

NX Manager CAM

Reduce complexity associated with managing NC programming and associated data

fact sheet

Siemens PLM Software

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► Summary

NX® Manager CAM software provides a manufacturing process based environment in which NX CAM users can manage NC programming data. It also replaces user directories on local disc drives and centralizes the storage of NC programming data. Users can save and retrieve the correct NC data configuration for a particular job, including part information, setup and tooling data, as well as shop documentation output from the NC programming session. NX Manager CAM also provides access to comprehensive libraries for both tooling and machining process templates.

Benefits

Fast, easy and effective access to correct CAM data saves time and reduces errors

Capture and re-use of proven methods results in higher levels of productivity, improves process repeatability and quality

Standard tooling and fixtures (selected from a common shared library) encourages use of standard, approved resources and improves utilization of existing assets

NC programmers can employ a purpose-built data management tool that eliminates the need for user file management in personal directories on local PC discs

Integration with Teamcenter® Manufacturing software avoids need to re-enter data into a separate planning system, saving time and reducing errors

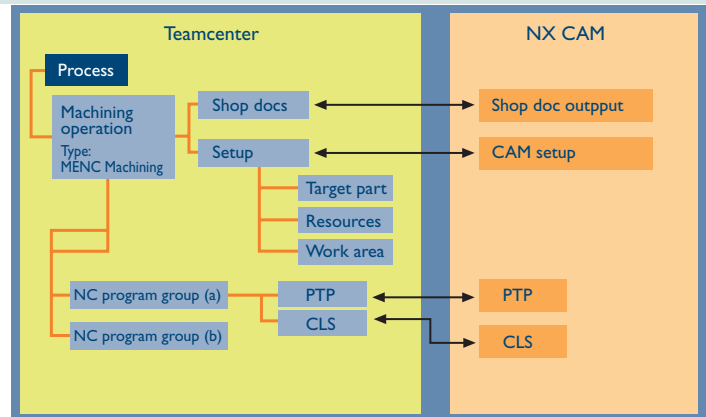
Reduce delays and costs with NX Manager CAM

NX Manager CAM provides a managed information environment that ensures that complex data required and created by a CAM session can be stored in an easily retrievable manner. This is in sharp contrast to the use of the standard operating system on a local PC where the user is forced to maintain data in complex directory structures.

It ensures that the right data, correct tooling and right NC 'tape' (or machine file that matches the required part) are deployed, resulting in reduced delays and costs due to typical information errors. Users can quickly locate and apply proven methods through access to setup and operation templates managed in Teamcenter (Siemens PLM Software's digital enterprise management solution). Additionally, NX Manager CAM automatically establishes machining operations to support a component process plan that is then managed in Teamcenter. NX Manager CAM is a direct, consistent extension of Teamcenter – the same proven software that manages NX product design data.

The solution also captures machining knowledge in a library of templates for machining processes, and manages CAM operations setups. Easy access to information reduces time and costs of re-establishing proven methods and leads to higher quality products and repeatable processes.

NX Manager CAM allows alternate users with appropriate authority to share access to the NC part programming data for re-use, with edit/review via a central database. This saves time and avoids errors that can occur when data has to be moved or copied between different computers. NX Manager CAM is fully compatible with NX Manager, used to manage part data and therefore allows



NX Manager CAM integrates NX CAM with Teamcenter

Features

Provides data management capability designed especially for NX CAM users

Automatically captures NX CAM output data for each operation under a manufacturing process

Stores NX CAM data under revision control

Connects NC programming session to tooling and fixture libraries managed within the resource manager

Provides machining process templates from the Resource Manager template library

Forms key element of a wider part manufacturing planning solution, combining NX and Teamcenter Manufacturing applications

Supported platforms

Hewlett-Packard: HP-UX
Microsoft: Windows XP
SGI – IRIX
Sun SOLARIS

easy access to part models and other design data. This is especially important where the NC programmer needs to share access to current design data.

Integration with the manufacturing elements of Teamcenter means that NC part programming can form an integral part of an overall part manufacturing planning solution – connecting the tasks of the manufacturing engineer, the tooling designer and the NC programmer – to improve communication and reduce time and errors.

Major capabilities

Intelligent management and storage of CAM data elements, such as setup templates, CLS files, postprocessor output files, shop documentation within a defined information structure. These elements are attached to the appropriate process and activity steps within Teamcenter

NX Manager CAM provides integration between NX CAM and Teamcenter to provide a managed information environment for the NC programmer. The functionality is a direct extension of the highly effective NX Manager Unigraphics technology that is widely used to manage NX design data. NX Manager CAM takes advantage of the specific manufacturing data types and structures available within the manufacturing elements of Teamcenter. The CAM operation data, including setup, tooling, part files and output data, can be automatically saved under a master process reference. All of the data is then under revision and change control and managed within the digital information backbone provided by Teamcenter. Importantly, the new manufacturing data structure preserves associativity between the component part file and the CAM data, and performs impact analysis whenever a design change is proposed.

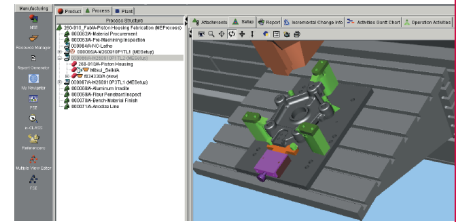
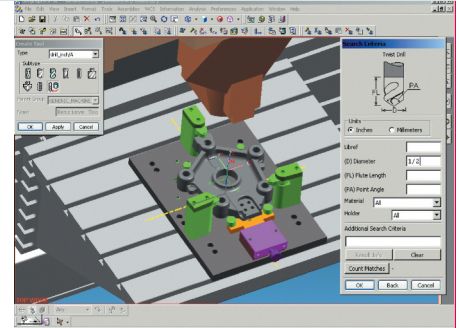
Selection and retrieval of tools from the in-class library within Teamcenter

NX Manager CAM adjusts the standard user interface options in NX CAM to provide user access to Teamcenter Manufacturing from within the NX CAM session for typical saving and retrieving of CAM session data. The NC programmer can access the Resource Manager libraries within Teamcenter. These libraries can include tools, tool assemblies and fixtures – classified via the Resource Manager application – that can be searched, retrieved and applied in the NX CAM session.

The automated save and retrieval of the CAM information between NX CAM session and Teamcenter driven by an extended set of user commands within the NX CAM environment

With the use of NX Manager CAM, NC programming data is stored as manufacturing process data with automatic population of the operations and subactivities within a master NC process. The user selects the output to be created in the NX CAM session, such as postprocessed file and shop documentation options, and this data is automatically saved as attachments to the NC operations within Teamcenter Manufacturing.

Teamcenter provides full data management functionality including access control, revision management, workflow and integrated visualization. The data created by the NX CAM session – connected to Teamcenter via NX Manager CAM – is fully compatible with these extended data management tools.



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