CONNECTICUT CENTER FOR ADVANCED TECHNOLOGY

Automated Production Programming Workflows and Technical Learnings

Digital Enterprise Technology for Maintenance and Sustainment Improvements

Prepared by
Patrick Jonak, Chris Miller, Nasir Mannan, Colette Ruden

February 2025

Table of Contents

01 Who We Are

Connecticut Center for Advanced Technology, Inc. (CCAT), a non-profit organization founded in 2004, is an innovative applied technology organization that leads regional and national partnerships that enable global industrial companies and the manufacturing supply chain across industry sectors in driving advancements and efficiencies to adopt leading-edge technologies, within Industry 4.0 disciplines. CCAT operates a Talent & Training Accelerator and three (3) Advanced Technology Centers: (i) Advanced Design, Automation & Metrology; (ii) Additive Technologies, Optimization & Machining; and (iii) Advanced Composites Technology Center. The Talent & Training Accelerator provides hands-on upskilling opportunities for the existing industrial base on digital transformation concepts and methods. Each center of excellence utilizes advanced manufacturing technologies for applied research and development with expert staff experienced in metals-based AM, 3D printing with polymers, automated fiber placement for composites manufacturing, reverse engineering, structured light scanning, precision machining, smart manufacturing with Artificial Intelligence and digital / IIoT technologies. In addition, CCAT is a member of America Makes, CESMII, MxD and ARM Institutes.

CCAT maintains an extensive network of partners throughout the regional Defense Industrial Base with a focus on small and medium-sized manufacturers. CCAT will leverage its network and Advanced Technology Centers to ensure deliverables focus on technologies that provide the greatest impact and return on investment.

02 Automating CMM Programming by Leveraging Model-Based Definition in Siemens Inspection

Siemens NX offers Coordinate Measurement Machine (CMM) programming capabilities that allow users to benefit from time savings of up to 90%. This is due to an automated programming process that uses the Product Manufacturing Information (PMI) of the Model-Based Definition (MBD) model, along with user-defined strategies that can be tailored to the manufacturer's best practices.

Siemens NX CMM Inspection Programming software has the option to use the programming language of the user's specific CMM or the Dimensional Measuring Interface Standard (DMIS) format. In addition to the benefits mentioned, users that also work within Siemens Teamcenter can maintain a central repository of all program revisions while offering revision control management.

This Teamcenter workflow ties the input MBD model of the CMM programming process, to the output CMM program, which in turn preserves the Digital Thread.

The following steps are set up by the user then employed in Siemens NX Inspection to generate a CMM program automatically using model-based definition:

1. CMM Templates
2. Defining Inspection Methods
3. Generate, Verify Program and Re-sequence paths
4. Collision Avoidance

Workflow tutorials are described in Appendix A.

*Please note that the following steps are created using Siemens NX Version 2206.

1. CMM Template

Creating CMM templates is the foundation for automating the CMM programming process. These templates are defined by a kinematic model of a specific CMM and its assigned probe(s) which are typically provided by the CMM manufacturer. Once setup, users can select the CMM template that exactly mimics the physical CMM setup allowing for accurate simulation and potentially eliminating all programming errors.

Key Features of Siemens CMM Templates:

- *Time Savings in Program Generation*: Siemens' automated programming process gives users the freedom to quickly make a safe and efficient program, with limited programming experience. The user-defined templates embody best practices created by experienced users, optimizing the inspection process.
- *Reduction in Programming Error:* With the CMM kinematic model loaded into a CMM template, Siemens NX can generate a CMM program that understands where the CMM axis limits are, where the probe is in relation to objects such as the part being inspected or the probe rack, and other physical considerations, to avoid collisions or potential safety risks to the users.
- *Customization*: There are a variety of ways that a user can customize the CMM templates. For example, the user can create various physical layouts the CMM may be configured to for different processes.
- *Programming Conveniences*:
    - Design Change Propagation – Within the Method View, features are categorized into different methods, each of which are editable and will propagate these changes to each of the geometries listed under that method.
    - Offline Programming – Programs can be seamlessly generated without having to connect the software to the hardware.
    - Simulation Functionality – Simulation tools give a visual representation of the program and provide options such as collision detection to let the user know exactly where in the program collisions may occur.

2. Link to Product Manufacturing Information (PMI)

By linking to PMI, the user can have NX automatically generate a complete CMM inspection program by utilizing the PMI to drive the program generation process. Siemens NX software reads the semantic PMI information, associates the characteristics with an appropriate inspection method, and creates the program steps for the CMM to inspect the feature.

Key Features of using the Link to PMI function:

- *Automation:* Linking to PMI is central to automating the CMM programming process. Siemens uses the assigned PMI, along with the CMM's kinematic model, to produce a program.

- *Efficiency:* By allowing Siemens NX to automatically generate the CMM program, an order of magnitude of time savings in programming can be realized. The Link to PMI function allows Siemens to identify features that need to be inspected without the need for a user to manually define each feature nor the need to apply an appropriate inspection method to each feature.
- *Design Cycle Enhancement:* The Link to PMI function shortens the overall production programming process which gives the manufacturer the benefit of producing usable programs in significantly shorter lead times. This ultimately saves manufacturers' cost by requiring less time and resources to generate a CMM program.

3. Defining Inspection Method

By defining an inspection method, the user can create methods for inspecting a feature type. For example, if the part that is to be inspected has a series of holes, some with tight tolerance requirements and some with looser tolerance requirements, a method can be created for each type. These methods can be predefined and reused in future programming generations. This allows for less-experienced users to categorize features into appropriate inspection methods that will stage downstream manufacturing or assembly of the component.

Key Features of defining Inspection Methods:

- *Flexibility in programmer experience:* Less-experienced users can create a program using pre-defined methods for inspection. Similarly, experienced users can generate a program in less time because the user can quickly categorize the features to a suitable pre-defined method rather than recreating each time.
- *Adhere to company inspection standards:* Depending on features, some manufacturers elect to inspect features in a specific way. Typically, this is done to inspect various parts in a consistent manner that has been internally approved. In other cases, a manufacturer's customer may require a specific feature type to be inspected in adherence to a customer specification. This is a means to control this requirement.
- *Utilize best practices: A* manufacturer may experience a unique feature that may or may not be shared amongst different product lines. This unique feature may require some special consideration to properly inspect. Through experience, a preferred way may be desired to be applied to accurately inspect a feature. Utilizing the Inspection Methods, this can be accomplished.

4. Generate, Verify Program and

Resequence Paths

The Generate, Verify Program and Resequence Path functions take the changes from modifying or creating inspection methods to generate the new tool paths for the selected operations, verifies there are no DMIS syntax, program, or tolerance errors and resequences them such that the overall program is optimized and reduces unnecessary probe movement to complete the inspection.

Key Features of Generate, Verify Program and Resequence Paths:

- *Automatically generate tool paths based on Inspection method definition:* Once an inspection method has been updated or created, a tool path needs to be generated to the inspection profile for the feature to be inspected.
- *Verify Program:* By verifying the program, Siemens NX software performs an analysis on the program to determine if the following are producing errors:
    - *DMIS Syntax:* Ensures the PMI that is applied to the model meets the standard and can be properly read for program generation.
    - *Program:* Ensures the program is without continuity errors and that the program generation is complete
    - *Tolerance:* Verifies that the tolerance bands applied to a feature can be logically read and without human error.
- *Optimization by resequencing:* Now that new inspection methods are created and the program has been verified, the Resequence Paths function will optimize probe movement and reduce probe tip changeout (if applicable) to shorten the program time.

5. Collision Avoidance

By using the Collision Avoidance function, NX will automatically analyze the generated inspection routine to identify collisions and will automatically insert transition points to prevent any collisions. Once the function is run and the analysis is complete, NX will populate a report stating transition points added and will include any collisions it could not resolve. Further, the user can simulate the process via the Simulate Machine option to visually see where the collisions are taking place.

Key Features of Collision Avoidance:

- *Safety:* In any manufacturing process, human safety is the top priority. By running the Collision Avoidance function, Siemens NX will analyze the CMM program and will prevent the Z-Slide or probe from coming in contact with an object outside of the required probe contact to inspect a feature. This reduces the risk of objects breaking off and potentially injuring the human inspector.
- *Eliminate equipment or product damage:* Similar to human safety,

eliminating damage to the equipment or product will prevent downtime on the equipment or prevent the need to scrap the part that is being inspected.

Leveraging Model-Based Definition (MBD) / PMI and Digital Thread

The benefits of leveraging MBD/PMI within the Digital Thread include enhanced and shortened design cycle times by utilizing incorporated product and process information, along with its associated inspection practices. This process accelerates decision-making for both experienced and less experienced users, enabling them to apply company standards or customer inspection method requirements that have been pre-defined, thereby reducing inspection errors or non-compliances.

Siemens NX software can automate the CMM programming process. This automation reduces programming time, frees up inspection equipment and ensures a fast response to new designs or design changes. Moreover, this process outputs a complete and validated CMM program through the simulation process, ensuring it is error-free, safe for the user, and protective of equipment and components.

03 Automating CNC Programming by Leveraging Model Based Definition in Siemens NX CAM

Siemens Computer-Aided Manufacturing (CAM), software is a part of Siemens' comprehensive suite of solutions designed to streamline and optimize manufacturing processes. It integrates seamlessly with Siemens Product Lifecycle Management (PLM) software and other Computer-Aided Design (CAD) systems to facilitate the efficient design and production of complex parts and assemblies. Siemens CAM solutions are renowned for their capability to support diverse manufacturing techniques, including milling, turning, and advanced multi-axis machining.

Siemens CAM solutions incorporate several advanced features designed to automate and optimize the CNC programming process. These include the setup template, feature-based machining (FBM), teach features, and operation sets. Together, these tools create a highly efficient, automated, and error-minimizing environment that leverages MBD and the digital thread.

Workflow tutorials are described in Appendix B.

CAM Setup Template

A "CAM setup template" in the context of Siemens CAM solutions refers to a predefined configuration that sets up the CAM system to handle various manufacturing tasks more efficiently. These templates are pre-configured data sets or files that include all necessary parameters, tool data, and operational sequences required to execute specific machining operations. The purpose of a CAM setup template is to standardize and expedite the process of programming CNC machines, reducing the time required to transition from design to production, and minimizing the potential for errors.

Key Features of Siemens CAM Setup Templates:

- *Standardization*: Templates provide standardized processes, ensuring consistent quality and output across multiple manufacturing cycles and setups. This is particularly beneficial for industries where precision and repeatability are critical.
- *Efficiency*: By using templates, manufacturers can significantly cut down on the time it takes to set up CAM operations. Since the essential settings are pre-loaded, CAM programmers can avoid starting from scratch for each new job, making the process quicker and more efficient.
- *Reduced Error*: With manual settings, the likelihood of errors increases, which can lead to faulty

parts and wastage. Templates mitigate this risk by providing tried and tested configurations, thereby enhancing overall reliability.

- *Customization*: Even though templates provide a standard setup, they are also customizable. This allows CAM programmers to tweak the templates according to specific project needs without deviating from the core parameters that define the operation's integrity.
- *Learning and Adoption*: For new users or less experienced operators, templates serve as learning tools. They help in understanding the intricate settings and parameters that govern efficient CAM operations.

Applications in Industries:
Siemens CAM setup templates are applicable in a variety of sectors such as aerospace, automotive, industrial machinery, and high-tech electronics, where CNC machining is prevalent. These templates are instrumental in producing complex part geometries with high precision on a consistent basis.

In practice, once a template is developed (often by a more experienced CAM programmer), it can be reused across similar manufacturing tasks or adapted for slightly different tasks, significantly contributing to lean manufacturing practices.

It is recommended to use CAM setup templates to enhance your operational efficiency and maintain consistency in product quality. These templates can be a part of the initial software package or can be created and customized according to specific user needs. Moreover, Siemens often provides training and resources to help users maximize the benefits of their CAM systems, including the effective use of templates.

Built-In Feature Based Machining
Siemens' built-in FBM is an advanced capability integrated within Siemens NX software. Feature-based machining automates many aspects of the planning and programming stages in CNC machining by utilizing intelligent algorithms that automatically recognize features of a part model and apply appropriate machining strategies based on those features.

Core Components of Feature-Based Machining

- *Feature Recognition*: This is the process where the software scans a part's 3D model, typically developed in a CAD system, and identifies standard machinable features such as holes, pockets, bosses, and slots. The recognition process can be fully automatic, where the software detects features without user input, or interactively, allowing the user to guide the recognition process.
- *Knowledge Database*: FBM works based on a predefined set of rules and machining strategies that are

linked to specific types of features. This knowledge base includes recommended tools, sequences, and cutting parameters optimal for each type of feature identified in the part.

- *Automatic Tool Path Generation***:** Once features are identified and categorized, Siemens NX automatically generates the tool paths needed to machine these features. This process is guided by best practices and predefined strategies in the software's database, ensuring efficient and error-free programming.
- *Customization and Extensibility***:** Users can modify existing rules or add new ones to the FBM's database to tailor the system's output to specific needs or to adapt to new types of parts and operations. This makes FBM highly adaptable to a wide range of industries and machining complexities.

Benefits of Siemens Feature-Based Machining

- *Reduced Programming Time*: Since many of the tasks are automated, from feature recognition to tool path creation, FBM substantially reduces the programming time required compared to traditional CAM methods.
- *Consistency and Standardization*: By using standardized machining strategies stored in its knowledge database, FBM ensures consistent and repeatable processes, which is crucial for maintaining quality across multiple parts and production cycles.
- *Improved Efficiency*: FBM optimizes the machining process by selecting the most appropriate tools and cutting strategies. This optimization can lead to significant savings in time and material, as well as prolonging tool life.
- *Error Reduction*: Automating the machining process reduces the likelihood of human errors in programming, which in turn decreases costly reworks and scrap rates.
- *Scalability and Learning*: As more features and their respective strategies are added to the database, the system becomes more robust and capable of handling an even greater variety of parts and complexities more efficiently.

Teach Features

Teach Features in Siemens NX CAM is a functionality that allows users to manually identify and define features on a part that have not been automatically recognized by the system's feature recognition capabilities. This can be particularly useful when dealing with complex parts or when the automatic feature recognition does not completely understand all design intents.

The process typically involves the following steps:

- *Manual Selection*: The user manually selects geometric elements on the part model that comprise a feature.

- *Feature Definition*: The user then defines what type of feature these elements represent (e.g., holes, pockets, slots).
- *Attribute Assignment*: Assigns attributes such as depth, diameter, and other relevant parameters to the manually defined feature.

This "teaching" of new features to the system helps in refining the CAM process, ensuring all necessary machining operations are correctly planned and executed.

Teach Operation Sets

Teach Operation Sets refers to the capability in Siemens NX to create and store sets of machining operations that can be easily reapplied to similar features in the same project or future projects. This facilitates the reuse of preferred machining strategies and reduces the time required for CAM programming on repetitive tasks.

Steps typically involve:

- *Operation Selection*: After defining the operations for a particular feature or set of features, these operations can be grouped into a set.
- *Operation Set Saving*: This set of operations can be saved with a specific naming convention that reflects its function or the type of feature it’s associated with.
- *Reapplication*: These saved operation sets can then be 'taught' to new features, either within the same part or in different parts, where the features are similar. This means that once a satisfactory set of operations is defined, it can be quickly applied to similar situations without the need to rebuild the operations from scratch.

Benefits of "Teach" Features and Operation Sets

- *Efficiency*: Significantly speeds up the programming process by eliminating repetitive work.
- *Consistency*: Ensures consistent machining practices are followed across similar features in different parts.
- *Customization and Flexibility*: Allows machining strategies to be tailored to specific manufacturing or company standards and still be applied automatically.
- *Learning and Adaptation*: The software learns from the operations and features defined by users, making it more robust and versatile over time.

These features in Siemens NX CAM assist in bridging the gap between automated machine programming and the unique needs of custom manufacturing environments. They improve the workflow by combining standardized approaches with flexibility and user input, ensuring both productivity and high-quality outcomes in complex machining tasks.

Leveraging MBD and Digital Thread

Both MBD and the Digital Thread are about ensuring that data flows seamlessly from design to manufacturing and onto other phases in the product lifecycle. Siemens CAM tools ensure that the transition from CAD to CAM is not only smooth but also retains all integral data which improves decision-making, efficiency, and traceability. By integrating these tools, Siemens CAM effectively addresses several of the traditional challenges associated with CNC programming. It makes the process faster, less susceptible to human error, and more closely integrated with the updated design data, thereby enhancing the overall efficiency and flexibility of manufacturing operation.

04 Automated Costing by Leveraging Model-Based Definition in Siemens

Siemens Product Cost Management (PCM) is software in the Siemens Teamcenter suite used for calculating costs associated with the manufacturing process, enabling manufacturers to effectively maximize profits.

Benefits of Siemens PCM

PCM allows for the creation of automated quoting workflows, reducing the time needed to cost a part. This workflow leverages PCM's ability to import data from a Bill of Materials (BOM). Once a BOM is created with a corresponding import configuration within PCM, the entire process of calculating a per-piece cost takes just minutes to complete. Along with time savings, the benefit of this workflow is its repeatability. The same part can be evaluated multiple times using alternative parameters, such as location or machine model, to determine the most cost-effective method of production. *The workflows in Appendix C demonstrate automated part quoting using a BOM in Siemens PCM v2312.*

Key Features in Siemens Product Cost Management

- *NX CAM Part Metadata*: Using NX CAM metadata allows for the costing process to be tied directly to the part model. This means that any changes to the manufacturing process within the G-code are automatically reflected in the metadata. Using the metadata as a source of truth allows for accurate and fast quoting.
- *Part BOM file*: The final step before importing the part into PCM will be to create a BOM file in Excel from the operation metadata exported from NX CAM. Ideally, the BOM will pull data directly from the exported file using Power Query, Microsoft's proprietary "data transformation and data preparation engine." Power Query can automatically extract, load and transform the metadata from NX CAM, giving substantial time saving over doing the process manually. An additional benefit of automating the BOM is that when the process within NX CAM changes, the BOM can be refreshed, with the changes automatically reflected. The full process of scripting the queries using Power Query in Excel can be achieved in many ways. For reference, linked is Microsoft's documentation for Power Query. This tutorial focuses on the initial data extracted from NX CAM and the final BOM structure to be imported, deferring to the reader on programming the data transformation. Nevertheless, the

process of creating the BOM can still be done manually, albeit without time savings.

05 Digital User Stories and Technical Training

CCAT engineers worked with technology partners and Department of Defense manufacturers to identify and document user story workflows capturing best practices for the development and assessment of digital transformation tools and strategies. These products also serve as self-paced software training videos focused on elements of MBD-workflows. Each video is publicly available, and the workflows are completed with representative parts to ensure these videos can be shared inside and outside of Department of Defense suppliers for technical training support.

Manufacturer Testimonials:

1. *Machining Optimization: Third Wave Systems & New England Airfoil Products*

CCAT partnered with a manufacturer, New England Air Foil Products Inc., to bring you a realistic and applicable user story in support of the department of defense and its digital transformation efforts. Their North American President, Nik Delic, joins us in sharing the return on investment, along with their Engineering Manager, Brian Jacobs, responsible for the integration and adoption of new technologies.

2. *Digital Transformation & Training: BNL Industries*

CCAT's engineering team has partnered with BNL industries to provide a user story focused on sharing the impact of online and in-person technical training offerings, along with recommendations from an existing Department of Defense manufacturer for others who are looking to jump start their digital transformation initiatives.

Technical Training Videos:

3. *Automated Inspection: M3DI & Polyworks*

CCAT partnered with M3DI to demonstrate automated programming for structured light scanning workflows with the consumption of model-based definition parts via Polyworks Inspector Software. This technology and software provide the ability for manufacturers to execute inspection programs, output completed inspection reports and upload the report to the inspection directory.

4. *Leveraging MBD with CNC Programming: TriMech & SolidCAM*

CCAT partnered with Trimech, to demonstrate SolidCAM, which runs completely inside SolidWorks and is a different program compared to SolidWorks CAM. By providing videos on both software solutions, our team aims to share the ability to leverage model-based definition and digital thread methodologies regardless of the solution your company has already invested in to support all manufacturers of all sizes.

5. Automated CNC Programming: SolidWorks CAM

To provide training materials to reach the majority of Department of Defense suppliers this video provides step by step details on using SolidWorks CAM to create automatic CAM programs with the pre-taught machining strategies. If your team uses Siemens platforms, check out our other two videos on automated CAM programming using Siemens NX.

6. Automated CNC Programming: Siemens NX CAM

This 2-part video features our Principal engineer Nasir Mannan in the usage of Siemens NX to create automated CAM workflows. This is achieved by developing the tool selection strategies and machining operations that Siemens can pull to create the CAM program.

7. Automated CNC Programming: Siemens NX with Custom Features and Operation Sets

This video is broken down into 3 sections, from creating templates to Siemens NX feature based machining capabilities and then onto custom feature development. The CCAT team has proven similar workflows on existing supplier parts to demonstrate time and cost savings with the implementation and usage of product manufacturing information.

8. Automated CMM Programming: Siemens NX

This video is directly focused on providing a training workflow created and demonstration in the Army Digital Proving Ground using the Siemens CMM module for automated programming of a CMM using a model-based definition part. You'll hear from our principal engineer Nasir Mannan and manufacturing applications engineer Chris Miller on this topic.

9. Automated Costing: Siemens Product Cost Management (PCM)

CCAT engineer, Patrick Jonak, demonstrates a full workflow in Siemens Product Cost Management software focused on automated part costing. This user story demonstrates the technical capabilities of the Siemens costing platform to support manufacturing costing.

10. Automated Welding: Siemens, Capvidia & Visual Components

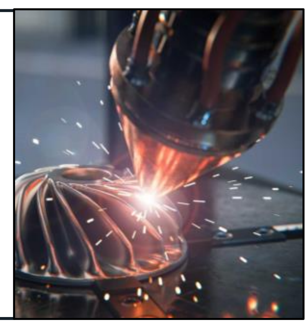
Two CCAT's engineers, Chris Miller and Maleeha Wahid are featured to share an example workflow for automated welding program development. They begin in Siemens NX with a Smit Bracket that highlights weldment call outs. Next, they move through a full workflow of robot programming, inspection, and comparison between the original CAD model and the final part.

06 Digital Transformation Resources List

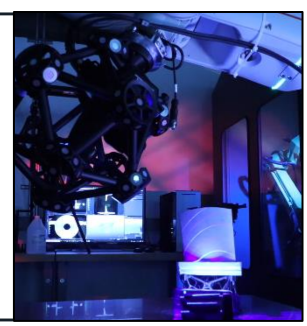
Hands-on Training
CCAT's Talent & Training Accelerator offers monthly training on 5 Career Pathways: Digital Design & Costing, Digital Manufacturing, Digital Inspection/Quality Assurance, Additive Manufacturing, CNC Operations and Digital Thread

On-Demand Technical Training
Step-by-step technical workflow training is offered publicly for reference via CCAT's YouTube playlist. Check out the training offerings: https://www.youtube.com/@CCATMedia/playlists

Regional Support
NIST supports Manufacturing Extension Partnerships (MEP) in all 50 states and Puerto Rico to provide services and resources to manufacturers. Check your local MEP out today at: nist.gov/mep/centers



Technical Demonstration Projects
CCAT's Advanced Technology Center offers demonstrations of existing technology with applicable parts and workflows. Contact our team today for a consultation!



Connect with us.

info@ccat.us

CCAT

@CCATInc

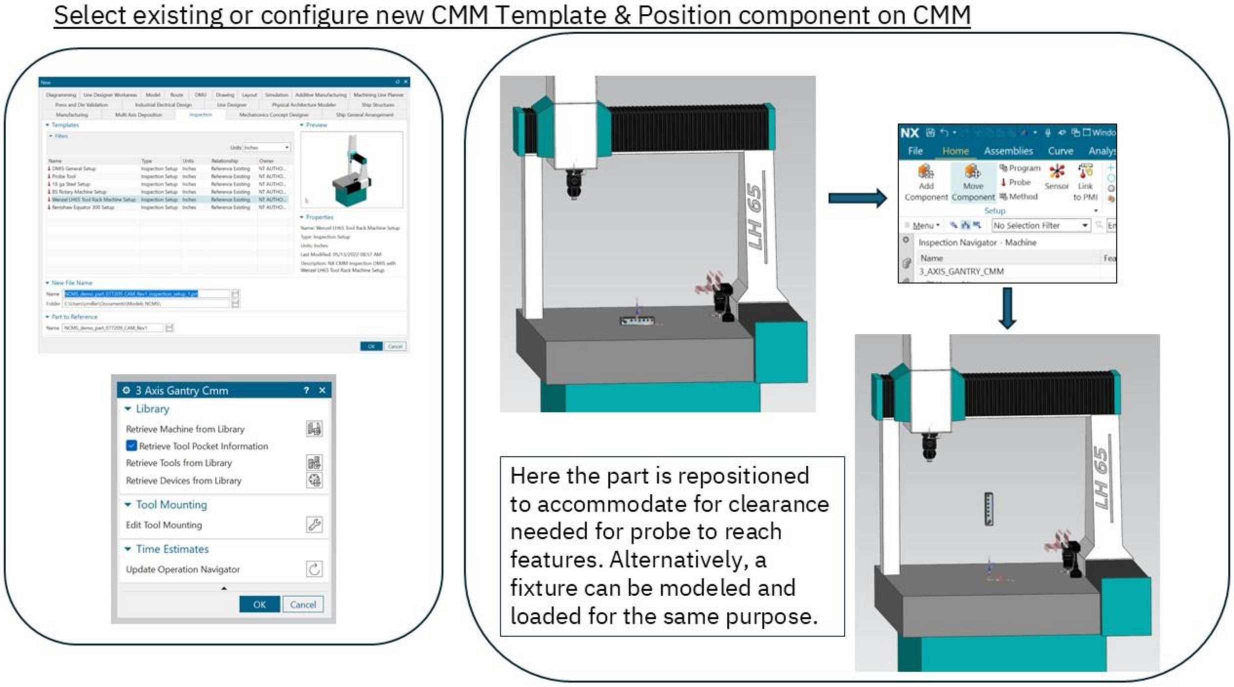
ccat.us

Appendix A: Automated CMM Step by Step Guides

CMM Template Tutorial:

1. Open the model file within Siemens NX
2. Select File > New...
3. Select the Inspection
4. From the list, select the appropriate option that matches the actual inspection environment.
5. Select the current units from the dropdown list
6. Select File Name and the Part to Reference, if applicable
7. Click OK
8. Once NX has loaded, the part will be loaded into the CMM Programming environment
9. From here, the user can use the Move Component function to place the part on the CMM where it will be inspected. If a fixture is required, this can be imported as well and the part can be fixed to this imported fixture.

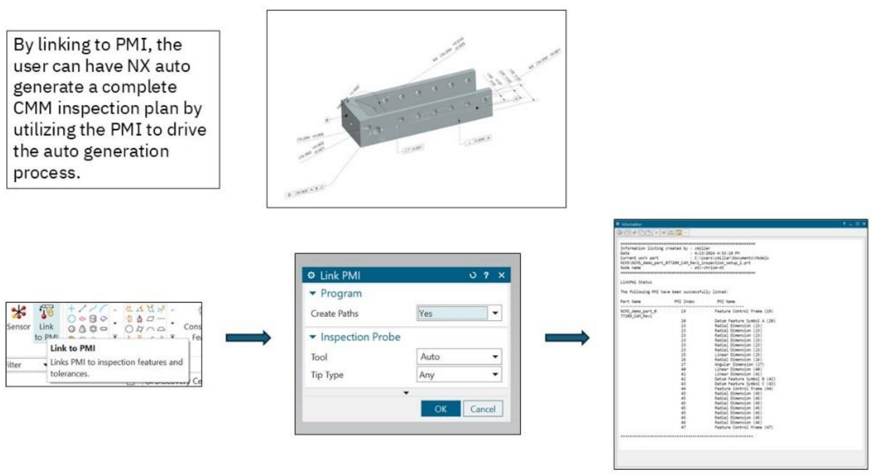
Select existing or configure new CMM Template & Position component on CMM



Link to PMI Tutorial:

1. On the top ribbon, ensure the Home tab is selected
2. Select Link to PMI and a dialog box will open
3. To have Siemens NX auto generate the program, select Yes in the dropdown box for Create Paths
4. Select the appropriate options Under the Inspection Probe section
5. Once complete with the dropdown box, click OK
6. A new dialog box will open stating the PMI that was imported successfully or not
7. If any PMI was not imported successfully, readdress how the PMI is applied to the model and repeat the Link to PMI process

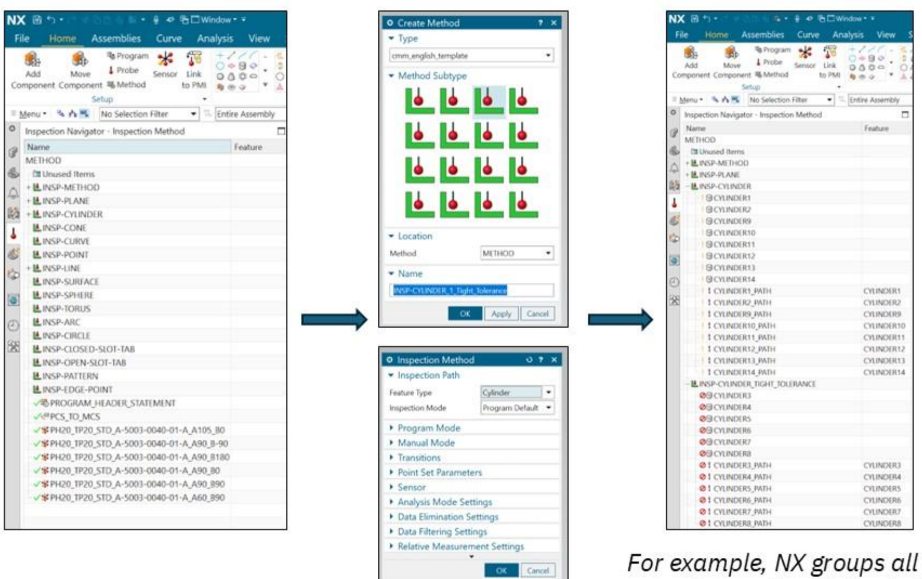
Link to PMI

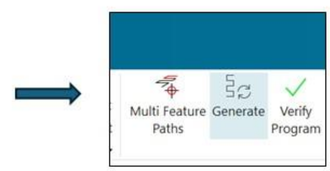


Inspection Methods Tutorial:

1. Enter the Method View to view the existing methods and the features that were previously assigned via the Link to PMI procedure
2. The user is able to do one of two options to modify how a feature is inspected:
   a. If the user decides that all features of a feature type should be inspected one specific way, double-click on the desired inspection method and make appropriate changes to the dialog box that opens
   b. If the user decides a new inspection method should be created to inspect a feature, select the Method function in the top ribbon. To locate this, be sure to be in the Home tab
      i. The user should select the Method Subtype that aligns with the feature type,
      ii. Select a location where the method should be stored under,
      iii. And provide a name to this new inspection method
3. Once the dialog box has opened, the user can modify the pertinent parameters that are best fit for the feature
4. After defining the inspection method parameters, click OK to save the method.
5. Now the user can select the feature ID and the associated feature path and drag-and-drop them into the new inspection method

(Optional) – Categorize features into Inspection Method based on tolerance requirements





By categorizing features into Inspection Methods, the user can determine the appropriate method type based on the tolerance requirements.

For example, NX groups all cylinder features together under the same Inspection Method. Perhaps the part has a few holes, some with loose tolerances and some with tighter tolerances. The user can create appropriate inspection methods for each tolerance value to accommodate for subsequent assembly, bonding, etc.

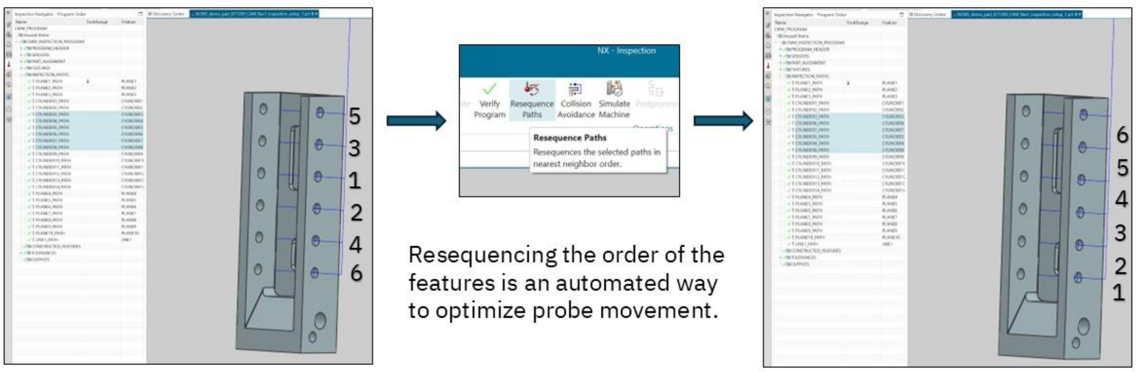
Generate, Verify Program, and Resequence Tutorial:

1. Once the features have been correctly categorized, ensure that the Home tab is

selected

2. Select the Generate button to create the tool paths
3. Once the Generate process is completed, select the Verify Program button to perform the analysis on the program checking for error
    a. Select a location for the verification file to be saved in for future reference
4. Select Resequence Paths which will optimize the tool paths to reduce inspection time

<u>(Optional) – Resequence Paths to Optimize Probe Movement</u>



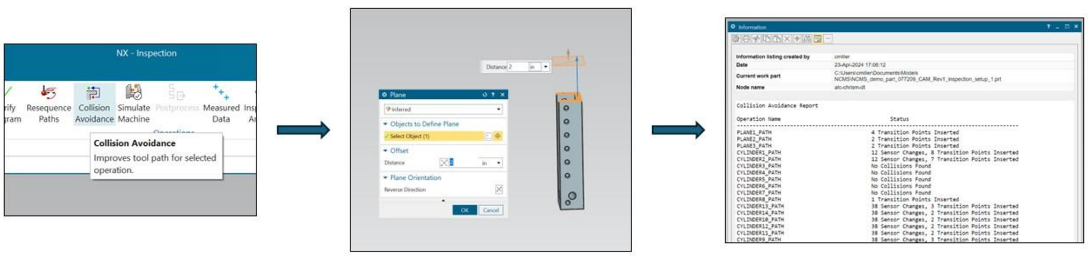
See the before and after images to see the order in which the holes would be inspected based on Resequencing Paths.

Note the number associated with the hole features in the images is to indicate order of operation. It does not indicate the assigned feature number shown in the tree diagram.

Collision Avoidance Tutorial:

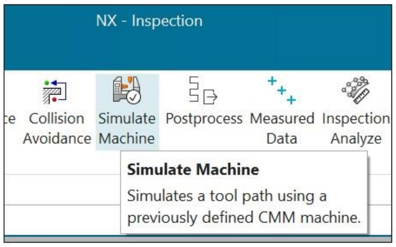
1. Select the Collision Avoidance button under the Home tab, a dialog box will open
2. Specify safety plane – this is typically the upper most surface so that probe head will move and articulate away from the part to maintain a safe process
3. Specify a clearance distance so that probe angle changes happen at a safe distance from the part so that a collision is avoided
4. The user can further select or deselect options under the Avoidance Methods section of the dialog box that meets inspection intentions
5. Similarly, the Collision Detection Settings can be modified to adjust the sensitivity on what is considered a collision

Collision Avoidance



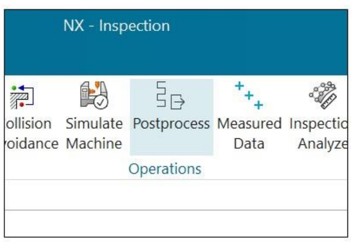
By using the Collision Avoidance function, NX will automatically analyze the generated inspection routine to identify collisions and will automatically insert transition points to prevent any collisions. Once the function is run and the analysis is complete, NX will populate a report stating transition points added and will include any collisions it could not resolve.

Collision Avoidance

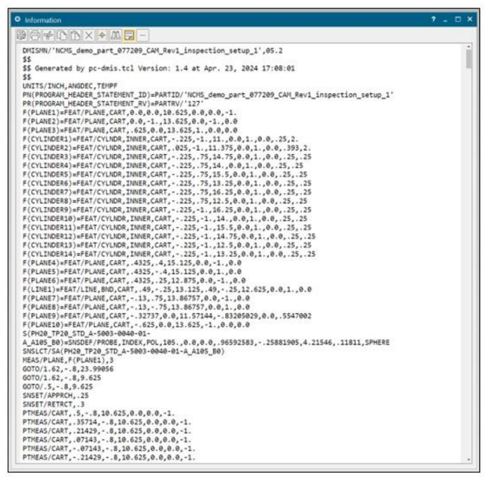


By Simulating the Machine, the user can watch an animated routine to show how the CMM can be expected to run live.

Postprocessing



After the user has confirmed the routine is complete, they can select the Postprocess function. This will output the entire inspection code which can be read into the associated CMM software.



Appendix B: Automated CNC Step by Step Guides

CAM Setup Template Tutorial:

The following step by step work instructions share the process to set up CAM templates in Siemens CAM:

1. Click “File > New”, select “Blank” and press “OK”.
2. In the ribbon bar, select “Application” and launch “Manufacturing”.
3. In the “machining environment” pop-up, the automatic selection can be kept. Generally, a good starting point for “CAM Session Configuration” is “cam_general” and a good starting point for “CAM Setup to Create” is “mill_contour”.
4. Edit the workpiece to select the desired material for the template. Add any tools/operations and modify any settings that you want saved when you load the template.
5. Within the operation navigator, right click on the added operations, hover over “Object”, and click “Template Settings”.
6. Checking the “Object can be used as template” box tells NX to load the selected item when using this file as a template. Checking the “Create if parent is created” will tell NX to create the selected item if the parent item is created (for example, creating a certain tool only if the operation that uses that tool is created).
7. Select the appropriate template settings in the tool view, program view, and geometry view as well.
8. Save and close the template file.
9. Open a part that you’d like to machine using the newly created template.
10. Launch “Manufacturing”, and at the bottom of the “Machining Environment” pop-up, click “Browse for a Setup Part”.
11. Select the saved template file and click “OK”.
12. All the template settings are loaded in. The workpiece and blank can be selected, and the operations can then be generated. Note, if operations that require geometry selection were added to the template, you must modify the operation and select the geometry on the workpiece.

Using Siemens CAM Built in Feature-Based Machining Tutorial:

The following step by step work instructions convey the process of using Siemens CAM built in Feature-Based Machining:

1. Navigate to the Machining Feature Navigator in the left-hand side toolbar.
2. Right click within the Machining Feature Navigator and select “Find Features”.

3. In the “Find Features” pop-up, set “Type” to “Parametric Recognition”, set the desired “Search Method” (“workpiece” is generally good if you want to search the entire part), and select the feature types you want found in “Features to Recognize” (deselect the ones you aren’t interested in identifying).
4. Click the “Find Features” icon in the bottom right of the window.
5. The “Recognized Features” box will be populated with features found that match the search criteria. You can remove any features you do not want, then select “OK”.
6. In the Machining Feature Navigator, select all the features, right click, and then click “Create Feature Process”. Under “Type” select rule based, make sure all the machining knowledge is checked, and then press “OK”.
7. Within the Operation Navigator, you will now see feature group(s) with associated operations. The newly created operations can then be generated.

Teach Features and Operation Sets Tutorial:

The following step by step work instructions convey the process of using Siemens CAM Teach Feature and Operation Sets functionality:

1. Navigate to the Machining Feature Navigator in the left-hand side toolbar.
2. Right click within the Machining Feature Navigator and select “Teach Features”.
3. Under “Select Feature Type”, highlight “CustomerFeature” and then click the icon next to “Add New Feature Type”. Type in a name for the feature you intend on teaching.
4. Under “Select Rule”, highlight “CustomerFeature” and then click the icon next to “Add New Rule”. The name you created for the new feature type should be automatically populated.
5. Select the faces that make up the feature you are teaching NX to recognize.
6. Click the icon in the bottom right of the window next to “Teach Recognition Rule”.
7. Right click within the Machining Feature Navigator and select “Find Features”.
8. In the “Features to Recognize” box, expand the “CustomerFeatures” group and notice the new feature you created. Make sure the box next to the new feature is checked, and then click the icon in the bottom right next to the “Find Features Text”. You should see the newly taught feature populate in the “Recognized Features” box. Click “OK”.
9. In the Machining Feature Navigator, highlight the newly created feature(s), right click, and select “Group Features”. In the “Group Features” dialog box, click the icon in the bottom right next to the “Create Feature Groups” text, then click “OK”.
10. Navigate to the Operation Navigator. In the Geometry View, you should see a new feature group named after the newly created feature.

11. Add in any tools and operations necessary to machine this feature.
12. Right click the feature group, hover over “Object”, then click “Teach Operation Sets”. Highlight the feature group within the “Operation Sets” box, then click the icon in the bottom right next to the “Teach Machining Rules” text. This tells NX to automatically generate the tools and operations within the feature group every time you “Create Feature Process” on this feature type.
13. To show what you’ve done so far, you can delete the feature group and related operations, navigate back over to the Machining Feature Navigator, highlight the newly taught feature(s), right click, and select “Create Feature Process”. In the pop-up under “Knowledge Libraries”, if you expand “OperationSets” you should see the taught feature and the associated feature group containing the operations you’ve taught NX to automatically generate. Make sure they are all checked off and click “OK”.
14. If you navigate back to the Operation Navigator, the feature group and the taught operation(s) were automatically generated.
15. Next, we are going to add parameters to the created feature and use these parameters to teach NX to use logic when machining the feature based on applied PMI. Navigate to the “Applications” ribbon and launch “Modeling”.
16. Add PMI to the feature you are teaching NX to recognize. Make sure the PMI you add is representative of the parameters you want access to in the “Teach Operation Sets” dialog. For the added PMI to be converted to a parameter, *you must add text in the right textbox of the PMI.* This text will become the name of the parameter. If adding a surface roughness, the text in the “Title” field will become the name of the parameter.
17. Navigate back to the “Applications” ribbon and launch “Manufacturing”.
18. In the Machining Feature Navigator, right click and select “Teach Features”. Select the feature type and the recognition rule of the feature you created, which we are now modifying to add parameters. Reselect the face(s) of the feature as well. If you are missing a grey box with additional options in the “Recognition Rule” field, click the gear icon at the top left of the “Teach Features” dialog box, and select “Teach Features (more)”. Then, under “Recognition Rule Options”, check the box next to “PMI Annotations”. This adds matching conditions from PMI annotations on the feature, assuming the PMI was added properly (as outlined previously). Click the “Teach Recognition Rule” icon, then click “OK”.
19. Now, go to the Machining Feature Navigator, and again “Find Features” and “Create Feature Process”, making sure the taught feature is checked.
20. Return to the Operation Navigator, right click on the created feature group corresponding to the newly taught feature, hover over “Object” and the click

"Teach Operation Sets". Notice, there are new parameters listed that correspond to the PMI added.

21. Now we are going to change the way NX machines the feature based on the PMI. In the Operation Navigator, create a new feature group (or copy the existing one and edit it) and add the operations desirable if certain PMI is present on the feature. For example, the original feature group may be desirable for loose tolerances, whereas the new feature group is desirable for tight tolerances. We will define when NX should generate each feature group, and any number of different groups can be created and then taught to generate based on different PMI conditions.
22. Make sure to right click on the new/copied feature group, click "Edit", and select the features associated with the group. These should be the same features associated with the automatically generated feature group.
23. For each feature group, right click, hover over "Object" and the click "Teach Operation Sets". Click on the feature group in the "Operation Sets" tab, then use the "Parameters" and "Conditions" to create a new condition that tells NX when to generate this feature group. For example, you may want NX to generate a given feature group when the tolerance range is less than a certain threshold value.
24. Once the condition is complete, click on the check mark icon next to the "Edit Condition" box. Then click the icon next to "Teach Machining Rules" in the bottom right.
25. Now if you delete the feature groups and "Create Feature Process" again, NX will generate whichever group(s) match the conditions. If you change the PMI and return to the Machining Feature Navigator, you'll see the icon next to the feature group has changed (the new icon signifies that associated PMI was changed). Right click the feature and click "Approve Changes". Then you can "Generate Feature Process" as normal.

Appendix C: Automated Product Costing Step-by-Step Guide

Exporting NX CAM part metadata tutorial

1. Find the part to be quoted within your directory.
2. Open the part with NX CAM.
3. Right-click anywhere in the “Operation Navigator” sidebar.
4. Select “Export to spreadsheet” option in the dropdown menu.
5. A dialog box will open, and you can select the name and location for saving your export file.
6. This will create a “live” Excel file, meaning any changes in the operations on the part within NX CAM will be automatically reflected in the file.

Creating a BOM file tutorial

1. Open the metadata file exported from NX CAM in Excel.
2. Select the top row of the file, and in the “Editing” section of the “Home” Tab click on “Sort & Filter”.
3. Find the column heading “Tool Number” and click on the dropdown next to it. In the menu, filter out the blank values.
4. Next copy the “Description” and “Time” columns into a new Excel spreadsheet. This will be the BOM file.
5. The final column structure of the BOM:
    a. Line Type: This is an identifier which will determine which column values will be populated per row. There should be 4 rows per operation, each with one identifier. These are the possible values for this column:
        i. Part
        ii. Detail (Detailed manufacturing step)
        iii. B (Labor)
        iv. M (Machine)
    b. Level: The BOM level of each row, starting at 0 for the part level, increasing by 1 for each subassembly level.
    c. Item number: The unique identifier for the part within PCM. Must be unique, cannot be used by any other part uploaded to PCM.
    d. Description: If the row’s “Line Type” is “Part” then it is the name of part. Else, it is the name of the operation.
    e. Region: If the row’s “Line Type” is “Part,” then it is the PCM identifier for the region the part is meant to be produced in, e.g. “Siemens.TCPCM.Region.Common.USAConnecticut” for Connecticut,

USA. Unique identifiers can be found in PCM using Full Text Search (FTS). Else, the column value should be blank.

f. Qty: Quantity of parts to be produced per batch, positive integer greater than 0. The value should be the same for all rows.
g. Qty Units: The unit of measure for quantity. This value should be “pcs” for all rows.
h. Currency: The currency used to quote the part. This value should be “USD” for all rows.
i. Annual Req: If the “Line Type” is “Part,” then it is the total number of parts to be produced per year. Else, the column value should be blank.
j. # Mfg Lots: If the “Line Type” is “Part,” then it is the number of lots to be produced per year. Else, the column value should be blank.
k. Life Time: If the “Line Type” is “Part,” then it is the total number of years the part is to stay in production. Else, the column value should be blank.
l. Procurement Type: If the “Line Type” is “Part,” then the value should be “purchase.” Else, the column value should be blank.
m. Calculation Quality: If the “Line Type” is “Part,” then the value should be “benchmark.” Else, the column value should be blank.
n. Cycle Time: If the “Line Type” is “Detail,” then the value should be the “Time” value from the NX CAM export converted to seconds. The NX CAM export is formatted as hh:mm:ss. Else, the column value should be blank.
o. Units: f the “Line Type” is “Detail,” then the value should be “s” for seconds. Else, the column value should be blank.
p. Labor Qualification: If the “Line Type” is “B” then the value should be PCM’s unique identifier for a machine operator “Siemens.TCPCM.MasterData.CostFactor.Common.MachineOpertor.” Else, the column value should be blank.
q. Labor name: If the “Line Type” is “B” then the value should be “Machine operator.” Else, the column value should be blank.
r. # Labors: If the “Line Type” is “B” then the value should be the number of laborers needed to complete all operations, i.e. the production line. Else, the column value should be blank.
s. # Atnd Systems: If the “Line Type” is “B” then the value should be the number of machines in the production line. Else, the column value should be blank.
t. Machine: If the “Line Type” is “M” then the value should be the unique identifier of the CNC machine used to produce the part. As with

"Region", and "Labor Qualification" unique identifier can be found with FTS. Usually, they correspond to the model-name of the machine. Else, the column value should be blank.

6. Please note, as mentioned previously, each operation will need 4 rows with each with its associated data, depending on the line type. Take care that the operations remain in the order they were exported out of NX CAM.

Glossary of Abbreviations

BOM	Bill of Materials
CAD	Computer Aided Design
CAM	Computer Aided Manufacturing
CCAT	Connecticut Center for Advanced Technology
CMM	Coordinate Measurement Machine
CNC	Computer Numerical Control
FBM	Feature Based Machining
MBD	Model-Based Definition
MEP	Manufacturing Extension Partnership
PCM	Product Cost Management
PLM	Product Lifecycle Management
PMI	Product Manufacturing Information
ROI	Return on Investment