SIEMENS

Introduction to NX CAM

Experience the power for yourself

Siemens PLM Software

Try NX CAM!

Get started with hands-on tutorials

www.siemens.com/plm/nx/cam

Use this DVD to install the NX CAM
software. In order to run the
software, you will need a valid trial
license.

Please contact your local sales
office or reseller to register for a
trial license.

*For information about the legal use
of Siemens software, please see
our guide to software piracy
prevention:*

*http://www.siemens.com/plm/
piracyprevention*

1st Edition
For use with NX 8.5

Contents

Try these hands-on examples in NX CAM!

NX™ CAM software has helped many of the world's leading manufacturers and job shops produce better parts faster. You can also achieve similar benefits by making use of the unique advantages that NX CAM offers.

This short book is designed to introduce you to a few of the many powerful capabilities available inside NX CAM. With three simple hands-on exercises (part files available for download from http://www.siemens.com/plm/nx/try-nx-cam) and instructions for accessing additional in-software learning resources, you can use this book to get started with NX CAM today!
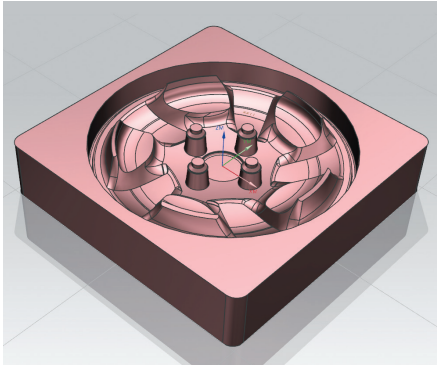
Create and verify a tool path
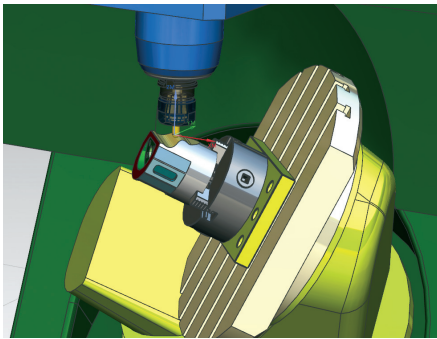
Run machine tool simulation

Automate NC programming with feature-based machining

3
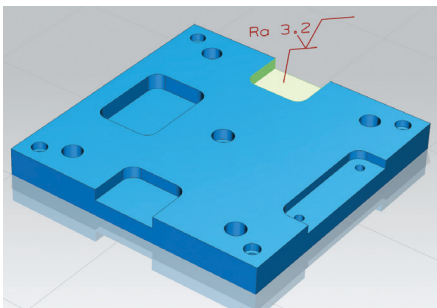
How do I use this book?

Begin by reading Chapter 1, Understanding NX CAM, to gain an overview of the key capabilities available in NX CAM. Then proceed to Chapter 2, Try NX CAM now!, where you'll be instructed to try NX CAM for yourself by following instructions to complete three hands-on examples. The hands-on examples* are simple exercises intended to introduce you to NX CAM:

- Example 1 Create and verify a tool path – tune a tool path's parameters to achieve a desired result
- Example 2 Run machine tool simulation – apply the G-code-driven machine tool simulation capabilities inside NX CAM to verify an NC program

- Example 3 Automate NC programming with feature-based machining – apply feature-based machining (FBM) to automatically recognize and create an optimized NC program

In order to run the hands-on examples, you will need to download the part files. Copy the example part files from: http://www.siemens.com/plm/nx/try-nx-cam.

When you're ready to learn more proceed to Chapter 3, Learn More About NX CAM, where you'll find information about additional resources to gain a more in-depth understanding of how to use the software including: in-software tutorials and online help documentation.

*In order to run the hands-on examples, you will need to have obtained access to the NX CAM software (version NX 8.5) from your local sales office or reseller.

4

Understanding NX CAM

What is NX CAM?

NX CAM is a complete set of NC programming software applications developed by Siemens PLM Software. Originally introduced as Unigraphics in 1973 for aerospace component machining, NX CAM's rich history of technology development has advanced it into a market-leading NC programming application for many industries. NX CAM is a key application in a broader set of software solutions for design (CAD), simulation (CAE) and manufacturing (CAM).

How does NX CAM help me make better parts faster?

More value in each key capability
NX CAM delivers real differences through key capabilities such as its advanced programming, postprocessing and simulation functionality. Each NX module provides more than the standard functions you expect from typical CAM packages. For example, the integrated machine tool simulation is driven by output from the NX postprocessor rather than just tool path data. As a result, NX facilitates a higher level of program validation right inside its CAM system.

NX for manufacturing
NX provides a complete set of NC programming capabilities in a single CAM system as well as an integrated set of manufacturing software applications. These

applications facilitate part modeling, tool deign and inspection programming – all based on proven NX technology.

Right for your industry

NX has been adopted across many industries, delivering proven capability for manufacturing in the aerospace, automotive, medical device, mold and die, and machinery industries. Whether you have a small machine shop with a few machine tools or a large team of manufacturing engineers who utilize numerous machine tools, NX provides a solution to match your business needs.

A leader in manufacturing

When the right design and manufacturing software is matched with the latest controllers, machine tools and other shop floor equipment, you can implement a process chain that delivers maximum performance for your business. Siemens is an established leader in advanced machine tool controller technology and drive equipment. This combination of software and manufacturing equipment expertise enables us to develop part manufacturing solutions that provide you with unique and powerful advantages.

6

Key capabilities in NX CAM

Advanced programming capabilities

NX CAM provides a wide range of functionality, from simple NC programming to multi-axis machining, enabling NC programmers to address many tasks using one system. The flexibility of NX CAM means that the most demanding jobs can be completed easily.

Fixed-axis milling

NX CAM provides a wide range of 2-axis and 3-axis machining capabilities for prismatic and freeform parts – ranging from manual tool path creation and editing to advanced, automated cutting methods. Optimized roughing methods maximize the material removal rate without overloading the tool.
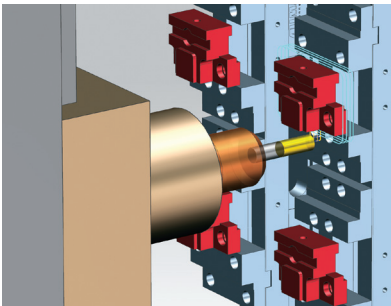
Fully automated rest milling removes uncut material from previous operations and eliminates air-cutting. A wide range of finishing techniques delivers superior surface finish quality. Automatic collision detection ensures safe machining of the most demanding geometry.

High-speed machining (HSM)

Successful high-speed roughing maintains high metal removal rates while managing tool loads. HSM finishing operations, such as the Streamline feature, produce smooth flowing cutting patterns that result in fine finishes at high feed rates.
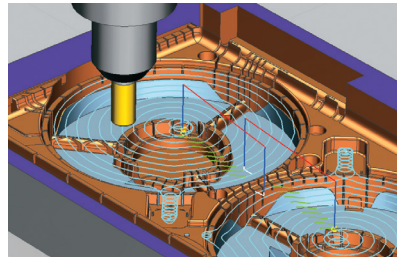

5-axis machining

Multi-axis machining in NX enables you to produce precision complex parts with fewer operations and setups – reducing cost and delivery times. NX CAM supports a range of methods for defining precisely controlled multi-axis tool paths on complex surfaces, with effective collision and gouge checking.

7
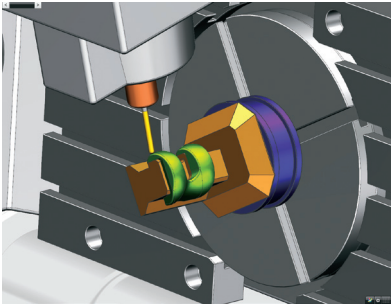
Fast and accurate roughing and finishing methods facilitate machining of complex parts, such as those typically machined for the aerospace industry. Z-level method with tilted tool facilitates the use of shorter tools to reduce tool deflection. Smooth flowing Streamline strategy is an ideal cutting method for multi-axis HSM finishing.

Turbomachinery milling
With NX, you can reduce programming effort by applying specialized 5-axis NC programming operations for complex multi-bladed rotational parts, such as blisks and impellers.
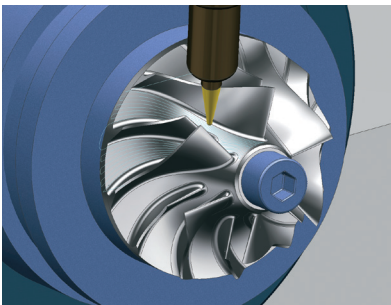
Turning
NX CAM provides a complete turning solution that is easy enough to use on simple programs and capable enough to tackle your toughest geometry in multi-spindle, multi-turret applications. Turning in NX can use either 2D part profiles or full solid models. It includes routines for roughing, multiple-pass finishing, grooving, thread cutting and centerline drilling.
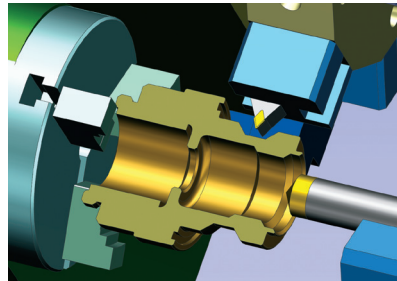
NX turning enables A-axis and B-axis tool control. In addition to rich functionality for common tasks, a special "teach mode" capability provides extra user control for fine finishing and special cutting situations.

Multi-function machines
NX offers a complete range of machining capabilities for the latest multi-function machines that support simultaneous multi-axis operations. Using a graphical display, the Synchronization

8

Manager provides interactive control for machining sequences across multiple channels.
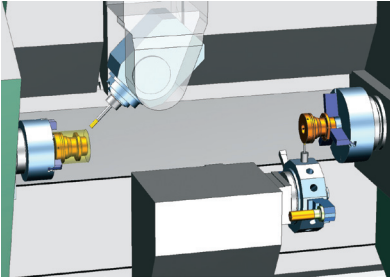
Keeping track of the in-process state of the workpiece is a critical component of effective mill-turning. NX CAM automatically generates an in-process workpiece (IPW) to provide seamless transfer of workpiece configurations between milling and turning.

Wire EDM

NX Wire EDM programming works from both wireframe and solid models, facilitating the cutting of parts in 2-axis and 4-axis modes. A wide range of wire operations is available, including multi-pass profiling, wire reversing and area removal.

Programming automation

Advanced feature-based machining provides extra value in programming automation. With feature-based machining, programming time can be reduced by as much as 90 percent.

Feature-based machining (FBM)

You can automatically create optimized machine programs directly from part design models by using feature-based machining in NX. FBM automatically recognizes and programs a wide range of machining feature types: prismatic, turning, wire EDM, colors and attributes.
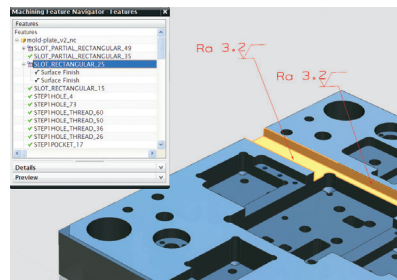
Each machining step is selected using configurable logic and criteria managed in a machining database provided as a part of the system. You can easily configure, add or modify the feature-based machining operations and how they are selected using a simple editor (Machining Knowledge Editor).

PMI-driven machining NX can read product and manufacturing information (PMI), such as tolerances and surface finish, that is attached to the model and drives the machining method selection. For example, a tight tolerance

9

might require a specific finishing process and tool. NX CAM can read the tolerance data added to the NX design model and use it to select the correct machining operations. In this way, the PMI data drives the NC programming and machining.

Machining data library

NX CAM provides a customizable machining database that enables you to manage and apply proven data to associative tool path operations. NX automatically applies the right feeds and speeds for a given operation and tool selection.

Machining wizards

For everyday common tasks, you can build process wizards using easy-to-follow steps. Wizards can make complex software settings based on simple user selections. Wizards are easily available from the NX menu.
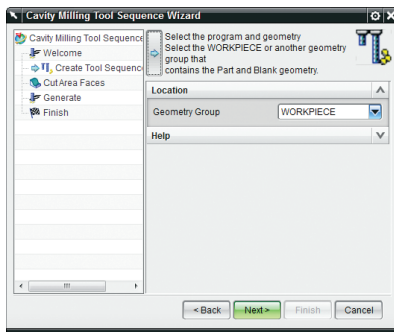
Process templates

NX CAM enables you to apply predefined rules-driven processes and setup templates to standardize and speed the programming task. Typically, processes are employed to ensure the use of preferred methods and tooling.

Postprocessing

Integrated postprocessor

NX includes its own postprocessor system, which is tightly connected to the core CAM system. You can easily generate the required NC code for almost any type of machine tool and controller configuration.

Postprocessor library

The postprocessor library is an online resource that contains a variety of posts that support a wide range of machine tools.

10

Post Builder

NX CAM includes a Post Builder capability that enables you to create and edit postprocessors. Using its graphical user interface, you can specify parameters for the required NC codes.

Optimized output for Siemens controllers

NX CAM provides an optimized Sinumerik postprocessor that automatically selects key controller settings based on machining operation data. NX CAM can also generate output for Fanuc, Heidenhain and other controller types.

Machining simulation

Multiple levels of NC program validation include G-code-driven simulation, which eliminates the need for separate simulation packages.
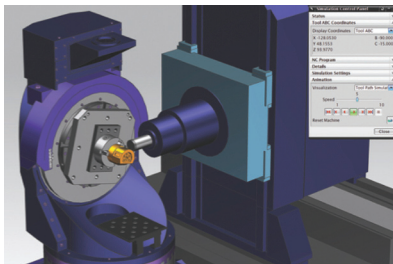
Machining process validation

A key NX CAM advantage facilitates integrated simulation and verification, which enables programmers to check tool paths within the NC programming session. Multiple levels of capability are available.

For example, the G-code-driven machine tool simulation displays the motion driven from the NC code output of the NX internal postprocessor. A 3D model of the machine – with the part, fixtures and tooling – moves in the way the machine tool will move as the G-code is processed.

When the Siemens virtual NC controller kernel (VNCK) is added to NX CAM, real controller software is used to drive your integrated machining simulation solution. The integrated solution facilitates the closest possible digital representation of actual machine tool motion with highly accurate speeds, accelerations, tool changes and cycle times.

Machine tool support kits

For advanced machine tools, support kits provide a complete solution that includes:

• Proven postprocessor
• 3D solid model of the machine tool
• Example parts, templates and documentation

Ease of use

For maximum productivity, users can work on the system graphically. For example, selecting and moving the 3D model of the tool to adjust a tool path is a fast and intuitive way to drive the system. The dialog boxes use graphics with clear annotation to indicate what values are required for menu input.

Graphical-driven programming

NX encourages the user to drive the system graphically as much as possible. It is much faster and more intuitive to control a tool by selecting and moving the 3D tool model on the screen than typing numbers into a menu.
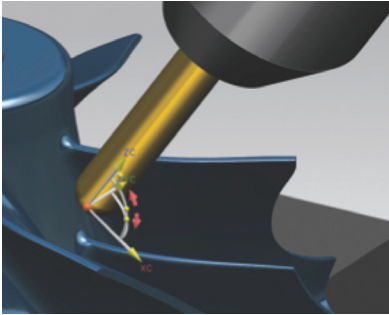
Intuitive user interface

You can increase productivity by leveraging the latest user interaction techniques and predefined programming environment.

Operation navigator

The operation navigator keeps critical information accessible to the programmer and facilitates re-use. A rich information environment displays the operation sequences and dependencies while tracking tool usage and facilitating associative part data.

Integrated workflow tutorials and documentation

NX provides step-by-step tutorials for various programming processes, including die, mill-turn and aerospace machining. These tutorials are directly accessible from NX.

The internet-like search engine in NX enables you to quickly find a command by searching against a keyword.
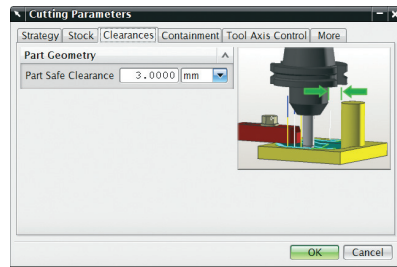
Integrated solution – the CAD/CAM advantage of NX

NX provides advanced CAD tools that can be used by the NC programmer for everything from modeling new parts to creating

12

setup drawings directly from 3D model data. For manufacturing, NX offers special applications alongside CAM, including tool design and inspection programming modules. The 3D model moves seamlessly between applications without data translation.

Connecting NX to Teamcenter® software for data and process management establishes a basis for an extended part manufacturing solution. All types of data from 3D part models to setup sheets, tool lists and CNC output files can be fully managed.

3D part model preparation

The latest CAD technology in NX enables the NC programmer to rapidly prepare part models, including third-party CAD models. With synchronous technology, you can directly edit the part model and prepare it for NC programming, including closing holes and gaps, offsetting faces and resizing part features. NX offers a set of specialized CAD functions that enables the NC programmer to quickly analyze the part before creating the NC operations.
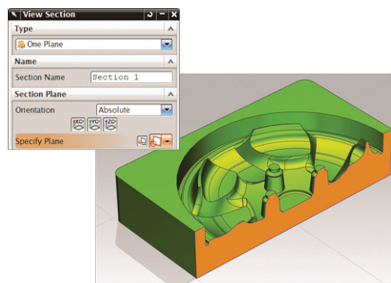
Master model concept

NX leverages the master model concept to facilitate concurrent design and NC programming by linking all functions, such as CAM and CMM, back to the single model definition of the part. As a result, the NC programmer can start programming a part before the designer is finished. Full associativity ensures subsequent updates of NC operations as the design model changes.

Modeling, assemblies and drafting

NX provides one of the most powerful sets of CAD functions available on the market today. These functions are packaged with NX CAM for the NC programmer who needs to model parts, stock shapes, fixtures or machine tools for simulation.

13
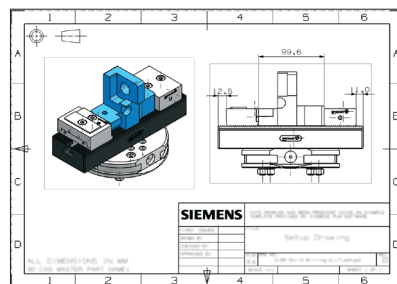
Try NX CAM!

You can start to learn NX CAM by following and completing the three hands-on examples in this chapter. Each example takes about 10 minutes to complete. These exercises are intended to help you form a first impression about NX CAM and to encourage you to explore NX more on your own. While these examples may seem quite simple, they are included here because they resemble familiar tasks you perform while NC programming.

In order to run the hands-on examples, you will need to download the part files. Copy the example part files from: http://www.siemens.com/plm/nx/try-nx-cam.
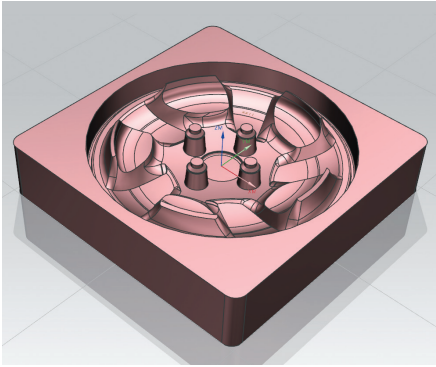
Hands-on example 1 – Create and verify a tool path
It is quite common to frequently adjust tool paths to achieve desired results. NX CAM provides the flexibility to tune tool paths to your specific needs. This example shows you how easy it is to make adjustments to tool path parameters and to visualize the results using tool path verification.
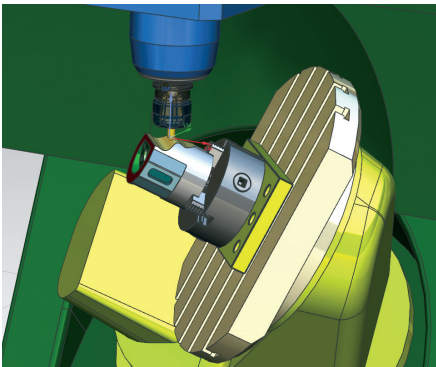
Hands-on example 2 – Run machine tool simulation
Machine tool simulation is often applied when needing to verify that an NC program will run safely on a specific machine. Instead of using a separate software application for this, you can simulate machining directly inside NX CAM. This example guides you through setting up and running a simple machine tool simulation.
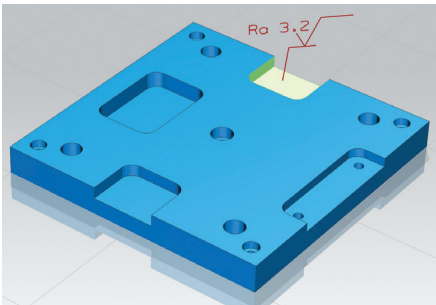
Hands-on example 3 – Automate NC programming with feature-based machining
You can save time and effort by automatically creating optimized NC programs directly from part models. The feature-based machining (FBM) in NX CAM can automatically recognize and program a wide range of feature types. This example shows you how to automatically create machining operations directly from a part model.

Hands-on example 1: Create and verify a tool path

In this example, you will change a roughing operation's cut pattern and use the verify function to see the effects of this change. Finally, you'll finish the example by regenerating the rest mill operation.

These instructions require that you run this example in a new NX session and use the example files that you will have copied onto your computer.

If you need help with this example, please contact your local sales representative.

Before you begin:

► Exit all NX sessions.

► In your internet browser, go to: **http://www.siemens.com/plm/nx/ try-nx-cam**.

► Copy the file **1-Create Tool Path.zip** to your computer's hard disk drive.

► Extract all of the contents of the **zip** file into a new folder **1-Create Tool Path**.

Begin by starting NX.

► Start the **NX 8.5** application.

The NX application window appears.
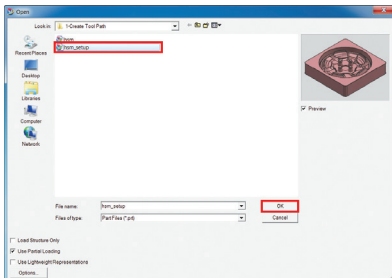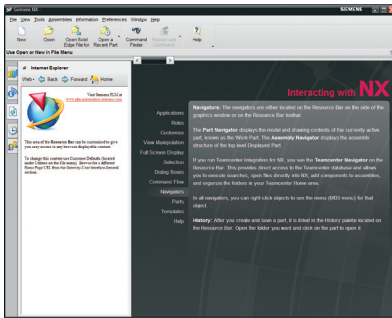
From NX, open the file for this example, hsm_setup.

► In the toolbar, choose **Open** .

► Look in the folder **1-Create Tool Path** you just created and select the file **hsm_setup**.

► With **hsm_setup** selected, click **OK**.
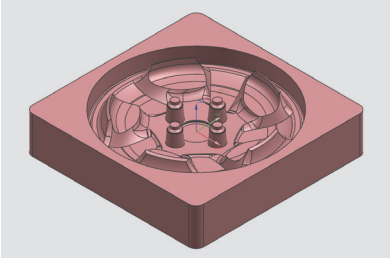
This part appears in the graphics window.
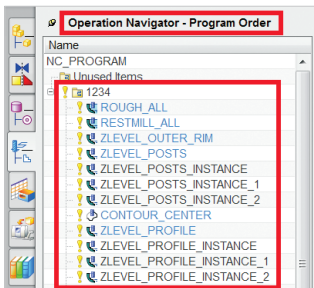
Use the Operation Navigator to define and control the machining process.

This example part has already been programmed. The machining operations for this part are listed in the **Operation Navigator** under the 1234 program.

The Operation Navigator is displaying the **Program Order** view which organizes operations according to the order in which they are executed on the machine tool.
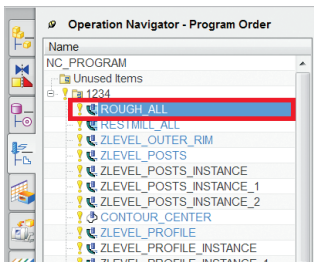
Display the tool path for the roughing operation.

▶ In the **Operation Navigato**r, click on the **ROUGH_ALL** operation.
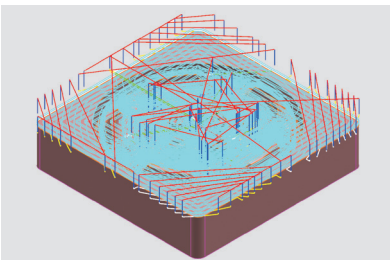
You immediately see the ROUGH_ALL operation tool path displayed in the graphics window.

Verify the tool path for the ROUGH_ALL operation.

Click **Verify Tool Path** in the toolbar.
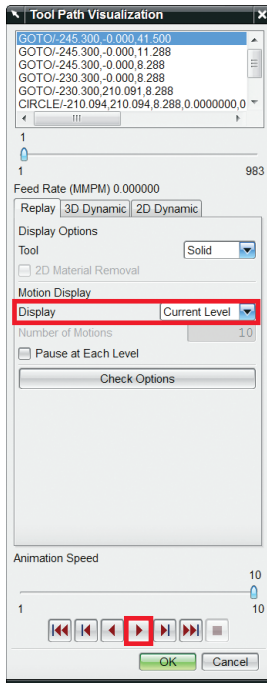
The Tool Path Visualization dialog box is displayed. This dialog box lets you control various verification parameters, such as check for gouges, check for tool and holder collisions, display material removal, and others.

▶ In the **Tool Path Visualization** dialog box, select **Current Level** from the **Display** list.

This displays one cut level at a time.

▶ Click Play ▶ .
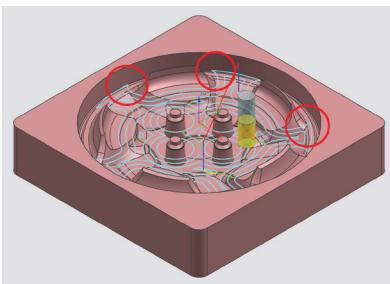
As the tool path verifies, note the sharp corners as the various cut levels are displayed. These sharp corners can overload the tool and may cause early tool failure.

You can change this tool path to a different pattern that avoids embedding the tool and results in a constant metal removal rate which is better for high-speed machining (HSM).

▶ Click **OK** to dismiss the Tool Path Visualization dialog box.
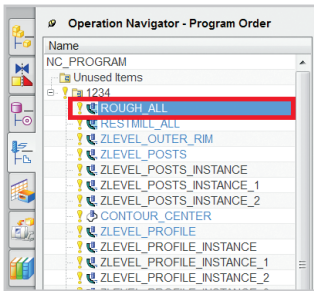
Change the cut pattern.

▶ In the Operation Navigator, double-click the **ROUGH_ALL** operation.
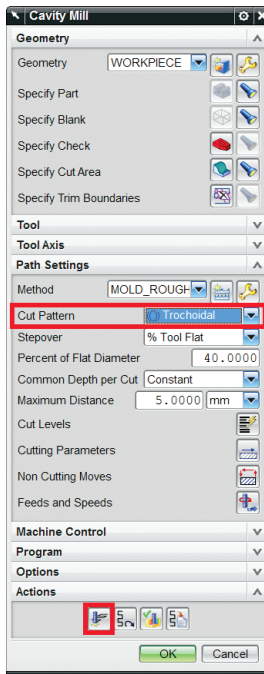
The Cavity Mill dialog box is displayed, which lets you specify the operation's machining parameters.

▶ Select Trochoidal from the **Cut Pattern** list.

Trochoidal tool paths can help improve roughing efficiency by maximizing the material removal rate without overloading the tool. This HSM strategy is especially useful for efficient milling of hard materials.

▶ Click Generate.

19
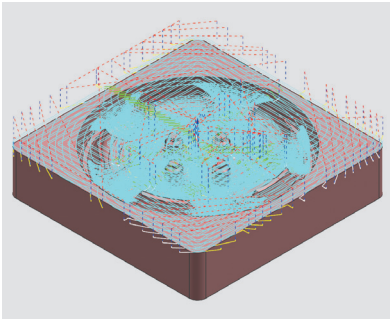
The new roughing operation with the trochoidal cut pattern is displayed.
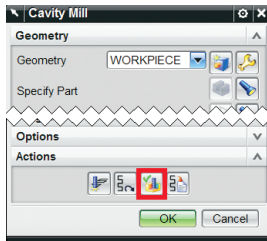
Verify the trochoidal roughing operation.

► Click Verify ⚒ at the bottom of the **Cavity Mill** dialog box.
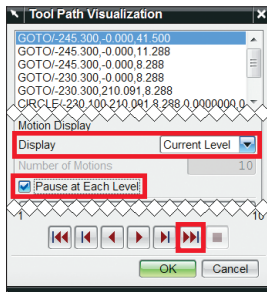
► In the **Tool Path Visualization** dialog box, select **Current Level** from the **Display** list.

► Be sure the P**ause at Each Level** check box is selected.

Pausing gives you a better view of the cut pattern at each level.

► Continue clicking **Forward to the Next Operation** ▶▶ at the bottom of the dialog box until each level of the tool path has been played.
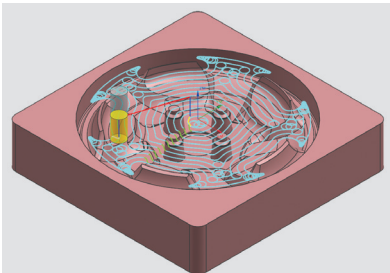
The trochoidal tool path at each cut level is displayed.
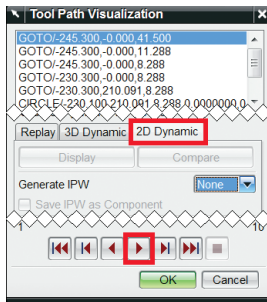
Verify the trochoidal operation with material removal.

► In the **Tool Path Visualization** dialog box, click on the **2D Dynamic** tab.

► Click Play ▶ .
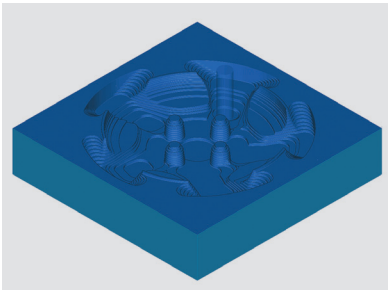
The material removal option provides visualization of the actual results you would expect when machining the part.

► After the verification completes, click **OK** to dismiss the **Tool Path Visualization** dialog box.

► Click **OK** to dismiss the **Cavity Mill** dialog box.
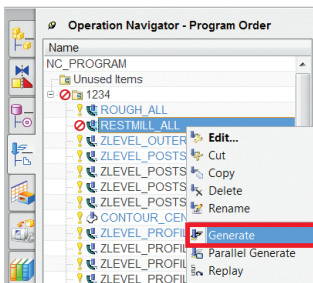
Regenerate the tool path for the RESTMILL_ALL operation.

The Regenerate status ⃠ symbol in the Operation Navigator indicates that the tool path for the RESTMILL_ALL operation needs to be regenerated. The tool path needs to be regenerated because the uncut material remaining from the previous ROUGH_ALL operation has changed.

► In the **Operation Navigator**, click the **RESTMILL_ALL** operation, then right-click and select **Generate** from the list.
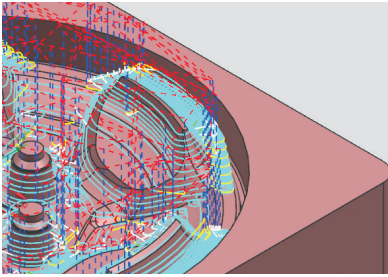
Generating the tool path for the rest milling operation takes a few moments as NX CAM calculates the uncut material left over from the previous operation.

NX CAM uses the in-process workpiece (IPW) to keep track of machined material. As a result, it can automatically create an optimized rest mill operation that targets uncut areas and results in a constant amount of remaining material for finishing.

As the tool path generates, it is displayed.

Finally, verify the rest mill operation.

▶ Click **Verify Tool Path** 📇 in the toolbar.
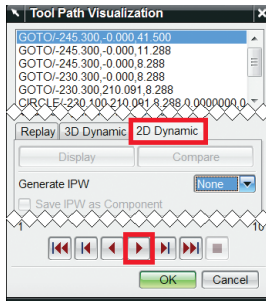
▶ In the **Tool Path Verification** dialog box, select the **2D Dynamic** tab.

▶ Click **Play** ▶ .
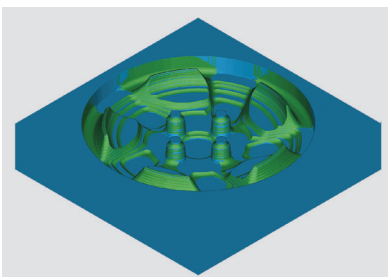
While the verification plays, note that the smaller tool used for the rest mill operation only machines the areas where the previous roughing operation left uncut material.

▶ After the verification completes, click **OK** to dismiss the **Tool Path Visualization** dialog box.

Close the file without saving it.

▶ Click **File** –> **Close** –> **All Parts**.

▶ In the Close All Files dialog box click **No – Close**.

Hands-on example 2: Run machine tool simulation

This hands-on example introduces you to the integrated machining simulation capability in NX CAM. By following these instructions, you will see how to apply G-code-driven machine tool simulation to a part that has already been programmed.

These instructions require that you run this example in a new NX session and use the example files that you will have copied onto your computer.

If you need help with this example, please contact your local sales representative.

Before you begin:

► Exit all NX sessions.

► In your internet browser, go to **http://www.siemens.com/plm/nx/try-nx-cam**.

► Copy the file **2-Machine Tool Simulation.zip** to your computer's hard disk drive.

► Extract all of the contents of the **zip** file into a new folder **2-Machine Tool Simulation**.
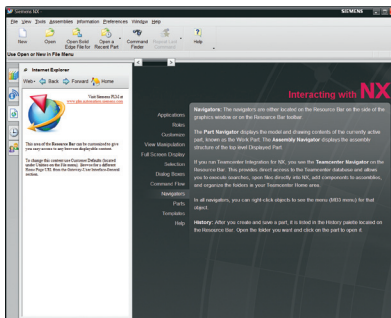
Begin by starting NX.

► Start the **NX 8.5** application.

The NX application window appears.
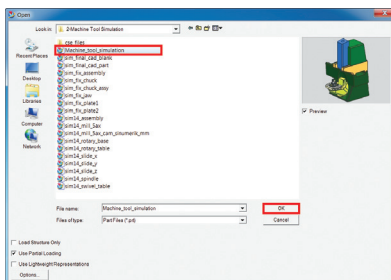
From NX, open the file for this example, **Machine_tool_simulation**.

► In the toolbar, choose **Open** .

► Look in the folder **2-Machine Tool Simulation** you just created and select the file **Machine_tool_simulation**.

► With Machine_tool_simulation selected, click **OK**.

23
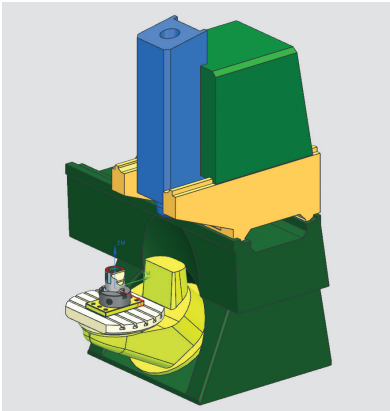
The machine tool and workpiece appear in the graphics window.

NX CAM includes a library of various machine tool models like this one. These models include kinematics and assembly constraints that are used for running simulations. You can even create your machine models in NX.
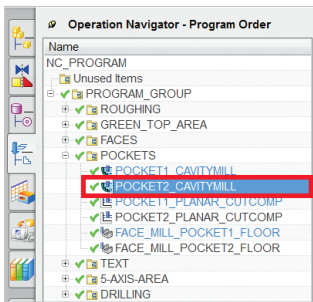
Simulate a simple pocketing operation.

G-code-driven machine tool simulation in NX CAM displays the actual motion of the machine driven from the NC code output from the internal NX CAM postprocessor.

► In the **Operation Navigator**, click the ⊞ to expand the **POCKETS** group.

► Click the **POCKET2_CAVITYMILL** operation.
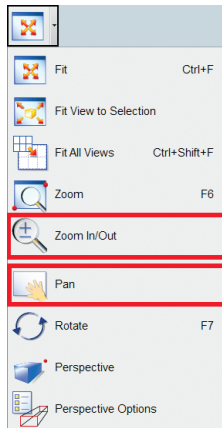
This operation is used to machine the slot on the front side of the part.

Zoom and pan to move the workpiece into the center of the graphics window.

► Click **Zoom In/Out** in the toolbar.

► **Place the cursor** over the part and then **click and drag** the cursor toward the bottom of the screen to zoom in.
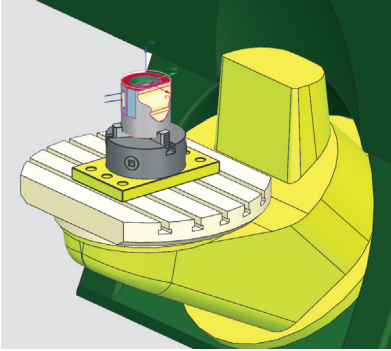
▶ Click and drag the cursor toward the top of the screen to zoom out.
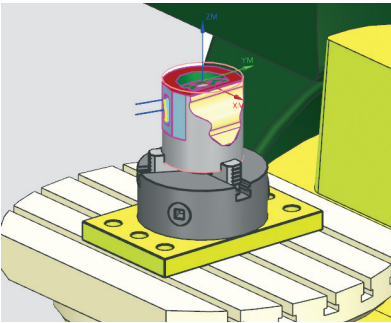
▶ Click **Pan** in the toolbar.

▶ Click and drag the cursor in any direction to move the part. Center the part in the graphics window.


▶ In the toolbar, click **Simulate Machine** .

25
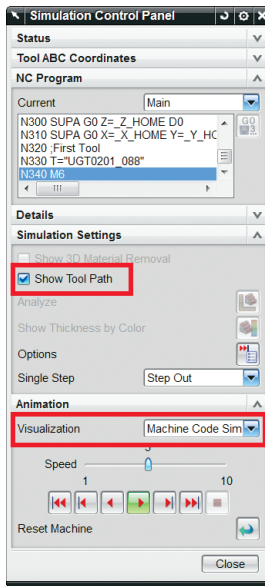
The Simulation Control Panel dialog box is displayed. This dialog box is used to set simulation parameters.
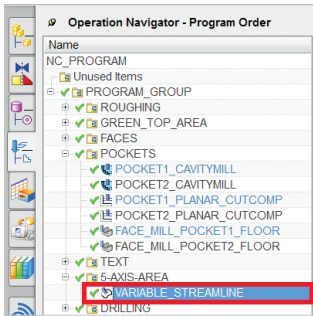
▶ Be sure the **Show Tool Path** check box is selected.

▶ Select **Machine Code Simulation** from the **Visualization** list.

▶ Click **Play** ▶ .
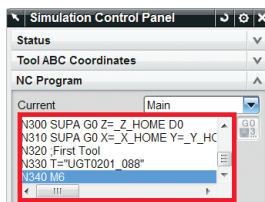
While the simulation runs, notice that the actual G-code is displayed in the NC Program window within the Simulation Control Panel dialog box.

▶ After the simulation completes, click **Close** to dismiss the **Simulation Control Panel** dialog box.

26
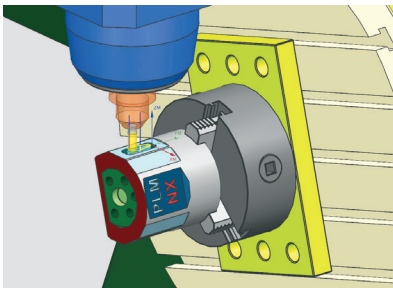
Simulate a simultaneous 5-axis operation.

► In the **Operation Navigator**, click the ⊞ to expand the **5-AXIS AREA** group.

► Click the **VARIABLE_STREAMLINE** operation.
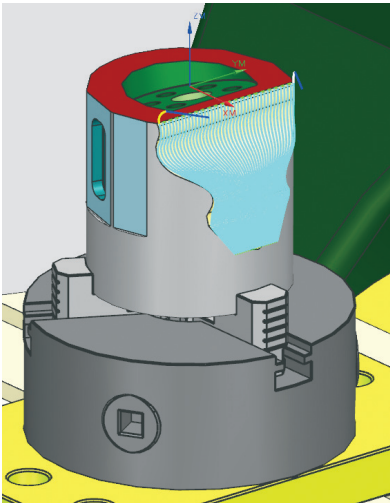
The tool path is displayed. See how it follows the natural shape of the freeform area of this part. This smooth flowing tool path is an ideal cutting strategy for multi-axis machining because it can produce a superior surface finish at a high feed rate.
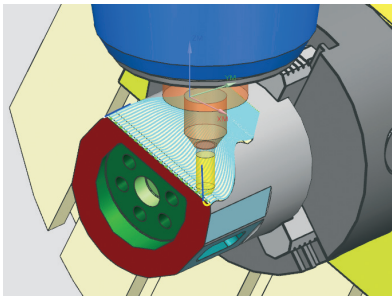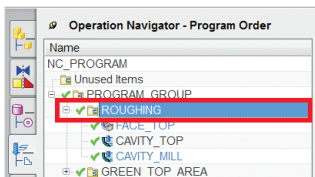

► In the toolbar, click **Simulate Machine**.
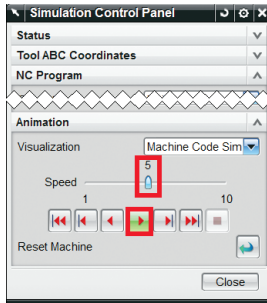
▶ In the **Simulation Control Panel** dialog box, drag the **Animation Speed** slider to 9.

▶ In the **Simulation Control Panel** dialog box, click **Play** ▶ .

You can see the G-code rotary angles update while the simulation runs in the NC Program window within the Simulation Control Panel dialog box.

▶ After the simulation completes, click **Close** to dismiss the **Simulation Control Panel** dialog box.

Run machine tool simulation with material removal.

▶ In the **Operation Navigator**, click the **ROUGHING** group.

▶ Click the ⊞ to expand the **ROUGHING** group.

See that there are three operations in this group. You can simulate the entire ROUGHING group (all three of the operations in sequence) by running the simulation while this group is selected.
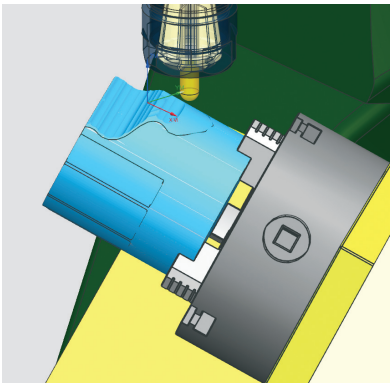

▶ In the toolbar, click **Simulate Machine** .

28
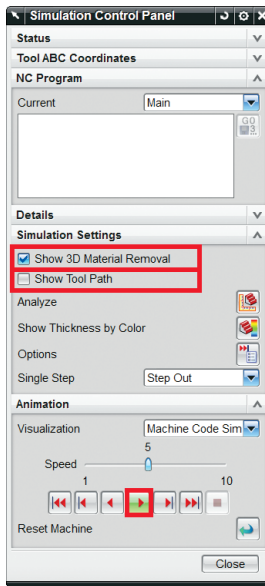
NX CAM can display material removal while checking for collisions and gouges during a G-code-driven simulation. Activate this capability by toggling on Show 3D Material Removal.

► Be sure the **Show 3D Material Removal** check box is selected.

► Be sure the **Show Tool Path** check box is unselected.

► Click **Play** ▶.

As the operations are simulated, the in-process workpiece (IPW) is passed from one operation to the next so that each subsequent operation can begin machining based on the IPW of the previous operation. This allows you to see an accurate representation of the workpiece's shape as the NC program progresses.

► After the simulation completes, click Close to dismiss the **Simulation Control Panel** dialog box.

Close the file without saving it.

► Click **File** –> **Close** –> **All Parts**.

► In the **Close All Files** dialog box click **No – Close**.

29

# Hands-on example 3: Automate NC programming with feature-based machining

This example introduces you to the feature-based machining (FBM) capabilities in NX CAM, which can help you save time by automating NC programming. Features such as holes and pockets can be recognized and then programmed using FBM. NX CAM makes intelligent decisions based on feature shape, size and product and manufacturing information (PMI) such as dimensional tolerances and surface finish. These decisions include tool selection, operation and program definition.

These instructions require that you run this example in a new NX session and use the example files that you will have copied onto your computer.

If you need help with this example, please contact your local sales representative.

Before you begin:

► Exit all NX sessions.

► In your internet browser, go to **http://www.siemens.com/plm/nx/ try-nx-cam**.

► Copy the file **3-Feature Based Machining.zip** file to your computer's hard disk drive.

► Extract all of the contents of the **zip** file into a new folder **3-Feature Based Machining**.
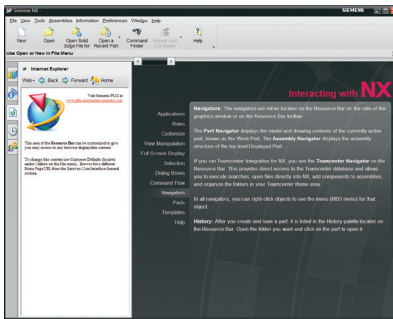
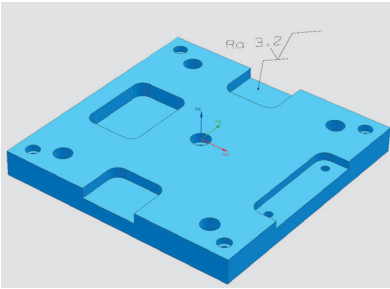Begin by starting NX.

► Start the **NX 8.5** application.

The NX application window appears.
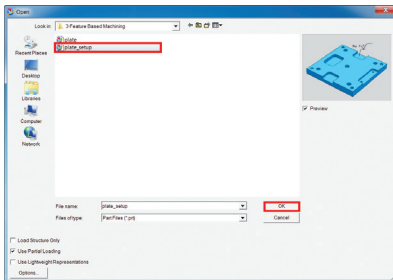
From NX, open the file for this example, **plate_setup**.

► In the toolbar, choose **Open** .

► Look in the folder **3-Feature Based Machining** you just created and select the file **plate_setup**.

► With **plate_setup** selected, click **OK**.

The plate appears in the graphics window. It is a solid body and does not contain any parametric features. NX CAM can recognize and create machining features directly from solid bodies.
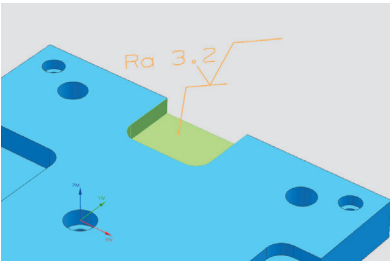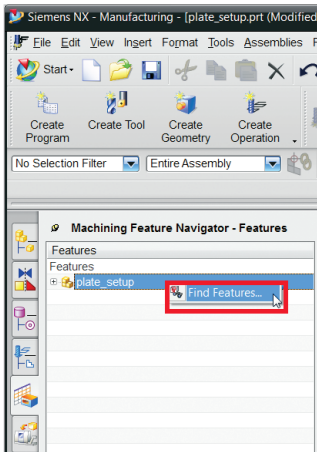

This plate also has surface finish PMI attached to the floors and walls of this pocket. NX CAM can recognize it and use it to automatically determine the selection and definition of machining operations.
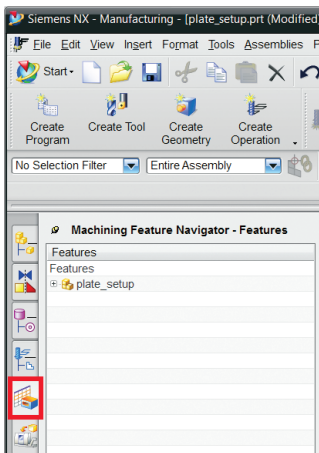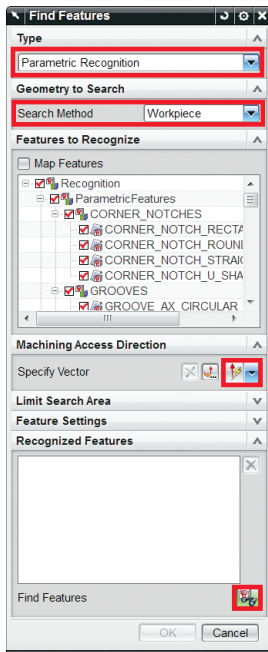

These steps recognize and create the features that will be machined.

▶ Click the **Machining Feature Navigator** tab located along the edge of the graphics window.

► In the **Machining Feature Navigator**, right-click on the **plate_setup** feature group, then choose **Find Features**.
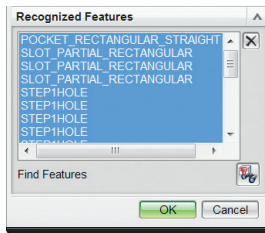

► In the **Find Features** dialog box, select **Parametric Recognition** from the **Type** list.

► Select **Workpiece** from the **Search Method** list.

Parametric Recognition searches the part for predefined geometric shapes and then creates machining features for the ones it recognizes. (You can teach NX CAM to recognize your unique features.)

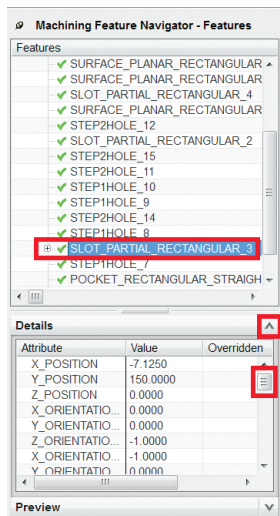► Select **Specify Vector** then select **ZC** from the Specify Vector list.

▶ Click **Find Features** 🏭 at the bottom of the dialog box.

The recognized features appear in the **Recognized Features** field of the **Find Features** dialog box.
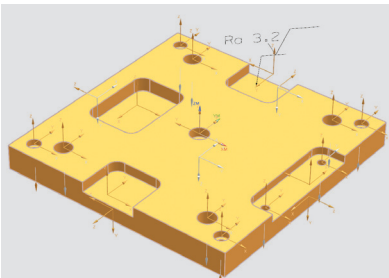
▶ Click **OK**.

The recognized features are created and displayed in the graphics window and also listed in the Machining Feature Navigator.

The machining features and their detailed attributes are used to create the machining operations.

▶ In the **Machining Feature Navigator**, click the **SLOT_PARTIAL_ RECTANGULAR_3** feature.

You can view more details about features by expanding the Details pane in the **Machining Feature Navigator**.

▶ Open the Details pane by clicking the arrow ⌄ (when closed).

▶ Drag the slider ▤ in the **Details** pane to scroll through information about the feature's attributes.

33

In addition to details about position, orientation, depth and size, this feature contains information about surface finish as designated by the **Ra 3.2** value for **BOTTOM_ROUGHNESS** and **SIDE ROUGHNESS**. This information was automatically read from the surface finish PMI attached to the pocket.

► Close the **Details** pane by clicking the arrow ◮ (when open).
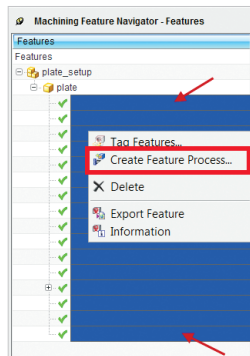
Create the machining operations for all of the recognized features.

Select all of the machining features.

► Select the first feature in the list, hold down the **SHIFT** key, and select the last feature in the list.

► While all of the features are selected, right-click on any of the features and select **Create Feature Process**.
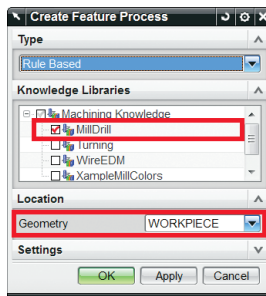
Select the appropriate Knowledge Library.

► Be sure that the **MillDrill** check box is selected.

► Select **WORKPIECE** from the **Geometry** list.

NX CAM includes Knowledge Libraries that contain rules for creating machining operations from features. These rules drive automatic selection of tools, operations and NC parameters. (You can also edit and define your own rules.)

► Click **OK**.

34
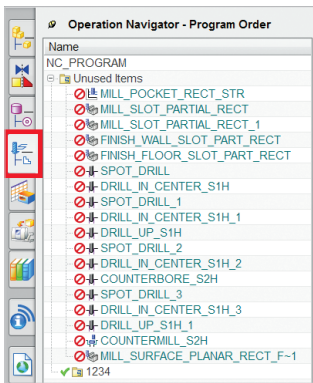
▶ Click the **Operations Navigator** tab located along the edge of the graphics window.

These are the machining operations that were automatically created. Each operation's details vary depending on its originating feature's geometric characteristics and attributes.
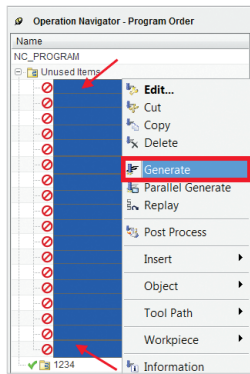

Generate the tool paths for all of the operations.

Tool paths for all of the operations need to be generated as indicated by the Regenerate status ⊘ symbol next to their names in the Operation Navigator.

Select all of the operations.

▶ Select the first operation in the list, hold down the **SHIFT** key and select the last operation in the list.

▶ Right-click on any of the highlighted operations and select **Generate**.
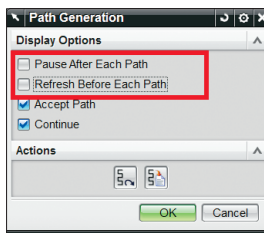
▶ In the **Path Generation** dialog box, be sure the **Pause After Each Path** and **Refresh Before Each Path** check boxes are unchecked.

▶ Click **OK**.

35
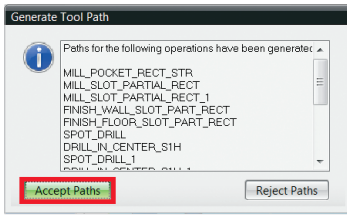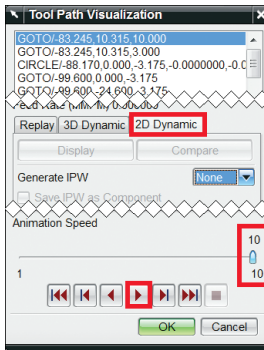
The tool paths are displayed as they are created.

▶ In the **Generate Tool Path** dialog box, click **Accept Paths**.

Verify the tool paths.
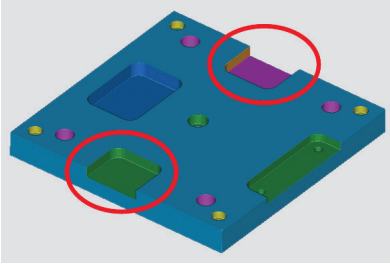
▶ Click **Verify Tool Path** in the toolbar.

▶ In the **Tool Path Verification** dialog box, select the **2D Dynamic** tab.

▶ In the **Tool Path Visualization** dialog box, drag the **Animation Speed** slider to **3**.

▶ Click **Play** ▶ .

As the tool paths verify, notice that the pocket with the surface finish PMI is machined using additional operations compared to the mirrored pocket on the other side of the part.

Based on the surface finish PMI, NX CAM's feature-based machining determined that this pocket required additional finishing operations for the walls and floors in order to achieve the specified surface finish.

► After verification is completed, click **OK** to dismiss the **Tool Path Visualization** dialog box.

NX CAM automatically created all of the operations required to machine the features in this part. If needed, you can reorder the sequence of operations in the **Operation Navigator** to suit your specific needs.

Close the file without saving it.

► Click **File** –> **Close** –> **All Parts**.

► In the **Close All Files** dialog box click **No – Close**.

Learn more about NX CAM

You can use the self-study learning tools directly inside the NX software to expand your understanding of the capabilities that NX CAM offers. Try the in-software Manufacturing Tutorials to gain a deeper understanding of NX CAM's tool path programming capabilities for fixed-axis milling, turning, multi-function machining, 5-axis machining, high-speed machining, postprocessing and other supporting capabilities.

If you need more information about a specific function, you can use NX Help to find how to use that specific feature or function in the software.
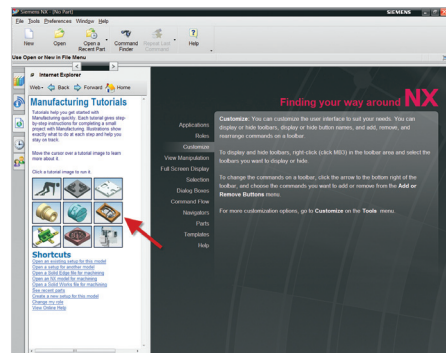
In-software manufacturing tutorials

To access the in-software tutorials, click on the Manufacturing Tutorials option beneath the Help button. You can start a tutorial by clicking on it in the embedded Internet Explorer window. Each tutorial provides a hands-on experience that guides you through completing an NC programming activity.
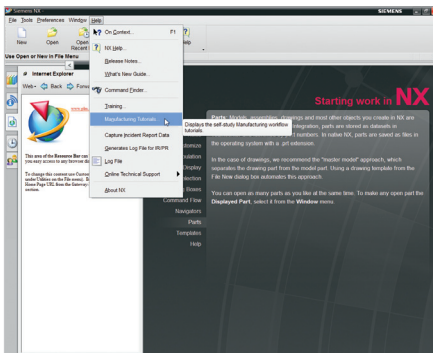

Click Manufacturing Tutorials beneath the Help option to display the self-study manufacturing tutorials.

Immediately after you start a tutorial, a separate browser window opens and displays the tutorial step-by-step instructions (depending on your browser's security option settings you may be required to "Allow Blocked Content..." in order to load the tutorial instructions into the newly opened browser window) and the part file that you need is automatically loaded into the NX session.

After the manufacturing tutorials are displayed in the Internet Explorer window inside NX, you can start a tutorial by clicking on it.
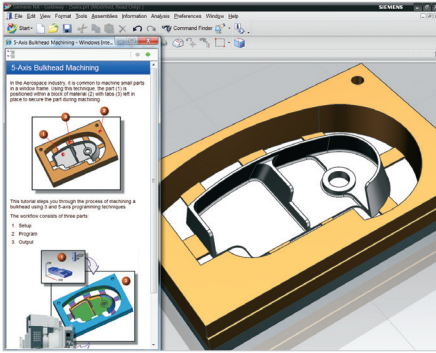
This is an example of what you'll see after you start the 5-axis Bulkhead Machining tutorial. A new browser window is opened and displays step-by-step instructions for completing the tutorial. The part file is also automatically loaded into NX.

Each tutorial gives step-by-step instructions for completing a small project with NX CAM. Illustrations show exactly what to do at each step and help you stay on track. All you need to do to complete the tutorial is follow each step of the instructions that opened in the browser window.
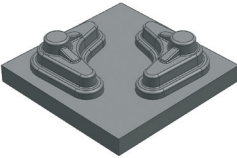
This table shows many of the in-software manufacturing tutorials that are included in NX:
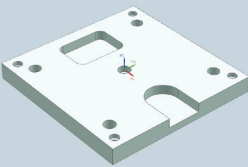
Manufacturing tutorials

Getting Started – This tutorial steps you through basic interactions with NX CAM. You learn how to use the Operation Navigator and how to perform common screen interactions.

Estimated time: 20 minutes

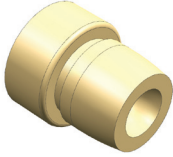Die Machining – This tutorial steps you through the process of machining a die.

Estimated time: 45 minutes

Plate Machining – This tutorial steps you through the process of machining a prismatic model. You first create operations individually. Then you create the program using feature-based machining.
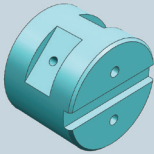
Estimated time: 60 minutes

Manufacturing tutorials

Turning – This tutorial steps you through the process of creating a turning program for a single spindle lathe.

Estimated time: 60 minutes

Mill Turn Machining – This tutorial steps you through the process of creating a multiple spindle mill-turn program. You create turning, milling and drilling operations and manage the in-process workpiece.
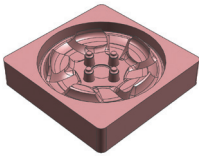
Estimated time: 60 minutes
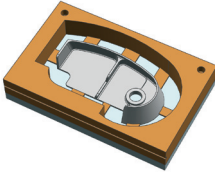
5-Axis Bulkhead Machining – This tutorial steps you through the process creating a multi-axis program for an aerospace part. You also mount the part to a 5-axis machine tool and simulate the machine tool movements for the program.

Estimated time: 60 minutes
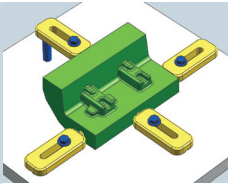
Mold Rework – This tutorial steps you through the process of remachining a damaged area by creating a repair program.

Estimate time: 30 minutes

High-Speed Machining – This tutorial steps you through the process of machining a wheel mold using high-speed machining techniques.

Estimated time: 60 minutes

Output – This tutorial steps you through how to output, customize and print shop documentation. Then you learn how to create and edit postprocessors with Post Builder and download postprocessors.

Estimated time: 45 minutes

NX Help

For an additional level of detailed feature and function information, you can use NX Help. You can access NX Help by clicking on the Help icon in the standard toolbar (you can also access NX Help under the Help menu).
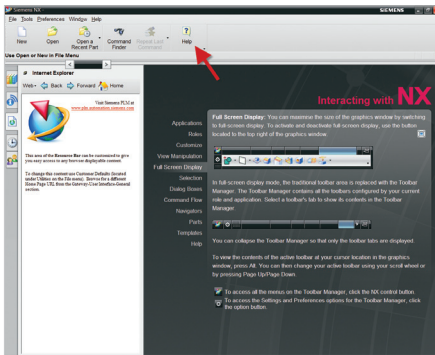
Access NX Help by clicking the Help icon.

Once you click on NX Help, you'll see a table of contents for the NX Help Library, which has links to each application or software area.

Example of what you'll see after you click on NX Help.

You'll find detailed technical information about NX CAM under the CAM (Manufacturing) section. The topic areas covered under this section include:

- Manufacturing General

- Manufacturing Milling

- Manufacturing Turning

- Integrated Simulation and Verification (ISV)

- Feature-based Machining

- Post Builder

- Manufacturing Miscellaneous

- Manufacturing Advanced Programming

- NX Post

- GPM Legacy Postprocessing

How do I get started with a production version of NX CAM?

Thank you for your interest in NX. If you would like to hear more about the way NX CAM can help you achieve your business goals or if you would like to purchase a production version of NX software, please contact your Siemens PLM Software representative or your Value Added Reseller today.

43

Beyond NX CAM – Siemens part manufacturing solutions

Siemens offers a wide range of software solutions that can help improve part manufacturing productivity. This diagram shows the various capabilities we offer in the areas of manufacturing planning and production.

For more information about any of these solutions, please follow this link to our website:

http://www.siemens.com/plm/nx/cam

Use this DVD to install NX Help.

Siemens Industry Software

Headquarters
Granite Park One
5800 Granite Parkway
Suite 600
Plano, TX 75024
USA
+1 972 987 3000

Americas
Granite Park One
5800 Granite Parkway
Suite 600
Plano, TX 75024
USA
+1 314 264 8499

Europe
Stephenson House
Sir William Siemens Square
Frimley, Camberley
Surrey, GU16 8QD
+44 (0) 1276 413200

Asia-Pacific
Suites 4301-4302, 43/F
AIA Kowloon Tower, Landmark East
100 How Ming Street
Kwun Tong, Kowloon
Hong Kong
+852 2230 3308

About Siemens PLM Software

Siemens PLM Software, a business unit of the Siemens Industry Automation Division, is a leading global provider of product life-cycle management (PLM) software and services with seven million licensed seats and more than 71,000 customers world-wide. Headquartered in Plano, Texas, Siemens PLM Software works collaboratively with companies to deliver open solutions that help them turn more ideas into successful products. For more information on Siemens PLM Software products and services, visit www.siemens.com/plm.