DASSAULT SYSTEMES

ENOVIA

CONNECTOR FOR NX

OBJECTIVE

Connector for NX™ provides a collaborative, design data management solution when combined with **Collaborative Innovation.** It allows designers to access and share each other's designs from within the native computer-aided design (CAD) user interface.

OVERVIEW

Connector for NX allows designers to effortlessly access, manage, share, and store NX data without leaving their preferred environment. **Connector for NX** facilitates the process workflow, increases data integrity and improves configuration management. Performance is optimized for checkout and check-in operations through understanding design structure requirements and accessing only required data.

The combination of **Connector for NX** and **Collaborative Innovation** provides a powerful solution to maintain the integrity and availability of NX data in the **3D**EXPERIENCE® platform. Users can search and browse the **3D**EXPERIENCE platform from within the NX application. Performance is optimized for checkout and check-in operations through understanding design structure requirements and accessing only required data.

HIGHLIGHTS

Key features and capabilities include:

Product Structure Data Integrity

Connector for NX controls the relationships between NX entities and how they are presented in the **3D**EXPERIENCE platform so that they are available to the entire organization. It manages the relationships between NX entities in the **3D**EXPERIENCE platform to ensure the proper data is loaded upon checkout.

Data Access and Storage Rights

Users attempting edits to designs they do not have locked are presented with a warning message that the local file is read-only. The user may then attempt to lock the design in order to save it to the **3D**EXPERIENCE platform or continue editing to perform what-if scenarios. Additionally, if editing is continued, the user is presented with a final warning at the time of check-in to either attempt to lock the design or save the design to a new revision stream. If neither is selected or the user does not have the proper access, the design changes will not be stored into the **3D**EXPERIENCE platform.

Design Recognition

Connector for NX provides an auto-recognition capability that gives the user confidence that on check-in, the file will be associated with the same revision stream from which it was checked out. This is valuable when design is performed externally, such as suppliers developing supporting systems.

Product Driven Designs

Connector for NX can provide automatic synchronization of a CAD structure to an engineering bill-of-material (EBOM). This link is critical to the correct representation of the intended design. Users may at any point in the design's maturity automatically create a corresponding engineering part structure. In addition, during this EBOM synchronization process, users may also transfer associated drawing balloon numbers to the corresponding EBOM. The CAD design is available immediately to the engineering community as a specification to the engineering part.

Maintaining Graphical Design Representations

The graphical representations stored with the 3D model may be of many different formats. The "derived output" formats are stored when the design is saved and are controlled by the user. For example, PDF files may be generated for drawings, while IGES or STEP files for 3D models could be generated for broader access to the design data without the CAD application.

Quick Access to Data

Connector for NX has been designed to allow users to manage their CAD files with minimal effort. The user interface allows users to remain in the context of the CAD application while performing daily routine tasks that interface with the **3D**EXPERIENCE platform. During a save operation, the user is presented with only preselected new and modified designs, which eliminates the need to traverse the structure to locate the desired items. Added flexibility is provided to save the active design ("Save Active"), or save all loaded designs "Save All". In addition, a "Quick Save" command saves the active in-session design without any further display or user interaction. In order to match revision processes users may manually input the revision sequence for the stored designs.

Opening designs is also easy. Users may search the database or access workspace folders and stored collections directly from the "Open" dialog. In order to encourage users to frequently update the system with their latest design iterations, **Connector for NX** maintains a list of recently accessed designs that users can quickly reference from the "Open" dialog for future download. Additional options such as 'download direct children" can provide users with potential performance gains during the "Open" operation.

Product Structure Maintenance

Connector for NX maps and maintains relationships among assemblies, parts, and other application-specific items as users check-in and check-out items, or users browse vault contents. In addition, previous iterations are stored to permit baselines and rollbacks. This also allows for virtual structure management through direct access to previous revisions/iterations stored in the database. An important aspect of structure maintenance is the synchronization of required attributes between the CAD files and the metadata stored in the **3D**EXPERIENCE platform. Bi-directional updates are automatically initiated by **NX** during "Open" and "Save" operations ensuring no metadata loss and immediate access by users.

Designers often utilize family tables of designs in their structures. **Connector for NX** manages and tracks the associative relationships of these instance designs. To avoid any confusion when saving modified instances, the family object is not displayed, only the affected instances. Relationships to the generic instance and family table are maintained in the database.

Revision Control

Organizations can maintain multiple revision trees of designs without manually creating new subdirectories or changing filenames. When **Connector for NX** users create a new revision, it is automatically saved as a new business object. The sequential revision code is added to the item name and it is related to the previous release. There is no need to propagate name changes interactively in the CAD application across files that reference the revised item.

Key Benefits:

- Maintain accurate representations of the intended design in the **3D**EXPERIENCE platform
- Achieve centralized management of all CAD files
- Control work-in-process, engineering changes, data, documents, and dynamic configurations
- Give non-engineering personnel direct, task-specific access to current CAD data
- Reduce the possibility of redundant, inaccurate, or out-of-date product information
- Increase information sharing while protecting intellectual property from unauthorized access
- Achieve ISO compliance
- Improve design control and business process management to realize truly functional product development and delivery

Design Team Collaboration

Connector for NX provides the critical connection between the mechanical CAD process and effective product development, and benefits from the following collaboration features in **Collaborative Innovation**:

- CAD structure and EBOM synchronization validation
- Advanced CAD structure management in the **3D**EXPERIENCE platform
- Design data workspaces
- Quick access to most-recently used design data
- Notification of design modifications of interest
- Update in-session reference designs as needed
- Derived output of multiple formats by design type for downstream processing

To ensure that designers do not overwrite each other's work, it is possible to query for the status of locally referenced designs from the context of the CAD tool. This display shows the design's type, name, current revision, current version, latest version available, and the user that has the design locked. Right mouse button commands allow users to lock, unlock and display design properties. Once this window is activated, it may stay on the desktop while the user continues to work in the CAD application. When a user changes work designs, the PLM status window may be 'refreshed" to update the display with the current active design status.

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