

NX 1872 Series – Monthly Update

Welcome to NX

August 2019

Dear Customer:

We are proud to introduce the latest release of our product development solution. With this release, we continue to provide innovative ways to deliver solutions that meet the next generation of your product design, development, and manufacturing challenges. The new version of NX is robust and powerful, and it delivers advanced technologies for product design, development, and manufacturing in a single, multidisciplinary platform. It preserves best-in-class customer deployment readiness, and builds on the productivity and stability achievements of the previous release.

Sincerely, your NX Release Team

CONTENTS

NX 1880	3
1880 UPDATES	3
Reenabling Automatic Updates Button.....	3
1880 PROBLEM REPORTS.....	3
1880 ENHANCEMENTS	4
Automation Designer: Electrical BOM	4
Coordinate Measuring Machine: Inspection Supplement	7
NX 1876.....	11
1876 UPDATES	11
Reenabling Automatic Updates Button.....	11
1876 PROBLEM REPORTS.....	11
1876 ENHANCEMENTS	12
Line Designer: Plant Navigator	12
Discrete Drivetrain: Additional Motion Bodies.....	14
Sheet Metal: Component Placement Enhancement	15
Global Technical Access Center	16
Installation assistance	16

NX 1880

1880 UPDATES

NX Automatic Update

Automatic updates are available.

Caveats

Reenabling Automatic Updates Button

If the login dialog that pops up on initiating the download of an update is closed:

1. An error message “Failed to download Updates” is displayed.
2. The “Automatic Updates” button is greyed out preventing further attempts to download the update.

To resolve this, a restart of the NX session will reenables the button for user interaction.

1880 PROBLEM REPORTS

For a detailed list of PR fixes, see Fixed_Problem_Reports.csv included with the release documents.

Please see the table below for a summary of the PR updates included in this release.

Applications	Count of PR Number
ROUTING_MECH	1
ADD_FIXED_PLANE	2
DESIGN	2
TRANSLATOR	2
CMM_INSPECTION	3
KDA	3
SHIP_DESIGN	4
SYSTEM	4
CAE	10
CAM	15
Total	46

1880 ENHANCEMENTS

Please see below for a list of enhancements included in this release.

ERs

- ER7774710 – Request to create the scanning path while rotating table
- ER9447573 – Add support for analog scanning probe filters via the GEOALG statements to NX CMM
- ER9449040 – Enhance DMIS and GD&T logical check in NX CMM similar to Inspector DMIS engine

Automation Designer: Electrical BOM

Introduction

Bill of Material

Use the **Bill of Material** command to create an electrical Bill of Material (BOM). An electrical BOM contains only electrical products, such as switches, DI/DO modules, and interface modules.

Teamcenter Compatibility

You can only use this feature with Teamcenter 12.2.


Automation Designer stores electrical BOMs in Teamcenter. Therefore, you can exchange them directly with ERP systems and suppliers. Procurement departments often use BOMs for ordering system components or generating reports.

An electrical BOM provides an overall part summary of one or more electrical products. You can create either an electrical BOM for your entire project, or a partial electrical BOM for parts of your project. You can create multiple electrical BOMs out of your project or you can create new revisions of the already existing electrical BOM.


Before you Start

The electrical BOM generates empty lines in Teamcenter if the products are checked out during electrical BOM generation.


Before you start with the generation of the electrical BOM, please make sure that you successfully save the project. This will ensure that all objects are checked-in and the generation can start.

Application	Automation Designer
Command Finder	Bill of Material 





Create an Electrical BOM


1. Choose **Collaboration** tab→**Teamcenter** group→**Bill of Material** 
2. In the **Create BOM in Teamcenter** dialog box, from the list, select **Create New**.
3. In the **Name and Attributes** group, double-click the **Value** cell in the **ID** row.
4. (Optional) Change the name.
5. In the **Source** group, from the **Scope of Bill of Material** list, select either **Entire Project** or **Partial**.
If you select **Partial**, in the aspect navigators, select one or more parent objects to define the scope of the electrical BOM.
6. In the **Location** group, browse to the location where you want to store the BOM.
7. Click **OK**.

Create a New Revision of an Existing BOM

1. Choose **Collaboration** tab→**Teamcenter** group→**Bill of Material**  .
2. In the **Create BOM in Teamcenter** dialog box, from the list, select **Create New Revision of Existing**.
3. In the **Existing Bill of Material** group, select the BOM for which you want to create a new revision.
4. In the **Source** group, from the **Scope of Bill of Material** list, select either **Entire Project** or **Partial**.
If you select **Partial**, in the aspect navigators, select one or more parent objects to define the scope of the electrical BOM.
5. Click **OK**.

Create BOM in Teamcenter Dialog Box

Creation list	<p>Create New Lets you create a new BOM.</p> <p>Create New Revision of Existing Lets you create a new revision of an existing BOM.</p>
Existing Bill of Material	
Available if you set the Creation list to Create New Revision of Existing .	
Project	Displays the name of the open project.
Existing bill of material list	Lists the existing BOMs in the project.
Name and Attributes	
Lets you determine the name and other attributes of the BOM.	
 <p>Secondary Attributes</p>	Opens the Attributes dialog box, where you can view and modify attributes of the BOM.
Source	
Scope of Bill of Material	<p>Entire Project Lets you include the entire project in the scope.</p> <p>Partial Lets you include part of the project in the scope.</p>
Partial Bill of Material	
Available if you set the Scope of Bill of Material list to Partial .	
 <p>Select Parent Objects</p> <p>Parent objects list</p>	<p>Lets you select parent objects to include in the partial bill of material.</p> <p>Lists the parent objects in the partial bill of material.</p>
 <p>Add to the List</p>	Lets you add parent objects to the list.
 <p>Remove from the List</p>	Lets you remove parent objects from the list.

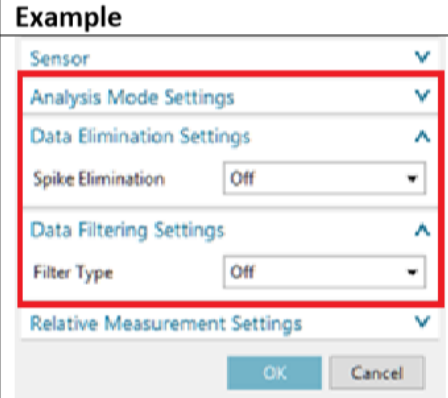
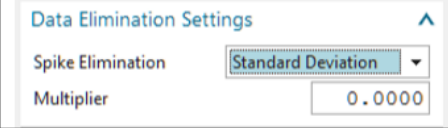
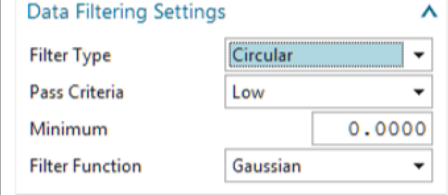
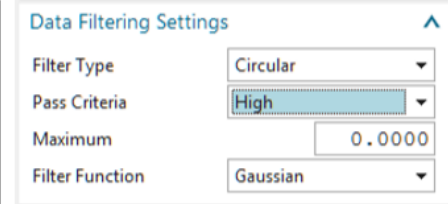
Location	
 Part File	Lets you choose a location to store the BOM. You can modify the location only if you are creating a new BOM, and not if you are creating a new revision.

Coordinate Measuring Machine: Inspection Supplement

Introduction

Data Elimination and Data Filtering

Data Elimination means the removal of measured data points, sometimes called ‘outliers’, that are outside a range of standard deviations from the mean deviation. Data Filtering means a high pass, low pass, or band pass filter is applied to the measured data in order to isolate form deviation from surface texture. You can enable these and set the parameters using new groups in the feature dialogs and the method dialog

Example	Meaning
	Data Elimination and Filtering are off.
	Data Elimination based on a standard deviation multiple.
	Low pass data filter
	High pass data filter

These settings are supported by the DMIS 5.2 post-processor using a statement of the following format:

```
GEOALG/CIRCLE, LSTSQR, STDDEV_LIMIT, 2., FILTER, CIRCULAR, HIGH, 15., GAUSS
```

Other post-processors do not currently support these settings.

Head Touch on Outer Circles and Cylinders

Simulation of inspection paths using the PH20 in Head Touch mode has been improved for the following feature types:

- CIRCLE/OUTER
- CYLNDR/OUTER
- GCURVE on the surface of a CYLNDR/OUTER

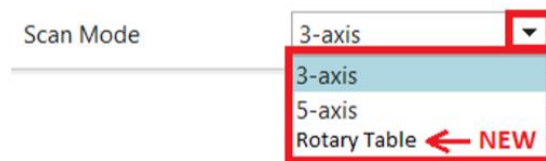
Verify Program

Inspection programs can now be verified before sending them to the shop floor for execution. Many common errors are detected and reported in the NX information window.

- Feature not measured or constructed
- Datum not defined before a tolerance that references it
- Size tolerance missing when using MMC or LMC modifiers

Arc Scan using Rotary Table

Arc Scans may now be performed by bringing the probe into contact with the part, then rotating the part with the rotary table while holding the probe at a stationary XYZ. You can specify this on the Scan Arc dialog using a new choice for Scan Mode:



This is supported by the DMIS 5.2 post-processor using a statement of the following format:

```
PAMEAS/P(CIRCLE1_PATH_Scan_Arc), ROTARY, RT(ROTARY1)
```

Other post-processors do not currently support this statement type.

DMIS Support – Support for Valisys Programs

Inspection programs written using Valisys (eM-ProbeCAD) may now be imported into NX CMM with good success. Less than 10% of operations will need to be edited by hand. Support for the following statement types was added in NX12.0.2 MP8:

- DATSET statement
- MACRO and CALL statements
- CONST/POINT,...,BF
- Convert 3 character datum letters VP1,VS1,VT1 to valid datum letters
- Option to ignore BND1,BND2,BND3,BND4,F(VBND and BOUND/
- Option to ignore Comments
- OUTPUT that isnt a T() or F() format
- Valisys GEOALG formats
- GSURF/GCURVE Features
- Width Tolerance
- OUTPUT/R()
- TEXT/OPER
- ROTAB/
- SAVE/FA

Link PMI Processing of Ordinate Dimensions

When processing ordinate dimensions, the Link PMI function now inserts operations into the CMM program to establish a Part Coordinate System (PCS) at the origin of the ordinate dimension. If several ordinate dimensions are based on the same origin, they will share the same PCS. The PCS column in the operation navigator will show the PCS assigned to the CORTOL tolerance.

Link PMI Processing of Hole Callouts

The Link PMI function now links PMI Hole Callouts into the appropriate set of CMM Inspection operations. The following hole types are supported:

- Cylindrical Through Hole
- Threaded Hole
- Counterbored Hole
- Countersunk Hole
- Blind hole (non through hole)
- Tapered Hole

Inspection features, tolerances, and paths are created to measure the following characteristics:

- Cylinder Diameter
- Counterbore Diameter
- Countersink Angle
- Counterbore Depth (Using plane features)
- Countersink Depth (Using constructed circle)
- Hole Depth (Using point features)
- Tapered Hole Cone Angle

Note: Hole depth is measured using point features in order to minimize inspection time.

Inspection Programming using Rotary Table

When programming a part involving a rotary table, the inspection part and rotary table will be displayed in the rotated position when viewing and editing inspection paths.

NX 1876

1876 UPDATES

NX Automatic Update

Automatic updates are available.

Caveats

Reenabling Automatic Updates Button

If the login dialog that pops up on initiating the download of an update is closed:

3. An error message “Failed to download Updates” is displayed.
4. The “Automatic Updates” button is greyed out preventing further attempts to download the update.

To resolve this, a restart of the NX session will reenble the button for user interaction.

1876 PROBLEM REPORTS

For a detailed list of PR fixes, see Fixed_Problem_Reports.csv included with the release documents.

Please see the table below for a summary of the PR updates included in this release.

Application	Count of PR Number
ASSEMBLIES	1
BETA_EAP	1
DESIGN	1
FLEX_PC_DESIGN	1
FLEXIBLE_PIPE	1
KDA	1
ROUTING_ELEC	1
SHIP_DESIGN	1
TC_FEATURES	1
TRANSLATOR	2
CAE	3
NXMANAGER	3
ROUTING_GENERAL	6
NX_SHEET_METAL	9
Total	32

1876 ENHANCEMENTS

Please see below for a list of enhancements included in this release.

ERs

- ER9071549 - FPC Design App - Attach bodies to flex and bring them along when flattening

Line Designer: Plant Navigator

Introduction

Creating End Items in the Plant Navigator

What is it?

In the Line Designer application, you can now create the end items for the components in the **Plant Navigator**. You can:

- Create end items in the **Plant Navigator**.
- Load the end items as a single component either as an item or as an occurrence, in the **Plant Navigator**. You can still see the geometry of all end items in the graphics window.
- Modify the end items for further design.
- Expand the end item structure using the **Expand End Items** option and collapse the end item structure using the **Create/Collapse End Item** option.
- Remove the end item state from the **Plant Navigator** using the **Remove End Item Assembly State** option.

Why should I use it?

You can now quickly navigate through the **Plant Navigator** tree structure. Also creating end items in a plant layout helps you reduce the number of BOM lines included in the structure, in the **Plant Navigator**, and you can create the desired structure of the plant layout.

Where do I find it?


Application	Line Designer
Plant Navigator	Right-click the component or work area object→ Create/Collapse End Item Expand End Items Remove End Item Assembly State

Working with end items in the Plant Navigator

When you load a plant layout with a large number of assemblies and components and want to navigate to an assembly, you need to scroll throughout the plant structure to that assembly, which takes more time. To simplify the layout structure and to focus on the assembly of interest, you can display these assemblies as a single Bill of Material (BOM) line using **Create/Collapse End Item** shortcut menu option.

Creating end items in a plant layout helps you reduce the number of BOM lines included in the structure in the **Plant Navigator**, and you can create the desired structure for the plant layout.

When you select the customer default **Load End Items Defined in Teamcenter**, you can easily navigate in the **Plant Navigator** tree structure of a plant layout using the **Create/Collapse End Item** shortcut menu option.

 = Load End Items Defined in Teamcenter [Note](#)

To find a customer default, choose **File** tab → **Utilities** → **Customer Defaults**, and

click **Find Default** .

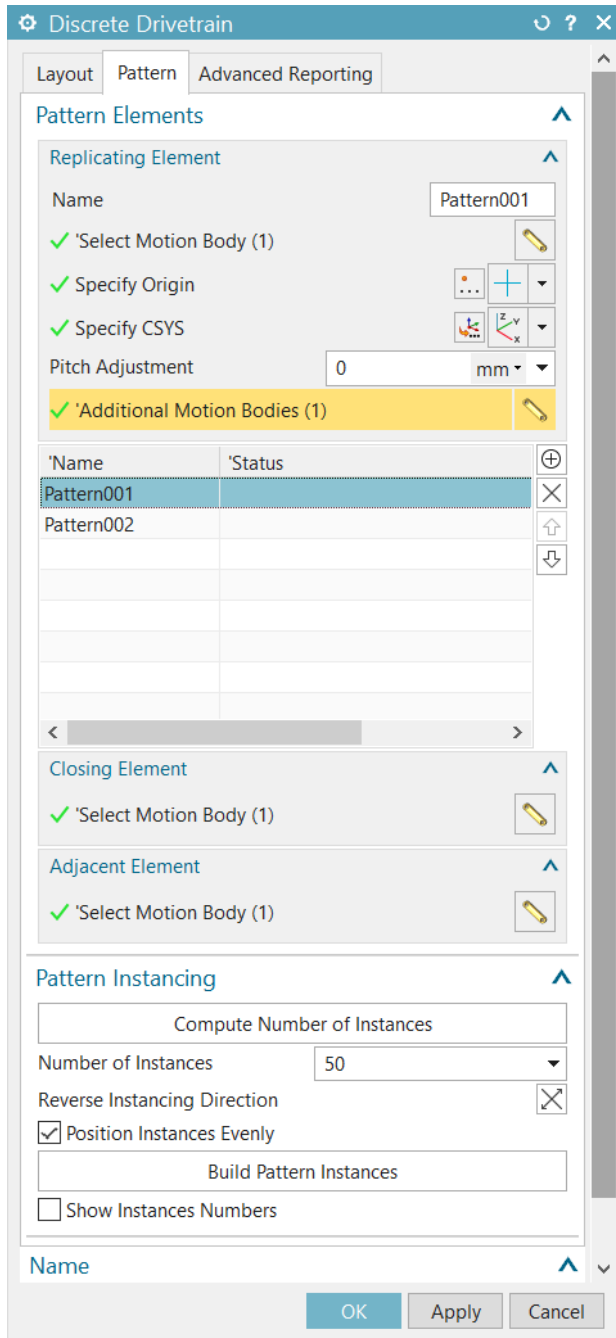
In Line Designer, when you load a plant layout, you can now:

- Create end items in the **Plant Navigator**. To avoid affecting instances of the component in other structures, you can use the **Occurrence** option to create end items.
- Load the end items as a single component either as an item or as an occurrence, in the **Plant Navigator**. You can still see the geometry of all the end items in the graphics window.
- Modify the end items for further design.
- Expand the end item structure using the **Expand End Items** option, and collapse the end item structure using the **Create/Collapse End Item** option.
- Remove the end item state from the **Plant Navigator** using the **Remove End Item Assembly State** option.

Discrete Drivetrain: Additional Motion Bodies

Introduction

Pattern Elements now support the definition of Additional Motion Bodies.



Additional Motion Bodies

The Motion Body selected in Select Motion Body is treated as the main Motion Body in this Pattern Element. Additional Motion Bodies in this list can be replicated and passively positioned as part of the Pattern Element along with the main Motion Body. These Additional Motion Bodies maintain their position and orientation relative to the main Motion Body in a pattern instance.

Additional Motion Bodies may be connected to the main Motion Body or to each other. Any Motion elements like joints, forces or constraints which connect them are also replicated when Build Pattern Instances is clicked.

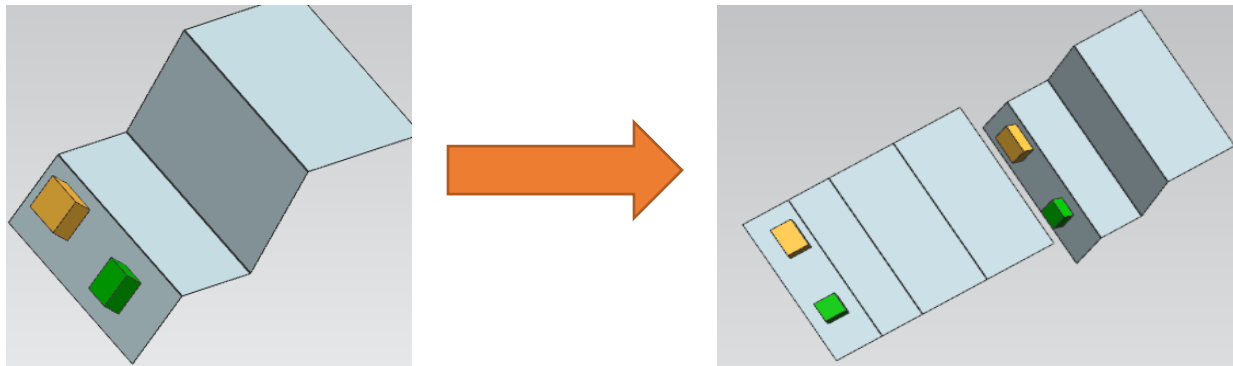
NOTE: Additional Motion Bodies may NOT be connected to a neighboring Pattern Element's main Motion Body or additional Motion Bodies. Such connections are not replicated when Build Pattern Instances is clicked.

Sheet Metal: Component Placement Enhancement

Introduction

Associate Object

When you create a Flat Solid in the Flexible Printed Circuit Design application with Transform Components option set to Body, the components associated with the faces of the formed body using the Associate Object command are copied, and these copies are transformed to the corresponding faces in the flattened body.



GLOBAL TECHNICAL ACCESS CENTER

Installation assistance

For additional installation assistance, or to report any problems, contact the Global Technical Access Center (GTAC).

Website:

<http://support.industrysoftware.automation.siemens.com/gtac.shtml>

Phone:

United States and Canada: 800-955-0000 or 714-952-5444

Outside the United States and Canada: Contact your local support office.