for the following parameters:

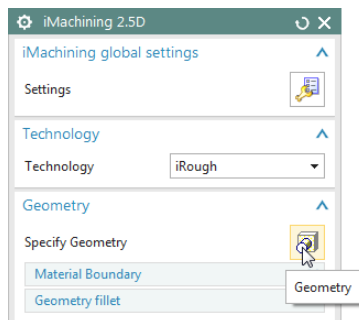
- **Diameter** – the diameter is used in conjunction with the specified cutting speed to determine the optimal spindle speed.
- **Lower Radius** – when a lower radius is defined, tool paths are automatically adjusted to avoid potentially dangerous cusps (uncut material) on planar floors.
- **Length** – chip thickness is modified according to the length minus the distance to which the tool is inserted into the holder (chip thickness is reduced as length increases).
- **Flute Length** – the flute length is used to automatically determine the number of steps that are needed to achieve the total cut depth.
- **Flutes** – the number of flutes determines the appropriate chip thickness per flute.

The tool material is used to further calculate machining data.

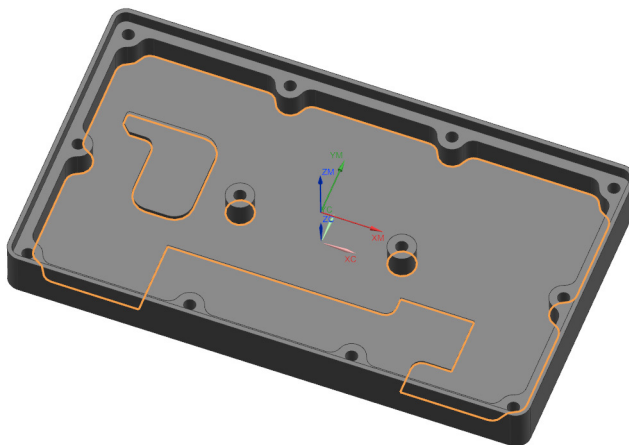
Geometry Definition

iMachining 2D geometry

The Geometry group, common to most NX CAM operations, is available within the iMachining 2D operation subtype.



iMachining 2D follows 2D boundaries to remove material along vertical walls, or walls that are normal (parallel) to the tool axis.



The material is removed in planar levels that are perpendicular to the fixed tool axis. The areas on the part to be machined can include planar islands and planar floors that are normal to the tool axis.

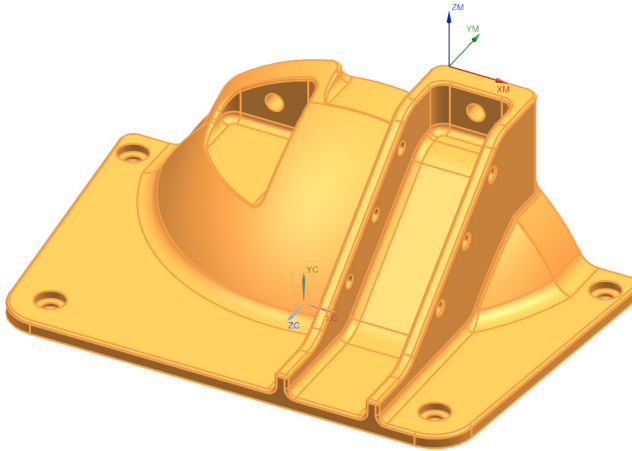
When defining 2D boundaries for use in iMachining 2D operations, the geometry can consist of a single chain or a combination of chains. For example, a single chain represents a pocket (or closed area). Combined chains represent a pocket with island or a pocket with internal open chain that is used for safe tool entry.

iMachining 2D recognizes a 2D boundary as a pocket that can be closed, open or semi-open (containing open edges). An unlimited number of pocket configurations are supported in iMachining 2D.

iMachining 3D geometry

The Geometry group, common to most NX CAM operations, is currently not available within the iMachining 3D operation subtype. All geometry elements must be created and inherited.

The solid body of the Part geometry must be defined for use in iMachining 3D operations.



Large volumes of material are removed in planar levels that are perpendicular to the fixed tool axis. The Part geometry can be contoured or planar.

iMachining 3D analyzes the Part geometry and recognizes all its features and depths. All the volumes that need to be removed are subdivided into milling regions. Using sophisticated analysis algorithms, the optimal ordering of those isolated milling regions is performed and roughing tool paths are generated in thick planar slices (Step down) followed by rest roughing tool paths in thin planar slices (Step-up). This sequence is repeated until the last milling region is machined.

It is important to note that the first iMachining 3D operation for producing a part also requires a Blank geometry as input. For subsequent operations, iMachining 3D utilizes the *In Process Workpiece* (IPW) of NX CAM to cut only the material remaining from previous operations. During calculation of the tool path, the IPW is also dynamically updated by each cutting move to reflect the exact geometric shape at every stage of the machining process so no time is wasted on air cutting.

User Parameters

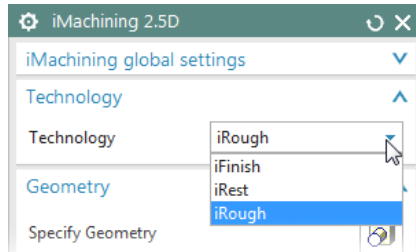
In the Settings group, clicking the  button displays the groups and parameters typical of most NX CAM operations as well as those specific to iMachining.

For both operation subtypes, the **Technology type** must first be specified to determine the functionality of the operation.

Technology types

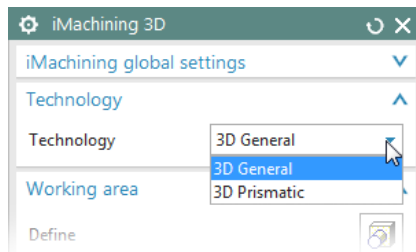
iMachining 2D

Currently, iMachining 2D enables you to define a roughing (**iRough**), rest machining (**iRest**) or finishing (**iFinish**) operation according to the specified Technology type. Each type has its own set of default offset values and supports a different range of tool path options.

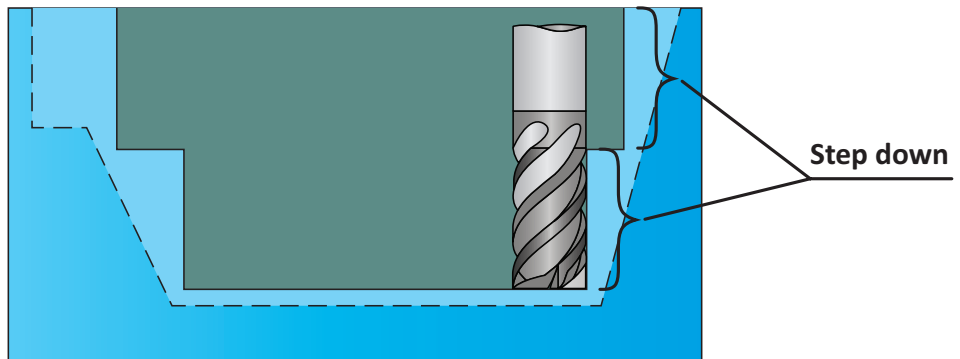


iMachining 3D

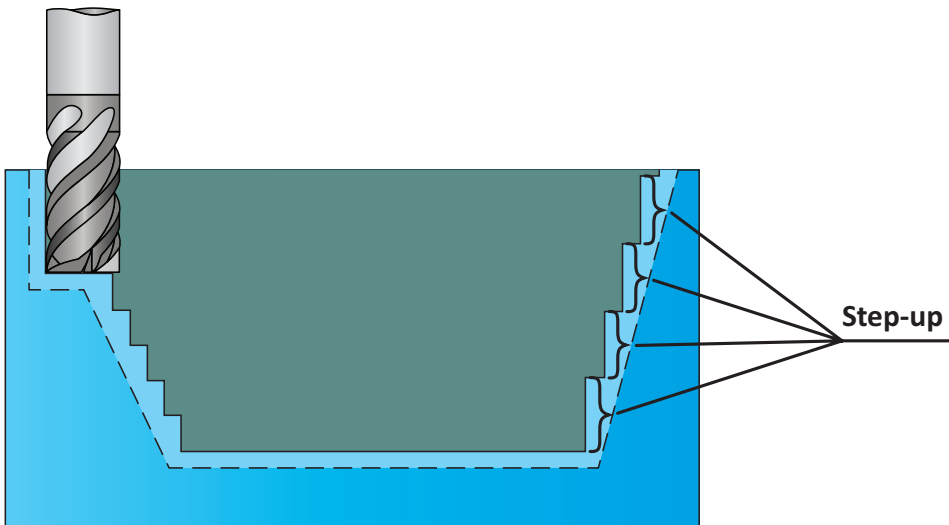
Currently, iMachining 3D enables you to define a roughing, rest machining and semi-finishing operation for either general shaped 3D parts (**3D General**) or 3D prismatic parts (**3D Prismatic**).



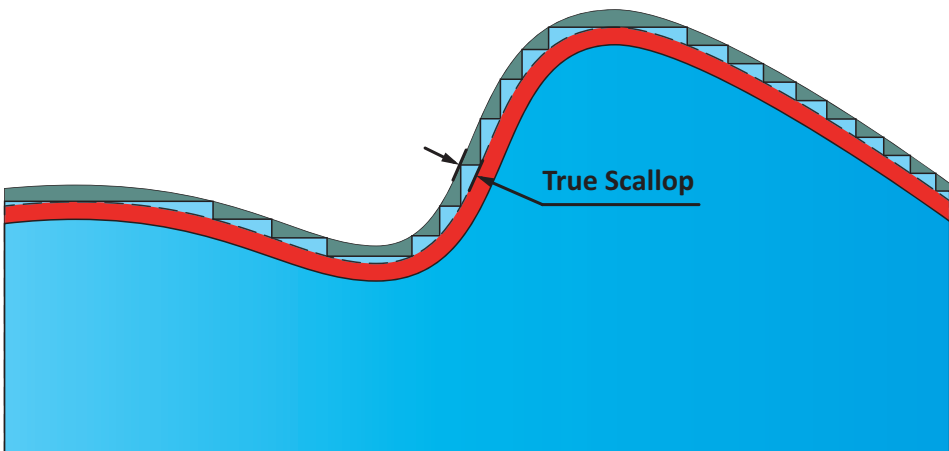
Using the proven algorithms of iMachining 2D and intelligent localized machining, roughing tool paths are generated first with deep Step down passes.



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.




According to the local contour of each individual surface of general shaped 3D parts, the height of the Step-up tool path passes change dynamically in order to maintain the specified **Scallop** value throughout the operation. Every scallop produced is therefore a **True Scallop**.



The **Scallop** parameter enables iMachining 3D to achieve its **minimum machining** feature on contoured surfaces.

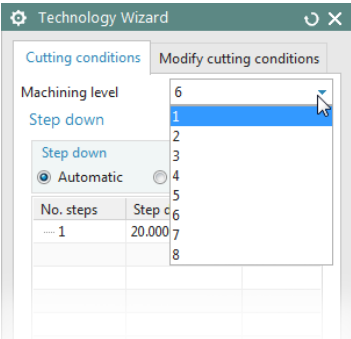
Wizard

The Technology Wizard is an algorithm for producing on-the-fly Cutting conditions for the active iMachining operation, taking into account the Geometry, Tool and Cut Levels definitions as well as the machine specifications and work material properties.

In the Wizard group, clicking the  button displays the Technology Wizard dialog box that enables you to view the current Cutting conditions.

Machining level

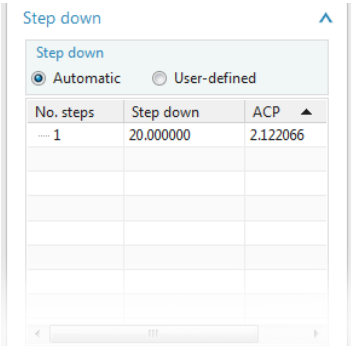
The Machining level enables you to select from calculated sets of Cutting conditions, which provides a convenient and intuitive way to control the Material Removal Rate (MRR). Each ascending number increases MRR and machining aggressiveness.



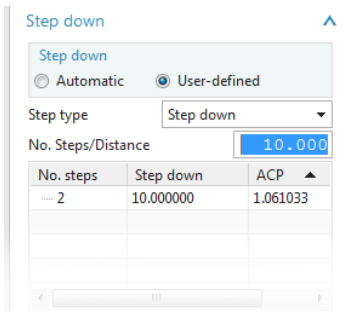
Step down

There are two methods which can be used for calculating Step down.

By default, the Wizard uses the **Automatic** option to calculate the optimal Step down value(s) according to the Tool information, Cut Levels and Floor offset (if any) defined for the operation.



When the **User-defined** option is chosen, the Step down can be defined by specifying a distance or by manually entering the number of steps that are needed to achieve the total cut depth.

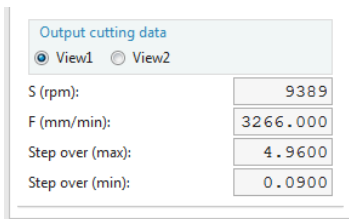


No. steps	Step down	ACP
2	10.000000	1.061033

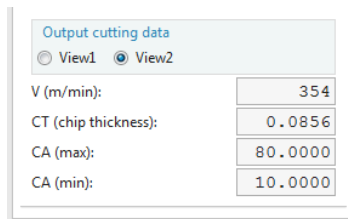
The output grid displays the No. steps, the Step down value and the number of Axial Contact Points (ACPs) calculated automatically by the Wizard. Rows are created for each Step down distance that is not the same.

Output cutting data

This section displays two sets of data related to the current Cutting conditions (the spindle speed and feed rate of the tool, the step over range, the material cutting speed, chip thickness (CT), and the cutting angle range).



S (rpm):	9389
F (mm/min):	3266.000
Step over (max):	4.9600
Step over (min):	0.0900

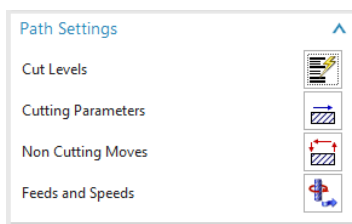


V (m/min):	354
CT (chip thickness):	0.0856
CA (max):	80.0000
CA (min):	10.0000

The Output cutting data should be monitored when selecting a Machining level.

Path Settings

The Path Settings group, common to most NX CAM operations, is available within iMachining operations.

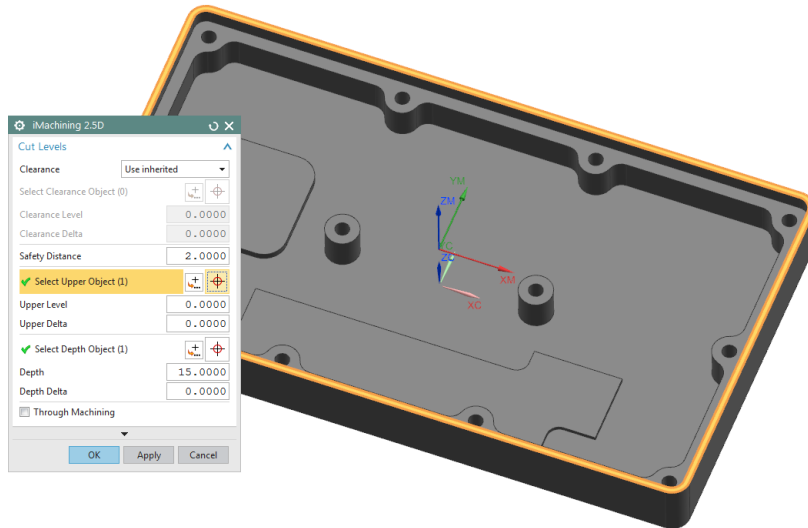


Cut Levels	
Cutting Parameters	
Non Cutting Moves	
Feeds and Speeds	

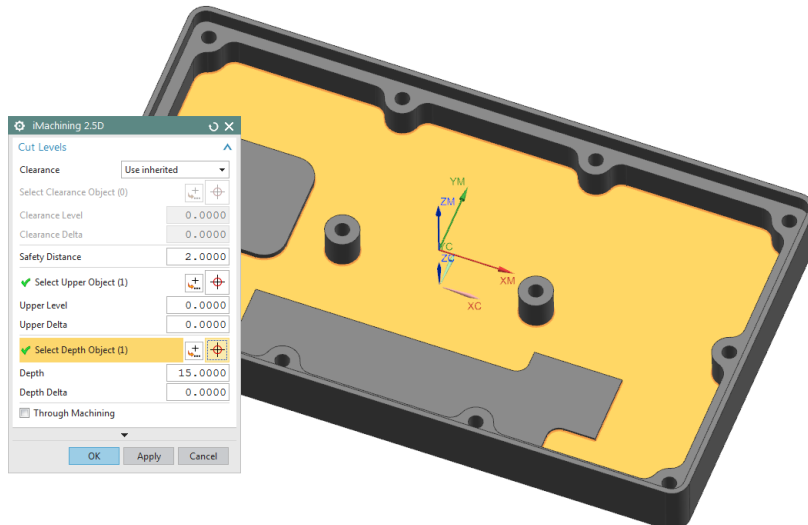
Cut Levels

In iMachining operations, the Cut Levels can be picked in the graphics window using the solid body data. There are only two required selections.

The **Select Upper Object** command enables you to specify what object to use for the Upper Level definition. This is the Z-level at which the machining starts.



The **Select Depth Object** command enables you to specify what object to use for the Lower Level (Depth) definition. This is the Z-level below which the tool does not machine in both operation subtypes.



Cutting Parameters

The functionality and available parameters within the Cutting Parameters dialog box is determined by the iMachining operation subtype and the specified Technology type. The Cutting Parameters enable you to define several technological parameters such as offsets and other data that are specific to iMachining 2D or iMachining 3D.

In iMachining 2D for example, the iRough Technology type displays the tabs and groups that are used in roughing calculations.

The screenshot shows the 'Cutting Parameters' dialog box with the 'iRough Data' tab selected. The 'Technology' section is active, showing 'Step down' set to 20.0000 and a 'Complete Z-Level' checkbox. The 'Offsets' section shows 'Wall / island offset' at 0.2400 and 'Floor offset' at 0.0000. The 'Cutting angles' section has 'Angle' selected, with 'Min cutting angle' at 10.0000 and 'Max cutting angle' at 64.8000. The 'Morphing spiral controls' section shows 'Efficiency' at 6 and 'Entry rate' at 2. The 'Rest material' section is empty. At the bottom are 'OK', 'Apply', and 'Cancel' buttons.

Section	Parameter	Value
Step down	Step down	20.0000
	Complete Z-Level	<input type="checkbox"/>
Offsets	Wall / island offset	0.2400
	Floor offset	0.0000
Cutting angles	Angle / Step over	Angle (selected)
	Min cutting angle	10.0000
	Max cutting angle	64.8000
Morphing spiral controls	Efficiency	6
	Entry rate	2
Rest material		

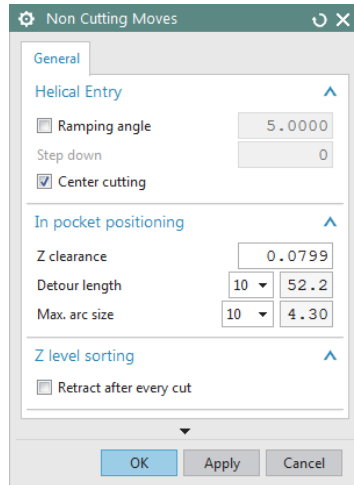
Since iMachining 3D generates tool paths according to the algorithms of iMachining 2D, many of the parameters are also common between both operation subtypes.

Non Cutting Moves

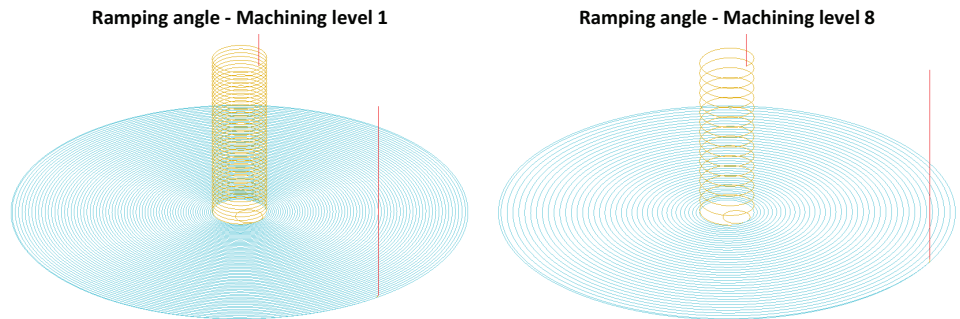
In iMachining operations, Non Cutting Moves define the approach and retreat of the tool as well as the linking between and sorting of tool paths.

The iMachining technology automatically calculates the best method to enter and exit the cut according to the geometry.

For example, when the geometry is closed in a roughing operation, the tool enters the material in a spiral movement according to the Ramping angle parameter in the Helical Entry group.



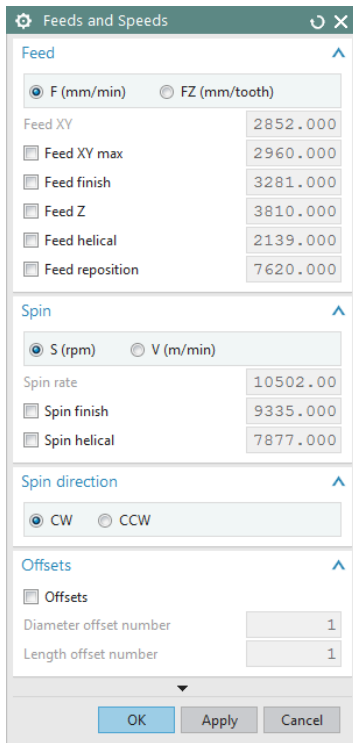
The Ramping angle parameter defines the aggressiveness of the descent angle by which the tool enters the material. By default, the Wizard automatically calculates the aggressiveness of the descent angle based on material hardness and the Machining level selection.



It is important to note that when using more aggressive descent angles, cooling can become a concern. Larger values will generate more heat and proper cooling should be applied when necessary.

Feeds and Speeds

When the Wizard is enabled, the Feed and Spin data for the tool are automatically calculated according to the selected set of Cutting conditions.



The image shows a software dialog box titled "Feeds and Speeds". It is organized into four main sections: "Feed", "Spin", "Spin direction", and "Offsets". Each section has a title bar with an upward arrow. The "Feed" section has two radio buttons: "F (mm/min)" (selected) and "FZ (mm/tooth)". Below are several parameters with checkboxes and numerical values: "Feed XY" (2852.000), "Feed XY max" (2960.000), "Feed finish" (3281.000), "Feed Z" (3810.000), "Feed helical" (2139.000), and "Feed reposition" (7620.000). The "Spin" section has two radio buttons: "S (rpm)" (selected) and "V (m/min)". Below are "Spin rate" (10502.00), "Spin finish" (9335.000), and "Spin helical" (7877.000). The "Spin direction" section has two radio buttons: "CW" (selected) and "CCW". The "Offsets" section has a checkbox for "Offsets" (unchecked) and two input fields: "Diameter offset number" (1) and "Length offset number" (1). At the bottom are three buttons: "OK", "Apply", and "Cancel".

Feed	
<input checked="" type="radio"/> F (mm/min)	<input type="radio"/> FZ (mm/tooth)
Feed XY	2852.000
<input type="checkbox"/> Feed XY max	2960.000
<input type="checkbox"/> Feed finish	3281.000
<input type="checkbox"/> Feed Z	3810.000
<input type="checkbox"/> Feed helical	2139.000
<input type="checkbox"/> Feed reposition	7620.000

Spin	
<input checked="" type="radio"/> S (rpm)	<input type="radio"/> V (m/min)
Spin rate	10502.00
<input type="checkbox"/> Spin finish	9335.000
<input type="checkbox"/> Spin helical	7877.000

Spin direction	
<input checked="" type="radio"/> CW	<input type="radio"/> CCW

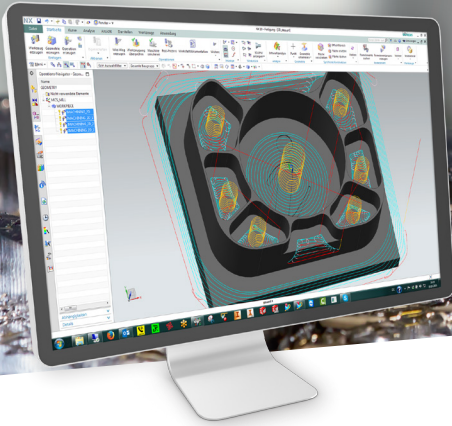
Offsets	
<input type="checkbox"/> Offsets	
Diameter offset number	1
Length offset number	1

OK Apply Cancel

Override check boxes are available for most parameters, but Feed XY and Spin rate are locked because they need to be synchronized when using the Wizard. If you would like to manually enter a preferred set of values, the Wizard can be disabled to open the fields for editing.

However, when using the iMachining technology, it is highly recommended to leave the Wizard on and to utilize the optimal feed rates and spindle speeds that are provided since these values are calculated according to many factors.

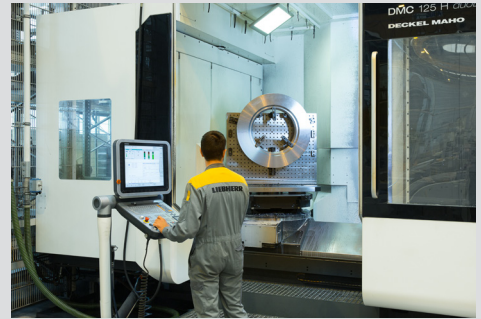
The Power of iMachining right
inside NX – a great Synergy!



*i*machining[®]

The Revolutionary Milling Technology integrated in **NX**
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