

CNC TRAINING

LEVEL 1 – INVENTOR HSM TRAINING

The Integrated CAM Solution for Autodesk Inventor

Inventor CAM is designed from the ground up to work inside Autodesk Inventor, providing a logical extension of the parametric environment into the CAM world. Experienced Autodesk Inventor users will feel right at home working with Inventor CAM and will be able to create high-quality toolpaths within minutes. New users will benefit from the unmatched modelling and simulation capabilities, mechanical design solutions and quickly extend any knowledge gained to the CAM process. This will result in improved design quality and reduced product development time.

Inventor CAM is available in several different levels. Depending on your needs Inventor CAM can generate high quality 2D, 3D, 5-axis milling, and turning toolpaths.

The Inventor CAM interface is designed for CAD users and machinists alike. If you know Autodesk Inventor, getting to know Inventor CAM will be easy and straightforward.

Autodesk Inventor, Inventor CAM Express, Inventor CAM Premium and Inventor CAM Ultimate - copyright Autodesk 2013-2019

Getting Started

Download free educational software links

Inventor - <https://www.autodesk.com/education/free-software/inventor-professional>

Inventor HSM - <https://www.autodesk.com/education/free-software/inventor-hsm-ultimate>

Intro to Inventor HSM

[Intro to Inventor HSM part 1](#)

[Intro to Inventor HSM part 2](#)

[Intro to Inventor HSM part 3](#)

Inventor CAM brings fully embedded Computer-Aided Manufacturing (CAM) capabilities into Autodesk Inventor and Autodesk Inventor LT. There are three levels of Inventor CAM and their capabilities are listed below:

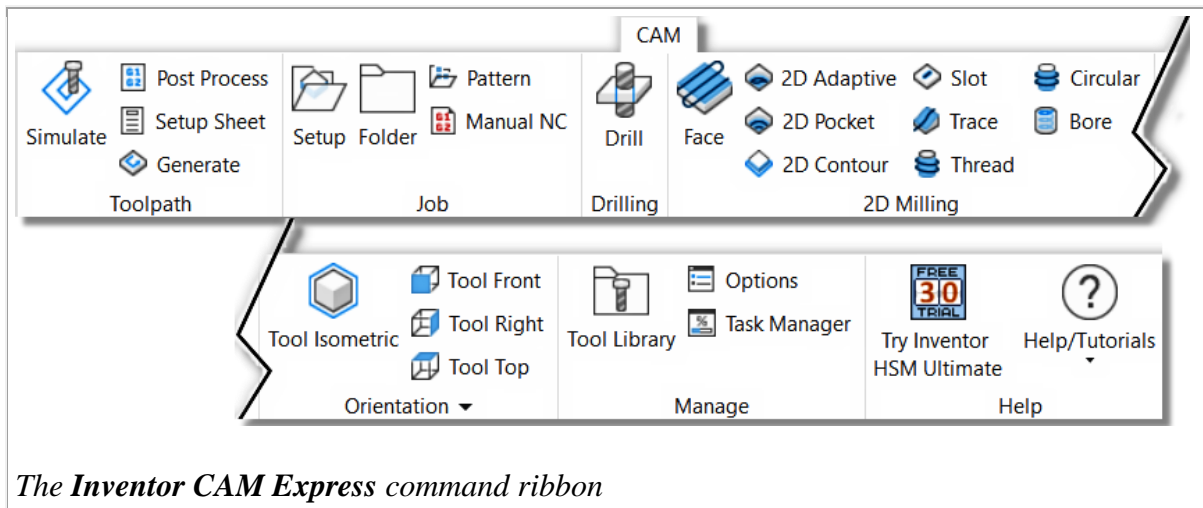
1. **Inventor CAM Express** - 2D/2.5 axis Milling functions for general machining.
2. **Inventor CAM Premium** - 2D and 3D Milling, 3+2 Milling and Turning functions.
3. **Inventor CAM Ultimate** - 2D, 3D, 3+2 Milling, 5x simultaneous Milling and Turning functions.

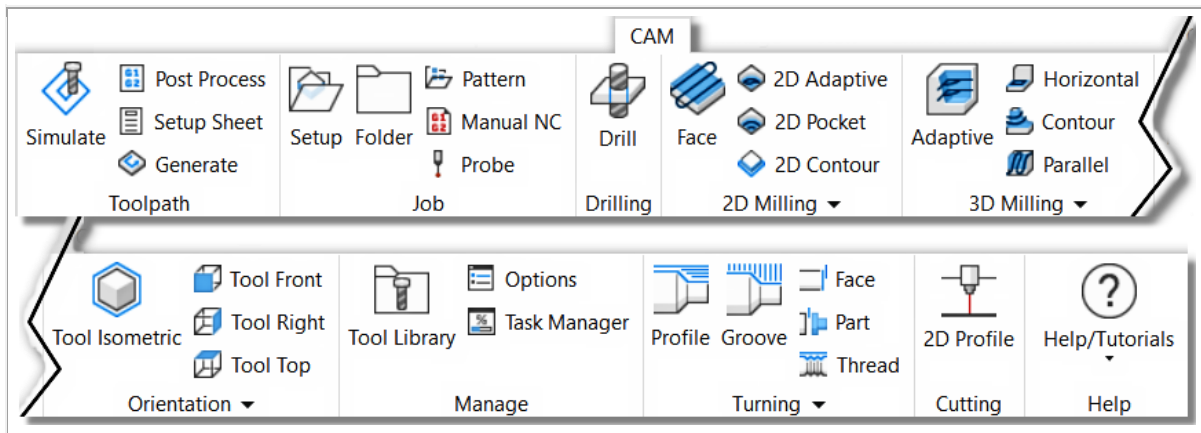
Inventor CAM should be installed on top of Autodesk Inventor. When you start Autodesk Inventor you will notice a CAM tab on the Inventor command ribbon. The commands on the CAM tab become visible, active, and ready for use after creating or opening an Inventor part or assembly file.

Restriction: Assemblies are *not* supported in Autodesk Inventor LT.

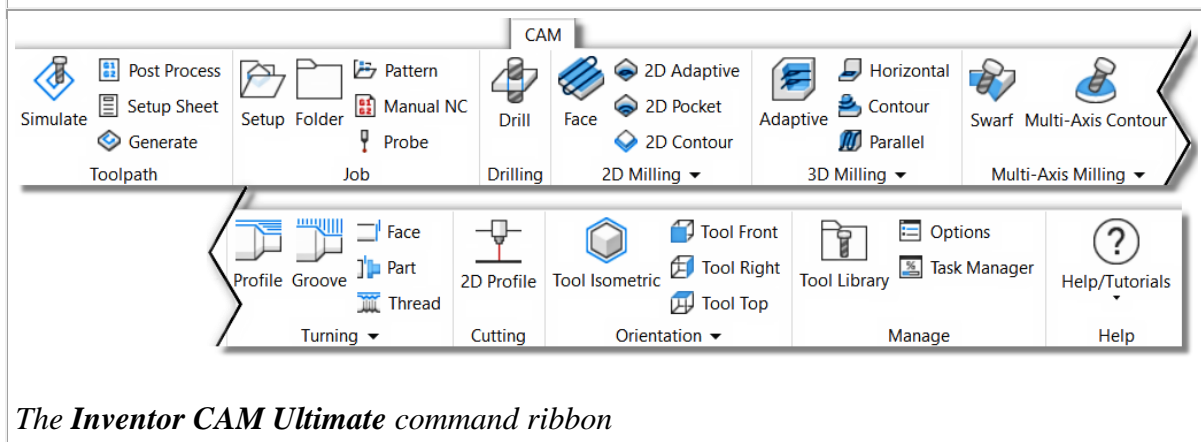
You can also load an existing file of any of the types supported by Autodesk Inventor or LT. These file types include CATIA, SolidWorks, NX, Pro Engineer, SAT, STEP, IGES, and many other industry-standard file formats.

The command ribbons and the commands they offer for the three different versions of Inventor CAM appear below:





The Inventor CAM Premium command ribbon



The Inventor CAM Ultimate command ribbon

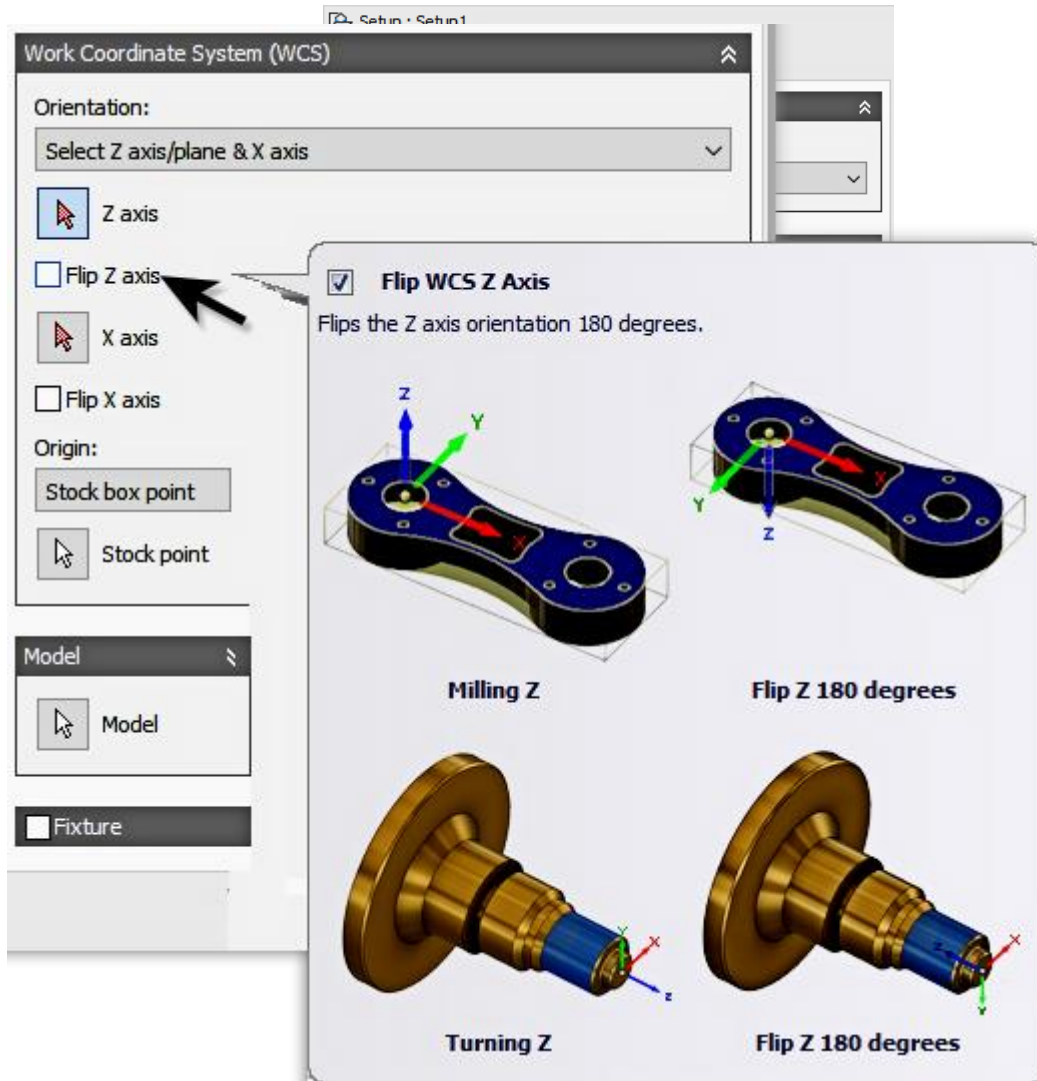
Successful Toolpath Creation

There are several steps you should follow to create your NC programmed part.

- **Setup** - Defines the part orientation, cutting plane, Stock size, XYZ Zero location and Work Coordinate System (WCS) offset.
- **Toolpaths** - Select the appropriate cutting strategy, cutting area, cutting tool and cutting steps.
- **Simulation** - Verify the toolpath meets your needs and cuts the correct areas. Edit the Toolpath Strategy if needed.
- **Post Process** - Select a Post that matches your machine/control and NC output for your machine.

Setup and ToolTips

Setup lets you select the type of machine you will be programming, set your stock size and XYZ zero position. Since you can machine any face on the part, use the WCS parameters to align the axis to your part. ToolTips are a powerful tool for learning about the system parameters. Some ToolTips will have a simple description of the parameter, other will have illustrations to make the point. Hover your cursor over the parameter to see the ToolTip appear. The Illustration on the right shows the ToolTip for the "Flip Z Axis" parameter.



Toolpath Strategies and The CAM Browser

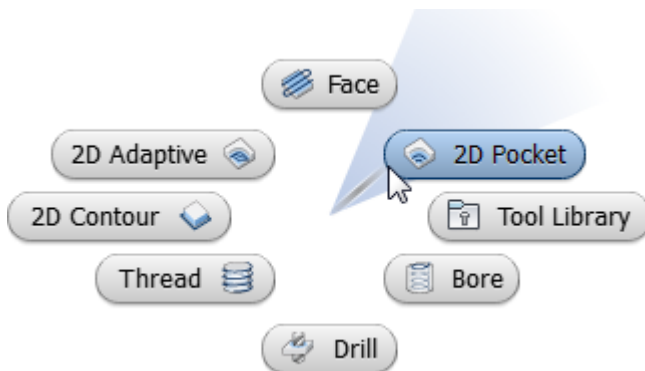
The **CAM Browser** is docked on the left side. It lets you view and modify the machining strategies associated with the current part. The **CAM Browser** becomes active once a part or assembly file is loaded and a toolpath strategy is selected from the CAM ribbon. This replaces the Autodesk Inventor **Model Browser**.

To create your first machining operation, simply select any of the toolpath strategies from the CAM toolbar. The type of toolpath required naturally depends on the geometry of your part. For a description of the individual machining strategies, please refer to the Inventor CAM Help topics: [2D Machining Strategies](#) and [3D Machining Strategies](#).

After creating your Setup, you can select a Toolpath Strategy by clicking the appropriate icon from the command ribbon.

In this example lets pick CAM tab > 2D Milling panel > 2D Pocket .

You can also right-click in an empty portion of the graphics window to display the Inventor marking menu and then select the appropriate 2D Toolpath Strategy.

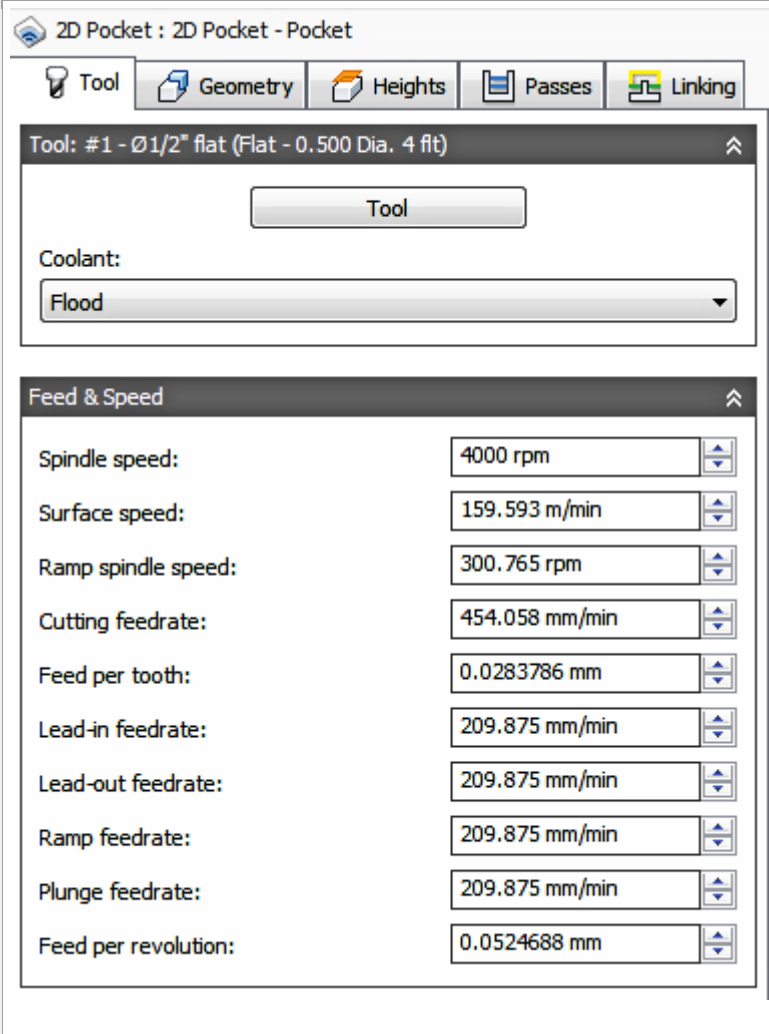
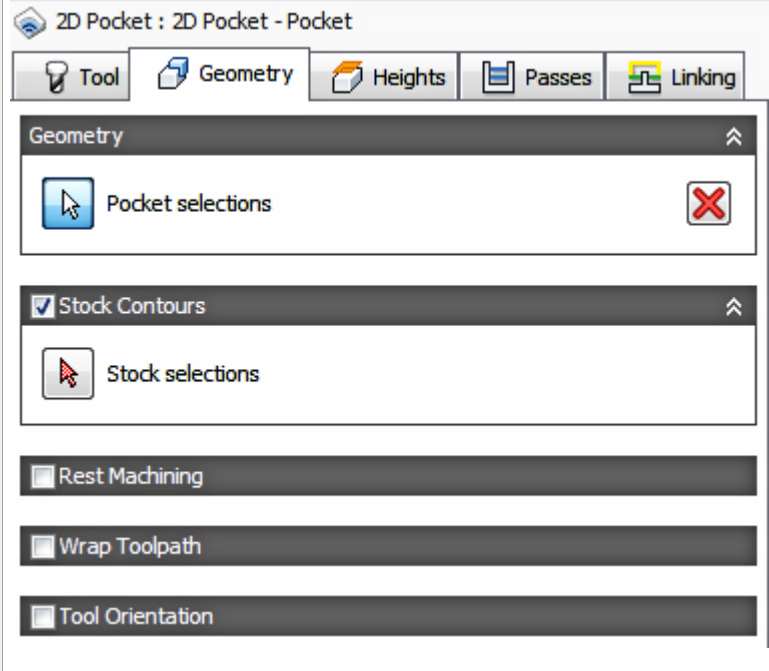


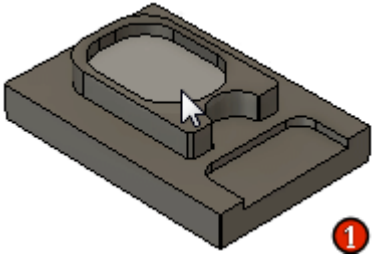
The Operation dialog box will display in the CAM Browser at the left side of the graphics window. In its title bar is the name of the strategy selected. Just to the right of the strategy name is the operation number. Since this is the first 2D pocket operation for the part, the name displays as *2D Pocket1*. The next 2D pocket operation will display as *2D Pocket2*, and so on. This naming convention applies to all setup and machining strategies in Inventor CAM.

Toolpath Dialogs

All toolpath dialogs follow a similar format. You will find 5 Tabs at the top of the dialog. This is an over view of each tab.

Note: Be sure to look at the ToolTips for additional parameter information.

	<p>Tool Tab</p> <ul style="list-style-type: none"> • Select a library tool or create a new tool • Set Coolant type • Set the Feeds & Speeds appropriate for your tool and material
	<p>Geometry Tab</p> <ul style="list-style-type: none"> • Select the area or edges to be machined • Select containment boundaries • Change the Tool Orientation for Indexing or 2+3 machining

	
<p>2D Pocket : 2D Pocket - Pocket</p> <p> <input type="checkbox"/> Tool <input checked="" type="checkbox"/> Geometry <input checked="" type="checkbox"/> Heights <input type="checkbox"/> Passes <input type="checkbox"/> Linking </p> <p>Clearance Height</p> <p>Retract height: [Dropdown]</p> <p>Clearance height offset: 10 mm</p> <hr/> <p>Retract Height</p> <p>Model top: [Dropdown]</p> <p>Retract height offset: 5 mm</p> <hr/> <p>Feed Height</p> <p>Top height: [Dropdown]</p> <p>Feed height offset: 3 mm</p> <hr/> <p>Top Height</p> <p>Model top: [Dropdown]</p> <p>Top offset: 0 mm</p> <hr/> <p>Bottom Height</p> <p>Selected contour(s): [Dropdown]</p> <p>Bottom offset: 0 mm</p>	<p>Heights Tab</p> <ul style="list-style-type: none"> • Clearance Height is for the tools approach position • Retract Height is the rapid position above the part • Feed Height is the starting feed position • Top Height is the top of the surface being machined • Bottom Height is the final cut depth <p>If the Face or Edge you selected is the final cutting depth, no depth position needs to be specified. Most times you do not need to make any adjustments, depending on the geometry selected for machining.</p>

CREATE CHANGE

2D Pocket : 2D Pocket - Pocket

Tool | Geometry | Heights | Passes | Linking

Passes

Tolerance:

Sideways compensation:

Minimum cutting radius:

Finishing passes

Preserve order

Both ways

Maximum stepover:

Use morphed spiral machining

Allow stepover cusps

Smoothing deviation:

Multiple Depths

Maximum roughing stepdown:

Finishing stepdowns:

Finishing stepdown:

Wall taper angle (deg):

Use even stepdowns

Order by depth

Order by step

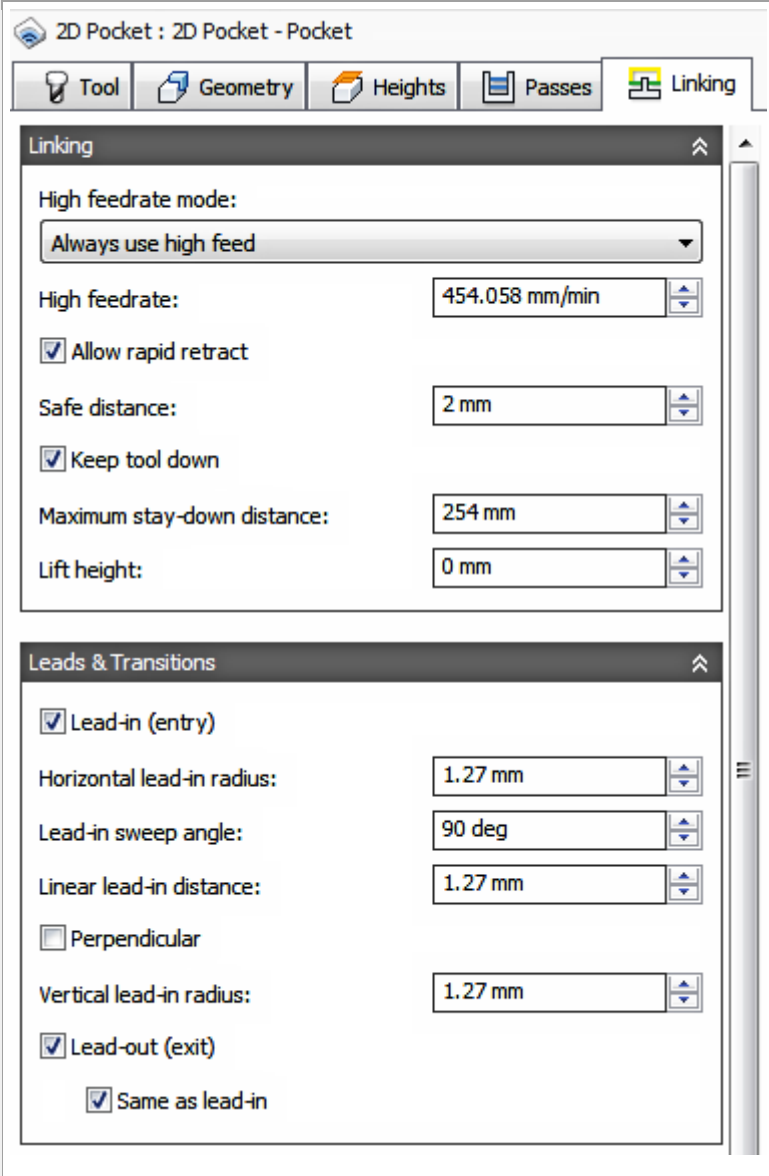
Stock to Leave

Smoothing

Feed Optimization

Passes Tab

- Passes controls the side cut parameters
- Multi Depths controls multiple step downs into the part
- Stock to leave for future cuts or finish passes
- Smoothing will filter multiple moves into a combined single move (line and arc filtering)



The screenshot shows the CAM Browser interface for a 2D Pocket toolpath. The 'Linking' tab is active, displaying the following settings:

- High feedrate mode: Always use high feed
- High feedrate: 454.058 mm/min
- Allow rapid retract
- Safe distance: 2 mm
- Keep tool down
- Maximum stay-down distance: 254 mm
- Lift height: 0 mm

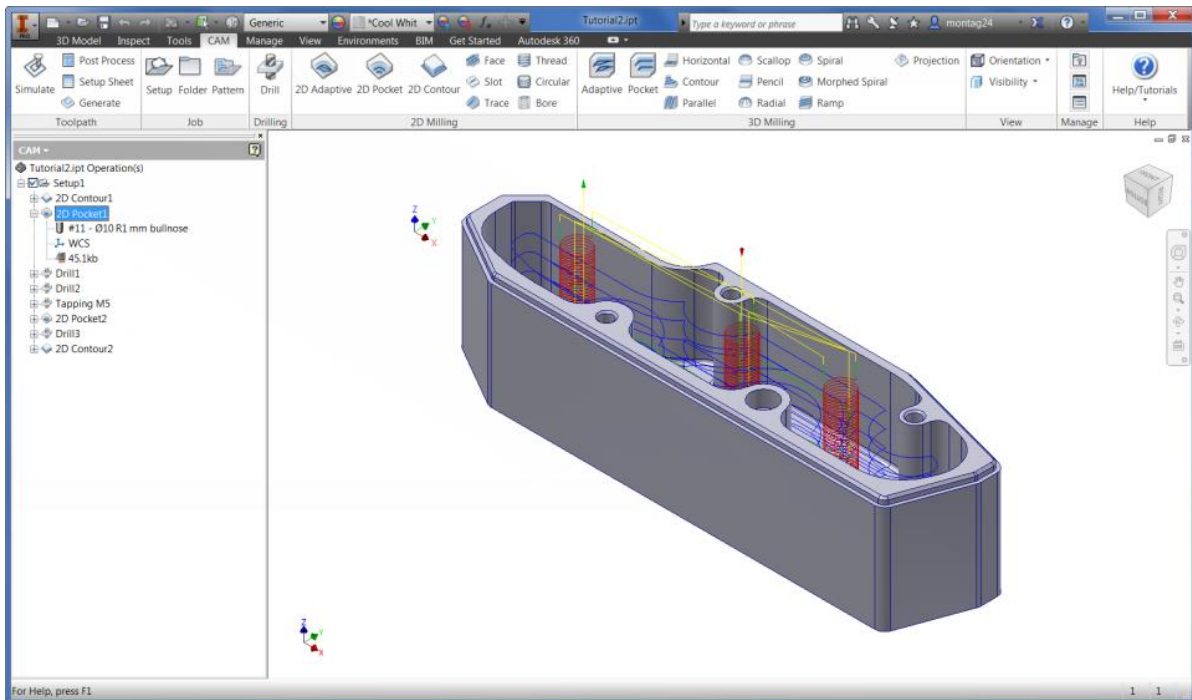
The 'Leads & Transitions' tab is also visible, displaying the following settings:

- Lead-in (entry)
- Horizontal lead-in radius: 1.27 mm
- Lead-in sweep angle: 90 deg
- Linear lead-in distance: 1.27 mm
- Perpendicular
- Vertical lead-in radius: 1.27 mm
- Lead-out (exit)
- Same as lead-in

To the right of the interface, the 'Linking Tab' is defined by the following list:

- Linking determines how the tool moves from one cut, to the next cut
- Sets the conditions for when, or if, the tool should retract
- Leads controls how the tool will lead onto, or off of the cut

A view of the CAM Browser containing toolpaths, with the 2D Pocket toolpath selected.



Toolpath Simulation

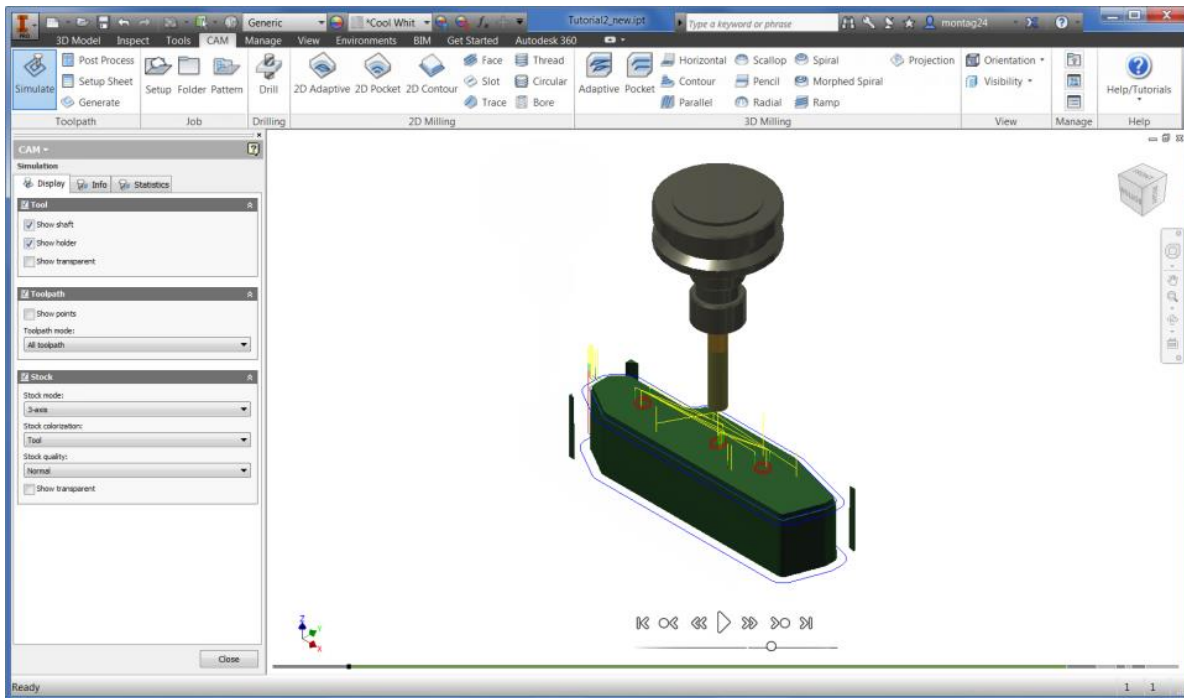
To verify the toolpath, select one or more operations from the CAM Browser (multiple operations can be selected by pressing and holding the **Ctrl** key while clicking with the mouse), and then

CAM tab ► Toolpath panel ► Simulate  on the CAM ribbon.

Stock Simulation

To invoke the solid simulation, enable the **Stock** check box in the **Simulation** dialog.

Use the player controls at the bottom of the graphics window to Play, Stop, Rewind or Step thru the toolpath simulation. The bottom slider controls the speed and direction (Forward or Backwards).



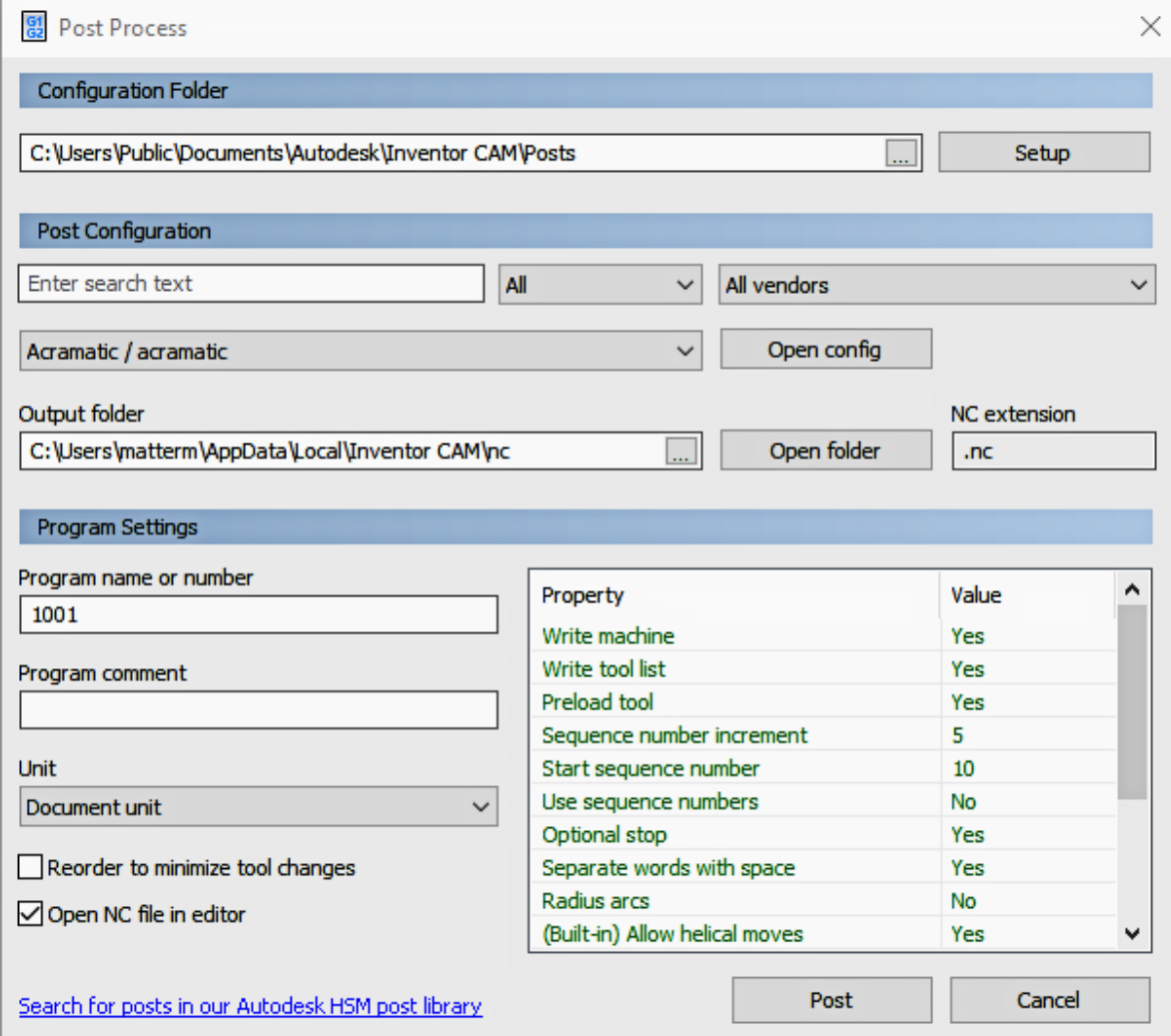
Post Processing

Inventor CAM ships with a number of customizable post processors that can be invoked by selecting one or more operations from the **CAM Browser**, and clicking

CAM tab ➤ Toolpath panel ➤ Post Process  on the CAM ribbon.

Search thru the list for your Machine brand or Control brand. If you don't see your machine or control, click the link at the bottom of the dialog to visit our on-line Post Processor library.

CREATE CHANGE



Post Process

Configuration Folder

C:\Users\Public\Documents\Autodesk\Inventor CAM\Posts Setup

Post Configuration

Enter search text All All vendors

Acramatic / acramatic Open config

Output folder NC extension

C:\Users\matterm\AppData\Local\Inventor CAM\nc Open folder .nc

Program Settings

Program name or number: 1001

Program comment:

Unit: Document unit

Reorder to minimize tool changes

Open NC file in editor

Property	Value
Write machine	Yes
Write tool list	Yes
Preload tool	Yes
Sequence number increment	5
Start sequence number	10
Use sequence numbers	No
Optional stop	Yes
Separate words with space	Yes
Radius arcs	No
(Built-in) Allow helical moves	Yes

[Search for posts in our Autodesk HSM post library](#) Post Cancel

Log Messages

If an operation in the **CAM Browser** is overlaid with an orange checkmark, it indicates that the operation could not be generated successfully. To see a description of the problem or error, right-click the operation and select **Show Log** from the pop-up context menu. The log is displayed in a dialog box and explains what went wrong.

Associativity and Regeneration


When you define operations in Inventor CAM, all relations to the model are *associative*. That means that if you change your model, you will not have to redefine any parameters and selections again - they will persist across model changes and rebuilds. You will, however, have to regenerate your operation whenever a part of the model is modified on which the operation depends.

When a modification of the model triggers invalidation of a toolpath, the regeneration symbol (i.e. a red cross) is overlaid on the corresponding operation and toolpath nodes in the **CAM**

Browser. If you try to use an invalidated toolpath, you are notified that it requires regeneration.

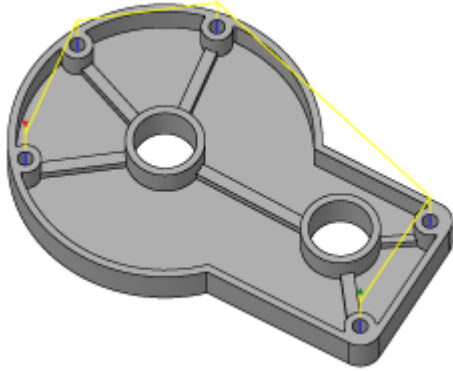
You can regenerate all your operations either at once, or individually, depending on whether you choose **Generate Toolpath (All)** from the right-click pop-up context menu of the **CAM Browser**, or choose **Generate Toolpath...** from the right-click pop-up context menu of a single operation/toolpath node.

Tip: Clicking CAM tab > Toolpath panel > Generate  from the CAM ribbon regenerates all operations that require regeneration.

While regenerating toolpaths, the Inventor CAM **Task Manager** dialog is shown. This shows the progress of any ongoing toolpath calculation, but can be hidden by clicking the **Hide** button so that you can continue working while the regeneration completes. Normally, 2D toolpaths generate in a matter of seconds, but some of the 3D strategies can take a considerable time to calculate, depending on the geometry and tolerances. If you have hidden the **Task Manager** dialog, you can restore its visibility by clicking CAM tab > Manage panel > Task Manager .

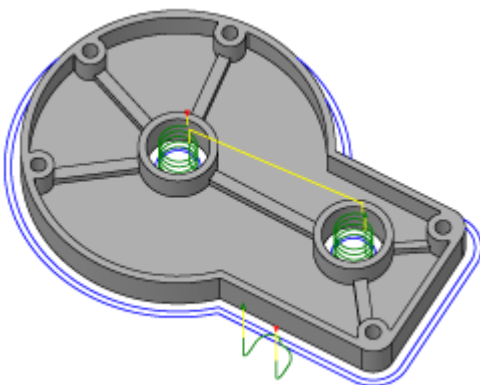
2D Machining Strategies

Drilling and Hole Making



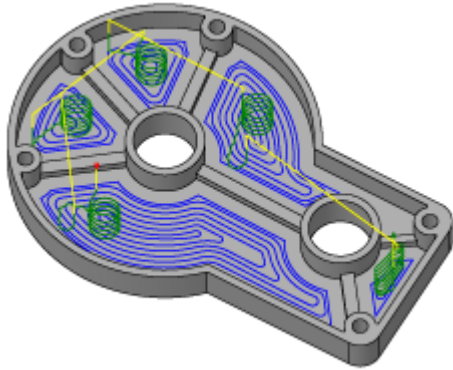
Inventor CAM includes a powerful **Drill** tool for generating drilling, counterboring and tapping operations. The **Circular** strategy is used for milling cylindrical pockets and islands, while the **Thread** operation is used for thread milling cylindrical pockets and islands. The **Bore** operation allows you to bore mill cylindrical pockets and islands by selecting the cylindrical geometry directly. All operations are optimized to minimize tool travel and overall cycle time. Both standard and customized cycles are supported for all point-to-point operations, including spot-drilling, deep drilling with chip break, etc.

Contouring



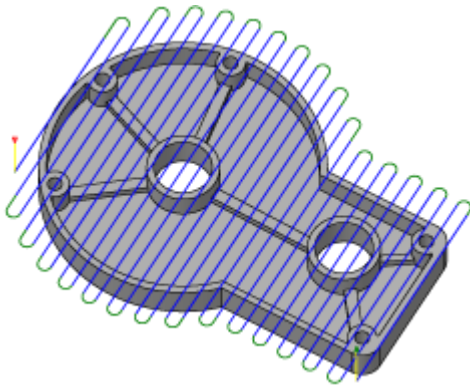
With the **Contouring** strategies, you can easily machine 2D and 3D contours with separate lead-in and lead-out, and with or without tool compensation. Choose multiple roughing and finishing passes and multiple depth cuts for any contour. Machine open and closed contours without creating additional geometry and eliminate sharp motion with corner smoothing.

Pocket



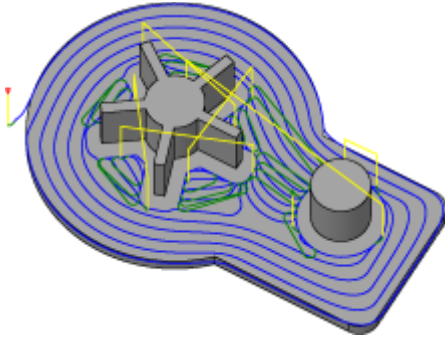
The **Pocket** toolpath is used for machining closed curves both with and without islands. The toolpath starts at the centre of the pocket and works its way outward. The entry can be selected anywhere on the model and includes possibilities for plunge, ramp, or at a pre-drilled position. The special high-speed option creates a smooth toolpath and allows you to specify a maximum tool engagement. As a result, the feedrate can be increased significantly, reducing the machining time and tool wear.

Facing



The **Facing** strategy is designed for quick part facing to prepare the raw stock for further machining. It can also be used for clearing flat areas in general.

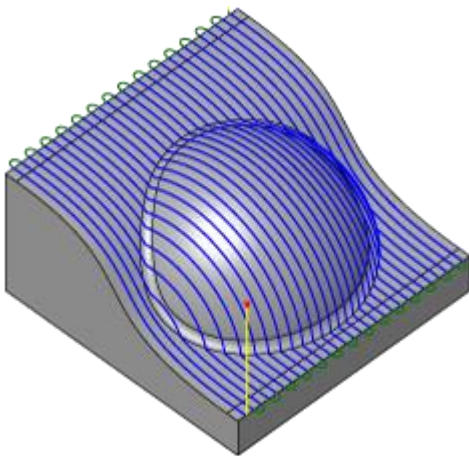
Adaptive Clearing



The **Adaptive Clearing** strategy creates a roughing/clearing toolpath inside closed curves both with and without islands. This strategy avoids full-width cuts by progressively shaving material off the remaining stock. The generated toolpath ensures that the cutting conditions remain constant with a stable load on the tool. As a result, the feedrate can be increased significantly, reducing the machining time by 40% or more which provides improved surface quality and less tool wear.

3D Machining Strategies

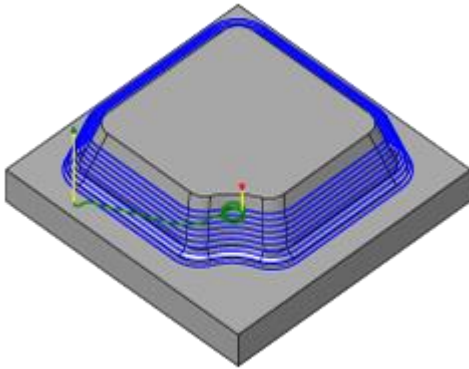
Parallel



Parallel passes are one of the most widely used finishing strategies. The passes are parallel in the XY plane and follow the surface in the Z direction. **Parallel** passes are best suited for shallow areas and down milling. To automatically detect shallow areas, the machining can be

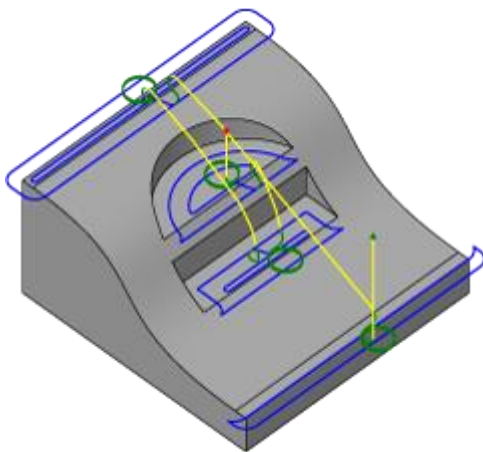
limited to a maximum angle between the tool tip and the surface. By selecting the down milling option, tool deflection can be minimized when machining complex surfaces.

Contour



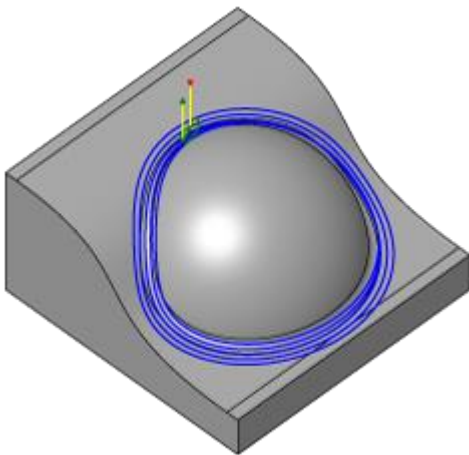
Contour passes is the best strategy for finishing steep walls, but can be used for semi-finish and finish machining on the more vertical areas of a part. If a slope angle is specified, for example 30 to 90 degrees, the steeper areas are machined, leaving the shallower areas up to 30 degrees for more appropriate strategies.

Horizontal Clearing



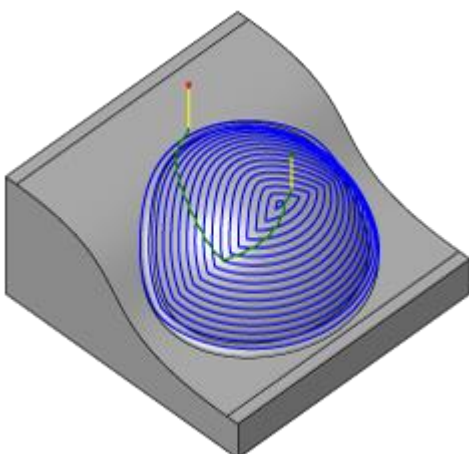
The **Horizontal Clearing** strategy automatically detects all the flat areas of the part and clears them with an offsetting path. When the flat area is shelved above the surrounding areas, the cutter moves beyond the flat areas to clean the edges. Using the optional maximum stepdown, horizontal faces can be machined in stages, making the horizontal clearing suitable for both semi-finishing and finishing.

Pencil



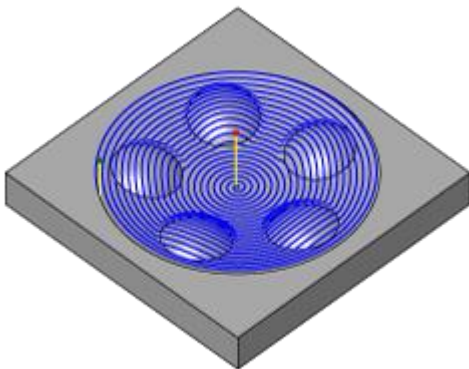
The **Pencil** strategy creates toolpaths along internal corners and fillets with small radii, removing material that no other tool can reach. Whether using single or multiple passes, the **Pencil** strategy is ideally suited for cleaning up after other finishing strategies.

Scallop/Constant Stepover



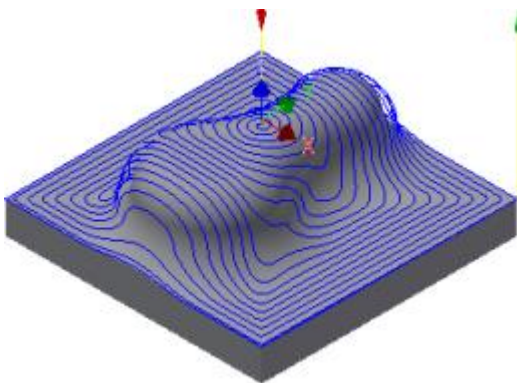
The **Scallop** strategy creates passes that are at a constant distance from one another by offsetting inward along the surface. The passes follow sloping and vertical walls to maintain the stepover. Although **Scallop** finishing can be used to finish an entire part, it is most commonly used for rest machining, following a combination of **Contour** and **Parallel** passes. Like the other finishing strategies, machining can be limited by a contact angle range.

Spiral



Spiral machining creates a spiral toolpath from a given centre point, generating a constant contact as it machines within a given boundary. It is ideally suited for use on round shallow parts using tool contact angles up to 40 degrees, in conjunction with **Contour** passes for the more vertical faces. The centre point of the detail to be machined is located automatically, or can be user-specified. This strategy also supports tool contact angles.

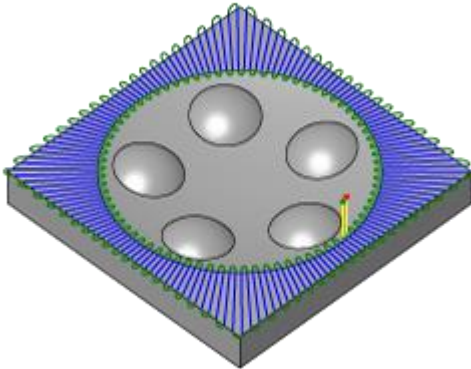
Morphed Spiral



The **Morphed Spiral** strategy is very similar to the **Spiral** strategy. However, a **Morphed Spiral** operation generates the spiral from the selected boundary as opposed to a **Spiral** operation which trims the generated passes to the machining boundary. This means that **Morphed Spiral** can be used for additional surfaces for which **Spiral** is not appropriate. It can also be very useful when machining free-form/organic surfaces. Although the **Scallop**

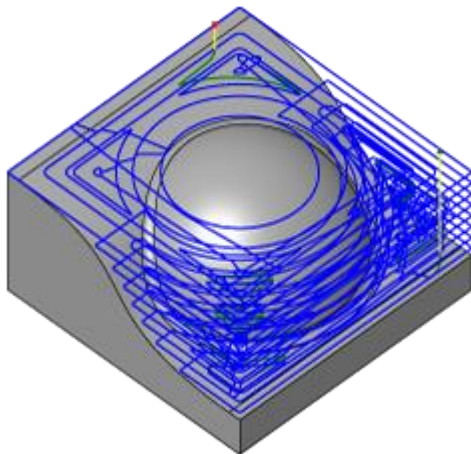
strategy is often used for these types of surfaces, both the sharp corners and the linking transitions between the generated passes can result in visible marks. The **Morphed Spiral** strategy generally provides a much smoother toolpath by avoiding these issues.

Radial



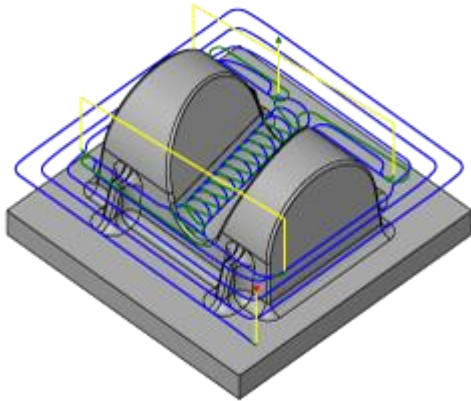
Like spiral machining, **Radial** machining also starts from a centre point, providing you with the ability to machine radial parts. It also provides the option to stop short of the centre of the radial passes, where they become very dense. The centre point of the detail to be machined is located automatically, or can be user-specified. This routine can also be used with tool contact angles.

Pocket



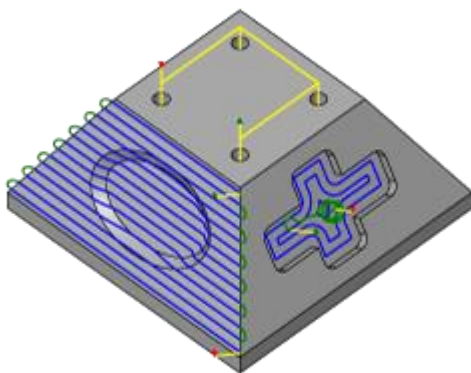
Pocket is the conventional roughing strategy for clearing large quantities of material effectively. The part is cleared layer by layer with smooth offset contours maintaining climb milling throughout the operation. To avoid plunging, the tool ramps down along a helical path between levels. To maintain a high feedrate, and thereby reducing machining time, sharp changes of direction are avoided by smoothing the tool motion.

Adaptive Clearing



Adaptive Clearing is an innovative roughing strategy that offers significant improvements compared to conventional roughing strategies. The strategy avoids full-width cuts by progressively shaving material off the remaining stock. The generated toolpath ensures that the cutting conditions remain constant with a stable load on the tool. As a result, the feedrate can be increased significantly, reducing the machining time by 40% or more.

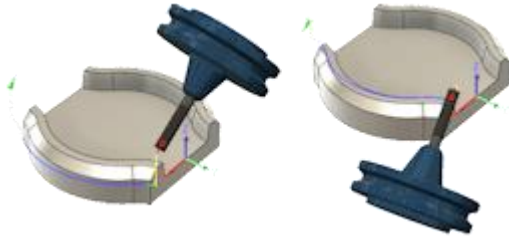
3+2 Machining



All 2D and 3D strategies support 3+2 machining (5-axis positioning) by rotating the part or the head of the machine tool through a combination of A, B, or C axis motions. Creating 3+2 operations is just a matter of selecting a work plane for the operation and Inventor CAM takes care of the rest. Once in position, all machining strategies are available, and are both tooling and holder gouge protected for all the strategies that normally support this.

Multi-Axis Machining Strategies

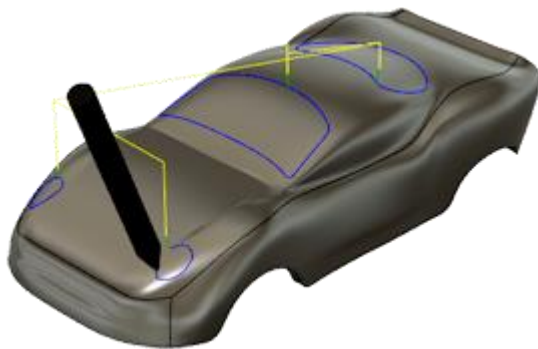
Swarf



Swarf 5 Axis Milling is a side cutting process. It's perfect for parts with bevelled edges and tapered walls. The toolpath is calculated as if there was a straight edge across the 2 edges that define the surface. As long as the face between those 2 edges is straight, you can use a Swarf cut to machine it.

Important: This functionality is only available in Inventor CAM Ultimate.

Multi-Axis Contour



Multi-Axis Contour is used to machine 3D curves that lie on the face of the model. These can be actual edges from the models faces, or 3D curves that have been projected onto the face of a model. Inventor CAM will create a toolpath that follows the selected curve/edge, while keeping normal/perpendicular to the surface of the model. Additional options will allow you to tilt the tool forwards or sideways. While you can compensate Left or Right of the selected curve, normally you would machine directly on Centre of the curve.

Important: This functionality is only available in Inventor CAM Ultimate.

Tutorials

This section includes a number of tutorials to help you familiarize yourself with Inventor CAM. Along with each tutorial you will find a corresponding example file. These example files are typically located here:

C:\Users\Public\Documents\Autodesk\Inventor CAM\Examples

[How to learn Inventor HSM – Basic – Chapter 1](#)

[How to learn Inventor HSM – Basic – Chapter 2](#)

[How to learn Inventor HSM – Basic – Chapter 3](#)

[How to learn Inventor HSM – Basic – Chapter 4](#)

Note: For more educational videos click on this [link](#).