NX 2406 Release Notes

NX 2406 Series

SIEMENS

About Siemens Digital Industries Software

Siemens Digital Industries Software is a global leader in the growing field of product lifecycle management (PLM), manufacturing operations management (MOM), and electronic design automation (EDA) software, hardware, and services. Siemens works with more than 100,000 customers, leading the digitalization of their planning and manufacturing processes. At Siemens Digital Industries Software, we blur the boundaries between industry domains by integrating the virtual and physical, hardware and software, design and manufacturing worlds. With the rapid pace of innovation, digitalization is no longer tomorrow’s idea. We take what the future promises tomorrow and make it real for our customers today. Where today meets tomorrow. Our culture encourages creativity, welcomes fresh thinking and focuses on growth, so our people, our business, and our customers can achieve their full potential.

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1. Welcome to NX 2406

Dear Customer:

We are delighted to announce the latest release of NX software that features new advanced capabilities, including AI, cloud capabilities, and immersive technology into the realm of design and manufacturing.

Dive into a world where artificial intelligence isn't just a tool, but a partner in design. NX leads the charge in innovation, employing AI to refine accuracy, expedite decisions, and introduce smart design recommendations that drive your projects to unparalleled efficiency.

Embrace the flexibility and collaboration that cloud technology brings to NX with cloud-based tools. Your teams can now access designs, share insights, and collaborate across disciplines from anywhere in the world. The cloud ensures that your data is secure, scalable, and always available, empowering you to respond swiftly to changes on the go.

Step into the future with NX's immersive technology. Our virtual and augmented reality features allow you to interact with your designs in a fully-immersive environment. This technology not only enhances the design experience, but also provides a tangible understanding of how your products will perform in the real world — all seen to scale.

NX's continuous release cycle continues to push the boundaries of product lifecycle management. Each update not only introduces cutting-edge security measures to provide maximum protection for your assets, but also ensures that NX remains scalable and flexible to meet your growing needs.

To further support your expansion, NX offers a range of add-on modules with trial options accessible via the NX Discovery Center, all purposefully created to integrate with your evolving design and manufacturing needs.

Join us in harnessing the power of AI, cloud, and immersive technology with NX.

Design

This release is a testament to our dedication to our users, converting valuable feedback into substantial improvements that enhance your NX experience. Here are some of the highlights:

AI-Enhanced Design

- **Performance Prediction**: NX's AI-driven tools offer swift analysis report generation with 3D visualizations, fully integrated with Teamcenter for enhanced collaboration.

Advanced Modeling

- **Lattice Structure enhancements**: The Lattice command now includes Voronoi enhancements and new methods for filtering and connecting rods.

- **Topology Optimization**: The environment boasts new milling support, automatic blending, and improved monitoring controls for streamlined topology optimization.

NX Integration with Building Information Modeling

- **BIM Design**: Enhanced navigation and property editing in BIM Design Navigator streamline the creation of stairs and guard rails in design assemblies.

Computer-Aided Manufacturing (CAM)

Key upgrades have been introduced to the NX for Manufacturing applications, including NX CAM and NX CAM On-Machine Probing.

3D Adaptive Roughing, the high-speed machining strategy in NX CAM, is now even more flexible, allowing programmers to determine the tool path start location to predrilled locations for safe tool engagement at high speeds.

The **Holemaking** operation, a favorite among many part manufacturers, now offers improved control of non-cutting moves (both within and between features), more robust local gouge checking, and quicker tool path calculation time.

The **Cloud Connect Tool Manager** is enhanced for more efficient management of machining data, easier tool holder creation, and direct access to several tool vendor catalogs from the application.

NX CAM On-Machine Probing now has advanced capabilities that allow programmers to configure and capture parameters for streamlined process optimization and reuse of best practices.

Additive Manufacturing

In NX 2406, the Siemens commitment to driving additive manufacturing forward continues. Here are some of the enhancements that enrich additive manufacturing and will save you time.

Fixed Plane Additive

NX has added a sub-nester to group and nest small fragile parts as a pre-nester before you create sinterboxes, brush facet selection to more easily define support regions on face-less convergent bodies, and multi-laser path preview enhancements to the Build Processor for EOS machines.

Multi-Axis Additive

NX now allows you to use multiple offset beads in planar additive thin-wall operations, which reduces the need to model auxiliary manufacturing geometry and new build styles and build rules. In addition, NX gives you a way to capture and control machine and material specific process parameters, known as the "secret sauce," for portions of a multi-axis additive operation.

Our additive manufacturing solutions continue to evolve and grow as we strive to meet the requirements of our customers for their use of this exciting new technology.

2. NX System Information

Customer support

Customers covered by valid maintenance agreements are eligible to receive customer support. If you are a customer of one of our partners or resellers, please contact them first to determine your first line support provider.

Our Customer Support team has the following mission:

- Provide solutions that ensure CUSTOMER SUCCESS with our products.
- Deliver an experience that customers VALUE and TRUST.
- Be there for the customer EVERY STEP OF THE WAY.
- CONTINUOUSLY IMPROVE how we serve our customers.

You can find your local Customer Support **contact information** and **open a support case** on **Support Center**.

Platforms

System requirements guidelines

Defining the minimum system requirements is difficult because key requirements, most notably memory, can vary dramatically from user to user. The following are general guidelines that you should consider before purchasing a system.

Processor performance

Although raw processor speed has a major impact on system performance, other factors also contribute to overall performance; for example, the type of disk drive (SSD, SCSI, ATA, or Serial ATA), disk speed, memory speed, graphics adapter, and bus speeds. The general rule is that "the faster the processor, the better the performance is," but this only applies when comparing like architectures. For example, it is difficult to arrive at performance expectations for an Intel processor when compared to an AMD processor just by looking at their respective processor speeds. There is also a general trend today to de-emphasize processor speeds and move to multi-core processors, which actually can have lower processor speeds.

NX takes advantage of modern processors, and if a processor does not have the required functionality, you will see the following error message and NX will not run on the installed CPU:

```
This processor is too old and not supported anymore
```

SMP

Symmetric Multiprocessing (SMP) is supported in NX mostly via Parasolid, although a small number of NX capabilities have some threading. In general, it is not possible to quote a figure for the general performance improvement achieved by using SMP, since this improvement depends on the nature of the operations you are performing. You need to evaluate your actual performance gains using your own models. Functional areas that are SMP enabled in Parasolid include:

- Validity checking
- Boolean operations
- Wireframe
- Rendering
- Hidden line rendering
- Closest approach
- Faceting
- Mass properties

SMP is enabled by default with the variable **UGII_SMP_ENABLE**, which is located in the *ugii_env_ug.dat* file.

Multi-Core

Multi-core processors are similar to SMP because there are two or more actual processor cores but they are delivered in single processor packages. Siemens Digital Industries Software has found that multi-core performance characteristics are similar to SMP. The one advantage of multi-core processors over SMP is that this technology has proliferated much faster than SMP and is now common in workstations, servers, and laptops.

Multi-core technology is complex and, depending on the configuration, can actually have a negative impact on performance. This is due to the potential conflict of multiple cores sharing system resources, such as cache, memory, and bus bandwidth, as well the need for the system to manage and control an increasing number of cores. Increasing the number of cores does **not** always translate into better performance. Although additional cores can improve NX performance, processor speed is still a vital measurement of NX performance.

Many systems enable you to turn off cores via the bios, which can enable you to compare performance with a different number of cores that are active. Some users may find that turning off some cores will actually improve performance. One micro-architecture (Intel) even does this automatically, shutting down unused cores and increasing the clock speed of the others.

The hardware vendors continue to improve their processor micro-architectures to better address the limitations of older multi-core technologies. New subsystems better integrate memory and other peripherals directly to the processors, resulting in major performance improvements. Buses are being eliminated, cores are better managed, and channel speeds continue to improve.

In summary:

- Turn SMP on only if you have an SMP system. Having it on in a single-processor system incurs a slight overhead.
- Turn SMP on if you have a multi-core system.
- Never assume that by simply adding more cores you will see better performance. Always test first.

Memory

The minimum recommended amount of memory to run native NX is 8GB. If you are running NX with Teamcenter (Teamcenter Integration for NX), the minimum recommended is 12 GB. However, because NX is capable of handling large assemblies and very complex parts, many of our customers use workstations with 32GB of RAM, and some even use 64GB, 96GB, or more.

For the optimum user experience and application performance, we recommend that you install as much RAM as can be installed in the client workstation that is running NX.

As a guideline, so that you get the best possible performance when using NX, ensure that the amount of physical memory (RAM) in your workstations is always larger than the amount of memory consumed by NX, plus all the other applications that are running at the same time on the same machine. This can be done by using a tool such as **Windows Task Manager** or **Resource Monitor**.

The minimum memory requirements varies and depends on various factors including:

- Complexity of the geometry within individual parts
- Size of the assemblies that are loaded
  - Less memory is required if the assemblies contain multiple instances of the same components.
  - More memory is required when large assemblies contain lots of unique components with complex geometry.
- Use of Teamcenter in addition to NX
  - Some additional memory is required when using Teamcenter 4-tier.
  - A even greater amount of memory is required when using Teamcenter 2-tier.

- Fully loading exact assemblies versus using lightweight assemblies and partial loading
- Generating CAM toolpaths for very complex geometry, such as an automotive engine block or gearbox
- Updating large assembly drawings

Graphics adapters

All the NX certified systems contain graphics adapters that meet all Siemens Digital Industries Software requirements and are fully supported by our hardware partners. The graphics adapters supported are carefully selected by working with our OEM partners as well as our graphics vendor partners. We do not recommend low-end, consumer, or game cards, since these graphics devices are developed for the DirectX market and are not well supported under OpenGL. Because a majority of platform/hardware problems are graphics related, it is critical that all the graphics adapters that NX supports are designed for OpenGL and have the highest level of support from our hardware vendors. We highly recommend that you only use supported graphics adapters and Siemens Digital Industries Software certified drivers.

For the latest information on certified graphics cards and driver versions, see **Supported hardware and graphics**.

Multiple monitors

Siemens Digital Industries Software supports multi-monitors but with limitations. These limitations were necessary due to the large number of possible configurations. Other combinations may work, but these conditions are tested and supported by Siemens Digital Industries Software. These guidelines could be extended or relaxed in the future.

The following is a summary of findings for the support of multiple monitors.

- NX 6.0.1 or higher - no older releases are supported.
- Two monitors only.
- LCD monitors only
- Run with native resolution and aspect ratio.
- Laptops are tested without docking stations or port replicators (direct connection only).
- Horizontal Scan mode (not Vertical) and only with identical monitors.
- Dual View (Nvidia) or Extended View (ATI) modes, but the user must have the display window entirely in either the primary or secondary monitor.

You do not have to comply with the configurations mentioned above, but Siemens must be able to duplicate the problem on the configurations in our labs before being able to investigate your issues.

Supported hardware and graphics

The list of currently supported hardware and graphics cards can be found on **Hardware and Software Certifications**. Click **NX-Graphics-Certification-Table** to open a spreadsheet that has tabs for supported systems and graphics.

Accelerated graphics requirement

NX has always required accelerated graphics, which makes a graphical processing unit (GPU) essential when running NX.

Previously, if a GPU was not present, the session would still run and depending on the commands that are used may even appear to have been a successful session. In those cases, a message would appear in the **syslog** file about the lack of a GPU being present, but the user often did not see this information, so it may have appeared that the GPU was only *recommended*, instead of *required*.

Advancements in the graphics libraries used in NX now require the use of a GPU.

Starting in NX 2306, if a GPU is not present NX displays the following message to the user, which is also written to the **syslog** file.

```
*****************************************************************
              Graphics Configuration Error
This is an unaccelerated graphics configuration. NX requires
accelerated graphics configurations. Perhaps the graphics device
is unsupported or perhaps an unsuitable device driver is installed.
Attempt to open any part may result in program termination.
*****************************************************************
```

If you ignore this message and continue to run the NX session, it could lead to undefined behavior or error messages.

NX central runtime directory

Central runtime directory concept

To help you find executables and libraries, and distinguish them from configuration files more easily, a central runtime directory that contains DLLs (or so's, or dylib's), executables, and JAR files is now used. The central runtime directory will be updated as NX supports more and different applications.

Scripts that are used to wrap executables have not been moved and they are still in their original kit locations with the configuration files.

NXBIN directory

The central runtime directory, NXBIN, is created on all platforms at the **UGII_BASE_DIR\nxbin** location during the NX installation. Depending on what was selected to be installed, libraries and executables from different kits are consolidated into the central runtime directory during the install process.

The new NX runtime directory does not contain all binaries and executables. Only the most frequently used files from the following directories have been moved:

- UGII
- UGMANAGER
- STEP203UG
- STEP214UG
- PVTRANS
- MACH

Other files will also be moved over time. However, some applications, such as NX Nastran, will not be moved.

While NXBIN does not contain all of the executables and DLL's, it does contain a large quantity and is a prime location to search for an executable or library.

UGII_ROOT_DIR obsoleted

UGII_ROOT_DIR has historically been used to find the following in the UGII directory:

- Libraries and executables
- Configuration files

With the creation of the central runtime directory, there are two different locations for these files (**nxbin** and **ugii**), so the concept of UGII_ROOT_DIR is obsoleted. Use the UGII_BASE_DIR variable instead when you write custom application code and scripts.

While the NX install no longer sets UGII_ROOT_DIR, the definition on the system will remain to support earlier releases. If your code or scripts rely on UGII_ROOT_DIR, modify them to ensure they still work in this release.

Teamcenter code and scripts

To provide backward compatibility with released versions of Teamcenter, all executables called directly by Teamcenter code or scripts are wrapped.

The wrapper executables just correct the settings of the UGII_BASE_DIR, PATH, and LD_LIBRARY_PATH variables that are set by the Teamcenter scripts and code, and then start up the real executable from the NXBIN directory. This enables support for the new runtime concept, while allowing NX to run with existing versions of Teamcenter.

These new wrapper executables must remain in the **ugii** and **ugmanager** directories since that is where Teamcenter code expects to find them. Two processes are shown for executables invoked from the **ugii** and **ugmanager** directories.

Executables are still in the UGII, UGMANAGER, STEP203UG, STEP214UG, and PVTRANS kits.

Teamcenter code migration to the new central runtime directory structure is scheduled in a future Teamcenter release.

NX Open programs on Linux

If you try to link an external NX Open C++ program using the **ufmenu** and **uflink** scripts with NX 2406 on SLES 15 SP4, Linux may fail with an error message such as:

```
/usr/x86_64-suse-linux/bin/ld: cannot find -lelf
```

For NX to properly build an NX Open program, you must have the correct RPM kits installed. You can use Yellowdog Updater Modified (YUM), or dandified YUM (DNF) as your package installer.

For NX 2406 and SUSE 15 SP4, the following packages are required:

```
elfutils-0.158-6.1.x86_64.rpm
libelf1-0.158-6.1.x86_64.rpm
libelf-devel-0.158-6.1.x86_64.rpm
```

For NX 2406 and RedHat 8.9, the following packages including their dependencies are required:

```
gcc-toolset-11-gcc-11.2.1-9.2.el8_6.x86_64
gcc-toolset-11-gcc-c++-11.2.1-9.2.el8_6.x86_64
gcc-toolset-11-elfutils-0.185-5.el8.x86_64
gcc-toolset-11-elfutils-devel-0.185-5.el8.x86_64
motif-2.3.4-19.el8.x86_64
motif-devel-2.3.4-19.el8.x86_64
ksh.x86_64
libnsl-2.28-236.el8.7.x86_64
redhat-lsb.x86_64
```

> Note:
>
> For SUSE, the *devel* packages are available from the optional SLE 15 SP4 SDK DVD/ISO.

Initializing the JVM

In some cases, NX is not able to create the Java Virtual Machine (JVM) properly on Windows. The root cause in these scenarios is insufficient memory to start the JVM. In most cases of insufficient memory Java is able to report back an error code indicating this problem. However, in some cases Java reports a generic error message that NX then displays. The typical error message is:

```
Can’t initialize the Java Virtual Machine (JVM)
```

When running a Java application, such as the Wave Browser or Interactive Class Editor, NX may give an unexpected error due to this problem.

If NX detects that there is not enough memory available for the JVM, an error message is given and information is provided in the syslog. The following is an example of the syslog information:

```
The JVM could not be created due to not enough memory.
The Java heap size must be contiguous and the largest contiguous block
available is outputted below.
Windows largest block free
================================
 Maximum block 267Mb
================================
Please note, this number is to be used as suggestion for setting the
heap size. It
is unlikely that you will be able to utilize the full amount.
If you need a heap size larger than what is possible you can try to use
the /3GB
switch or its equivalent, if available for the Operating System you are
on.
Otherwise your other option is to use Remoting. Please consult the NX
Open
Programmer's Guide for more information on this topic.
```

Reset the size of the Java heap

To remedy this problem, you can reset the size of the Java heap that NX uses. Choose **File→Execute→Override Java Parameters** to open the dialog box and set UGII_JVM_OPTIONS to the size of the Java heap. You can experiment with the size of heap that you need, but if the JVM is already started then you cannot change the UGII_JVM_OPTIONS setting.

It is recommended that you use both the **–Xmx** and **–Xms** options together. Both of these are needed as Java may determine default values for the heap size that are not possible with the machine’s current memory load. For example:

```
UGII_JVM_OPTIONS=–Xmx50M –Xms50M
```

When trying to determine the size of the heap, keep in mind that if the heap size is too small, a Java program you are trying to run may not run. This could be due to the amount of memory available on the machine, or due to multiple Java processes running. This can be the case with the Wave Browser where there is a client and server process.

Once you find a value that works, you can modify the UGII_JVM_OPTIONS value in the **ugii_env.dat** file so you don't have to reset it in the **NX Open Java Parameters** dialog box each time you start an NX session.

NX variables in the ugii_env.dat file

Standard and modified environment variables

The *ugii_env_ug.dat* file contains the standard environment variables for NX. You can override these variables with the **ugii_env.dat** file. This file is essentially empty when delivered. You can define new values for the environment variables or define new ones in the **ugii_env.dat** file and they take precedence over the values defined in the **ugii_env_ug.dat** file.

You can make all, some, or none of the changes to the variables in the **ugii_env_ug.dat** file. However, it is recommended that you use the **ugii_env.dat** file to define all of the values in use at your site.

Both the **ugii_env_ug.dat** and **ugii_env.dat** files are located at `<UGII_BASE_DIR>\ugii`.

Other variables and the ugii_env.dat file

There are other variables in addition to the variables defined in the **ugii_env.dat** file which are evaluated by NX. For the same variable:

- User variables override system variables.
- User or system variables override variables in the **ugii_env_ug.dat** and **ugii_env.dat** files.
- Variables in the **ugii_env.dat** file override variables in the **ugii_env_ug.dat** file.

Using the ugii_env.dat file

The following apply when using the **ugii_env.dat** file:

- Define the variables before the #include statement.
- The first variable defined is used. If you have the variable defined twice in the file, only the first one is used.

  > Note:
  >
  > This also applies to the **ugii_env_ug.dat** file.

- You cannot have spaces in the variable names.

Designate a single ugii_env.dat file for all users

You can place the **ugii_env.dat** file in a central location for all users to access.

For each user, set the environment variable **UGII_ENV_FILE** to the location of the file. For example:

```
UGII_ENV_FILE=G:\common\version_env_vars.corp_standards
```

> Note:
>
> The file can have any name but must be a fully qualified path.

Statements you can use in the ugii_env.dat file

The statements that you can use in the **ugii_env.dat** file are shown below with examples:

- `#if/#else/#endif`

  Checks for specific conditions and then defines variables based on those conditions. Conditions that can be checked are:

  - FILE - Check for the existence of a file

    ```
    #if FILE ${UGII_BASE_DIR}\UGII\html_files\start_${UGII_LANG}.html
    UGII_CAST_HOME=${UGII_BASE_DIR}\UGII\html_files\start_$
    {UGII_LANG}.html
    #else
    UGII_CAST_HOME=${UGII_BASE_DIR}\UGII\html_files\start_english.html
    #endif
    ```

  - platform - Check for a specific platform. Possible values:

    x64wnt

    lnx64

    macosx

    ```
    #if lnx64
    UGII_CAM_THREAD_MILL=${UGII_CAM_THREAD_MILL_DIR}thrm.so
    #endif
    ```

  - $variable = “value” - Check for a specific value for a previously defined environment variable

    ```
    #if $UGII_LANG = "simpl_chinese"
    UGII_COUNTRY=prc
    UGII_COUNTRY_TEMPLATES=$UGII_BASE_DIR\localization\
    $UGII_COUNTRY\simpl_chinese
    #endif
    ```

  - $variable != “value” - Check for a previously defined environment variable that does not have the specified value.

```
#if ${UGII_PACKAGE_DIRECTORY} != ""
#if FILE $UGII_PACKAGE_DIRECTORY\ugii_package_env.dat
#include $UGII_PACKAGE_DIRECTORY\ugii_package_env.dat
#endif
#endif
```

- `#include`

  Includes a specified environment file at the current location

  ```
  #include $UGII_PACKAGE_DIRECTORY\ugii_package_env.dat
  ```

- `export`

  Exports a variable so that downstream processes spawned by NX can also use it.

  ```
  export SPLM_LICENSE_SERVER=28000@cinxflex
  ```

  If the variable is already defined in one of the **.dat** files, such as:

  ```
  UGII_NX_NASTRAN="${UGII_BASE_DIR}\nxnastran\bin\nastran.exe"
  ```

  you can add the statement to a **.dat** file you control so that it can be exported:

  ```
  export UGII_NX_NASTRAN
  ```

Operating system

Operating system requirements

This section documents operating system and service pack requirements.

Minimum Operating Systems

The following operating systems are the minimum recommended for NX. Newer versions and service packs are available as they are released.

O.S.	Version
Microsoft Windows (64-bit)	Microsoft Windows 10 Pro and Enterprise editions
Linux (64-bit)	SuSE Linux Enterprise Server/Desktop 15 SP 4 Red Hat Enterprise Linux Server/Desktop 8.9

Linux

Starting in NX 1847, NX is no longer supported on the Linux operating system for interactive NX (NX running with UI), which includes graphics integration. It is only supported for running solvers and NX Open batch programs with no user interface calls.

To run NX Open programs or binaries provided by Siemens, you will be required to purchase a license to run batch programs on Linux. Contact Customer Support for the new NX Linux product.

Batch NX programs run on Linux are dependent on the availability of Motif in the installation.

If you are using the SCL compiler on Red Hat Enterprise Linux, you need to do some additional setup. See **Programming Tools Product Notes** for more information.

Mac OS X

Starting in NX 1847, the Mac OS X operating system is no longer supported.

Windows 10

Windows 10 is the minimum supported release for NX 2406. The supported versions of Windows 10 are the Pro and Enterprise editions utilizing Semi-Annual Channel (SAC) updates.

Windows 10 is also supported for NX 10.0.3 and later versions up to NX 2406. Any caveats or special instructions are provided in the SFB announcing support.

Windows 11

Windows 11 is supported for NX 2007 and later versions up to NX 2406. The supported editions are Pro and Enterprise utilizing Semi-Annual Channel (SAC) updates. Any caveats or special instructions are provided in the SFB announcing support.

Linux Distributions

NX is developed using Linux standards and accepted Linux development conventions. Quality assurance testing is performed on the two versions of Linux listed in the table above. Siemens PLM Software cannot guarantee operation, performance, compatibility, or support on any other distributions.

The following is a list of information about the development environment (for reference only):

Kernel	kernel 4.18.0-513.9.1.el8_9.x86_64 glibc-devel-2.28-211
C, C++ Compiler	gcc 11.2.1, g++ 11.2.1
Java Development version	Eclipse Adoptium (JDK) Temurin Development Kit with Hotspot 11.0.14.1+9 (x64)

Java

Java Runtime Environment

NX is certified to use Java Runtime Environment (JRE) 11.

While many Java releases are available from several providers, we recommend that you use only the Long Term Support (LTS) releases, such as JRE 11, 14, and 17. Any version of Java should work as long as it is equal to or higher than the certified version. The Java version can be obtained from any Java provider.

For additional Java information, contact Customer Support.

> Note:
>
> If you are running a Siemens product other than NX, you may need to run a different version of Java according to the requirements for that product.

Java is used for the following NX applications and products:

- NX Relations Browser
- Manufacturing – Process Studio Author
- Command line version of the following translators:
  - CATIA V4
  - CATIA V5
  - Dxfdwg
  - IGES
  - NX Pro E
  - Step AP203
  - Step AP214
- Quality Dashboard
- Validation Rule Editor
- Simcenter 3D Batch Mesher

- Simcenter 3D Response Analysis Function Tools
- Simcenter 3D Report Writer
- Siemens NX Launcher
- Customer written NX Open Java programs

Java requirements for NX Open

NX Open for Java **.jar** files are created with Eclipse Adoptium (JDK) Temurin Development Kit with Hotspot 11.0.14.1+9 (x64). You can use Java 11 or later for your NX Open programs. The Java version can be obtained from any Java provider.

Linux

NX supports SuSE Linux and Red Hat Linux, both 64-bit only. The minimum supported version of SuSE Enterprise (Desktop/Server) is 15 SP4 and Red Hat Enterprise (Desktop/Server) is 8.9. Newer versions will be supported via certification.

See **Supported hardware and graphics** for details about supported hardware configurations as well as for the latest graphics drivers.

Installing the .NET framework

The NXOpen for .NET API is a Windows-specific project that leverages the Microsoft .NET Framework component. Before you can execute a custom .NET application, you must install .NET Framework 4.8. In addition, to replay a .NET journal, the .NET Framework 4.8 must be installed.

To download the .NET Framework, use the links on this page: **Microsoft .NET Framework 4.8 installer**.

Visual Studio runtime redistributable

To run NX 2406, the MS Visual Studio 2019 runtime redistributable is required. In addition, the MS Visual Studio 2013 runtime redistributable is required to properly setup some system **dll** files for some of the older executables.

You can download the Visual Studio 2019 redistributable **here**.

You can download the Visual Studio 2013 redistributable **here**.

> Note:
>
> If you have MS Visual Studio 2019 and 2013 installed, you don't need to download and install the redistributable.

Configuration files

Starting in NX 12, the NX configuration files on Windows are written to `C:\users\<name>\AppData\Local\Siemens`.

Browser requirements

Browser requirements

The NX suite of documentation (Help, What's new Guide, and Release Notes) is provided in an HTML format that is displayed in your local Web browser.

There are three ways to access the NX documentation:

- Access directly on the internet from **Support Center** (requires login).
- Install the **Secure Documentation Proxy**, which allows access to documentation on Support Center without logging in or creating a Webkey account. This eliminates the need and overhead of installing documentation locally while still providing secure access. Download the documentation proxy **here**.
- Install the Siemens Digital Industries Software Documentation Server, which sets up a web server on each workstation or on your company intranet. For requirements, see **Siemens Documentation Server for Windows** for Windows. Download the documentation server **here**.

The latest versions of web browsers are recommended to comply with the latest browser security updates.

Windows browser support

- Google Chrome
- Microsoft Edge
- Firefox

Downloading browsers

These browsers are free and can be downloaded from the following Web sites:

Chrome — **http://www.google.com/chrome**

Edge — **http://www.microsoft.com**

Firefox — **http://www.mozilla.org**

Licensing Caveats

General licensing caveats

License server upgrade

NX 2406 requires Siemens License Server version 2.1 or later.

Using SPLM_LICENSE_SERVER

Some Siemens products use the new **SALT_LICENSE_SERVER** license server environment variable, which is referred to in the licensing installation guide. NX 2406 continues to use the **SPLM_LICENSE_SERVER** environment variable instead of the new variable.

License conversion

Starting in NX 2212, the license you receive from Siemens must be installed using the Siemens License Server installer. For details, see the *Updating the License File* section in the license server installation guide.

During installation, the Siemens License Server installer defaults to port number 29000, which was 28000 in previous releases. You can change the port number while running the installer. The port number value is used in the value for the **SPLM_LICENSE_SERVER** environment variable.

License files and license server

The following are applicable to license files and the license server:

- NX requires and tests for the latest version of the **ugslmd** vendor daemon. This vendor daemon is supplied with NX and must be installed and initiated prior to starting NX. If an older daemon is utilized, a generic NX License Error dialog box is displayed during start-up. In addition, the syslog will contain an error message that the daemon version is older than the client.

- Merging of license files is not supported. For example, you can not merge a pre-TC 2007 MP3 or pre-NX 5 license file, which uses the **uglmd** license daemon, with a TC 2007 MP3, NX 5, or NX 6 license file, which uses the **ugslmd** daemon.

NX Borrowing is version specific

NX license borrowing is version specific due to dependencies within the third party licensing software used by Siemens PLM Software. To ensure that NX works on a borrowed license, you should always utilize the borrow tool supplied with that NX version. For example, a license borrowed using the NX 2406 borrowing tool will work for NX 2406 but cannot be used to run NX 2312. In addition, you cannot borrow licenses for two NX versions simultaneously on the same workstation.

Licensing caveats for Windows

The following caveats are applicable to Windows platforms only.

License Option tool

The **License Option** tool should only be used to borrow NX licenses, even though it may display other licenses. The **License Option** tool does not filter features in the license file that are of an earlier version than NX, such as Teamcenter lifecycle visualization, so these licenses are displayed in the tool. If you try to borrow a license for an application other than NX, you get an error when you launch the non-NX application.

LMTOOLS fails to restart service

You may encounter a problem where the license service fails to start after using **LMTOOLS** to set locations for the license file or server debug log file, or for both.

Cause: The license service is run as as **user=LocalService**, and **LMTOOLS** may have insufficient modification rights to do the following:

- Read the license file.
- Create and append to the server debug log.

Solution: Do one of the following:

- Use the default locations that the Siemens License Server Installer uses:
  - For the license file: C:\ProgramData\Siemens\License Server\<vendor>.lic
  - For the server debug log: C:\ProgramData\Siemens\License Server\<vendor>.log
- Specify a directory location that **user=LocalService** can modify.

  To determine whether a directory has LocalService permissions, go to the directory location in Windows **File Explorer**, right-click the folder, choose **Properties**, and click the **Security** tab. Search the **Group or user names** list for **LOCAL SERVICE**.

Licensing caveats for Linux

Additional software to support licensing

SuSE and Red Hat require the following to be installed:

- LSB 4.0

Product compatibility - supported release combinations

Teamcenter and NX

The following table lists the version combinations of Teamcenter, Active Workspace, and NX that are supported. The following conventions are used:

- TC is Teamcenter
- AW is Active Workspace
- The Teamcenter release that is compatible with NX is identified in the table. If no patch is specified in the table cell, then the base release is the minimum required.

  If a patch is required, then the Teamcenter patch level specified in a table cell is the minimum required for that Teamcenter release. Later patch levels in that release are also applicable, unless otherwise specified. For example, *TC 11.4.0_patch_5* is the minimum patch level in the Teamcenter 11.4.0 release, and also includes *patch_6*, *patch_7* and so on, that is, *TC 11.4.0_patch_5 or later*.
- The NX release identifies the minimum update in that series, where applicable. For example, the NX 1953 series specifies the NX 1961 update as the minimum.
- To connect to any supported Teamcenter release that is configured for use with single sign on (SSO), the NX 2007, NX 2206, NX 2212, and NX 2306 series requires that the Teamcenter security services session agent from Teamcenter 13.3 or later is installed.

> Note:
>
> Starting in Teamcenter 2312:
>
> - The Teamcenter release numbering follows the same numbering convention as NX releases, as in YYMM (year and month).
> - Active Workspace is an integral part of Teamcenter and not an independent application. The table specifies compatible NX and Teamcenter combinations without specific reference to Active Workspace. It is a deployment decision whether to include Active Workspace as part of your installation based on the requirements at your site.

Teamcenter and NX X

The following is applicable for Teamcenter and NX X:

- NX X includes built-in data management.
- NX X is compatible with Teamcenter X.

- NX X is compatible with Teamcenter installed at your site according to the release compatibility in the table below.

Compatibility table

	NX 12.0.2	*NX 1847 Series*	*NX 1872 Series*	*NX 1899 Series*	*NX 1926 Series*	*NX 1953 Series*	*NX 1980 Series*	*NX 2007 Series*	*NX 2206 Series*	*NX 2212 Series*	*NX 2306 Series*	*NX 2312 Series*	*NX 2406 Series*
Teamcenter UA *10.1*	☑ TC 10.1.7 AW 3.3.1 AW 3.4												
Teamcenter UA *11.4*	☑ TC 11.4.0_patch_5 AW 3.4 AW 4.0.4 AW 4.1 AW 4.2	☑ TC 11.4.0_patch_5 AW 4.0.4 AW 4.1 AW 4.2	☑ TC 11.4.0_patch_15 AW 4.0.4 AW 4.1 AW 4.2										
Teamcenter UA *11.5*	☑ AW 4.0.4 AW 4.1 AW 4.2	☑ AW 4.0.4 AW 4.1 AW 4.2	☑ TC 11.5.0_patch_10 AW 4.0.4 AW 4.1 AW 4.2 AW 4.3.3	☑ TC 11.5.0_patch_10 AW 4.0.4 AW 4.1 AW 4.2 AW 4.3.3	☑ TC 11.5.0_patch_10 AW 4.1 AW 4.2 AW 4.3.3	☑ TC 11.5.0_patch_10 AW 4.1 AW 4.2 AW 4.3.3							
Teamcente	☑ NX 12.0.	☑ NX 1851	☑ TC 11.6.	☑ TC 11.6.	☑ TC 11.6.	☑ TC 11.6.	☑ TC 11.6.						

r UA *11.6*	2 MP5 TC 11.6. 0_patch_1 AW 4.1 AW 4.2	update TC 11.6. 0_patch_1 AW 4.1 AW 4.2	0_patch_4 AW 4.1 AW 4.2 AW 4.3.3	0_patch_4 AW 4.1 AW 4.2 AW 4.3.3	0_patch_4 AW 4.1 AW 4.2 AW 4.3.3	0_patch_4 AW 4.1 AW 4.2 AW 4.3.3	0_patch_4 AW 4.1 AW 4.2 AW 4.3.3		
Team center r UA *12.1*	☑ NX 12.0. 2 MP5 TC 12.1. 0_patch_1 AW 4.1 AW 4.2	☑ NX 1851 update TC 12.1. 0_patch_1 AW 4.1 AW 4.2	☑ TC 12.1. 0_patch_4 AW 4.1 AW 4.2 AW 4.3.3	☑ TC 12.1. 0_patch_4 AW 4.1 AW 4.2 AW 4.3.3 AW 5.0	☑ TC 12.1. 0_patch_4 AW 4.1 AW 4.2 AW 4.3.3 AW 5.0	☑ TC 12.1. 0_patch_4 AW 4.1 AW 4.2 AW 4.3.3 AW 5.0	☑ TC 12.1. 0_patch_4 AW 4.1 AW 4.2 AW 4.3.3 AW 5.0		
Team center r UA *12.2*	☑ NX 12.0. 2 MP5 AW 4.2	☑ NX 1851 update AW 4.2	☑ AW 4.2 AW 4.3.3	☑ AW 4.2 AW 4.3.3 AW 5.0	☑ AW 4.2 AW 4.3.3 AW 5.0 AW 5.1	☑ AW 4.2 AW 4.3.3 AW 5.0 AW 5.1	☑ AW 4.2 AW 4.3.3 AW 5.0 AW 5.1	☑ AW 4.2 AW 4.3.3 AW 5.0 AW 5.1	
Team center r UA *12.3*			☑ TC 12.3. 0_patch_1 AW 4.3.3	☑ TC 12.3. 0_patch_1 AW 4.3.3 AW 5.0	☑ TC 12.3. 0_patch_1 AW 4.3.3 AW 5.0 AW 5.1	☑ TC 12.3. 0_patch_1 AW 4.3.3 AW 5.0 AW 5.1 AW 5.2	☑ TC 12.3. 0_patch_1 AW 4.3.3 AW 5.0 AW 5.1 AW 5.2	☑ TC 12.3. 0_patch_1 AW 4.3.3 AW 5.0 AW 5.1 AW 5.2	☑ TC 12.3. 0_patch_1 AW 4.3.3 AW 5.0 AW 5.1 AW 5.2

Teamcenter UA *12.4*				☑ AW 5.0	☑ AW 5.0 AW 5.1	☑ AW 5.0 AW 5.1 AW 5.2	☑ AW 5.0 AW 5.1 AW 5.2	☑ AW 5.0 AW 5.1 AW 5.2	☑ AW 5.0 AW 5.1 AW 5.2	☑ AW 5.0 AW 5.1 AW 5.2			
Teamcenter UA *13.0*					☑ AW 5.0								
Teamcenter UA *13.1*					☑ AW 5.1	☑ AW 5.1 AW 5.2	☑ AW 5.1 AW 5.2 AW 6.0	☑ AW 5.1 AW 5.2 AW 6.0 AW 6.1	☑ AW 5.1 AW 5.2 AW 6.0 AW 6.1	☑ AW 5.1 AW 5.2 AW 6.0 AW 6.1	☑ AW 5.1 AW 5.2 AW 6.0 AW 6.1		
Teamcenter UA *13.2*						☑ NX 1961 update TC 13.2.0_patch_8 AW 5.2	☑ TC 13.2.0_patch_8 AW 5.2 AW 6.0	☑ TC 13.2.0_patch_8 AW 5.2 AW 6.0 AW 6.1	☑ TC 13.2.0_patch_8 AW 5.2 AW 6.0 AW 6.1 AW 6.2	☑ TC 13.2.0_patch_8 AW 5.2 AW 6.0 AW 6.1 AW 6.2	☑ TC 13.2.0_patch_8 AW 5.2 AW 6.0 AW 6.1 AW 6.2	☑ TC 13.2.0_patch_8 AW 5.2 AW 6.0 AW 6.1 AW 6.2	
Teamcenter UA *13.3*							☑ TC 13.3.0_patch_3 AW 6.0	☑ TC 13.3.0_patch_3 AW 6.0 AW 6.1	☑ TC 13.3.0_patch_3 AW 6.0 AW 6.1	☑ TC 13.3.0_patch_3 AW 6.0 AW 6.1 AW	☑ TC 13.3.0_patch_3 AW 6.0 AW 6.1 AW	☑ TC 13.3.0_patch_3 AW 6.0 AW 6.1 AW	☑ TC 13.3.0_patch_3 AW 6.0 AW 6.1 AW 6.2 AW 6.3

			AW 6.2	6.2 AW 6.3	6.2 AW 6.3	6.2 AW 6.3
Teamcenter UA *14.0*		☑ TC 14.0.0_patch_2 AW 6.0				
Teamcenter UA *14.1*		☑ AW 6.1	☑ AW 6.1 AW 6.2	☑ AW 6.1 AW 6.2 AW 6.3	☑ AW 6.1 AW 6.2 AW 6.3	☑ AW 6.1 AW 6.2 AW 6.3	☑ AW 6.1 AW 6.2 AW 6.3
Teamcenter UA *14.2*			☑ AW 6.2	☑ AW 6.2 AW 6.3	☑ AW 6.2 AW 6.3	☑ AW 6.2 AW 6.3	☑ AW 6.2 AW 6.3
Teamcenter UA *14.3*				☑ AW 6.3	☑ AW 6.3	☑ AW 6.3	☑ AW 6.3
Teamcenter *2312*					☑ (1)	☑ (2)	☑
Teamcenter *2406*						☑	☑

Note:

(1) When you use Active Workspace Application Connect, NX 2306 is not compatible with Teamcenter 2312.

(2) When you use Active Workspace Application Connect, only NX 2312 and higher is compatible with Teamcenter 2312. Previous versions of NX are not compatible with Teamcenter 2312 for Active Workspace Application Connect.

For information on version compatibility for Teamcenter and Teamcenter lifecycle visualization, see the Teamcenter documentation.

NX compatibility with Spreadsheet

NX version ▼	*Platform►*			*Windows*			
	Spreadsheet software version►			MS Excel 365 Desktop	*MS Excel 2019*	*MS Excel 2016*	*MS Excel 2013 and previous releases*
	Platform ▼	*Operating system▼*	*Operating system version▼*		*For both 32–bit and 64-bit*		
NX 2406	*Windows*	*Windows 10 and Windows 11 64–bit*	10 Pro and Enterprise	Certified and supported	Certified and supported	Certified and supported	Not supported

Note:

- The 64-bit versions of Microsoft Excel are supported starting in NX 12.
- Microsoft Starter Edition is not supported by NX due to lack of support for Add-in's, Macro's, Math and Equation Editing.
- If you launch a spreadsheet command such as **Spreadsheet** or **Part Family** on a system having Excel 2003 or older version as the default spreadsheet application, NX displays an error message and does not proceed with the launched command.

NX support for the currently supported versions of Microsoft Office will be discontinued as per the following dates:

Microsoft Office version	*End of support*
2007	October 11, 2017
2010	October 13, 2020
2013	November 20, 2021
2016	October 14, 2025
2019	October 14, 2025

NX applications unsupported on specific platforms

The applications listed are not supported on the specified platforms.

Application	Functionality	Platform not supported on:
Tooling & Fixture Design	NX EasyFill Analysis - Advanced	Linux
NX Translators & Open Tools	NX Open Python Author	Linux
NX Translators & Open Tools	Content Migration Manager for SolidWorks	Linux
NX Translators & Open Tools	CMM Drawing Repair Assistant	Linux
Electro-Mechanical Design	ELMA	Linux
Electro-Mechanical Design	NX Piping Fabrication Drawings & PMI	Linux
Electro-Mechanical Design	Mechatronics Concept Designer	Linux
Mechanical Design	NX Lattice Structures Design	Linux
Mechanical Design	NX Drafting	Linux
Mechanical Design	NX Additive Design with Convergent	Linux
Mechancial Design	NX Ship Structure PMI Creation	Linux
Mechanical Design	NX Ship Dimensions & Adv Annotation	Linux
Mechanical Design	Drawing Automation for NX (version 10 and later)	Linux
Mechanical Design	Layout for NX	Linux
Mechanical Design	NX Aerospace Design	Linux
Mechanical Design	NX Lattice Structures Design	Linux
Gateway	Advanced Studio Rendering Style Mode	Linux
Pre/Post	Abaqus OBD result file reading	Linux
Pre/Post	Topology Optimization and Shape Optimization	Linux Redhat
Motion	Motion Mechatronics co-simulation	Linux

NX applications and features not installed by default

During installation, typical applications and features that are needed to run a standard installation of NX are automatically selected and installed by default. Due to your selections and the licensing requirements at your site, some applications and features may not be available. The following are not installed by default and may affect your expected operation of NX.

Localization

When the **Localizations** add-on product is not installed, NX cannot be launched in a non-English language unless the localization add-on for that language is installed.

Simcenter 3D

When the **Simcenter Nastran** add-on product is not installed, you cannot use the following:

- Stress Wizard
- Vibration Wizard
- Design Simulation
- Simcenter Nastran

Programming Tools

When the **Programming Tools** add-on product is not installed:

- The compilation of NX Open applications may not work as expected.
- NX Open Wizards are not available.

Support for touch enabled devices

In NX 2406, you can interact and manipulate 3D models and control the overall user interface using touch screen and stylus. The support for touch enabled hardware follows a slightly different support model than what is provided through the NX certification program. We have tested NX on a number of Windows based touch screen laptops, monitors, and tablets. However, support for these devices and other comparable systems is limited as they are not true workstations and do not qualify for our full certification program. Support for these devices is limited as follows:

- Graphics performance issues are not supported as most of these devices do not offer the level of driver support available on workstations.
- Graphics quality and display issues are not supported unless they can be duplicated on a certified workstation.
- Any issue determined to be caused by the graphics driver is not supported.
- Performance issues with NX are not supported on these systems unless reproducible on certified workstations.

Caveats aside, we have tested and used for development a variety of touch based systems from HP, Dell, Microsoft, and others and have found them acceptable for general usage of NX.

Additional Notes

- NX supports touch devices on the Windows operating system.
- Only stylus configurations support pre-highlighting. The most commonly used devices supporting stylus are the Surface Pros.

Legacy commands and commands to be retired

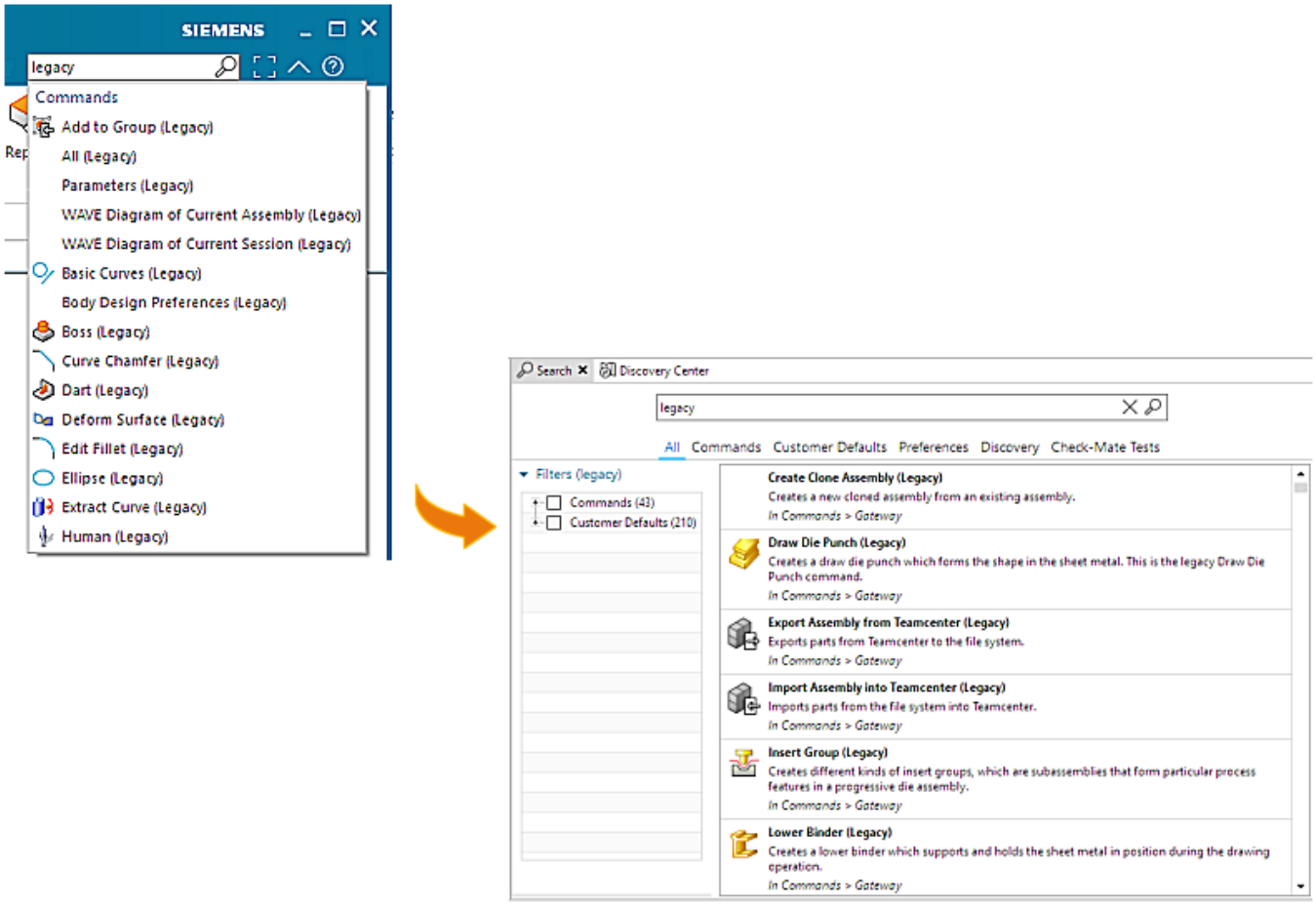
Commands that have **(Legacy)** or **(to be retired)** in their name are outdated, unsupported commands. In many cases, newer commands that provide more robust and enhanced functionality, or support improved workflows, exist.

Legacy commands remain in NX to sustain legacy workflows and products. Commands to be retired will be removed in a future version of NX, typically within two to four NX versions.

By default, both legacy commands and commands to be retired are removed from the Menu, the Ribbon bar, shortcut toolbars, and shortcut menus. However, they do appear in the **Search** box, and you can add them to your Ribbon bar, Quick Access Toolbar, or Border bars using the **Customize** command.

Finding legacy commands and commands to be retired

In the **Search** box in the top right corner of the NX window, type **legacy** or **to be retired** to see a list of the corresponding commands. Press Return to display the commands in the **Search** window.



Retired commands in NX

The following table contains a list of retired commands since NX 12, and the recommended replacement commands.

Retired command	Replacement command	Release retired
Instance Feature	**Pattern Feature**	NX 12
Export PNG	**Export Image**	NX 12.0.1
Export JPEG	**Export Image**	NX 12.0.1
Export GIF	**Export Image**	NX 12.0.1
Export TIFF	**Export Image**	NX 12.0.1
Export BMP	**Export Image**	NX 12.0.1
Materials/Textures	**Show Usage**	NX 12.0.1
Soft Blend (Closed)	**Face Blend**	NX 1847
Show/Remove Constraints (in Legacy Sketcher)	**Constraint Navigator** (in NX Sketcher)	NX 1847
Display Dimension as PMI	**Display as PMI**	NX 1847
Delete Sketch Object (old)	**Delete Sketch Object**	NX 1847
Ray Traced Studio (old)	**Ray Traced Studio**	NX 1847
Ray Traced Studio Editor (old)	**Ray Traced Studio Editor**	NX 1847
Material Editor	**Studio Material Editor**	NX 1847
HQI	**Ray Traced Studio**	NX 1847
Artistic Image	None. Use **Ray Traced Studio** to realistically render your part.	NX 1847
Visualization Preferences (non-block based)	**Visualization Preferences**	NX 1847
Visualization Performance Preferences	**Visualization Preferences**	NX 1847
Facet Settings	**Visualization Preferences**	NX 1847
Fillet Surface (closed)	**Face Blend**	NX 1872
Join Curve	**Composite Curve**	NX 1872
Isoparametric Trim/Divide	**Snip Surface**	NX 1872
Move Defining Point	**Transform**	NX 1872
Move Pole	**Transform**	NX 1872
Tolerance Feature Associated Objects	PMI commands	NX 1872

Retired command	Replacement command	Release retired
Tolerance Feature Search	PMI commands	NX 1872
Tolerance Feature Update Report	PMI commands	NX 1872
Convert Non-associative Tolerances	PMI commands	NX 1872
Delete Tolerance Feature	PMI commands	NX 1872
Geometry Tolerancing	PMI commands	NX 1872
Geometric Tolerancering Preferences	PMI commands	NX 1872
List All Tolerance Features	PMI commands	NX 1872
Master Model Tolerancing	PMI commands	NX 1872
Validate Tolerance Specification	PMI commands	NX 1872
Navigate	**Fly-through**	NX 1899
Navigation Options	**Fly-through Options**	NX 1899
Edit Text	Commands that let you directly edit the text of objects.	NX 1899
Basic Lights	**Scene Preferences**	NX 1899
Edit Background	**Scene Preferences**	NX 1899
Scene Editor	**Scene Preferences**	NX 1899
Measure Distance	**Measure**	NX 1926
Simple Distance	**Measure**	NX 1926
Simple Diameter	**Measure**	NX 1926
Simple Radius	**Measure**	NX 1926
Measure Angle	**Measure**	NX 1926
Simple Angle	**Measure**	NX 1926
Measure Length	**Measure**	NX 1926
Simple Length	**Measure**	NX 1926
Measure Point	**Measure**	NX 1926
Minimum Radius	**Measure**	NX 1926
Measure Face	**Measure**	NX 1926
Measure Body	**Measure**	NX 1926
Measure Extreme	**Measure**	NX 1980
Merge Overlapping Facet Bodies	**Mesh from Cloud**	NX 2206

Retired command	Replacement command	Release retired
Hide Similar Component	**Select Similar Component** and then use **Show** or **Hide**.	NX 2212
Show Similar Component	**Select Similar Component** and then use **Show** or **Hide**.	NX 2212
Find Object	**Search** panel and **Filter** shortcut menu options in the **Part Navigator**	NX 2212
Model Compare	**Compare Body**	NX 2306
Interactive Class Editor (ICE)	Microsoft Visual Code plus the Knowledge Fusion extension. See %UGII_BASE_DIR% \design_tools\NXKF\README.md for more information.	NX 2312
View Layout Setting	None. You can just select the viewport where you want to display a graph or contour plot.	NX 2406

3. NX Caveats and Product Notes

Teamcenter Integration for NX

Caveats

Opening in Visualization

In **Active Workspace**, when you select an item and **Open in Visualization**, a browser is launched with an error message. The selected item is not displayed.

Listing order for JT assemblies in Assembly Navigator

The listing order of items you have set in the **Assembly Navigator** is not maintained for JT assemblies if you reopen the assembly. To get the set listing order for the JT assemblies, you need to save the top JT assembly with the master dataset.

Assembly arrangements in Active Workspace

NX does not load the assembly arrangement from **Active Workspace**. The arrangement that was active at the last save of the assembly into Teamcenter is loaded into NX.

Running NX Relations Browser

If you have used the NX Relations Browser in previous NX versions, you may get an error when launching it. For example:

```
An error has occurrred. See the log file
C:\Users\<userid>\NXBrowser\metadata\log.
```

This can be due to old files interferring with the new NX version. Delete the C:\Users\<userid>\NXBrowser directory, then restart the NX Relations Browser.

Publishing product interfaces

Starting in NX 2306, the default setting for publishing product interfaces to Teamcenter is **None** instead of **Relations**. To get the publishing of product interfaces to remain as it was in previous releases, you may need to reset the **Product Interfaces** customer default.

Partitions in Teamcenter Integration for NX

When you use structure only loading, you cannot see the partition memberships in BVR partitions.

Fundamentals

Product Notes

NX Virtual Reality is retired

You can use the new Immersive Explorer application to access the functions of the NX Virtual Reality Review commands.

For assistance with the migration of the NX Virtual Reality Collaboration functionality, contact Customer Support.

Environment variable NX_RTS_IRAY

The environment variable NX_RTS_IRAY is removed in NX 2212.

Scenario...	...Action required
Your administrator has set this environment variable in NX11 or so, and you do not have any legacy part.	No further action required. In NX 2212, you get the latest functionality of NX for high-end rendering.
You have a specific library of visual materials that you use, and the edited materials are not captured in the released part files.	Review your user library against the latest visual materials that are provided in NX 2212 library. We recommend upgrading the original library and getting the new capability of NX.
You have created a material library or used those materials or modified versions of these materials in released parts. These parts have been brought into new assemblies, but there is no clear route to migrating the original materials in the released parts to the latest visual materials.	Contact Customer Support.

Logic Editor task environment

The **Execute NXOpen** node now supports C++ programs.

Unit Manager settings

When you run the *ug_convert_part* tool to convert an NX 12 part from mm to inch, the settings in the **Unit Manager** dialog box do not display the converted units. This is as intended.

The **Units Manager** dialog box displays the units that you have selected as the default units for data entry and object information output. Those units are independent of the underlying part units (metric or inch).

Navigate tool is retired

The **Navigate** tool is retired. You can explore your design in 3D space using **Fly-through**.

Caveats

Display performance issues

If you are using an Intel GPU or an entry level GPU from NVIDIA or AMD, you may experience performance issues for **Shaded** display style.

To prevent this, set the **Finish Effect** visualization preference to **Part Shininess**. You can also use the **Finish Effect** customer default.

We also recommend upgrading the graphics driver to the latest version.

Display of materials and decals

- For an Siemens Visual Material (SVM) with box mapping and color texture set to an image on your local machine, when you apply this material to a face for which the normal is aligned with the Z-axis, the display of the color texture image might be incorrect.
- When you apply a visual material with a cylindrical mapping projection and set **Appearance** to **Visual Material**, NX might display black texture on the surface.
- For a decal with a non-transparent color, there might be differences in the display of the bump effect in **Shaded** and **Studio** display styles.
- When the SVM material, **Coated Satin Metal**, is applied to a block and when you set the work view to **Left** or **Right**, the NX displays the material in black.

Siemens Visual Material (SVM) format

When pre-NX 2206 materials are displayed or referenced, NX converts these to the SVM format. The exceptions are as follows:

- Edited Iray+ materials with non-SVM base type, for example Plastic, Opaque, Aluminum, Matte and so on. These remain in the part as Iray+ materials.

  Note that in NX 2206, Iray+ materials that are modified to SVM base type remain as Iray+ materials.
- Iray+ materials created from imported MDL, for example, NVIDIA Materials. These remain as Iray+ materials.
- Unedited legacy materials with a name that does not have a matching SVM of the same name in the **System Studio Materials** palette. NX translates these to an SVM using the undefined material appearance.
- Edited legacy materials. NX translates these to an SVM using the undefined material appearance.

The **Advanced Studio** display and the **Ray Traced Studio** display do not currently support the following SVM parameters:

- **Textures** node→**Base** tab→**Displacement** texture
- **Base** node→**Color** group→**Micro-Surface Shadow**

Ray traced rendering

On GPUs with less than 4GB of memory (for example, Desktop: NVIDIA T500-2GB|T400-2GB, Quadro P620|P600|P400; Mobile: NVIDIA T500, Quadro P620|P520; VDI/Data-Center: GRID vGPU 2Q configuration), when you use the **Ray Traced Studio** command, NX might close abruptly when you do the following:

- Use a command or functionality that creates a thumbnail preview of the visual materials, for example, editing visual materials, importing MDL materials, opening pre-NX 2206 parts that contain visual materials.

Decal Sticker dialog box

When you use the **Decal Sticker** dialog box, dragging and dropping sample decals from system studio materials palette may not always appear in correct location.

For a decal with **Reflectivity** set to **From Visual Material**, when you edit this decal, in the **Decal Sticker** dialog box, **Reflectivity** shows **Matte**.

Extended reality files

For export to extended reality files, NX does not support:

- Scene Lights, Advanced Lights, and Image-based lighting from high end rendering modes.
- PMI objects.
- Raster images.

Speech Recognition

You can use all languages to execute voice commands in NX except Traditional Chinese.

> Note:
>
> When you execute commands using speech recognition, you can move forward through a dialog box by saying **next**. This is supported only for the group-based dialog boxes, such as **Extrude** and **Blend**.

This navigation in the dialog boxes is not supported for other commands, such as, **Open** and **Curve**.

Immersive Explorer

- When the Auto-rotate Model option is active and you use the Relative to World coordinate system, certain material textures may appear to move during the rotation.

- With some VR headsets, ambient occlusion can accentuate the tessellated edges of faces on models, causing them to appear more prominent.

  To mitigate this, switch off **Ambient Shadows**.

Bookmarks

If you create a bookmark file with Ray Traced Studio mode enabled, when the bookmark is applied Ray Traced Studio mode is not in effect.

If you create a bookmark file when displaying a View Section with clipping disabled, when the bookmark is applied the section may be incorrectly clipped.

If you try to apply a bookmark file when

1. the bookmark file is not for the current displayed part and

2. the number of views in the layout at the time when the bookmark file was created is not the same as the number of views in the current displayed part

then the number of views in the layout may be wrong after the bookmark file is applied. Applying the same bookmark file a second time corrects the number of views.

3MF files with lattice bodies

3MF currently doesn't support ball specifications. If you import or export a 3MF file that has lattices with balls or specified ball diameters, the balls are lost during the import or export operation.

Documentation notes

Export to Omniverse

Use the **Export to Omniverse** command to export your entire part into a USDZ file which you can upload to the NVIDIA Omniverse database.

Where do I find it?

Prerequisite	You must have a license for exporting to extended reality file formats.
 Command	**Export to Omniverse** 

Customer Defaults

Customer defaults

Customer defaults allow you to customize the operation of NX. The initial settings and parameters of many functions and dialog boxes are controlled by customer defaults. You can view them in NX by choosing **File→Utilities→Customer Defaults**.

For a description of the new or modified customer defaults, see the **Customer Defaults What's Changed Report**. You must have a valid Siemens Digital Industries Software sign in account to access this report.

For additional information on customer defaults, see Customizing the NX installation→Customer Defaults in the NX online help. For a listing and description of all customer defaults, see the Customer Defaults help.

Design (CAD)

Sketch

Caveats

Sketch Navigator

- The Activate Parent command in the 2D Component context menu does not work.

Silhouette Curve

- If you select a convergent face as input, Sketch creates only partial Silhouette Curves.

- If you edit a silhouette curve, Sketch removes any persistent relations or dimensions that are affected by the edit of the Silhouette.

Pattern and Mirror with Silhouette Curve

- If you pattern or mirror a Sketch feature that contains Silhouette Curves, the output will incorrectly contain the source curves.

Assemblies

Product notes

Multi-user design notifications

In Teamcenter Integration for NX, you can follow parts using multi-user design notifications. This means, you receive notifications when another user saves the parts that you follow or checks them in or out.

You must install the eventing service when you configure Teamcenter Integration for NX. In Teamcenter 13.3 or later versions, the eventing service is supported on both Windows and Linux. In Teamcenter 13.1 and 13.2, the eventing service is supported only on a Linux host.

Caveats

Synchronous Assemblies

The synchronous assembly version of the **Move** command does not support:

- **Angle** and **Distance** constraints with limits
- Component patterns
- Mating conditions
- Joints or couplers
- Parametric surfaces and curves, except for b-spline surfaces, which are supported

Copying and pasting components

You cannot copy and paste:

- Joints or couplers
- Constraints that reference deformed components

WAVE

You may encounter performance issues when working with large assembly structures.

When you make changes to the assembly hierarchy in the **Assembly Navigator**, such as adding or deleting a component, the **Interpart Link Browser** hierarchy becomes chaotic.

Animation Designer

Caveats

Kineo solver limitations

You may experience unexpected behavior using the Kineo solver when:

- You have gear objects without a fixed axis, such as planetary gears.
- You have a single rigid group constrained by multiple curves.

Modeling

Caveats

Deviation Gauge

If you try to reset tolerances for **Deviation Gauge** when using faceted bodies, you will run into an error.

Vector Constructor

The following are fully supported in IP1100:

- **Assemblies**
  - **Assembly Sequence**
  - **Mirror Assembly**
- **Routing**
  - **Qualify Part**

The following will be supported in the future:

- **Assemblies**

  - **Move Component**
  - **Assembly Constraints**
  - **Pattern Component**
- **Routing**
  - **Move Routing Object**
  - **Line Path**
  - **Spline Path**
  - **Edit Point**
- **Measure** command

Contribution Model

GPU based preview for Lattice and Implicit is only supported for fully shaded style. Existing convergent preview will be used in other display styles.

Thread Enhancements-Visual Feature Thread

• Display of the dashed circles for Threads in Shaded mode does not update correctly when you turn Show Decal Stickers on or off. You need to hover the mouse over the Symbolic Thread or Threaded Hole feature in the graphics window to update the display.

• For master model drawing files from previous releases, user will need to load the master part first, so that the Thread Decals are applied to the thread faces before loading the non-master drawing part file.

Measure

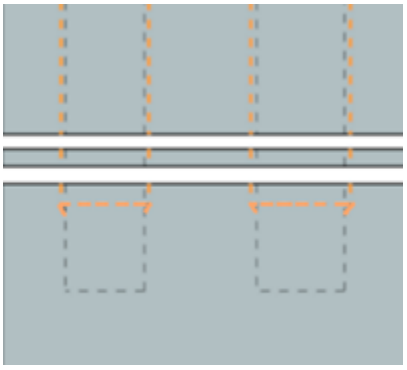
You may have some lag to calculate Inscribed Circle when pick-up two curves. Performance maybe be improved if more objects are selected.

Drafting

Product Notes

Thread enhancement

When you use the **Subtract** command to create a thread feature through multiple target bodies in the same part, NX supports displaying these threads in any drawing view.



See **Subtract command enhancements** in the *What's New Guide* for more information.

Product and Manufacturing Information (PMI)

Product notes

Specialized PMI commands

You can access the following commands using the **Customize** dialog box. These commands remain available with the appended text **to be retired**, and they will be phased out in a future release.

- **General Note**
- **Specific Note**
- **User Defined**
- **String Note**
- **Number Note**
- **Integer Note**
- **URL Note**
- **Enterprise Identification**
- **Material Specification**
- **Part Identification**
- **Process Specification**

The **Coordinate Note (Legacy)** and **Locator Designator (Legacy)** are available in the command finder.

Model Based Definition

Product notes

PMI Position command enhancements

The **PMI Position** command now supports the repositioning of angular dimensions.

The command is also enhanced such that:

- PMI objects are more evenly distributed around the model to provide optimal placement.
- PMI objects are arranged to avoid contact and overlapping.

Out-of-the-box rules

The following MBD logical rules are now available out of the box:

- **Create PMI Context**
- **Create PMI Context from List**
- **Add PMI to PMI Context**
- **Create Linear Dimension**
- **Create Angular Dimension**
- **Show Component in Model Review**

The **Find PMI by PMI Context** rule is now available out of the box for MBD Query.

Technical Data Package

Product notes

Creating technical data packages in batch mode

A login file is now required when using the batch file executable for creating technical data packages. See the *PublishTDPREADME* file in the *{UGII_BASE_DIR}\tdp\batch* directory for more information.

Performance Predictor

Product notes

The Performance Predictor stress results are improved. Results are now provided at more locations, resulting in a smoother plot. Your simulation results may vary slightly as a result.

When performing a linear statics analysis, NX now creates a minimum factor of safety result measurement alongside the displacement measurements and stress result measurements.

Coating

Product Notes

Coating Navigator columns

NX2406 does not support the following columns that are displayed in the **Coating Navigator**.

- **Linear Length**
- **Raw Linear Length**

To hide these columns, use the following steps.

1. Right-click in the background of the **Coating Navigator** and choose **Column Configuration→Configure Columns**.
2. In the **Configure Columns** dialog box, under **Column Configuration**, clear the check boxes for **Linear Length** and **Raw Linear Length**.
3. Click **OK**.

NX hides these columns.

Caveats

Coating parameters

Coatings that are applied on linked faces and sheet bodies are not included in calculations of mass properties.

Visualization material for coating

When you create a coating layer, you can select a visualization material to apply to the coating layer from the **Studio Material List** dialog box. If this dialog box displays an invalid visualization material, we

recommend that you ignore this material and select a valid visualization material to apply to the coating layer.

Integrated Material Management (IMM) database

When you assign a physical material for the coating from the IMM database, NX does not save that material as a non-geometric component (NGC). As a result, no NGC is saved in Teamcenter or added to any bill of materials.

Using special characters in material names

When you create a coating, NX does not create an NGC if the name of the material contains any one of the following special characters.

- Less than (<)
- Greater than (>)
- Colon (:)
- Double quotes (" ")
- Forward slash (/)
- Backslash (\)
- Vertical bar (|)
- Question mark (?)
- Asterisk (*)

Reusing a coating stack

You can reuse a coating stack and apply it to only one coating region. If you want to modify the region on which the reusable coating stack is applied, you must edit each layer in the coating stack.

Routing

Product notes

Capital and VeSys-supported MCAD Integrations

Refer to the **Siemens Community forums** for Capital and NX version compatibility.

Caveats

- You cannot place a port on a convergent face.
- You cannot place overstock on the convergent face of a fitting.

Routed System Designer

Product notes

Spline Path behavior

- To ensure that a spline's end point remains connected to another object, set the **Create Interpart Link**  option when selecting the connecting point. You can also set the customer default **Allow Associative Interpart Modeling**.
- Any spline you create in an Electrical Routed System Designer part must be included in a wiring component. If a wiring component is not present in the part, a message advises you that you must create one by clicking **OK** in the message box. The new wiring component appears in the **Assembly Navigator** with the prefix *rtw-* attached to its name.
- To start or end a line or spline from a specific point between the end points of another spline, first subdivide the existing spline and then use the resulting routing control point (RCP) to define the new line or spline start or end point.
- Use the **Point on Curve** snap point option to start a spline path at any arbitrary point along the spline. Set the **Tangency on Selection** option, formerly called **Tangent to Path**, to maintain tangency between the two splines.
- With the **Spline Path** command, only RCPs are available on an interpart routing segment selection.
- You cannot apply the **Disconnect Components** command to a spline point. To disconnect one spline component from another, right-click the spline in the graphics window and choose **Spline Path** to open the command in edit mode. In the dialog box, from the point list table, right-click the connected point and choose **Disconnect**.
- In a multiple-level assembly context, you can only disconnect a spline point that references another object. In a single-level assembly, you can disconnect both spline points, that is, the one that holds the reference and the one that is referenced by another spline point.
- Tangency support is now available between splines for interpart connections.
- While specifying a spline point at a location where multiple objects connect, a spline point that is already connected to another routing object is not selectable. This helps you connect subsequently created splines to the same driving object.

- When you create a spline, NX might generate a WAVE linked feature, depending on the type of object selection you make, and whether you are performing a single-level mechanical selection in which the stock component and spline component share a common parent part, or a varied context selection, in which the stock component and spline component do not share a common parent part.

  - For all interpart selections to a non-routing object, a WAVE linked feature is created.

  - For all interpart selections to an RCP, no WAVE linked feature is created.

  - For interpart selections to a port, a WAVE linked feature is created in a varied context selection, but no WAVE linked feature is created for a single-level mechanical selection.

Inheriting feature template (FTS) stock attributes

When you assign FTS stock to a bare target path (one with no stock currently assigned to it) that is connected to another path with existing FTS stock, you can let NX automatically select a stock size that matches the size attributes of the existing FTS stock or PTS component on the connected path.

For this to occur, the following three criteria must be met:

- The FTS attributes of the new stock must share identical attribute display names with the existing stock or fitting, for example, **Height**, **Width**, or **Diameter**.

- The template attribute(s) must not be Read-Only. This is specified in the Feature Template Author utility.

- Smart initialization must be turned on for the attribute(s). This is also specified in the Feature Template Author utility.

If any of the above criteria are not met, then any FTS stock you apply or edit assumes the default attribute values specified in the **Routing Part Library**. Likewise, if only some of the attribute display names match, then those non-matching attributes retain their default values.

Assigning FTS stock to a new path connected to an existing path with FTS stock on it

When you connect a new path to an existing path that has FTS stock assigned to it, the stock on the new path will automatically match the size attributes of the stock on the existing path. This includes whether you connect the new path to the existing path RCP or to one of its stock or fitting ports, as well as to any point position along the existing path. If you elect to change the stock profile from rectangular to flat oval or vice versa, the stock on the new path still retains the size attributes of the stock on the connected path.

Assigning or editing FTS stock on a standalone path

When you assign FTS stock to a standalone path, the stock adopts its default attribute values defined in the **Routing Part Library**. You can use the **Edit Template Attributes** to modify its size. If you select a different profile for the standalone stock, it still retains its current size attributes.

Modifying the material attribute of an FTS stock

If you modify the material attribute of a standalone or connected stock, its current size attributes are unaffected.

Capital and VeSys-supported MCAD Integrations

Refer to the **Siemens Community forums** for Capital and NX version compatibility.

Caveats

Automatic elbow placement in Mechanical Routed System Designer

If you attempt to create a line path originating from one of the free end RCPs of an existing stock component with the **Elbow** corner option selected, the operation raises the following error message:

```
Incorrect object for this operation.
```

This issue will be addressed in an MU release.

Ship Structure

Caveats

Sketch

A new Sketch solver was introduced in NX 2007 and is the default, but it is not recommended for ship design workflows. We recommend you set the **Use the new solver and UI for sketching** early access feature to **Off**.

Manufacturing XML Output

The following caveats apply when you use the **Export DXF File** option to export a DXF file to use it in the external nesting software FNEST of FORAN.

- The name of the DXF file that NX generates is too long for FNEST to process.
- FNEST cannot process the following weld bevel code that is in the exported DXF files:

```
Joint_((R0_4)_(T2_4)_(A2_45)_(T3_6)_(A3_0)_(T4_4)_(A4_45))
```

Sheet Metal

Caveats

Patterning of a gusset on non-cylindrical bends

When a gusset is on a non-cylindrical bend, you cannot create a pattern of the gusset.

> Note:
>
> This is applicable only for gussets created using **Automatic Profile**.

To create a pattern of a gusset that exists on a non-cylindrical bend:

1. Extract the curve of the mold edge of the non-cylindrical bend.

   For example, use **Offset Curve in Face** to extract the curve.

2. Use the extracted curve for defining the plane location and create a gusset.

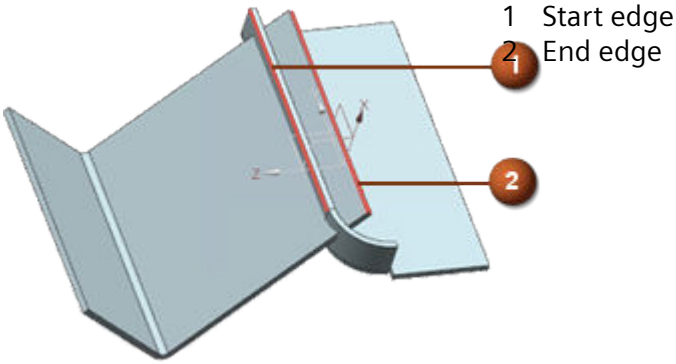
3. Pattern the gusset using the extracted curve as a path.

   > Tip:
   >
   > To do this, use **Along** in the **Layout** list.

Bridge Bend

You cannot create a bridge bend if the two sheet metal bodies intersect in the following way:

- A body from which you select an end edge intersects with the non-planar web face of the parent body of start edge.



Convert to sheet metal

If you create a fold transition between the planar web faces of chained bends and convert the body to a sheet metal part, the conversion may be partially successful.

Creating multiple flanges

While creating multiple flanges, if adjacent flange edges are not in the same set, you cannot apply an internal miter.

Unite in Sheet Metal

If you unite bodies that consist of Sheet Metal and advanced Sheet Metal features, the united body may not support subsequent Sheet Metal operations.

Material and Bend tables

The tool selection options are not available for the following commands and options.

- **Hem Flange**
- **Lofted Flange**
- **Convert to Sheet Metal**
- **Bridge Bend**→**Type** group→**Z or U Transition**

  Bridge Bend→**Type** group→**Fold Transition**→**Bend Properties** group→**Define By** list→**Length** option
- **Advanced Flange**→**Type** group→**To Reference**

PCB Exchange

Caveats

Simcenter FLOEFD Smart PCB solution

- Component heat loads that are set in PCB Exchange are not supported in FLOEFD.
- FLOEFD **Smart PCB** is created with default materials.

Flexible Printed Circuit Design

Caveats

Unite in Flexible Printed Circuit Design

If you unite multi-thickness bodies, the united body may not conform to the selected layer scheme.

Human Modeling

Product Notes

Improved Human Scaling

The **Human Control Panel** is updated with additional end point control for the human hands, for more accurate and easy reach studies.

You can now use the following reference points for end point control:

- **Palm Center**
- **Index Finger Tip**
- **Middle Finger Tip**
- **Thumb Tip**

This helps you create accurate reach studies easily. It is especially useful in occupant layout studies where the focus is on the reach of a human hand to a display console and to several controls.

Improved scaling of human hand

The **Human Hands** dialog box is updated with the **NHANES_SURVEY_DATA** database to scale human hands.

This database provides consistency between full body scaling and human hand scaling. It also provides flexibility to use the disembodied hands, both to check the clearance and for grasp studies.

Documentation Notes

Hands-on learning aids in the online help

The online help now includes a limited number of hands-on procedures with part files to download, which let you work through detailed instructions on your own. These hands-on procedures are a good

learning tool to try out new functionality and complex workflows. The procedure topics are titled **Hands-on: <Procedure name>**.

Vehicle Design and Validation

Caveats

- Eyellipse features created in versions earlier than NX 2007 do not work in the **Direct Vision** command in NX 2304. You must create a new eyellipse feature in NX 2007 or later.
- You cannot create a monocular direct vision feature for the user-defined eyellipse. You can create a user-defined direct vision feature for such points.
- We recommend that you avoid selection of vision devices, such as direct vision, mirror, and camera, that are defined before you open the **Interception** dialog box.

  If you select these devices and then open the dialog box, NX cannot select an obstruction for the ground or wall clearance. In such cases, restart NX.
- When cross section of View Volume Camera is less than the size of the obstruction body, it does not get trimmed.
- The view volume between the vision device and the obstruction body does not get created.

Caveats for early access features

Environment painting

- Sometimes the creation of an environment painting feature fails.
- When you select a sheet body, a cylinder or a sphere, for the environment or environment painting NX creates an incorrect result.

Standard environment data

- The environment feature for a user-defined flow is without bodies and for other standards it has its own bodies. Thus, you cannot hide a user-defined environment from **Part Navigator**. Alternatively, you can hide the original bodies, or the WAVE-links that are generated if the body selection is within the work part or the entire assembly respectively.
- You cannot edit the standard name selection for an environment feature from user-defined to other standards or vice versa after you create the feature.

VDV mirror

In NX 2306, you cannot edit a mirror feature that you created in NX 2306 or earlier if it has Class I standards.

Manufacturing (CAM)

Manufacturing Product Notes

The Manufacturing product notes describe product changes that are not included in the *What's New in NX* documentation.

Tool path and template changes

Template change to mill_contour

The **Quick Roughing**  operation subtype is available under **mill_contour**. This offset roughing operation has a simple user interface, and you can program it quickly.

Quick Roughing  is recommended for quick programming and machining of simpler prismatic parts, mold cavity and cores, dies, casting, and forging with contoured shapes using planar cut levels.

Mill Turn Workpiece

For facing head (U axis Turning head) scenarios, you must continue to use separate Milling and Turning Workpiece geometry objects.

Manufacturing templates

All SimXX example machines are removed from the **New** dialog box (**File→New→Manufacturing** tab→**Templates** group).

Updated machines are available on **Post Hub**.

Tool path changes

A general reminder: There are ongoing changes in the processors to fix problems, add enhancements, and improve reliability. In many cases, you may see some differences between the new path and the old path when you generate an operation from a previous release. If you rely on automatic methods, these changes should be acceptable. The end result of the new path should be comparable to, or better than, the previous path.

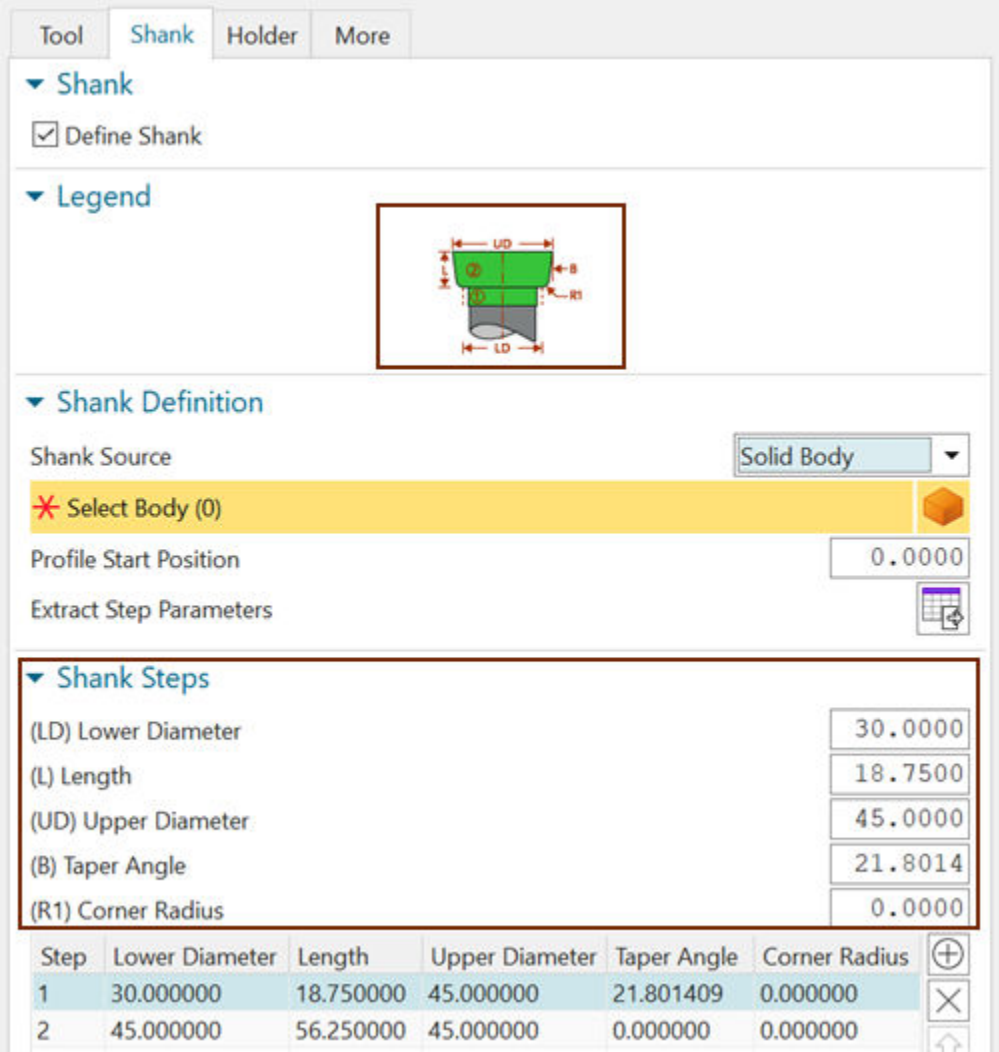
To prevent accidental changes to tool paths in part files from a previous release, use the **Lock Tool Paths During Version Upgrade** customer default. This option automatically locks all operations with an edit status of **Complete** or **Repost**.

Ongoing dialog box changes

As a result of customer feedback for the explorer tree format dialog boxes, there are ongoing changes to parameter locations. To find a specific parameter, enter the parameter name in the box. As you type, NX displays a list of matching entries that you can select from.

Tool templates

All milling and drilling tool subtypes are now enhanced with additional parameters and also support multi-step shanks.



Customized templates

If you create your own template files, you must back up the files *before* you upgrade to the current release. Copy the template files from the templates folder to a separate location to prevent the installation from overriding your defined templates.

After the upgrade installation is done, copy your template files back to the same location where you had earlier saved them.

Merging customized templates

You can merge your customized templates with the templates included in this release in the following ways:

- Start with the new default templates and apply your customizations. This is recommended to ensure you receive the PR fixes that apply to default values and presentation.
- Re-file your customized templates in the new release, review the changes listed for the release, and implement the applicable ones in your templates.

You can use the **Compare Objects** command to help you merge your customized templates with templates in the new release.

General changes

Toolholder item changes

The following toolholder items (**Create Tool**  **→Retrieve Device from Library**  **→Library Class Selection→Toolholder**) are removed and it is recommended that you do not use them.

- **sim10_turn_tool_block**
- **sim10_id_tool_block**
- **sim10_blank_tool_block**
- **sim10_axial_mill**
- **sim10_radial_mill**
- **sim11_toolblock_holder**
- **sim15_axial_mill**
- **sim15_axial_mill_high**
- **sim15_radial_mill**
- **sim15_turn_block**
- **sim15_turn_block_duo**

Machine Tool changes

The standard machine tools supplied with NX have revised postprocessors and kinematics models. Review all existing Manufacturing setups which use a standard machine tool from a release prior to NX 8. If necessary, retrieve the machine tool again.

Multi-Axis Additive Manufacturing

Planar Additive Thinwall

Planar Additive Thinwall lets you create multiple beads using the **Profile**  type of **Cut Pattern**.

4x +1 Auto Tilt

The **Align Tilt From Point** tool axis option is renamed to **4x +1 Auto Tilt** as **4x +1 Auto Tilt** better represents what the option actually does.

This option creates a 4-axis operation where the tool automatically tilts, but does not rotate. Use this option to create smoother output on round parts.

Tape tool

The new tape tool assumes the end of the tape is squared off, whereas laser and extruder tools assume the end of the bead is rounded.

When you use the tape tool in Freeform Additive Coating operations with containment, the tool trims the tape where it touches the containment, and does not overlap the containment. Laser and extruder tools trim the bead when the center of the tool touches the containment.

Tool Axis Laydown

The **Tool Axis Laydown** tool axis option is removed in the following operations:

- **Planar ZigZag**
- **Planar Additive Follow Inward Outward**
- **Planar Spiral**
- **Planar Thinwall**

4x +1 Auto Tilt

The **4x +1 Auto Tilt** tool axis option is introduced to create smoother output on round parts.

4x +1 Auto Tilt is added in the following operations:

- **3D Freeform Buildup**
- **Gradient Buildup**
- **3D Freeform Thinwall**

NX Machine Tool Simulation (NX MTS)

ISV renamed to NX Machine Tool Simulation

Beginning NX 2306, Integration Simulation and Verification (ISV) is renamed to NX Machine Tool Simulation (NX MTS).

NX MTS using CSE simulated controllers with Python parser

- The CSE Python parser is supported for Windows only.

  This means that the standard examples supplied with NX will not run on non-Windows operating systems. As a workaround: Inside the MachineConfigurator, in the **Global Settings** dialog box, change the **Implementation** setting for the MCF from **Python** to **Library**.

  > Note:
  >
  > The Machine Configurator is supported for Windows only.

- For existing simulations using Python parser with encrypted files from older versions, for example NX 9 or NX 10, we recommend that you use the un-encrypted **.py* files from the latest release.

Run MyVirtual Machine

You can now boot the Sinumerik ONE virtual controller after you close the **Create MyVirtual Machine** HMI manually. This enables you to boot the Sinumerik ONE virtual controller without restarting NX.

Feature-based Machining (FBM)

First Help for FBM Configurator

When you click F1 or the **Help**  icon on the **FBM Configurator** dialog box→**Machining Rules** tab, the help opens in the embedded browser within NX, displaying short example videos of machining rule scenarios.

With First Help, you can:

- Learn more effectively from visual content and videos.
- See examples that help you get started with the command.
- Link from the navigation bar to additional help topics.
- Click the feedback button and let us know what you think.

Persistent state of FBM Configurator dialog box

By default, NX now retains the previously accessed state of the **FBM Configurator** dialog box in an active session. So when you reopen the **FBM Configurator** dialog box, NX displays the dialog box as it was before you closed it.

Robotic Machining

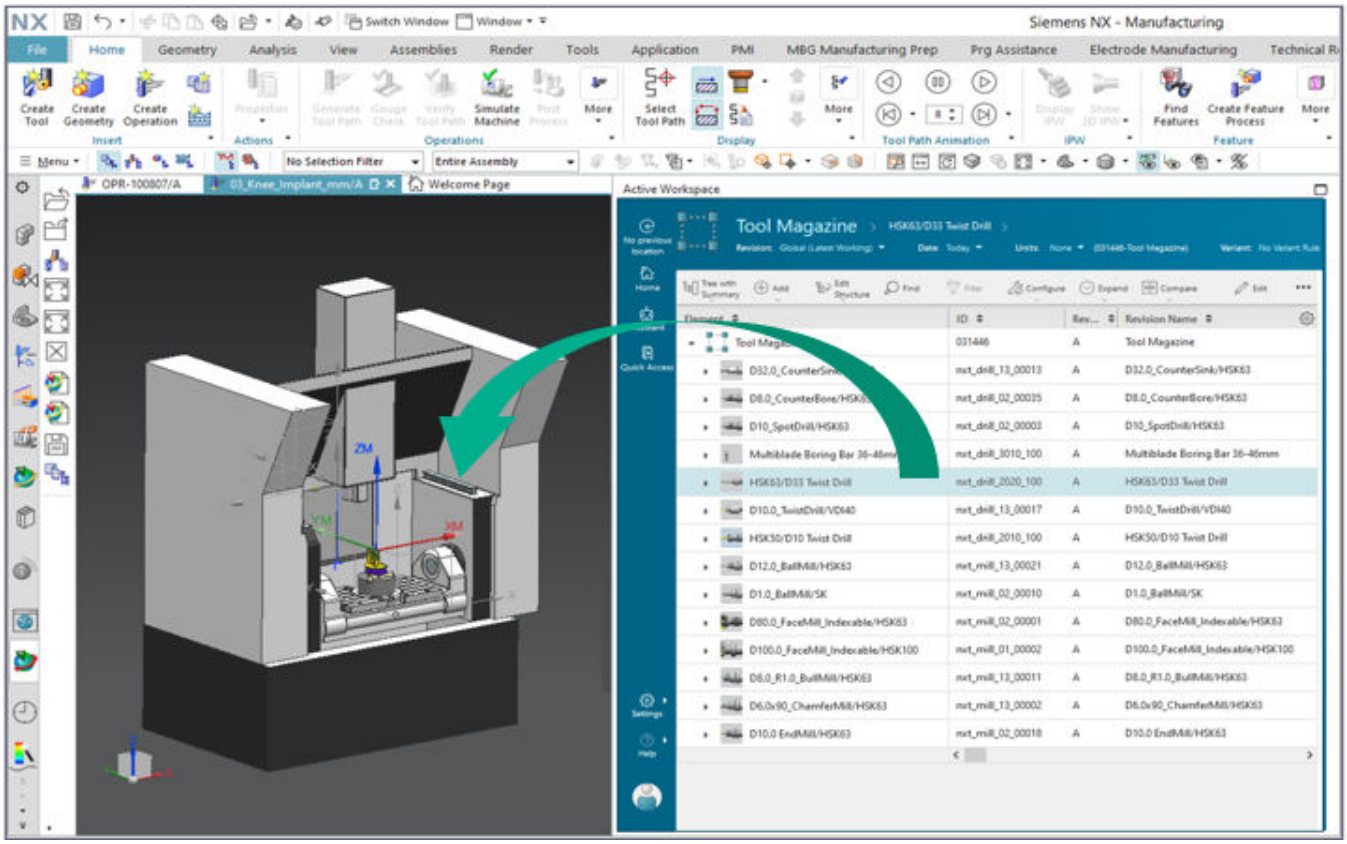
Display Tool in Machine

The **Display Tool in Machine** option is moved from within the **Display** group→**More** options to directly in the **Display** group options for greater visibility.

NX CAM to Teamcenter integration

Drag machines and tools directly from Active Workspace to NX CAM

You can now drag resources from the Active Workspace web user interface to the NX CAM session. This is an easy way to retrieve NC tools and machines.



Manufacturing critical maintenance and retirement notices

Planned retirement of classic ONT and CFP

The classic Operation Navigator (ONT) and the classic Create Feature Process (CFP) features are no longer maintained and will be retired in a future release.

Planned retirement of robot samples

In a future release, the robot samples available in the installed machines folder of the NX installation will no longer be maintained. While you can still use them, there will be no updates and no issues will be corrected. New and updated samples will be made available on **Post Hub** in a future release.

Retirement of Machining Knowledge Editor (MKE)

The Machining Knowledge Editor (MKE) is retired in NX 2306. The functionality is available in the Feature-based Machining (FBM) Configurator.

Planned retirement of Machine Tool Driver simulation support

In a future release, the Machine Tool Driver (MTD) based simulation in NX CAM will be under critical maintenance. There will be no updates and no issues will be corrected. At that time, you can still

run machine simulation built on MTD technique, but Siemens Digital Industries Software strongly recommends that you update existing kits to the CSE or virtual controller based simulation.

Planned retirement of Post Builder application

In a future release, the Post Builder application will be under critical maintenance. There will be no updates and no issues will be corrected. At that time, you can still run postprocessors built in Post Builder, but we strongly recommend that you switch to the Post Configurator platform.

Verify

Currently, the **Verify** command is hidden by default.

You can use the **Show Verify (Legacy)** option in **Customer Defaults** to control the visibility as needed.

There are newer display and animation commands on the Ribbon bar to validate tool paths.

Planned retirement of Tool Path Based Simulation support for setups without a machine

Beginning in NX 2206, the **Tool Path Based Simulation** command will not animate the tool path if the CAM setup does not contain a machine.

Use the display and animation commands on the Ribbon bar when the CAM setup does not contain a machine.

Template part files

The following template part files were deleted in NX 1980:

- Die Mold (Essentials)
- Turning (Essentials)
- Machinery (Essentials)
- Multi-Axis (Essentials)
- Mill Turn (Essentials)

Retirement of 2D Dynamic module

In the **Tool Path Visualization (Legacy)** dialog box, the **2D Dynamic** tab is removed.

Retirement of NX Manufacturing Tutorials from all platforms

The Manufacturing Tutorials were retired in NX 1980.

The new *Manufacturing Quick Courses* replaces the existing tutorials. To view the *Manufacturing Quick Courses* content:

1. Launch NX.

   > Note:
   >
   > The *Manufacturing Quick Courses* tab is available on the *Discovery Center* when you launch the complete NX suite or the NX Manufacturing application.

2. Click the *Manufacturing Quick Courses* tab on the *Discovery Center*.

Retirement of Solid Edge CAM Pro Manufacturing Tutorials from all platforms

The Manufacturing Tutorials were retired in Solid Edge.

The *Manufacturing Quick Courses* content replaces the tutorials.

Planned retirement of Manufacturing Wizard Builder

The Manufacturing Wizard Builder is on critical maintenance. It is a plug-in for the Process Studio Author (PSTUDIO) application, which is no longer being developed. The 32 bit version is included in the NX installation, but will be discontinued in a future release.

The location in the Windows start menu has changed. Choose **Start→All Programs→Siemens NX→NX Manufacturing - Process Studio Author**.

There are no plans to discontinue CAM Wizards, which are xml files based on our block based UI architecture.

Point To Point planned retirement from all platforms

Point To Point is in critical maintenance. The **Drill** template has been hidden by default, but is still available.

Manufacturing documentation notes

Hands-on learning aids in the online help

The online help now includes a limited number of hands-on procedures with part files to download, which let you work through detailed instructions on your own. These hands-on procedures are a good learning tool to try out new functionality and complex workflows. The procedure topics are titled **Hands-on: <Procedure name>**.

> Note:
>
> Make sure you are not in the **Welcome** role when you work through a hands-on procedure.

Manufacturing caveats

General caveats

Toolpath Merge

The planar merge group has issues with **Parallel Generate**  .

Transition Path

When you generate transition path on instanced operations, the path could be generated incorrectly.

Mass Edit

When you perform a mass edit of parameters in multiple operations, some parameters cannot be updated in all operations and are updated only in the first operation selected for mass edit. For example, multiple stepover distance parameters, range definition parameters, and notes.

Allow Editing of Library Tools customer default

If you select the **Allow Editing of Library Tools** check box in the **Customer Defaults** dialog box (**Manufacturing→Tools→General→Libraries** tab), it may lead to an inconsistency between the parameters of the tool modified in the current session and the tool in the library.

Reflect Component

NX CAM does not support the **Reflect Component** assembly command. If you use any geometry from a reflected component for your operation, the results may be unpredictable.

NX CAM plans to support the **Reflect Component** command in a future release.

NX process

When you exit NX, it may close the UI without stopping the process. There is no data on how widespread this problem is.

Tilt Tool Axis

Problem	Workaround
Shortest 2D distance to curve For longer tool paths, the shortest distance calculation can become unsynchronized.	Use the **3D** shortest distance option.
The tilted tool path may have non cutting moves with lifts in regions where they are not needed, and the moves can cause gouges.	None

IPW

- When you combine long motions with very small cut depths, the IPW may not be displayed clearly in the graphics window because of shading issues. This display issue will be fixed in a future release.

- A **Generic Motion** (GMC) operation does not change the machine mode state of the in-process workpiece (IPW). Adding a **Machine Control** subop with a **Set Modes** event at the beginning of the GMC operation does not change the machine mode state of the IPW. The previous machine mode, either milling or turning, remains active.

  If the preceding operation does not have the required machine mode, you must add an operation with the required machine mode before the **Generic Motion** operation. For example, if the Generic Motion operation is for turning, add a turning operation. The new operation must generate tool path, but does not need to cut material.

  > Note:
  >
  > This is not a limitation in NX Machine Tool Simulation (NX MTS) when you use the CSE driver. The machine state is defined by the workpiece spindle setting, which is either turning or milling.

Milling caveats

Fixed Axis Guiding Curves operation - bull nose tools

A **Fixed Axis Guiding Curves** operation could have spikes in the tool path when all of the following conditions apply:

- The tool is a bull nose.

- The drive curve is not defined on the boundary of a cut area.

- The cut area includes transitions between steep and non-steep areas.

The spikes are caused by inconsistent contact points as the bull nose tool transitions between the bottom tooltip and the lower corner of the tool.

As a workaround, do one of the following:

1. Reselect the drive curve to define it from the cut area boundary.

2. Define the tool axis so that the bull nose tool can touch the cut area consistently.

3. Choose another bull nose tool that has a different lower radius.

Rotary Floor milling

Problem	Workaround
The **Min. Lead Angle** does not influence the tool path.	None

Multi-Axis Additive Manufacturing

Build Style Rules

Each rule must have a unique name. Duplicate rule names in the same file are not supported.

Planar Additive Thinwall

Multiple beads are supported only with the **Profile**  type of **Cut Pattern**. The **Helical**  type of **Cut Pattern** is not supported for multiple beads.

Medial Axis infill

Medial Axis infill is currently not intended for parts with junctions in the body, such as *T-junctions* or *Cross Junctions*.

Freeform Additive Buildup Gradient

The **Freeform Additive Buildup Gradient**  operation is only applicable to the laser additive deposition of metal powder.

Turning caveats

NX MTS tool-path based simulation

NX Machine Tool Simulation (NX MTS) tool path based simulation does not support the dynamic thread in-process workpiece (IPW). Material is not removed in this case.

Tool rapid IPW collision checks

In threading operations, checks for collisions between the tool rapid and the IPW are ignored.

Using Verify for 3D Dynamic material removal

Check Tool and Holder does not perform holder collision checks for helical moves during thread turning.

Performance regression

If you select an operation after a thread turning operation, you may experience performance regression.

Multi-channel Turning

The **Rough Turn Multi-Channel** operation has the following limitations:

- When you use the following settings, the retract movement of the primary operation can cause reduced spindle speed of the secondary operation.

  Main node→**Multi-Channel Turning** group

  Mode = **Merged**

  Feeds and Speeds node→**Spindle Speed** group

  Output Mode = **SMM**

- In the **Tool Settings, Axis & CutCom** node→**Tool Orientation** group:
  - You must set the **Tracking Point** option to **Automatic**. Do not use the default setting of **Rotate With Holder**.
  - When you set the **Reorient Tool Holder** option to **Fixed**, the fixed value is passed to the secondary operation. This can lead to an unwanted tool path for the secondary operation.

Hole machining caveats

Tapping operations

Drilling  and **Tapping**  are distinct operation subtypes. Although tapping cycles are available in the **Drilling** operation dialog box, we recommend that you do not use a **Drilling** operation combined with a tapping cycle. The tapping cycles will be removed from the **Drilling** operation in a future release.

In a **Tapping** operation, you can set feature geometry parameters, such as pitch, that output the required mom variables for tapping. If you use one of the tapping cycles in a **Drilling** operation:

- The operation will not contain the necessary feature geometry parameters and in-process feature volumes for tapping.
- You will have the legacy Point to Point output where the pitch is driven by feed rate.

Back Counter Sinking operation

The **Gouge Checking** option reports false gouges.

NX Machine Tool Simulation (NX MTS) caveats

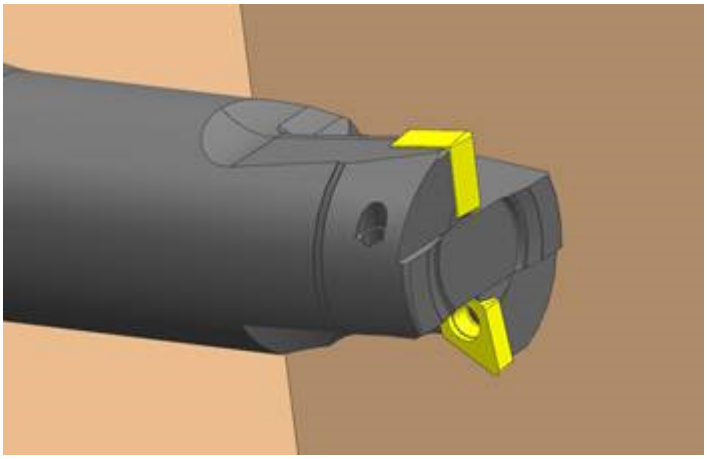
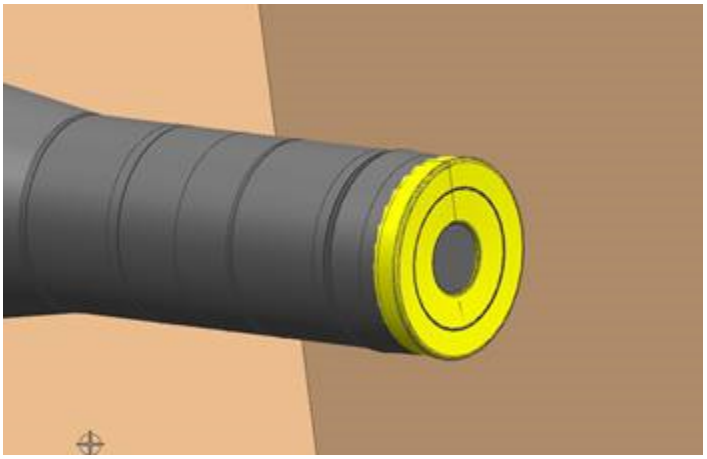
NX MTS tool-path based simulation

NX Machine Tool Simulation (NX MTS) tool path based simulation does not support the dynamic thread in-process workpiece (IPW). Material is not removed in this case.

Multi tool collision warnings

Multiple function tools (multi tools) can be used in both spinning mode for operations such as milling or drilling, and non-spinning mode for operations such as turning.

If the cutting portions of individual cutters on a multi tool overlap, the inactive cutter can cause misleading collision messages.

Non-spinning display	Spinning display
	

Simulation

- In certain cases the reported minimum distance of a clearance violation is not the closest distance.
- Selecting a location on the tool path is now supported in NX MTS for both CSE-based simulation and tool path simulation. Selecting tool path segments involving cycles, such as drilling cycles, may not work in some cases.

  There are also some cases in which the selected positions of rapid motions are not displayed correctly.
- When the simulation runs in the history buffer, and the spindle speed **Output Mode** for an operation is set to **SFM** or **SMM** to maintain constant surface speed, the simulation behavior can be unreliable.
- When you use the **Machine Code Based Simulation** option for synchronized operations, **Show Tool Path** is supported, but **Path Segment Selection** is not supported.
- When you use the **External Program Simulation** option, and you replace a machine, the software removes the selected external main program from the **Program Manager** dialog box. You then lose all changes in the edited main NC code program.

  As a workaround, save your main program before replacing the machine.

Virtual NC Kernel (VNCK)

- When it reaches a breakpoint, the simulation may fail to stop or may not stop at the exact position specified.
- VNCK 4.95.1.1 does not support *scissor kinematics* due to missing PathOutputBCS/BCS2 event messages. This issue will be resolved in future releases, but you may need to update the SRAM file.

- VNCK version 4.5.5 is officially supported by Windows 10.
- Although the VNCK versions 4.4 and 4.7.4 are not officially released for Windows 10, they usually run without any problems when you use **HMI Operate**.

If the VNCK does not boot after installing, please ensure that Windows Data Execution Prevention is not active for *vncksl.exe* and *vplc3172dp.exe*.

1. Press the Windows key.
2. In the search box, type **advanced system**, and press Enter.
3. In the **System Properties** dialog box, click the **Advanced** tab.
4. Under **Performance**, click **Settings**.
5. Click the **Data Execution Prevention** tab.

   If the **Turn on DEP for all programs and services except those I select** option is selected, you must exclude the *vncksl.exe* and *vplc3172dp.exe* files.

To exclude the *vncksl.exe* and *vplc3172dp.exe* files:

1. Click Add.
2. Select the *vncksl.exe* and *vplc3172dp.exe* executable files.

   The default location for both files is the VNCK installation folder.
3. Click **OK**.

> Note:
>
> You must exclude the *vncksl.exe* and *vplc3172dp.exe* files for all installed VNCKs.

VNCK - NCK Power On Reset

In the Human Machine Interface (HMI), if you execute an **NCK Power On Reset** (warm restart) while the simulation is stopped for any reason, the VNCK state and the simulation state in NX are not the same.

If you try to continue the simulation, or execute an **NCReset**, **NCStop**, or **NCStart** command, NX displays the following **Machine Tool Simulation** message and aborts the action.

```
VNCK executed an externally triggered 'NCK Power On Reset (warm
restart)' while being in 'SIM Stop' mode. This is an unsupported use
```

```
case which created an inconsistent state. Shutdown and reboot VNCK to
heal this.
```

If this happens you must shut down and reboot the VNCK.

HMI Advanced

- HMI Advanced is not supported, because it is not supported on Windows 10.

Show Machine Axis Positions dialog box

When the setup has a multi-function machine and you use the dynamic manipulator to change the tool axis for a fixed-axis operation, NX does not update the **Show Machine Axis Positions** dialog box. To avoid confusion, use the **Show Machine Axis Positions** customer default to suppress the dialog box.

1. Choose **File** tab→**Utilities**→**Customer Defaults**.
2. In the **Customer Defaults** dialog box, choose **Manufacturing**→**User Interface**.
3. Click the **Dialog Boxes** tab, and in the **Visibility** group, clear the **Show Machine Axis Positions Dialog** ☐ check box.

Post Configurator

DEF File Editor

The editor removes comments from the main definition file, including any existing comments.

TCPM prepositioning plane

When this option is activated, the plane output for the prepositioning is wrong in the following case:

- There is only one MCS in the **Geometry View** of the **Operation Navigator**, and
- The MCS is defined as **Local**, and
- **Special Output** is set to **None**.

Postprocessing tool paths

Postprocessing a synchronized group

To directly postprocess a synchronized group without entering the Synchronization Manager, you must use the default postprocessor registered with the machine. NX does not generate synchronized NC codes if you browse for a postprocessor.

On-Machine Probing

Product Notes

Parameters dialog box enhancements

The **Parameters** dialog box is updated to improve the probing cycles. You can:

- Edit multiple parameters simultaneously.
- Select a point that is associative to the part.

Dialog box enhancements

The dialog boxes are updated for more user-friendly performance.

The dialog boxes now include the following features:

- Prediction is available.
- User-defined entries (UDEs) are now supported.
- The dialog box can be customized.
- The dialog boxes now support API.

Preset Parameters

You can now preset all the parameter options using a central settings file. The file is stored at this default location:

UGII_CAM_PROBING_DIR\settings\settings.json

Use the settings file to:

- Filter and sort the displayed measuring cycles. This is specifically for a loaded machine or controller, or for a generic machine.
- Preset the parameters.
- Control the tracking point to use for tool path calculation. This depends on the machine side calibration.
- Preset and display the work offset parameters from G54 and G599 for Sinumerik and Renishaw.

The full documentation for **On-Machine Probing**  settings is stored at this default location:

UGII_CAM_PROBING_DIR\documentation\NX CAM On-Machine Probing-Settings Guide.pdf.

> Note:
>
> You can customize all settings according to individually definable conditions.

Automatic tool path generation

Now you can automatically generate a tool path each time you select a probing cycle or modify a parameter. The calculated tool path is available in NX for the current session and not only for a temporary preview.

Cycle Packages

- Several changes and error corrections are completed for the Sinumerik cycle and the Heidenhain cycle.

Documentation Notes

On-Machine Probing online help

On-Machine Probing is now available in online help.

Hands-on learning aids in the online help

The online help now includes a limited number of hands-on procedures with part files to download, which let you work through detailed instructions on your own. These hands-on procedures are a good learning tool to try out new functionality and complex workflows. The procedure topics are titled **Hands-on: <Procedure name>**.

Caveats

- The combination of an inch part with the metric tool, or a metric part with the inch tool, creates an incorrect tool path.

- Currently there is no setting to configure the machine specific combination.

Additive Manufacturing

Product Notes

Standard (3MF, STL and CLI) build processor

From NX 2406 onwards, the 3MF, STL, and CLI printers use the **Siemens Build Processor Server**. Siemens Digital Industries Software strongly recommends that you select the **Standard** build processor when you install the **Siemens Build Processor Server**. This provides full 3MF, STL, and CLI printer and build strategy configuration capabilities, including configurable output grouping, for all these printers.

Using NX Additive Manufacturing with the **Siemens Build Processor Server** but without the **Standard** build processor provides only the hard-coded **STL-File Printer – Parts and Supports** and the **CLI – Local File Printer**, and does not provide the 3MF printer.

Running NX Additive Manufacturing without activating the connection to the **Siemens Build Processor Server** from the **Customer Defaults** is useful when you only use one or more Materialise build processors, and it provides the extra hard-coded **3MF-File Printer** and the **STL-File Printer - Build Tray**.

These hard-coded printers will be removed in NX 2412.

Siemens Build Processor Server – installation upgrade

The dedicated Build Processor Server installation detects whether a previous NX 2312 installation exists, and upgrades to the latest NX 2406 release. It reuses the existing configuration and does not ask for any details.

> Note:
>
> You cannot install multiple versions of the build processor server on the same computer or VM.

The upgrade installation automatically restarts the Windows Services for the previously configured build processors. Build processor profile databases from the *%UG_APP_ADDITIVE_XBPS_DATA_DIR%* data directory are automatically migrated to the NX 2406 format and a backup copy is also created.

However, an EOS, EOS_PSW, Renishaw, and TruTops Build Processors installation upgrade does not restart the corresponding Windows Services. You must delete the *eos_db.xml*, *eos_psw_db.xml*, *renishaw_db.xml*, and *trutops_db.xml* files from the *%UG_APP_ADDITIVE_XBPS_DATA_DIR%* data directory and restart the corresponding Windows Services. This automatically creates new **_db.xml* files.

This procedure is also recommended when you want to update EOSPRINT to a new version, for example from 2.10 to 2.14.

1. Stop the Windows Service for the EOS build processor.

2. Delete the *eos_db.xml* file.

3. Change the *%UG_APP_ADDITIVE_EOSPRINTAPI_DLLS%* environment variable to point to the installation directory of the new EOSPRINT version.

4. Restart the Windows Service.

Printer types and printer instances

With the introduction of printer types and printer instances in NX 2406, the **Select 3D Printer** dialog box does not show your previous printers until you create new printer instances for them using the **Build Profile Library** navigator in NX Additive Manufacturing.

Siemens Build Processor Server installer

The **Siemens Build Processor Server** installer is available for download on the **Siemens Support Center** website. Click **NX→Downloads→Major Releases** tab→**NX 2406** version→**Additional Downloads** tab. For detailed instructions, see Install and set up Siemens Build Processor Server.

Installation executable files for Materialise BPI and BPs

You can download the *DemoBuildProcessor-x64.exe*, *HPBuildProcessor-x64.exe*, and *BuildProcessorInterface.exe* files from the **Siemens Support Center** website (**Downloads→Additional Downloads** tab→**NX Additive Manufacturing**). These files are removed from the *$UGII_BASE_DIR\mach\auxiliary\mfgam* location.

Additive Manufacturing Templates

If you created your own Additive Manufacturing template files (**File→New**) in an earlier version of NX, you must do this before you upgrade to the next NX version: copy the template files from the templates folder (`%UGII_CAM_BASE_DIR%\mach\templates\`) to a separate location to prevent the installation from overriding your defined templates.

After the upgrade installation is done, copy your template files to the `%UGII_CAM_BASE_DIR%\mach\templates` location.

Support Structure Profile Library am.xml data

If you have custom support profiles in your current NX version and you want to upgrade to a newer NX version, you must ensure that NX migrates your *am.xml* data to the upgraded version so that your custom profiles are available in the upgraded NX version. For detailed instructions, see Migrating the am.xml file.

Documentation Notes

Hands-on learning aids in the online help

The online help now includes a limited number of hands-on procedures with part files to download, which let you work through detailed instructions on your own. These hands-on procedures are a good learning tool to try out new functionality and complex workflows. The procedure topics are titled **Hands-on: <Procedure name>**.

Caveats

Renishaw build process – build strategy limitation

From NX 2406 onwards, Renishaw printers use the Siemens Build Processor Server. When you apply a build strategy that has different layer thickness values for parts and supports, it results in an error and the following message is displayed, "`Part/Support material layer thickness is not compatible with build material layer thickness or layer height.`"

This will be resolved in a later NX 2406 maintenance release.

Reichenbacher and SLM solution build processors

For Reichenbacher and SLM build processors, the following **Printer** parameters (**Build Profile Library**  **→Profile Parameters** panel) are either not yet supported or have known limitations:

- **Inch** (**Printer** parameter**→Platform** category)—Inch units are not yet supported. All build profile parameter dimensions are currently interpreted in metric units. For example, a hatching distance of 0.06 means 0.06 mm.

- **Scaling** (**Printer** parameter**→Scaling** category)—**Scaling** values are not yet supported.

Platform Filter Mode

In the **Create Supports** dialog box, the **Platform Filter Mode** options do not work for **Volume**, **Tree** and **Hybrid** types of support structures.

Dragging a part to remove it from a part group

Currently, you cannot remove a part from a part group by dragging it to the build tray. You can continue to use the **Remove** command on the shortcut menu.

Journaling and automation support

Currently, journaling and automation support is not available for defining mesh resolution per part in the **Build Tray** panel (**Mesh Properties** column configuration) of the **Additive Manufacturing Navigator**.

Print marks

- Support structure calculation on print marks is not currently working.

- Geometric bodies of print marks on all the non-nested parts get removed when you use the **Add Overflow Build Tray** command to add another build tray for the non-nested parts. Generate the geometric bodies for all print marks on the parts added to another build tray.

Support structures anchored to a lower part or user-defined support structures

> Note:
>
> Support structures are anchored to a lower part or to user-defined support structures only if you have set the **Prevent creating supports intersecting other parts in the build tray** feature toggle to **On** in the **Early Access Feaures** dialog box.

- If you use the **Pattern Supports** command for support structures that are extended to a lower part or user-defined support structures and pattern them to a part that does not have any lower part or user-defined support structures, the patterned support structures are extended only to the length of the original support structure instead of the build tray. Use the **Regenerate Supports** command to properly update the support structures.

- If you use the **Create Supports** command to add line and point support structures using the automatic support generation process, then the generated line and point support structures do not get extended only to the lower part or user-defined support structure. Instead the support structures pass through them to the build tray. Use the **Regenerate Supports** command to anchor the line or point support structures to the lower part or user-defined support structures in the build tray.

- If you use the **Move Part** command to move along the Z-axis of 3D printer an upper part that has an existing support structures anchored to a lower part or an user-defined support structure, the support structures are displayed as out-of-date in the **Additive Manufacturing Navigator**. Use the **Regenerate Supports** command to properly update the anchored support structures.

- If you use the **Move Part** command to move along the X or Y axis of 3D printer an upper part that has an existing support structures anchored to a lower part or an user-defined support structure, the support structures are not automatically updated or displayed as out-of-date in the **Additive Manufacturing Navigator**. Delete the support structures and create them again to generate the support structures according to the updated orientation of the part.

User-defined supports

When you move a part that has user-defined supports, the part is moved, but the user-defined supports do not move unless your user-defined supports reference part geometry, for example, edges or faces of the part.

Siemens recommends adding the part and its user-defined supports to a development build tray, and then adding this development build tray to the production build tray. This proves more efficient during

build preparation and printing, depending on the build processor, and ensures full support for moving and patterning parts.

Heat Sink support attribute

The **Heat Sink** attribute is currently not visible for gusset support structures.

CMM Inspection Programming

CMM Inspection - New Features

Line Profile tolerance with variable width tolerance zone

NX 2206

You can now create a line profile tolerance operation with a variable width tolerance zone that is defined by two curves. These curves will be nominal features only.

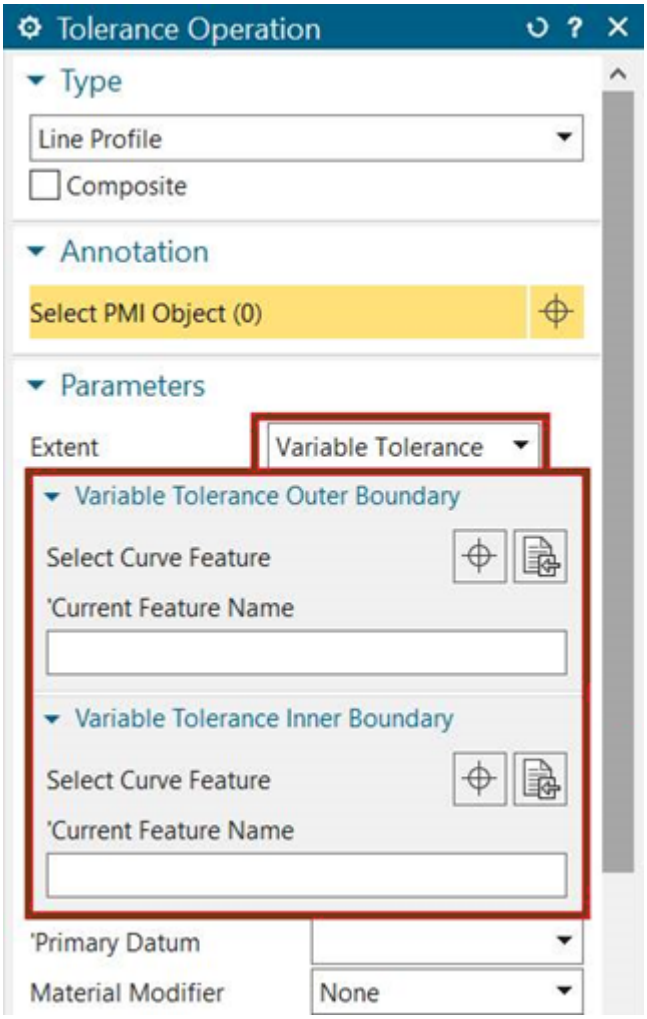
Example:

1. Choose **Home** tab→**Insert** group→**Tolerance**.

2. In the **Tolerance Operation** dialog box, set the following parameters:

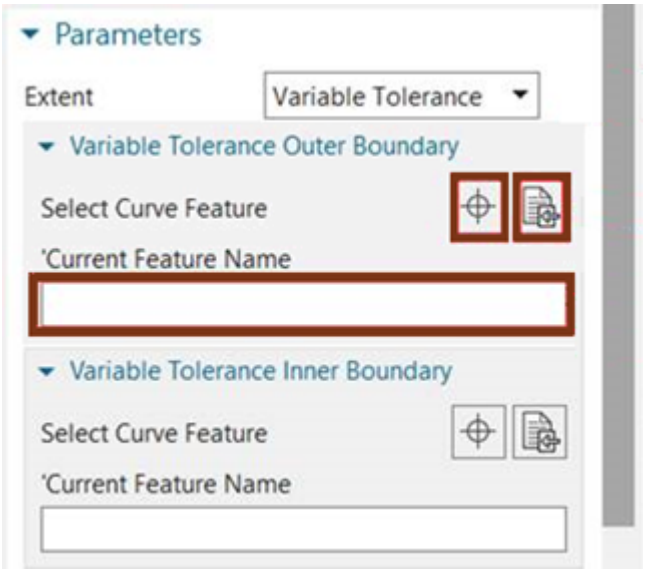
   - **Type** = **Line Profile**

   - **Extent** = **Variable Tolerance**



3. In the **Variable Tolerance Outer Boundary** group, specify the outer curve using one of three methods:

   - Click  and select the curve in the graphics window.
   - Click  to open a list of existing curve features and select from the list.
   - Type the name of an existing curve feature in the **Current Feature Name** box.



4. In the Variable Tolerance Inner Boundary group, specify the inner boundary in the same manner you specified the outer boundary.

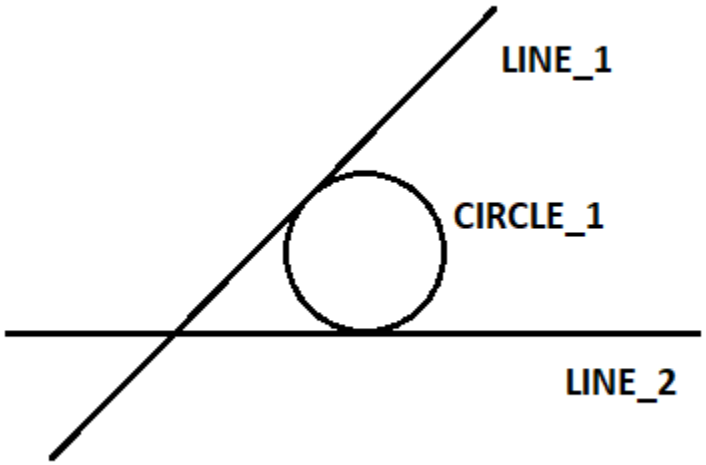
5. If required, specify datums and an operation name, in the same manner as you do when **Extent** is not set to **Variable Tolerance**.

6. Click **OK**.

Construct a circle from lines

NX 1973, NX 2007

You can now construct a circle of a specified diameter that is tangent to two lines.



A constructed circle of fixed diameter is typically used during alignment to simulate the gauge pin of a fixture.

Example:

1. Choose **Home** tab→**Insert** group→**Constructed Feature**  .

2. In the **Constructed Feature** dialog box, set the following parameters:

   Constructed Feature group

   Feature Type = **Circle**

   Construction Form = **Tangent To**

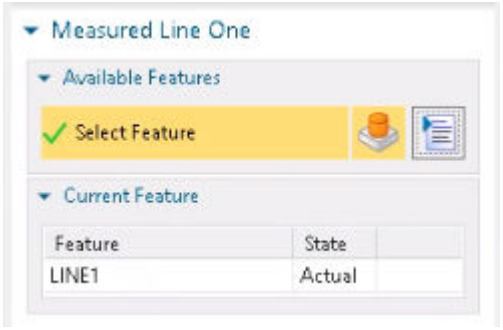
   Select Design Feature = **Create New**

   Design Feature Name = **new_circle**

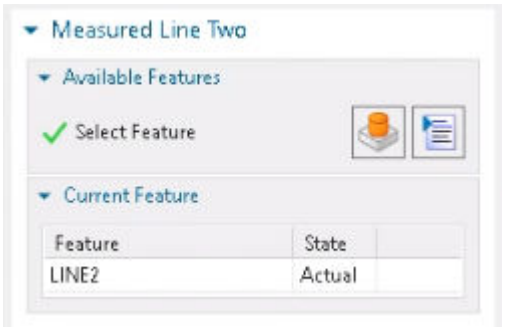
   Circle Diameter group

   Diameter = **0.55**

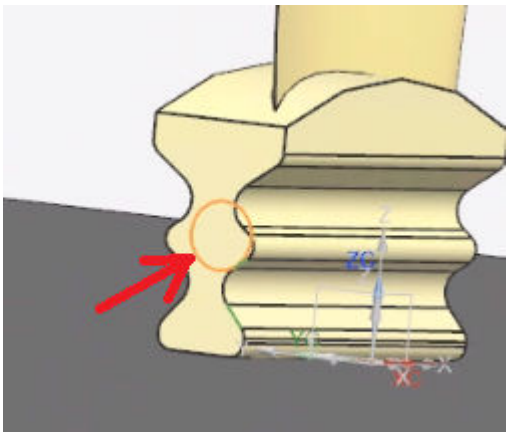
3. Select the first line.



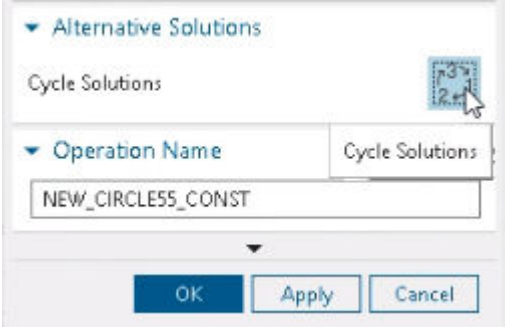
4. Select the second line.



5. View the computed solution.



6. Select the desired solution.



7. Enter an operation name.

8. Click **OK** to save the constructed feature.

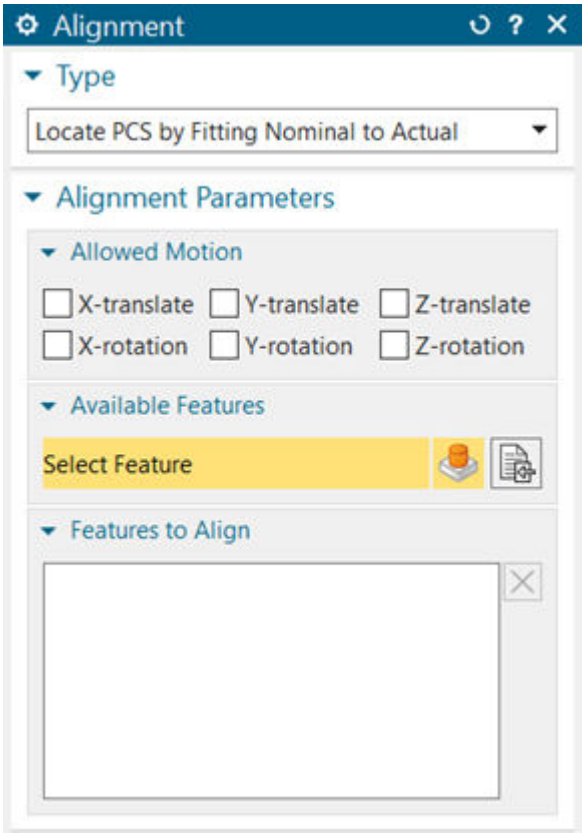
Alignment operation improvements

NX 1980

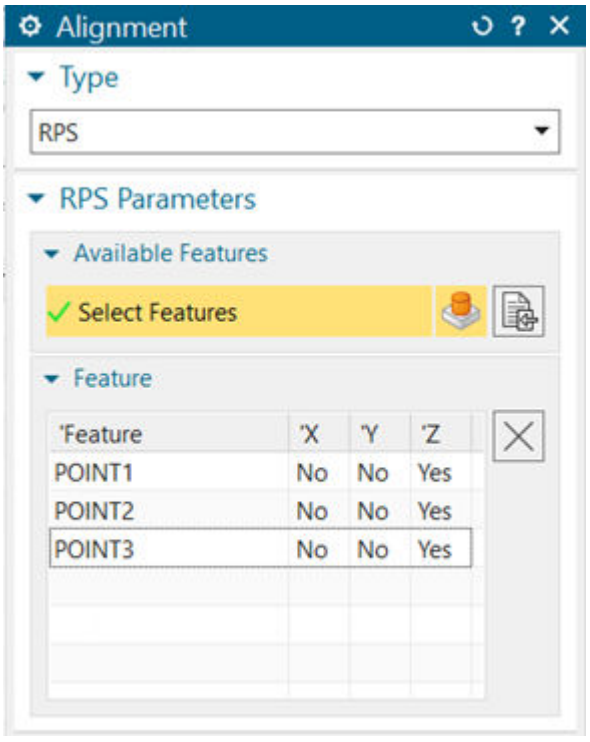
CMM now supports two new alignment types:

- **Locate PCS by Fitting Nominal to Actual**
- **RPS** (Reference Point System)

Locate PCS by Fitting Nominal to Actual	To create a locate alignment, do the following: 1. Select the check box ☑ for each motion that is allowed. 2. Select each feature to align. The features to align must be point reducible features. The exact capabilities of the operation are determined by your CMM controller.



RPS



To create an RPS alignment, do the following:

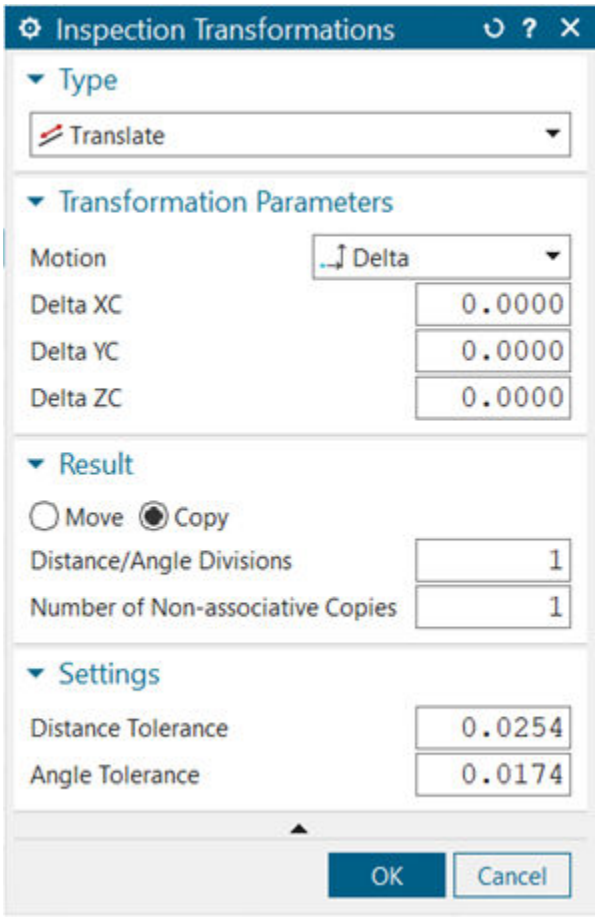
1. Select the features to align.

2. For each feature, set the direction values to **Yes** or **No** to specify which direction of motion the feature controls.

The features used for an RPS alignment are usually point features. The RPS alignment is typically used when aligning to a free-form surface such as the surface of a turbine blade. The exact capabilities of the operation are determined by your CMM controller.

Copy and Transform

NX 1980



You can now move or copy inspection operations using a transformation. NX CMM supports the following transformation types:

- **Translate**
- **Rotate About a Point**
- **Rotate About a Line**
- **Mirror Through a Plane**

To transform one or more operations, do the following:

1. In the **Inspection Navigator**, select the operations.
2. Right-click and choose **Object→Transform**.

From the **Inspection Transformations** dialog box, you can do the following:

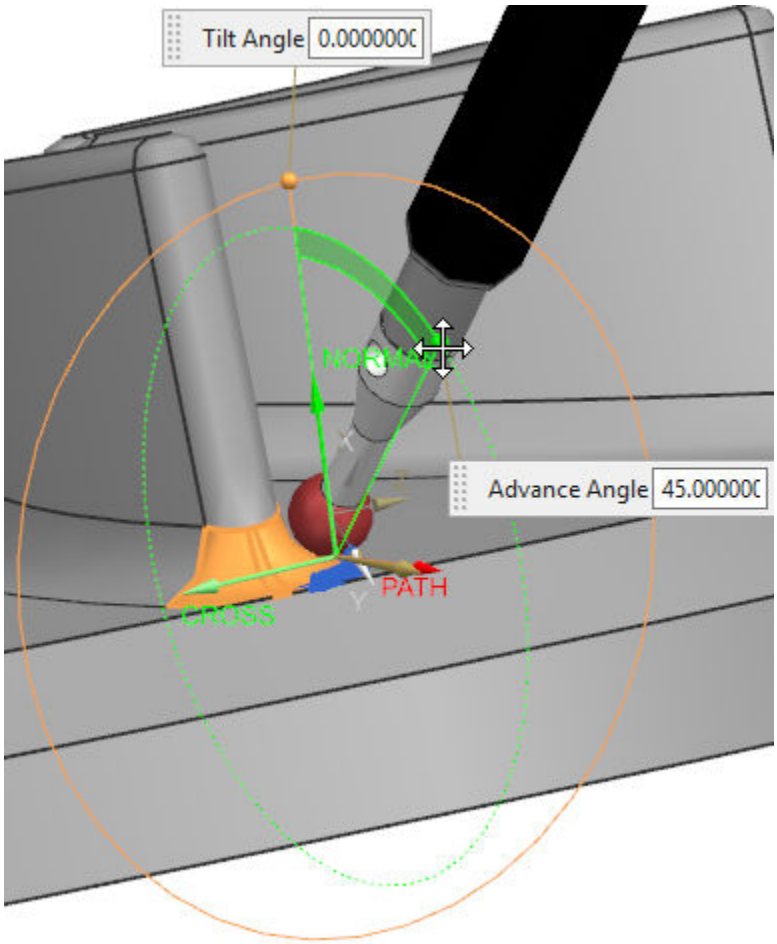
- Specify the transformation to apply.
- Choose whether you want to move the operations or make copies.
- Specify the tolerances for matching the transformed operations to the part model.

For additional information about transformation options, see *Transformations dialog box* in the *Manufacturing General* document.

Enhancement to 5-axis Scan Curve

NX 1953

You can now drag the probe for a **5-axis Scan Curve** sub-operation in the graphics window to change the probe angles. This enhancement makes it much easier to create 5-axis scan paths on complex surfaces.

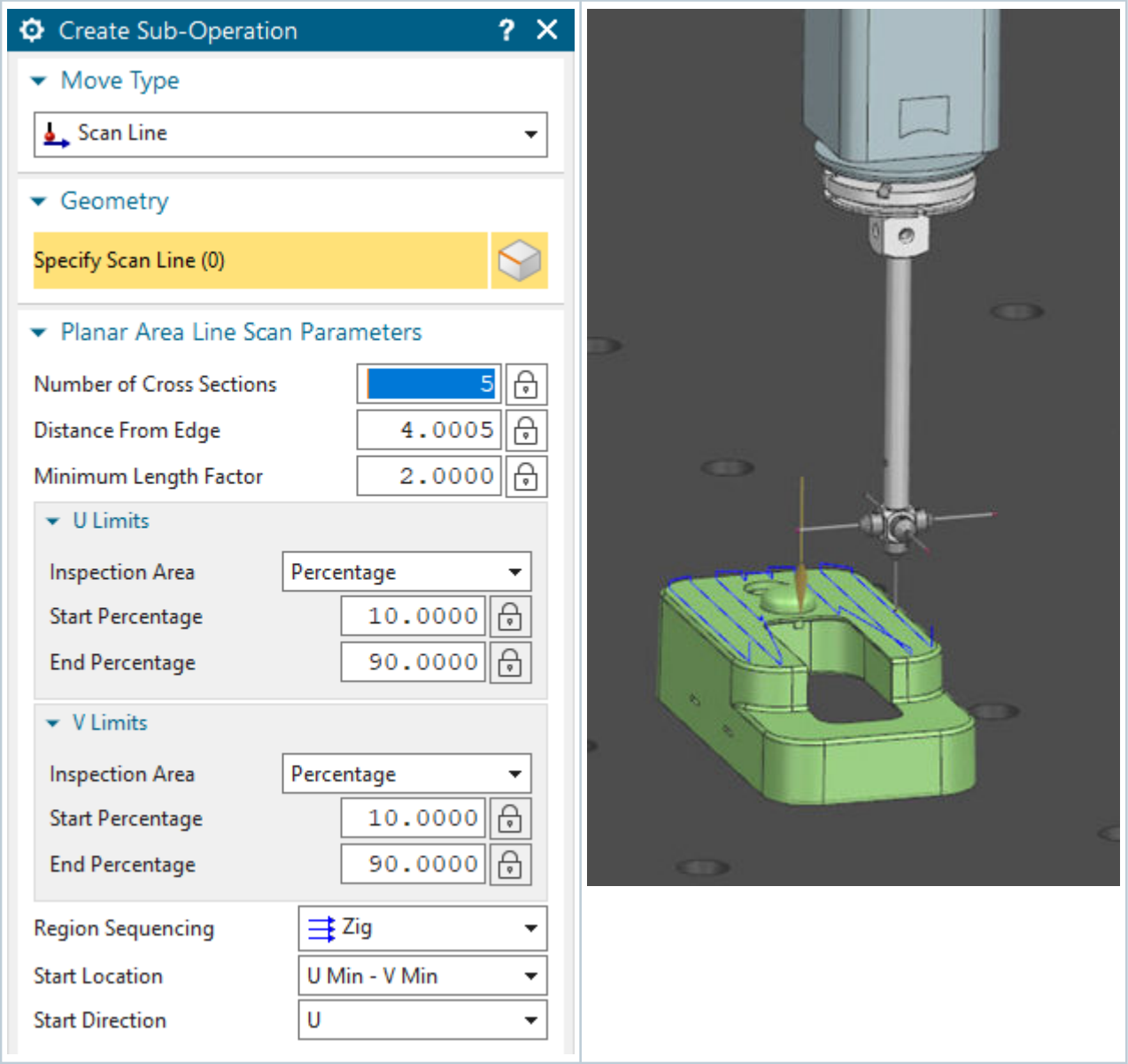


Scan lines on a plane

NX 1953

Use the new **Scan Line** sub-operation type to easily create scan lines on a planar face. Previous releases required using the Modeling application to manually create the lines for these type of paths.

The **Scan Line** parameters are similar to the U and V parameters for the **Point Set** sub-operation type.

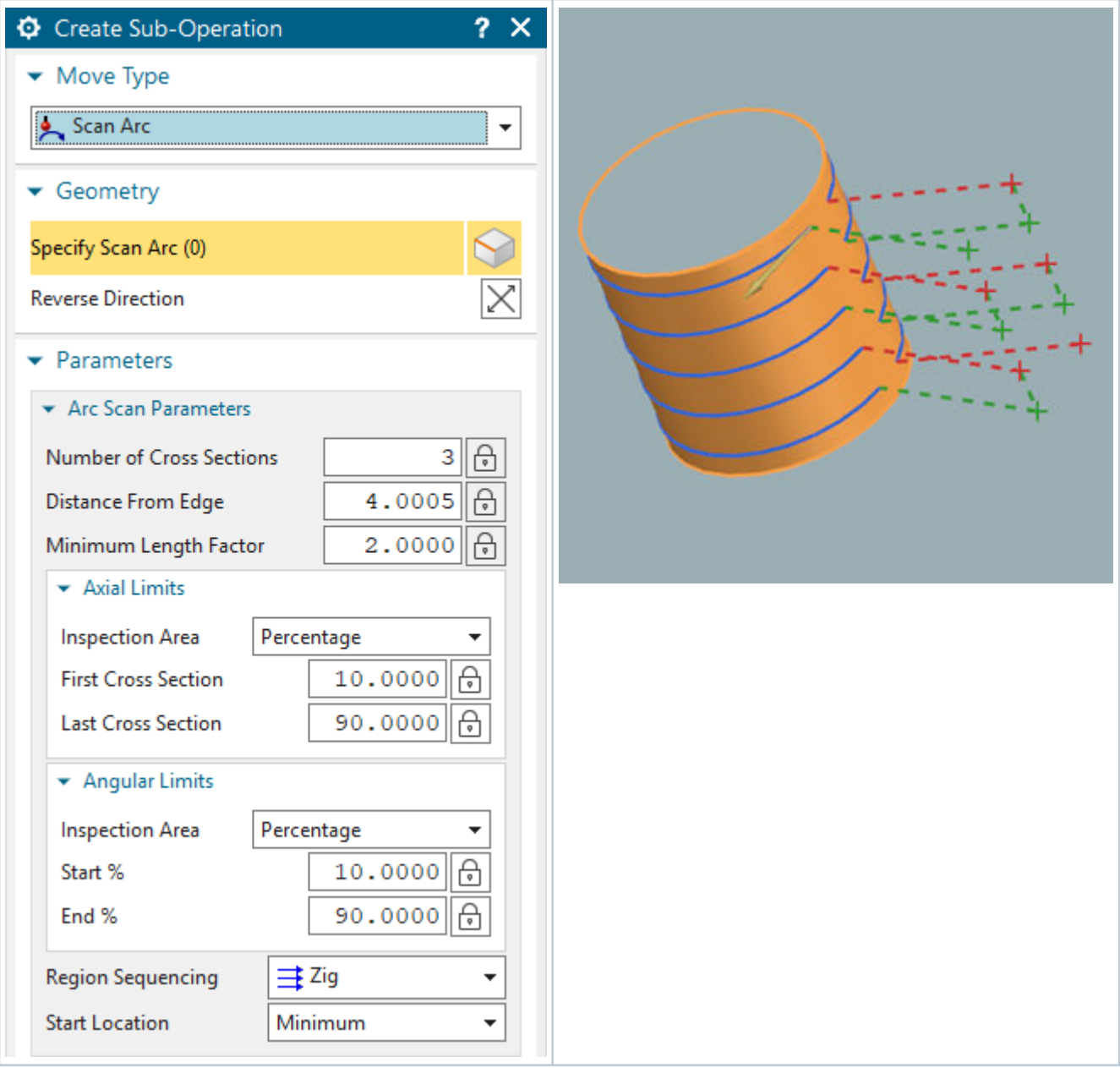


Scan arcs on a curve

NX 1953

Use the new **Scan Arc** sub-operation type to easily create scan arcs on cylinders and cones. Previous releases required using the Modeling application to manually create the arcs for these type of paths.

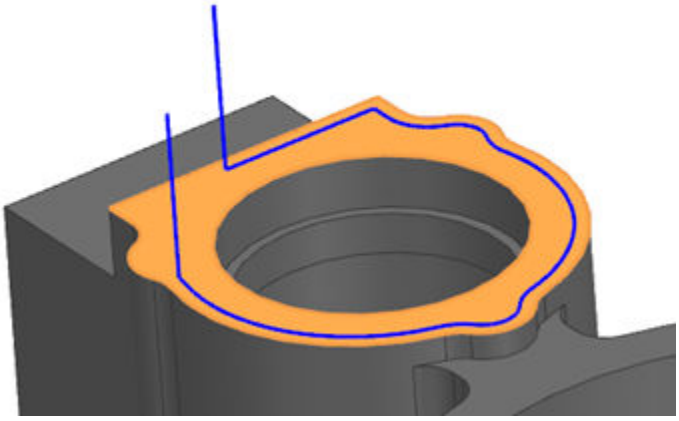
The **Scan Arc** parameters are similar to the parameters for the **Point Set** sub-operation type.



Scan the boundary of a plane

NX 1926

You can now easily create a scan path that follows the boundary of a planar face. Previous releases required using the Modeling application to manually create the boundary curves for these type of paths.



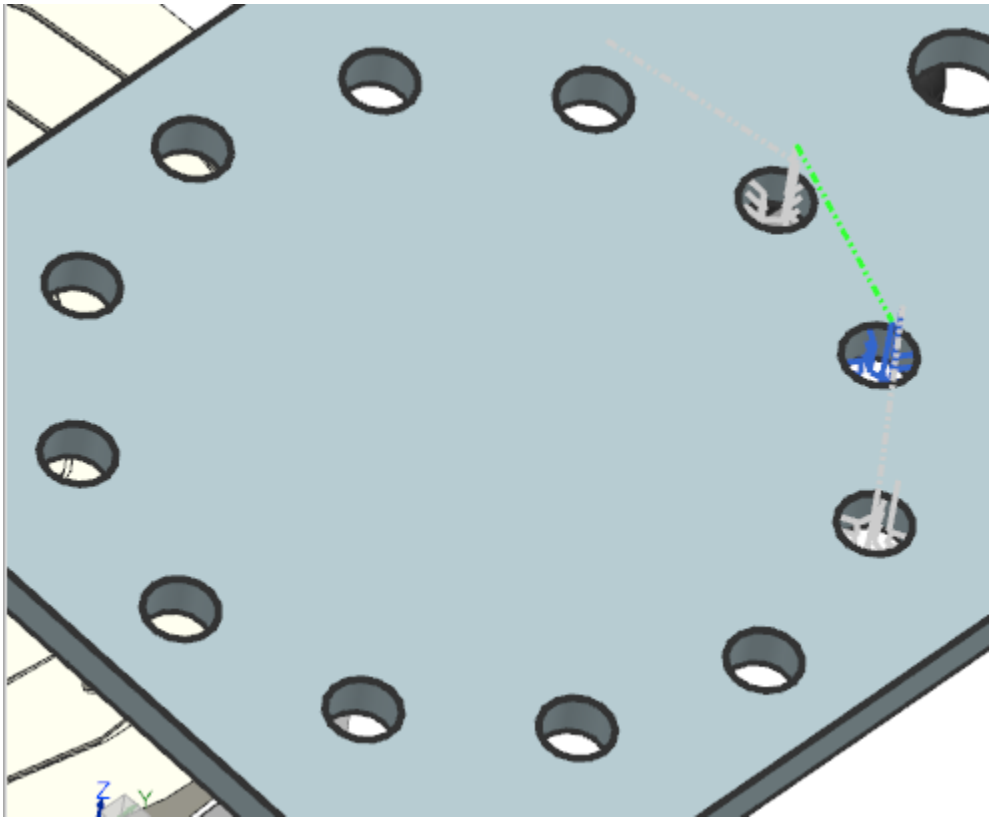
To create the scan path, do the following:

1. Choose **Home** tab→**Insert** group→**Inspection Path**.
2. In the **Inspection Path** dialog box, click **Select Feature** .
3. Select the feature to inspect.
4. Click **Add Sub-Operation** .
5. In the **Create Sub-Operation** dialog box, from the **Move Type** list, select **Scan Curve**.
6. In the **Geometry** group, from the **Curve Selection** list, select **Boundary**.
7. Specify the scan curve parameters, and click **OK**.
8. Click **OK** to save the inspection path.

Display Nearby Paths

NX 1926

NX can now display other paths in the CMM program as you create and edit an inspection path. In the following example, the path being editing is displayed in blue. The paths before and after the active tool path are displayed in gray.



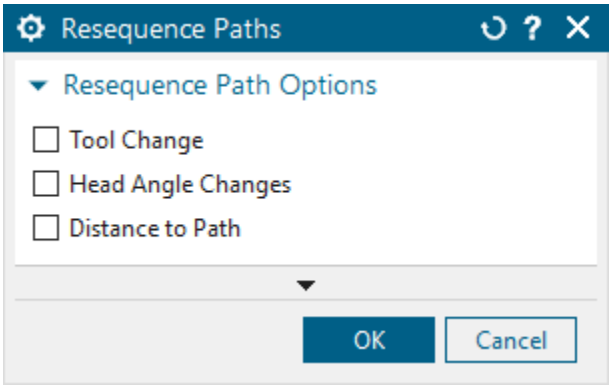
To change the path display, right-click an object in the **Inspection Navigator**, and choose **Path Display**, then one of the display options.

- **Show Selected Paths Only**
- **Show Nearest Paths**
- **Show All Paths**

Resequence Paths

NX 1926

The **Resequence Paths** command now has sequencing options.

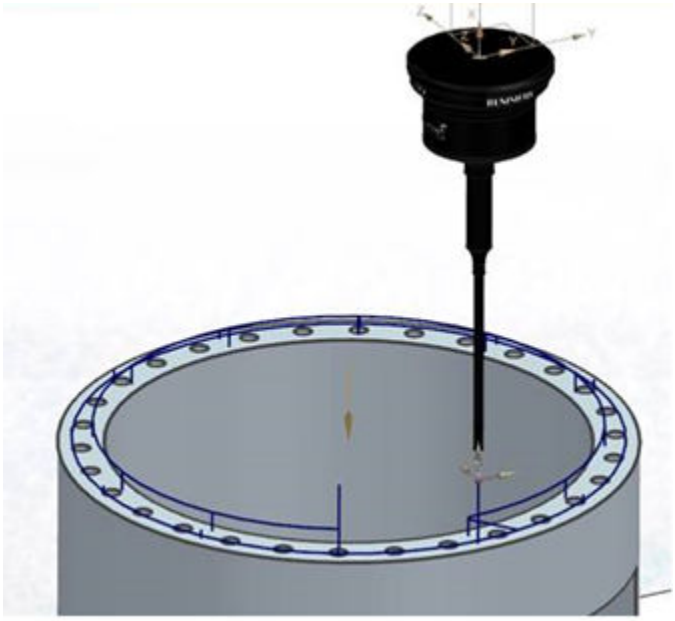


- **Head Angle Changes** considers head angle changes when resequencing inspection paths.
- **Tool Change** considers tool changes when resequencing inspection paths.
- **Distance to Path** gives the same results as previous versions of NX.

Circular Arc Transition Moves

NX 1899

When programming inspection paths for round parts, you can specify the Transition Motion as Circular. This will produce transition moves between inspection points that follow the curvature of the part.

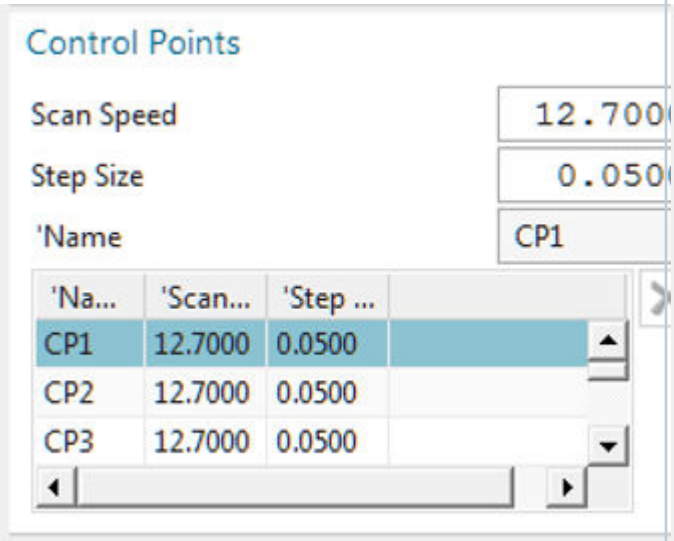


Curve Scanning Improvements

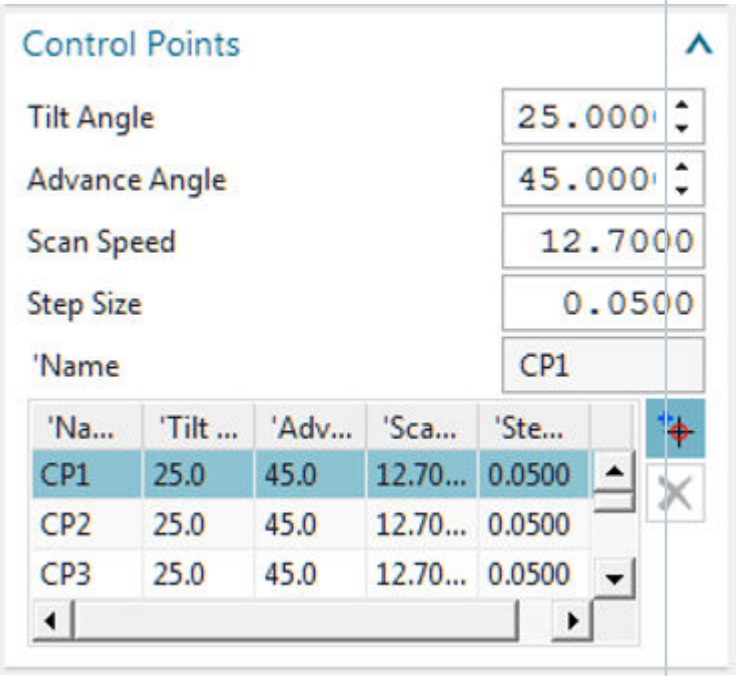
NX 12.0.2 MP12, NX 1899

A curve that is defined in NX using multiple segments can now be inspected as a continuous scan with control over the scan speed and point density for each individual segment. In the **Curve Scan** sub-operation dialog box, a list of control points is presented. The list is pre-populated with control points at the beginning and end of each curve segment. You can edit scan speed and point density at each control point.

3-axis curve scan



5-axis curve scan



These settings are implemented in the DMIS 3.0, DMIS 5.2, and MODUS (Equator) postprocessors using DMIS statements in the following format:

```
P(CURVE_SCAN_1)=PATH/CURVE,<point data for curve1>
P(CURVE_SCAN_2)=PATH/CURVE,<point data for curve2>
P(CURVE_SCAN_3)=PATH/CURVE,<point data for curve3>
PAMEAS/DISTANCE,<dist1>,SCNVEL,<speed1>,P(CURVE_SCAN_1),$
DISTANCE,<dist2>,SCNVEL,<speed2>,P(CURVE_SCAN_2),$
DISTANCE,<dist3>,SCNVEL,<speed3>,P(CURVE_SCAN_3)
```

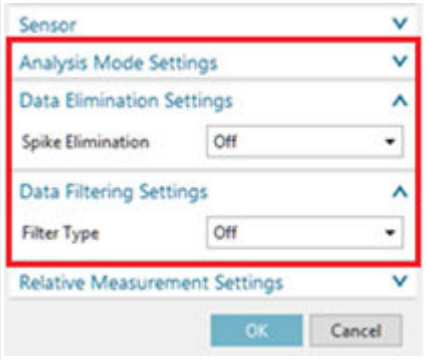
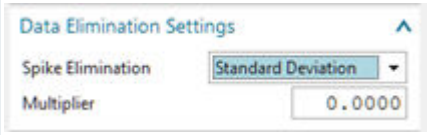
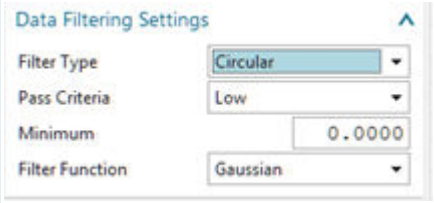
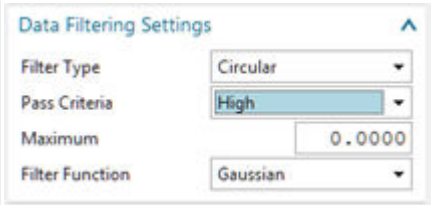
Other postprocessors do not currently support these settings.

Data Elimination and Data Filtering

NX 12.0.2 MP10, NX 1880, NX 1899

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 dialog boxes and the method dialog box.

Example	Meaning
	Data elimination and filtering are turned 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 postprocessor using a statement in the following format:

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

Other postprocessors do not currently support these settings.

Head Touch on Outer Circles and Cylinders

NX 12.0.2 MP10, NX 1880, NX 1899

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

NX 12.0.2 MP10, NX 1880, NX 1899

You can now verify inspection programs 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

NX 12.0.2 MP8, NX 1880, NX 1899

You can now perform arc scans 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 box using the new **Rotary Table** choice for **Scan Mode**.

This is supported by the DMIS 5.2 postprocessor using a statement in the following format:

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

Other postprocessors do not currently support this statement type.

Support for Valisys programs

NX 12.0.2 MP8, NX 1880, NX 1899

Inspection programs written using Valisys (eM-ProbeCAD) may now be imported into NX CMM with good success. Fewer than 10% of operations will need to be edited by hand. Support for the following statement types was added in NX 12.0.2 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 isn't a T() or F() format
- Valisys GEOALG formats
- GSURF/GCURVE Features
- Width Tolerance
- OUTPUT/R()
- TEXT/OPER
- ROTAB/
- SAVE/FA

CMM release notes

PAX file change

The .pax files are now located in the *UGII_BASE_DIR\cmm_inspection\templates* folder.

VALISYS critical maintenance and retirement notice

The VALISYS application is unsupported as of July 21, 2016. Use the NX CMM Inspection Programming application instead of VALISYS Programming and use CMM Inspection Execution instead of NX-Inspect.

Linking to PMI information

All geometry referenced by PMI must be in the current reference set, and if it is not, then NX:

- Issues warning messages in the information window.
- May still create features, but will not create paths because there is no access to the feature geometry.

Caveats

Renaming or deleting the SENSORS group

In the **Inspection Navigator**, do not delete or rename the **SENSORS** group as this may cause machine simulations within the **Inspection Path** dialog box to fail.

Machine simulation

When you create an inspection path or simulate a program, you may see the following `INI Programs` message:

```
In order to handle INI files in the Program Manager, define the channels
in the Machine Tool Builder.
```

You can ignore this message for NX CMM Inspection applications. It has no effect on the machine simulation.

Manufacturing Planning

Line Designer

Product Notes

Template files

To work in Line Designer managed mode, you must import the standard part files delivered with Line Designer into the Teamcenter database, so you can access these files in the **New Item** dialog box. If you do not perform this step, the **New Item** dialog box displays only blank templates.

A setup script for installing Line Designer templates in Teamcenter is included with NX:
%UGII_BASE_DIR%\MANUFACTURING_PLANNING\templates\tcin_linedesigner_template_setup.bat

For information about how to use the script, see *Installing File New templates* in the NX help at the following location:

Home→
Teamcenter Integration for NX→
System Setup/Administration→
Server Setup/Administration→
Installing/creating/modifying templates→
Installing File New templates

Platform support

Line Designer is currently supported only on the Windows platform.

Data Upgrade

Due to changes in the connector design in NX 11 and NX 12, equipment that uses connectors created in the previous versions should be upgraded to NX 12.0.2 MP5 or later format.

For more information, see *Using library components with connectors from NX 12.0.2 MP5 or earlier* in the Line Designer caveats.

Manufacturing Planning Additions

Manufacturing Planning Additions includes Teamcenter extensions to support the following:

FactoryCAD Library Migration	Migrating libraries from FactoryCAD to Line Designer also requires a Teamcenter extension to support new required dataset types. This extension is included in the Manufacturing Planning Additions package to Teamcenter installation (*Manufacturing Planning Additions 2402*).

Find the *MfgPlanningAdditions x.x* package on **Customer Support**. Log on using your webkey credentials.

If you need to request credentials, see **WebKey Account Management** in the **Support Services→Global Support Resources→Support links** section, on the **Support page**.

In Support Center, find the ManufacturingPlanningAdditions package in the following section:

Support Center→
Teamcenter→
Downloads→
Additional Downloads→
Manufacturing Planning Additions→
Select a version

From the **Select a version** list, choose **Manufacturing Planning Additions 2402**, then click **Manufacturing Planning Additions 2402**. From the **Filter by OS/Language** list, choose **Windows** or **Linux**, and search for the correct version for your server configuration.

Find instructions for applying the Manufacturing Planning Additions in the file *Teamcenter_Manufacturing_Planning_Additions 2402.pdf*.

Factory resource samples

Installation of the sample factory resources, conveyors, and robots from the manufacturing resource sample library to Teamcenter is highly recommended.

Factory Resources

Factory Conveyors

Factory Robots

Instructions for installing the samples are located in the following section of the Teamcenter HTML documentation:

Home→
Installing Teamcenter→
Teamcenter server Installation on Windows→
Adding Features→
Installing and configuring the Manufacturing Resource Library

Manufacturing Resource Library version compatibility

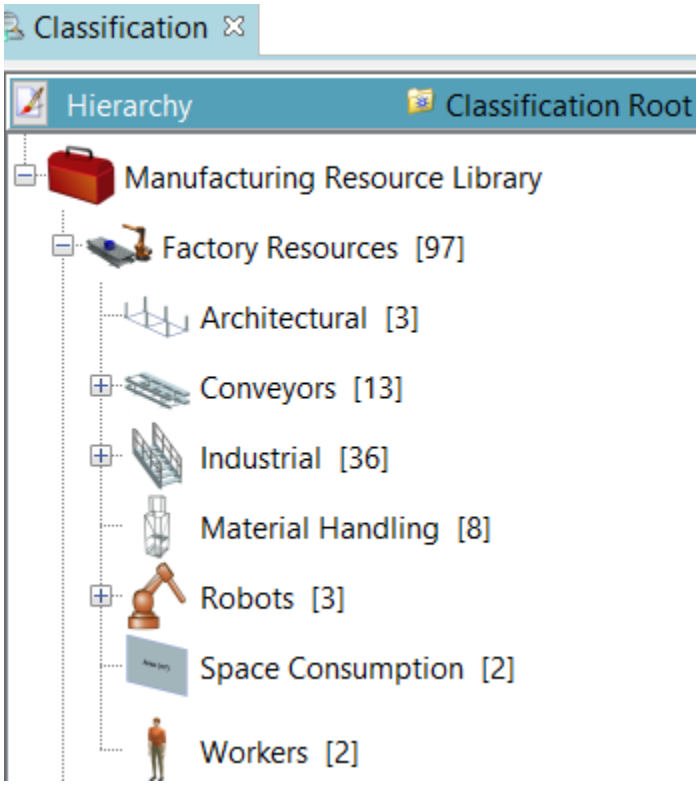
Line Designer supports the use of factory resources with compatible versions of Teamcenter and Manufacturing Resource Library (MRL). You should find the latest version of MRL in the latest Teamcenter release.

> Note:
>
> You should always upgrade the Manufacturing Resource Library components from the previous release to the new release.

Manufacturing Resource Library and Factory Resources

Line Designer supports the use of three types of Factory resources,

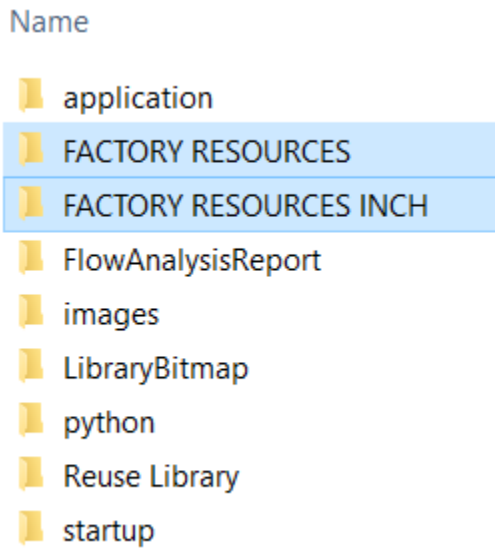
- Manufacturing Resource Library (MRL) – Installed using MRL. For Managed mode, Teamcenter has predefined classification structure. In Teamcenter, you can import and classify the resources, map the attributes, and so on.



- The **FACTORY RESOURCES** and **FACTORY RESOURCES INCH** folders are available in native NX with the NX installation directory that contains the resource from the MRL setup.

  The location of the file = *~\kits\MANUFACTURING_PLANNING\line_designer*

  > Note:
  >
  > Files in MRL and FACTORY RESOURCES are mostly the same, but vary depending on whether they are in the NX managed and native mode. MRL files are in the TC installation directory, whereas the FACTORY RESOURCES and FACTORY RESOURCES INCH files are in the NX installation directory.

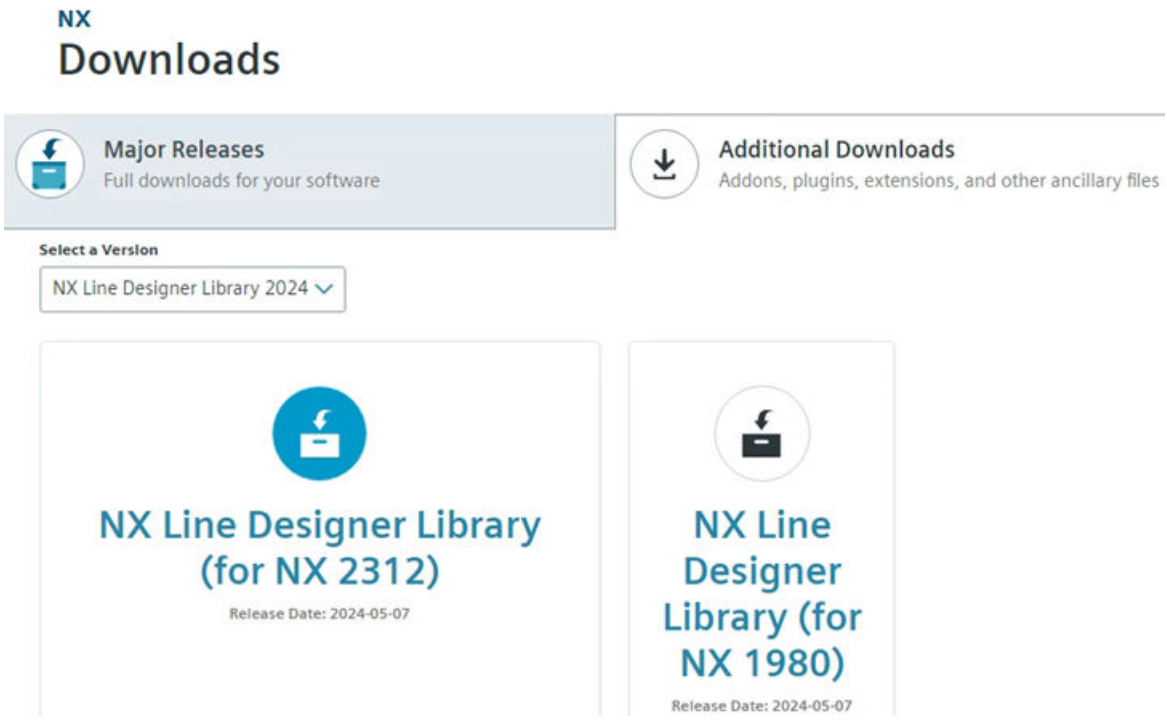


- Additional Factory Resources – In Support Center, find the NX Line Designer Library package in the following section:

  Support Center→
  NX→
  Downloads→
  Additional Downloads→
  NX Line Designer Library→
  NX Line Designer Library XXXX→
  NX Line Designer Library (for NX XXXX, where XXXX is the NX release number)

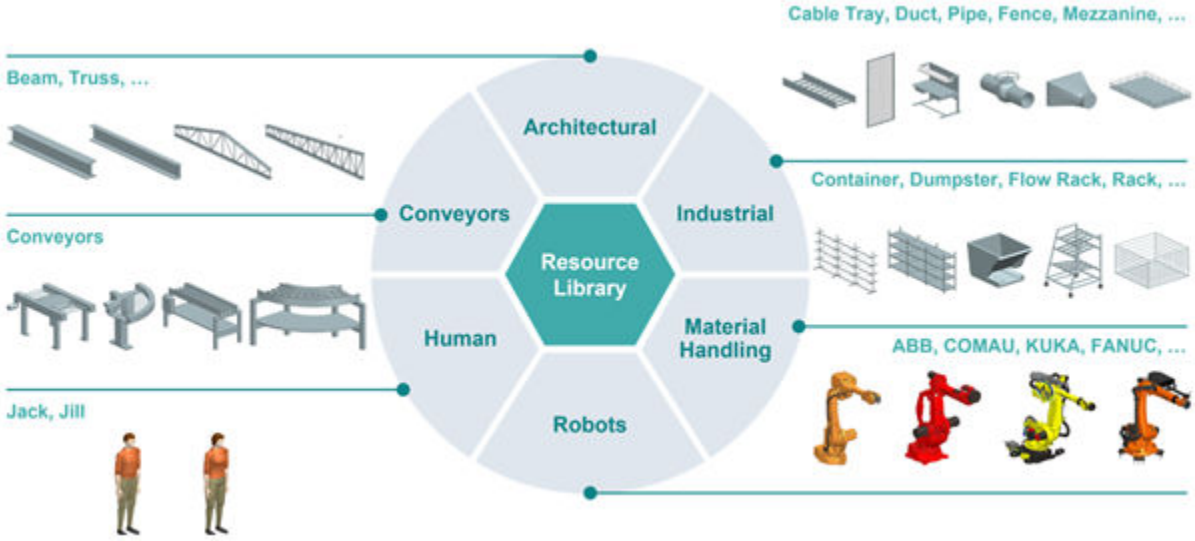
  Where XXXX is the latest release number. For example, 2024 or 2023.

NX Line Designer Library contains the information mentioned in the below image.



Overview of NX Line Designer Library

- Configurable Parametric Resources made up of Product Template Studio and Part Families
- 149 – PTS parts
- 86 – Standard parts (robots and its controllers)
- 19 – Part Family templates
- 1791 – Part Family members
- Available in Imperial / Metrics units
- Available in NX native and managed mode
- Resource authored with standard practice
- Geometry categorized into layers
- Includes Reference Sets (2D_TOP_VIEW)
- Includes connectors, kinematics (If relevant)
- Includes Plant Simulation Class (If relevant)



Saving the connection information in Teamcenter

By default, NX saves the connector connection information in Teamcenter, using the environment variable:

LD_DO_NOT_SAVE_CONNECTION_TC = 0

To save the connector connection information locally and not in Teamcenter, set the environment variable:

LD_DO_NOT_SAVE_CONNECTION_TC = 1

> Note:
>
> You must always set the variable for all users, to save the connector connection information locally and not in Teamcenter.

Paste duplicate of the components

To use the **Paste Duplicate** of the components in NX in Teamcenter 12.3 and later, set the Teamcenter preference:

ME_defer_save_in_clone = false

Overview of Minimal loading of plant structures

- Loading of parts with connectors in minimal load is supported from NX 2312 onwards only when you save the connectors in 2312.

- In assembly structure, when you load the parts with connectors, Line Designer loads them minimally only when you save them in 2312.
- When you load an assembly having write access parts with old connectors, NX modifies these parts for connector conversion. NX displays a message to the user after loading.
- When you drag and drop the Reuse Library components with access, NX convertes them.
- From 2406 onwards, parts with connectors and the immediate parent node which contains resources that are inter-connected using connectors are loaded minimally.
- In the managed mode, parts with kinematics are loaded partially, and part family members are loaded fully.

Documentation Notes

Note the following about the Line Designer documentation:

- In the release notes, you can find Line Designer in the **Manufacturing Planning** section.
- Line Designer is now available to work in native NX. Please refer to the **Line Designer Help** documentation for more details.

  > Note:
  >
  > End Items support is currently not available in Line Designer in native NX.

- You can find the Line Designer customer defaults in the **Installing Line Designer** section, in **Line Designer system administration**. Please refer to the **Line Designer Admin** documentation for more details.
- When you create a connection in **Line Designer** and view it in **Process Simulate**, you must add the early access feature, listed below, to the *features.txt* file in the **Process Simulate** install location.

  For example, Install location = *~\apps\PLM\Tecnomatix_x_x\eMPower\features.txt*

  Early Access Feature name = PS_ON_TC_GLOBAL_ATTACHMENT

Caveats

Configured components in the Plant Navigator

Configured out components from the plant structure appear as loaded components with no graphics in the **Plant Navigator**.

Using library components with connectors from NX 12.0.2 MP5 or earlier

While placing or disconnecting components in a layout, or while dragging components to another layout, you may experience incorrect behavior if those components have connectors created in NX 11 or earlier.

Recommendation Before using the library objects in NX 12.0.2 MP5 or later Line Designer layouts, you should upgrade any part files of the library objects that include connectors created before NX 12.0.2 MP5. To do this:

- Open the library objects in NX.
- Open the **Edit Connector** dialog box for each connector.
- Click **OK**.
- Save the library objects in the later version.

Creating occurrence poses

When you create occurrence poses for the components, NX shows no modifications for the parent component in the tree structure, in the **Plant Navigator**. Also, if the state of the component for which you create occurrence poses is **Released**, you may observe an issue that you can still create and save the occurrence pose.

Deleting the connected components in a layout

When you work in managed NX, and you load a plant layout and then delete any component that is connected to another component, NX deletes all the connections in the layout.

This issue is fixed in Teamcenter 11.6.0.10, Teamcenter 12.2.0.5, and Teamcenter 12.3 and later.

Study Manager

Currently, Study Manager in NX does not work with **Active Workspace** versions from 6.1.6 to 6.2.3.

Using load options for resources in the Plant Navigator

If you make any positional changes in the Layout in Process Simulate and load the layout in Line Designer with the Structure Only load option, the resources loaded one at a time in Line Designer reflects incorrect positions.

Modifying the kinematic components

In a kinematic component, when you jog the component joints with no **HOME** pose, NX displays the component as modified.

Paste-Duplicate and Unmount Tool command

When you paste duplicate the station with Robot-Tool mount relation, the **Unmount Tool** command does not work for cloned tool.

Load Line, Study Manager, and Structure Only load option

When you load a structure using the load line command or study manager in which resources are added, or child nodes are created, the newly created BOM lines are not visible when you open the plant node of this structure using **Structure Only-Minimal Load** option

Fixture Planner

Product Notes

Template files

The **Open** dialog box displays only installed templates. You must import the standard template part files delivered with Fixture Planner into the Teamcenter database, so you can access these files in the **Open** dialog box. Other templates are not configured to work with Fixture Planner.

A setup script for installing Fixture Planner templates in Teamcenter is included with NX:
%UGII_BASE_DIR%\MANUFACTURING_PLANNING\templates\tcin_fixtureplanner_template_setup.bat

For information about how to use the script, see *Installing File New templates* in the NX help at the following location:

Home→
Teamcenter Integration for NX→
System Setup/Administration→
Server Setup/Administration→
Installing/creating/modifying templates→
Installing File New templates

Platform support

Fixture Planner is currently supported only on the Windows platform.

Manufacturing Planning Additions installation

Using Fixture Planner with Teamcenter also requires you to install the Manufacturing Planning Additions package to the Teamcenter server installation (for example, Manufacturing Planning Additions 2402).

Find the *MfgPlanningAdditions 2402* package on **Customer Support**. Log on using your webkey credentials.

If you need to request credentials, see **WebKey Account Management** in the **Support Services→Global Support Resources→Support links** section, on the **Support page**.

In Support Center, find the Manufacturing Planning Additions package in the following section:

Support Center→
Teamcenter→
Downloads→
Additional Downloads→
Manufacturing Planning Additions →
Select a version

From the **Select a version** list, choose **Manufacturing Planning Additions 2402**, then click **Manufacturing Planning Additions 2402**. From the **Filter by OS/Language** list, choose **Windows** or **Linux**, and search for the correct version for your server configuration.

Find instructions for applying the Manufacturing Planning Additions in the file *Teamcenter_Manufacturing_Planning_Additions 2402.pdf*.

Caveats

Loading the study with assembly load options

Currently, Fixture Planner, does not support the **Minimal Load** option in the **Assembly Load Options**. We recommend that you select the **Partial Load – Lightweight Display** for optimal performance.

Assembly Line Planner

Product Notes

Platform support

Assembly Line Planner is currently supported only on the Windows platform.

Manufacturing Resource Library version compatibility

Assembly Line Planner supports the use of factory resources with compatible versions of Teamcenter and Manufacturing Resource Library (MRL). You should find the latest version of MRL in the latest Teamcenter release.

> Note:
>
> You should always upgrade the Manufacturing Resource Library components from the previous release to the new release.

Manufacturing Planning Additions installation

Using Assembly Line Planner with Teamcenter also requires you to install the Manufacturing Planning Additions package to the Teamcenter server installation (for example, Manufacturing Planning Additions 2402).

Find the *MfgPlanningAdditions 2402* package on **Customer Support**. Log on using your webkey credentials.

If you need to request credentials, see **WebKey Account Management** in the **Support Services→Global Support Resources→Support links** section, on the **Support page**.

In Support Center, find the Manufacturing Planning Additions package in the following section:

Support Center→
Teamcenter→
Downloads→
Additional Downloads→
Manufacturing Planning Additions →
Select a version

From the **Select a version** list, choose **Manufacturing Planning Additions 2402**, then select **Manufacturing Planning Additions 2402**. From the **Filter by OS/Language** list, choose **Windows** or **Linux**, and search for the correct version for your server configuration.

Find instructions for applying the Manufacturing Planning Additions in the file *Teamcenter_Manufacturing_Planning_Additions 2402.pdf*.

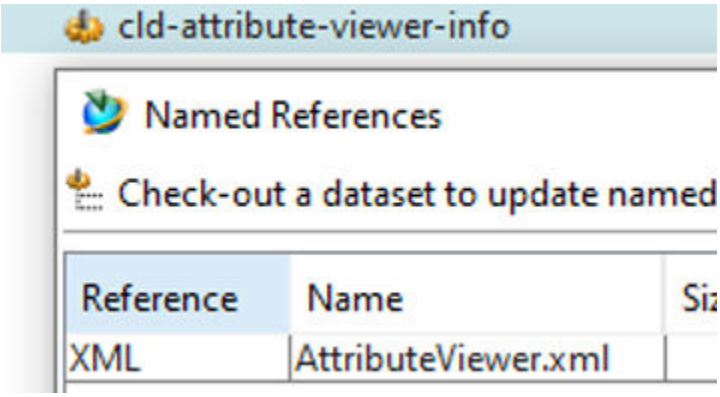
Caveats

Defining the different Teamcenter data structures

Assembly Line Planner depends on the recommended BiW data model. Please refer to the recommended data model section in the documentation when you set up your Teamcenter environment.

Setting up the Attribute Viewer

In Assembly Line Planner, you can use the new Attribute Viewer to review the attributes on an object of your choice. The definition of the Attribute Viewer needs to be done in a document saved as xml file named *cld-attribute-viewer-info*, containing the file *AttributeViewer.xml*.



Example of the syntax in the attribute viewer:

```
<?xml version="1.0" encoding="UTF-8"?>
<AttributesView>
    <Item type="Item" inherit="false">
        <Attribute name="object_name" category="Overview"
propertyOn="Item"/>
        <Attribute name="object_name" category="Overview"
propertyOn="ItemRevision"/>
        <Attribute name="object_type" category="Overview"
propertyOn="ItemRevision"/>
    </Item>
</AttributesView>
```

Data translation

Product Notes

SolidWorks to NX translator

You can now read SolidWorks 2024 files.

NX-to-JT translator

NX 2406 simplifies JT tessellation settings with an orientation towards getting the same display as NX. The JT tessellation settings in the default *tessUG.config* file shipped with this NX version reuse values from NX. The **Export JT** dialog box, the **JT Configuration** dialog box, and the *tessUG.config* file that produce a JT file from an interactive NX session or the command-line application **ugtopv** are now

preconfigured to use NX Resolution and automatically generate other LODs to optimize visualization as needed.

You may notice a change in density of the JT tessellation mesh compared to the previous versions. For some users, the tessellation fidelity and the related trade-off in the JT file size are important considerations.

You can customize JT tessellation by modifying the first level of detail (LOD) and allowing the software to automatically generate visualization LODs as needed. You can choose a different pre-configured NX resolution like **Fine** or **Coarse** or create a user-defined LOD with any of the individual LOD parameters described in JT configuration file—Levels of detail section. You can specify other LODs instead of letting the software automatically generate them. You can label LODs for different uses and for later identification.

LOD chordal parameters can be specified in absolute units, or as a percentage value relative to the entire scene. **Absolute** gives more predictable tessellation, preferred for analysis and inspection of individual parts. **Relative** helps to optimize visualization where fidelity of the smaller parts or details is less important.

The **Export JT** dialog box now has a preview window so you can see the result of tessellation settings before they are applied.

In the **Export JT** and **JT Configuration** dialog boxes, the options **Include PMI** and **Include Precise Geometry** are now in the **Data to Export** tab.

Note that all previously documented LOD parameters remain available using the *tessUG.config* customization if you wish to continue with manual settings or upgrade at a later date.

Updates for the default tessUG.config JT configuration file

- To ensure that you are using an up-to-date configuration file, we highly recommend using the *tessUG.config* configuration file included with NX and customizing it per your current settings.

- The level of detail (LOD) section in the default *tessUG.config* file is updated in NX2406. By default, the translator now uses the current NX resolution setting saved in the NX part to produce tessellation levels of detail in a JT file. In your workflow, if you are producing JT files using the default *tessUG.config* file, you may notice some difference in file size and number of facets in levels of detail (LODs) in a JT produced from NX2406 as compared to previous NX versions. If you want to tessellate the objects using your existing LOD settings defined with prescribed values of chordal, angular tolerance, you can take the default *tessUG.config* file contained in this release and modify the LOD section in the *tessUG.config* file as per your existing LOD settings.

- The `autoLowLODgeneration` configuration option in the **EAITranslator** section is removed and the `smartLODgeneration` option is enabled in the default *tessUG.config* file, which is contained in NX. Automatic generation of additional levels of detail is controlled by the `smartLODgeneration = true` setting in the configuration file.

- The value of the `chordalOption` configuration option in the **EAITranslator** section is set as `chordalOption = "PER_LOD"`. For the levels of detail (LOD) produced using prescribed parameters of chordal and angular tolerance values, this setting allows you to produce levels of detail as per the individual `ChordalOption` parameter set within the level. For example, you can specify `ChordalOption = "ABSOLUTE"` in LOD 1 to produce tessellation for geometrical accuracy and `ChordalOption = "RELATIVE"` in LOD 2 and LOD 3 to produce tessellation in these levels for visualization use cases.

CATIA V5 translator

You can now import CATIA V5-6R2023 SP2 files to NX.

ACIS translator

You can now read and write ACIS version up to 2024.

DXF/DWG translator

You can now read and write AutoCAD DXF/DWG version up to 2024.

Creo Import translator

You can now import Creo files up to Creo version 10.0.

Caveats

G3D translator

The G3D translator only supports plain triangle mesh (view with Type 0).

MCAD Translators

Solidworks import translator enhancements

If Solidworks part has different colour applied in assembly context than the part color, such overridden colour of the part is lost when Solidworks assembly is imported in to NX. The component is displayed in the base color of the Solidworks part in NX.

Support for imported PMIs

The following NX translators do not support the export of imported PMI objects:

- 2D Exchange
- DXF/DWG

Exporting convergent bodies

- The NX ACIS and CATIA V4 translators do not allow selection of convergent bodies.
- The NX STEP AP203, STEP AP214, and IGES translators do not support exporting of the convergent bodies. These translators add a message "*Convergent/Facet bodies are not processed*" to the log file.

Support for Teamcenter settings

The following translators do not support Teamcenter settings other than variant rule, revision rules, and effectivity:

- STEP AP203, AP214, and AP242
- IGES
- DXF/DWG
- 2D Exchange

Limited support for new line styles

The following NX translators do not support the new OOTB and shipbuilding line styles, and write them as Solid line style:

- CATIA V5
- IGES
- STEP AP203, AP214, and AP242

Internationalization caveat

File import or export by the following translators may not work if you set the NX temporary directory `UGII_TMP_DIR` to a folder containing non-locale characters.

- DXF/DWG
- IGES
- STEP
- 2D Exchange (export only)

NX to JT translator caveats

For NX parts that contain symbolic thread features:

If you associate a PMI object to a single curve of a symbolic thread feature in an NX part, it gets associated to all the curves of that symbolic thread feature in the JT file. If you select a PMI object that is associated to a single curve in thread feature PMI in the JT file in Teamcenter Visualization, all of the curves of thread feature PMI are highlighted.

Exporting line width applied on symbolic thread curves caveat

Different line widths applied on the curves of symbolic thread features are not exported to JT files. When viewed in Teamcenter Visualization, such curves are displayed with the **Thin** line width.

JT Support for raster images

If raster images are applied on the face of the object, you may notice some graphical interference of raster image with the face of a body on which it is applied while viewing the JT in Teamcenter Visualization.

Custom symbol presentation improvements in JT

In Teamcenter Visualization, orientation of custom symbols stacked with other PMIs may not match exactly with NX when view is rotated.

DXF/DWG translator caveats

DXF/DWG — Dimension export caveats

These caveats are applicable when you export a file using the **3D** option in the **AutoCAD DXF/DWG Export Wizard** dialog box.

- Dimensions associated with external references are exported as non-associative dimensions to the DXF/DWG file.
- NX Radius dimensions associated with ellipse or spline object are translated as AutoCAD block reference.
- The dimension associated between NX sheet object and View port object may be translated as overridden text of AutoCAD dimension.
- Narrow dimensions are exported as non-associative dimensions to the DXF/DWG file.
- Chamfer and Thickness dimensions are exported as block reference to the DXF/DWG file.
- Angular dimensions created with vector option are exported as block reference to the DXF/DWG file.
- Dimension with fits tolerance having fit tolerance style other than **Fit Symbol** is exported as block reference in AutoCAD.
- Dimension text location may not match with NX for the dimensions created with oriented text.
- Dimension line breaks and foreshortening symbols are not supported when dimension is exported as group or block.
- If a dimension text is on the extended dimension line, the exported dimension has to be updated to view any foreshortening symbols on it.

DXF/DWG — MText import caveats

You cannot import:

- MText paragraph tabs to NX.
- Euro symbol (created using %%128 in MText) to NX.

DXF/DWG — Architecture data import caveats

- You can not import:
  - View dependent data
  - Texture data
  - Corner windows
  - Wall cleanups
  - Body modifiers on staircase

- Associativity defined between the architecture objects is not maintained in NX. For example, door on wall will not move along if wall is moved.

DXF/DWG — Solids import caveats

You cannot import:

- Material, color, and transparency applied on solids in the DXF/DWG file.
- Solids data at location outside the NX supported bounding box.

DXF/DWG translator- Image translation caveats

- When you import DXF/DWG model data to NX drawing view, the translator does not support image translation.
- When you import DXF/DWG file that contains image rotated by angle which is not in a multiple of 90, the image is imported with zero degree rotation.

  Note:

  In NX, the images are rotated in steps of 90 degrees only.

- When you import DXF/DWG file with images having other than .jpeg, .tiff, and .png formats to NX, the translator does not support importing of such images.
- Images are not imported if exported as CGM.
- Image translation is not supported if the DXF/DWG file is imported to workpart.

DXF/DWG translator- 3D Workflow caveats

When you export View Breaks using 3D workflow, the translator does not support an export of the following:

- NX Object clipping and view break symbols.
- Geometry clipped inside a view boundary.
- A section line arrow in Break view.

Dimension export using 3D workflow

- When a dimension is exported as a block reference, the foreshortening symbol is not supported.
- 3D workflow does not support the inclusion of before/after appended text in the export of the inspection dimension. It always includes all the appended text in the frame.
- The translator does not support a tolerance separator in the output DXF/DWG file.

Text export using 3D workflow

- When you export text as lines using 3D workflow, the translator ignores the text symbol aspect ratio.

DXF/DWG translator- Associative dims to polyline and block caveats

You cannot import dimension as real associative in following scenarios:

- Blocks are imported as custom symbols or as a part and if dimensions are associated with block reference in the DXF/DWG file.
- Dimensions are associated to 2D and 3D polyline.
- Dimensions that have following Object snap points:
  - Tangent
  - Perpendicular
  - Nearest
  - Apparent Intersection
  - Parallel
  - Intersection
- Dimensions are associated to arc segment of the polyline.

DXF/DWG translator- Support for drafting objects caveats

You can not export:

- Drawings created in the **Layout** application using the **2D** option.
- Custom symbols used in a PMI table.
- Section views.

DXF/DWG Support for Hole table

If the multiple sections of the hole table are created on different drawing sheets, the continuation text will be lost.

DXF/DWG support for importing model and layout data

- You cannot import DXF/DWG model data and layout data to a drawing view while importing to a work part.

- When you import DXF/DWG files with model data to drawings with **Import Curves as Sketch Curves** ☑, it results in increased translation time.

2D Exchange caveats

2D Exchange – Dimension export caveats

These caveats are applicable for both, when you export a file using the **NX Part file** option in the **2D Exchange Options** dialog box or when you export a file using the **2D** option in the **AutoCAD DXF/DWG Export Wizard** dialog box.

Following dimensions are exported with the Override Dimension Text:

- Feature Parameter Dimensions
- True Length Dimensions
- Dimensions in scaled view and output set to **Modeling** (applies only when you export a file using the **NX Part file** option in the **2D Exchange Options** dialog box)
- Dimensions associated to:
  - Drafting Intersection point
  - Offset center point
  - Section line (in scaled view)
- The dimensions where associated object type changes in the flattened part. For example, circle projected as line.

Following Data will be exported as grouped geometry in the 2D part file and as a block in the DXF or DWG file.

- Linear and Radial callouts
- Retained dimensions
- Component level dimensions and PMI dimensions
- Inherited ordinate PMI dimensions
- Dual dimension in scaled views and output set to **Modeling**.
- Dimension with hole and shaft tolerance in scaled views and output set to **Modeling**.

- Dimensions associated with:
  - Blanked objects
  - 3D and Symmetric centerlines
  - Faces
  - Two object intersection (applies to ordinate dimension only)
  - Target Points
- Dimension created in plane other than view plane.
- PMI Partial Bolt Circle centerline.

2D Exchange translator- Image translation caveats

- 2D Exchange Translator does not translate image, if image is not projected on 2D. This happens when image is in XY plane and trimetric view is exported.
- 2D Exchange Translator supports translation of images only if input is being exported to Part File and not to IGES).

2D Exchange translator- Dimension export caveats

- When Dimension and a Leader are associated with the symmetrical centerline, the translator exports them as a grouped geometry.
- When a radial dimension is associated with a break view geometry, the translator exports it as a group geometry.
- When a perpendicular and angular dimension is associated with a break view section line, translator exports them as a group geometry.

2D Exchange translator - Foreshortening Symbol export caveats

A translator does not support export of foreshortening symbol when dimension is exported as a group in a 2D part.

2D Exchange translator – Leader caveats

A leader with an extension line is converted as a grouped geometry when it is associated with a spline in a 2D part.

2D Exchange translator – Weld symbol caveats

When you export a weld symbol containing arcs, the translator exports it as a grouped geometry.

2D Exchange translator – Support for drafting objects caveats

You can not export:

- Drawings created in the **Layout** application
- Custom symbols used in a PMI table.
- Section view while exporting Model views.

2D Exchange Support for Hole table

If the multiple sections of the hole table are created on different drawing sheets, the continuation text will be lost.

2D Exchange: Export to part

Drafting Datum feature symbol: The 2D Exchange translator does not support export of text formatting such as strikethrough, double strikethrough, and double underline, applied to datum feature symbol text.

DXF/DWG and 2D Exchange - Convergent body support

- If you export convergent bodies as polyline mesh, it exports only edges.
- Colors are not honored when you import 3D faces as convergent bodies or JT facets.
- Colors applied to face of convergent body is not honored on export.
- You cannot export model views with convergent bodies with 2D Exchange (or using 2D option on DXF/DWG export dialog box).

STEP translator caveats

- NX always exports kinematic information with respect to the absolute coordinate system. NX does not export kinematic information with respect to any other coordinate system that you specify.
- Line fonts for PMI objects are not supported for import or export.
- PMI objects in section views are not imported from or exported to STEP 203/214.
- Translating hidden components and objects:
  - When you export NX components as nested external references to STEP AP203 or AP214, the hidden components will not be marked as invisible.
  - When you export NX components as basic or nested external references to STEP 242, the hidden components will not be marked as invisible.
  - The translator does not export hidden objects at the assembly level, even if the `PROCESS_HIDDEN_OBJECTS` is set to `Yes`.

  - When you export NX data to a STEP AP203/214 file, objects on invisible layers will always be translated, even if the `PROCESS_HIDDEN_OBJECTS` is set to `No`.
  - When you export NX data to a STEP AP242 file:
  - ■ The translator excludes hidden points, even if the `PROCESS_HIDDEN_OBJECTS` is set to `Yes`.
    ■ The translator excludes hidden points and objects on invisible layers, even if the `PROCESS_HIDDEN_OBJECTS` is set to `Yes`.
- Kinematics defined between different bodies within single part cannot be exported to STEP AP242.
- When you import kinematics information from AP242 XML files, the translator names the joints as **Rigid Group**. To use such joints in simulation, you must rename them per your requirements.
- When you export or save NX data to an STPX file, the STEP translator does not export nested external references.
- When you export NX data containing PMI section views to a STEP AP242 file, the following are not supported:
  - Section views where only a few objects or components are clipped by a section cut.

    Note:

    If you export an NX part or assembly with this type of section view to a STEP file, then the output STEP is displayed with all objects or components clipped by the section plane when imported in the receiving system. This behavior is due to a limitation in the STEP schema as it does not support these types of section views.
  - Section plane display properties
  - Section view crosshatches
- When you export NX data to a STEP AP203, AP214, or AP242 file, the translator does not export NX layer categories.
- When you export data to a STEP AP242 file, the translator does not export:
  - Layer mapping and layer mask.
  - Color overrides for faces, wireframes, points, and coordinate systems.
  - Components and body colors, when you export components as external references.
  - Color overrides at the assembly level, when you export selected objects.

  - Color overrides at the assembly level, when you export NX files to XML (*.stpx, *.stpxz).

- The STEP AP203 and AP214 translators do not export and import assembly color overrides.
- Due to STEP schema limitations, chamfer dimensions from NX are exported as presentation data.
- When you import tessellated PMI from STEP AP242 files, vertices that create the PMI display are not optimized and this can result in an increase in the NX file size.

CATIA V5 translator caveats

- When you use the **Simplify** option in the **Import CATIA V5 File** dialog box, NX modifies the underlying geometry within a specified tolerance. This does not guarantee a result with continuity for all surfaces.
- The translator exports NX files to CATIAV5 R20 files.
- If an attempt to export a newly created part in Teamcenter Integration for NX mode fails, close and open that part file and re-run the translation.
- You cannot import CATIA V5 R7 and earlier version of files.
- Color is supported on a per face basis.

  Color is not supported when you export an NX convergent body or face to a CATIAV5 point cloud mesh.

- You can only import CATIA V5 “Lines and Curves” into NX using default **Linetype** and **Thickness** values.
- You cannot export NX parts with file name containing international characters.
- The CATIAV5 translator does not keep a log of failed export of password protected data during translation.
- During import, the translator will look for component files at and below any directories specified through Assembly Search Directories.
- The translator looks for the default settings file in the following sequence:

  1. Settings file specified using the d= option if you launch the translator from command line prompt.
  2. Settings file specified using the NX_CATIAV5_DEF environment variable.
  3. Settings file specified using customer defaults.

4. Searches for the settings file in the directory specified by the environment variable UGII_CATIAV5_DIR. If this is not defined on your system, you can set it to %UGII_BASE_DIR% \catiav5 on Windows or $UGII_BASE_DIR/catiav5 on Linux.

ACIS translator caveat

Heal Bodies has been enabled by default in ACIS Import to remove invalidities in the imported data.

IGES translator caveat

- When you import trimming surfaces to NX, the trimming curves that are outside the surface limits are not snapped to the surface boundary if:
  - Any portion of the trimming curves is outside the surface limits.
  - Any curve is outside the surface limits by more than the distance specified by the general **Distance Tolerance** Modeling preference.
- NX Drafting symbols are not exported.

IFC translator caveats

The IFC translator does not import or export assembly color overrides.

OBJ translator caveats

- The OBJ translator does not import or export assembly color overrides.
- The OBJ translator does not export vertex normals.

ACIS translator caveats

When you import ACIS files, few solid bodies from ACIS file may get imported as collection of sheet bodies, and sheet bodies with multiple faces may get split into multiple sheet bodies. This will also result in increased translation time.

Caveat for all translators

The password protected parts or assembly components are not exported to other file formats.

System administration

Plotting and Printing

Caveats

Mapping errors

When you configure UGII_TMP_DIR and other storage environment parameters to mapped drives or UNC paths, you may have issues with plot and print operations.

We recommend that you must set the value of UGII_TMP_DIR as a directory.

Example:

You can set the value to either a UNC or a drive\directory as follows.

UGII_TMP_DIR = \\hostname\directory

UGII_TMP_DIR = T:\directory

You must not set UGII_TMP_DIR to the root of the mapped drive.

Example:

Correct mapping

UGII_TMP_DIR=\\testhost1\Temp

UGII_TMP_DIR=T:\Temp

Incorrect mapping

UGII_TMP_DIR=\\testhost1\

UGII_TMP_DIR=T:\

You must ensure that the temp folder exists and you have write-access to it. For example, if UGII_TEMP_DIR=\\hostname\Temp then the folder \temp on the host "hostname" must exist and you must have write-access to it.

For mapped drives or UNC paths, you might get errors when you access folders on remote servers. This may happen when you use virtual configurations such as on the cloud. This may happen due to latency, timeout or sleep issue (such as when a storage device is configured to go to sleep after a pre-set period of inactivity or no access). If this happens, to troubleshoot, point them to an existing location on a local drive.

Legacy format libraries

The following legacy format libraries for the Plot feature are only available on Support Center. To obtain these libraries, contact Customer Support.

- *Abs-hpgl.zip*

  This contains *sdi_cgm_hpgl.dll* and *cgmhpgl2.dll* and is required for plotting absolute coordinate HPGL files.

- *Cals.zip*

  This contains *scals.dll* and is needed for plotting absolute coordinate HPGL files.

After you obtain these libraries, unzip the contents to *%UGII_BASE_DIR% \NXPLOT\bin*.

Mechatronics Concept Designer

Product Notes

Developing Inverse kinematics

If you use the **Inverse Kinematics** command to create a motion solution for 6 axis robotic applications where the last 3 joints intersect with one point you can:

- View and select poses from a list of potential solutions.
- Select a motion type to define the movement between poses.

Signaling with long integer data types

If you use the **External Signal Configuration** command with the **LInt** and **ULInt** data types to communicate you can only transfer the first 32-bits of the 64-bit integer or unsigned 64-bit integer, respectively.

You can use these data types for the following protocols:

- **OPC DA**
- **OPC UA**
- **PLCSIM Adv**

Industrial Electrical Design

Product Notes

When using NX Industrial Electrical Design with Teamcenter, you must set it up as a four-tier, client-server application with Teamcenter as the server.

Software requirements for Industrial Electrical Design in NX 2406

Product	Supported version	Notes
Teamcenter	• 13.3 • 14.1 • 14.2 • 14.3 • 2312 • 2406	If you are upgrading Teamcenter, see *Teamcenter Upgrade* in the Teamcenter documentation.
EPLAN	• 2.9 SP1 • EPLAN Electric P8® V2022 • EPLAN Electric P8® V2023	You must have an EPLAN runtime license EADN:Application 0193. Industrial Electrical Design supports only .zw9 files. Note: When working with EPLAN Electric P8® V2022 version 2020.0.3.347 or higher of EPLAN, a License Client is required.
TIA Portal	• 17.0.0 • 18.0.0 • 19.0.0	You must have a valid TIA Portal license. • STEP7 Professional Combo V16, V17, V18 • (Optional) STEP7 Safety Advanced Combo V17, V18, V19

Product	Supported version	Notes
		Note: If you want to export safety software blocks to TIA Portal, you need a valid license for this component. • (Optional) Drive Support: SINAMICS Startdrive V17, V18, V19 • (Optional) SINUMERIK Toolbox V17, V18, V19
ECLASS ADVANCED	8 or later	You must have a valid eCl@ss license. The terms of use of the eCl@ss standard state: This content contains eCl@ss. The use of the eCl@ss standard requires a license. Please register and order in the Download Portal.

Industrial Electrical Design supports the versions of Windows that NX supports.

Caveats

Installation and Configuration

- 4 part template files of NX Industrial Electrical Design in Teamcenter are outdated. If you use the PRT/PAX files in Teamcenter by setting the preference **TC_NX_FileNewPAXFiles**, these files need to be replaced with the correct versions from the installation of NX Industrial Electrical Design in the folder *%installation directory of NX%\automation_designer\templates*.

  This is fixed with the following patch levels of Teamcenter: 13.3.0.8, 14.1.0.7, 14.2.0.2, 14.3.

  If you have earlier versions of Teamcenter the datasets of the following item revisions need to be deleted and re-uploaded:

item Revision	Dataset content from
AME_DM_FileNewTemplate/A	AME_DM_FileNewTemplate.pax
AME_DM_GenericTemplates/A	AME_DM_GenericTemplates.pax
AME_FileNewTemplate /A (text)	AME_FileNewTemplate.pax
AME_FileNewTemplate/A (ugmaster)	AME_FileNewTemplate.prt

  (AD-47464)

- Creating Teamcenter BOM using Teamcenter versions before Teamcenter 14.3 on Linux servers fails. To fix the issue apply the following commands at the server:

  business_model_updater -u=infodba -p=infodba -g=dba -mode=upgrade -update=all -file=RemoveLOV.xml

  business_model_updater -u=infodba -p=infodba -g=dba -mode=upgrade -update=all -file=AddLOV.xml

  The argument files **RemoveLOV.xml** and **AddLOV.xml** are contained in the NX ELD client installation in folder [NX client installation path]\automation_designer\tc.

  (AD-47365)

- If you upgrade your existing ELD content to version 12, please note the following:

  The Java utility for ELD content fails in some cases where the proper java environment is not configured. Please check your java settings. As a workaround, you can run the import process manually following these steps:

1. Extract Contents.zip to a folder and ensure that there are no any spaces in the file path. The user performing this step must have administrative access.

2. Start Teamcenter command shell and go to path where you have saved aed0_* in Contents folder.

3. Run the following command in the Teamcenter shell.

   <aed0_content_updater> -u=<user> {p = password | pf = path-to-password-file} -g=<group> -location=<path to folder aed0Content that is unzipped from Contents.zip> -file= <aed0Content.csv File Path> -optionset=TIELLBCZImportOptionSetDefault -rd=no

   For example, in this installation process, we use the following command:

   <aed0_content_updater_14> -u=library_maintainer -p=library_maintainer -g=AutomationGroup.Engineering -location=<path to folder aed0content> -file=<path to file>\aed0Content.csv -optionset=TIELLBCZImportOptionSetDefault -rd=no

   > Note:
   >
   > Ensure that *infodba* is not the user that is installing the content. You must use the user name as <library_maintainer> who is in the relevant role as ELD library admin.

- In rare cases, after upgrading ELD Content to version 12, creating ELD project can report an error about a library mismatch and requires version 12. To fix this, make sure the library upgrade completed successfully. In the Classification Admin application in Teamcenter, edit the classification folder name, for example: Industrial Electrical Design [11.0.4], under Classification Root. Change the string name to **Industrial Electrical Design [12.0.1]** and save.

General

- Creating a new ELD project or item such as a product, symbol, or template, causes the error message *The preference does not exist in the database*. When creating a project, click **OK** in the message box, and continue working with the project. When creating a new item, restart the NX session, create an ELD project, confirm the error message, and then continue creating the item.

  You can install the NX feature Tooling Design & NX Join as a workaround for this issue.

  A fix will be provided with a maintenance update.

  (AD-48408)

Collaboration

- When you repopen an ELD project that is mapped to a mechanical assembly, an internal error occurs. If you work with an imprecise assemblies, do not upgrade NX. If you don't work with imprecise assemblies, disable the environment variable UGII_UGMGR_ENABLE_IMPRECISE_SRC_ITEM_ASSEMBLY.

A fix will be provided with a maintenance update.

(AD-48410)

- If you collaborate with a BVR assembly, and mapped components of the assembly are revised in a Teamcenter version older than 2312, the mapping is not propagated to the new revision. As a workaround, create a mapping to the new component.

  (AD-47807)

Automation

- The sdx-file import of CreateMyConfig 6.0 release version has known issues that block the transfer of the engineering result from Industrial Electrical Design to CMC:

  - The products S120M, CU310-2PN and CU310-2DP after export from Industrial Electrical Design are not imported successfully in CreateMyConfig.

    (AD-8763, AD-8779, AD-22967)

  - The encoder number value as defined in Industrial Electrical Design is not respected by CMC. You use the encoder number for establishing the mechanical connection between Motor and Encoder.

    (AD-8764)

Electrical Design

- The print processing does not downscale images that are inserted into table cells. Before you use a form sheet in a project, verify that any image in the form sheet is smaller than the table cell and it is printed correctly. If the image is larger than the table cell, the display on screen shows a downscaled version of the image, but the image will not be printed.

Aspect Name

- - With NX 2312 release, aspect naming is enhanced for explicit protection of aspect names. To make this available in search, every Engineering Object needs to have been loaded once. Fully expand each aspect navigator to do this.

  - If you display the property *Aspect Name set by* in a query, you need to change this to property *Naming rules apply*.

Programming Tools

NX Open differences between releases

You can determine what's changed for the APIs used in your NX Open programs and how it impacts existing programs.

NX Open APIs

The NX Open differences between releases and deprecated items replacements are published in **What's changed in NX Configuration and NX Open** in Support Center. On that page, select **NX Open APIs** and view the changes that have occurred for the release and any that might require changes to your existing code. The **What's New** tab provides new items for the release.

You can compare your release to another one to see the amount of change that has occurred between the two releases. In the **Comparing NX <your release> and** listing, select the release you want for comparison, and click **Compare**. You can get comparisons for up to two years of releases.

NXOpenReporter toolset

The **NXOpenReporter** toolset is a collection of command line tools to manage the migration of NX Open applications to the newer version of NX.

Use these tools to generate the reports of all the changed, deprecated, or deleted NX Open APIs, classes and types that are used in the given NX Open application. This helps you determine the effort that is required to update NX Open applications to a newer version of NX.

You can generate the following reports in a csv file format.

Overview report

Provides an overall status of the evaluated application, such as whether the application requires any rework or code change, any future work in case of deprecation, or if a recompile is enough to upgrade the application.

Detailed report

Includes a list of all affected NX Open APIs that are used in the application and the action that is required for each affected NX Open API. As a developer, you can use this report to migrate the NX Open application to make it compatible with a newer version of NX.

Usage report

Provides a list of all the NX Open APIs that are used in the evaluated NX Open applications or journals. This report is optional, and you can use it to find the most used NX Open APIs in an application.

Currently you can use the **NXOpenReporter** toolset to generate reports for NX Open APIs in .NET, C++, Python, and Java.

These tools help with the following:

- Better deployment planning when you upgrade NX by removing uncertainties regarding the NX Open customization readiness and robustness.

- Save the time that is required for pre-deployment readiness, development, and validation.

You can access and download the **NXOpenReporter** toolset in **NX Configuration and Customization Assistant** in Support Center. Click on the **NXOpen Reporter** tab for more information and download instructions. For more information about the toolset, see the user manual that is shipped along with it.

To use the **NXOpenReporter** toolset, you must have .NET 4.6.2 or later installed, and the **UGII_BASE_DIR** environment variable must be set.

Release upgrades

A primary goal of NX Open is to maintain your automation investments. This is done by adopting policies which minimize the amount of required code changes by you to migrate your applications to new releases of NX. The following contains a description of these policies and how they impact your ability to support the users of your applications, and the steps you should take to successfully move your application to new releases of NX.

NX Open API change policy

NX maintains the following three primary policies to protect your investments:

- **API stability**

  Stability ensures that the API that has been released will continue to work. This means that the API is still present in future releases and the behavior of the API doesn't change. API changes should be designed to minimize any changes to your source code. For example, if the capabilities of a method are expanded which requires new parameters, then NX may maintain and optionally deprecate the original method and add a new method that includes the new capabilities. By doing this, existing applications do not require code changes unless they want to take advantage of the new capabilities.

- **Binary compatibility**

  Compatibility means that applications that are built against a specified release continue to work with subsequent releases. This requires not only that API stability is maintained but also is binary compatible. For example, re-ordering the values of enum members so that API stability is maintained. In this case, the values have changed so existing automation programs will be using the wrong values. In other cases, deleting a method or modifying the inheritance hierarchy of a class will likely cause run-time errors for existing programs.

  The other aspect of binary compatibility is the compiler. When the compiler NX uses changes, this may break binary compatibility of automation programs. While some language bindings are more tolerant of being backwards compatible, such as Java, others are not as tolerant, such as C++.

  NX libraries will be forward compatible for all releases in a release family. If you compile and link your application with the libraries shipped with a functional release of NX, such as NX 1847, then your application should continue to run with all future monthly releases in that release family (NX 1847). This compatibility means you do not need to recompile, relink, and reship your applications to customers running various monthly releases of NX. Note that NX is not backwards compatible, which means you can't compile and link an application in NX 1847 and expect it to work in NX 12.

  NX will only change the compiler used (breaking binary compatibility) in a functional release and will not change it more than once a year. During the functional release, the compiler may change and APIs may be retired per the deprecation policy for deletion.

- **Deprecation policy**

If an API change requires you to make source code changes, you are given notice one year in advance, if possible. For example, if a method is going to be replaced by a new method, the original method is marked as deprecated in NX 1847, which is the release family. The original method is maintained through the current release family and through the next release family also, after which the API may be retired and removed.

The what's changed reports in **Support Center** and, if possible, compile time warnings are used to warn you of changes made in the current release and changes coming in the next release. The deprecated items topic in **Support Center** provides a list of the deprecated methods and their replacement methods.

For more information about API information in **Support Center**, see **NX Open Differences between releases**.

Note:

NX development makes every effort to follow the above policies. However, there may be instances when the policies must be violated due to the type of changes required. When this type of change is made, it is published on the **Changes That May Require Code Changes** tab in **Support Center**. This includes information on the change that happened and suggests changes that could be made to the code.

NX maintains and publishes stability metrics that show that for at least the last 10 years the stability of the NX Open APIs is above 96% for any given release, while the average is above 99%.

Exploring changes to the NX Open API between releases

NX Open changes are published in **Support Center**. In the **NXOpen APIs** section for your release, you can view API changes in the **NXOpen APIs What's Changed Reports**.

By default, this webpage contains the reports of the current release and the last functional release. In addition, you can select a release to compare with your release to get a comparison between the two releases. You can get comparisons for up to two years of releases.

Each page presented provides a point-to-point comparison of the APIs. The **What's New** tab on the page presents the new API items added between the two selected releases. The **Changes That May Require Code Changes** tab provides information that shows the changes that violate API stability and (or) binary compatibility, in accordance with the rules specified in the above policies.

For example, removing API items violates our API stability and binary compatibility policy unless the compiler changes. For each entry, we provide remediation suggestions, if needed, and possibly the reason for the change. An example of a change for which the only remedy is to recompile, is if a class has a new base class inserted into it's hierarchy. In this case, you don't need to change your code, but you do have to recompile your automation code that uses the API. This is contrasted by changes to a method signature to add an additional parameter. In this case, there will be a description of what the new parameter's purpose is and how to use it.

Also, the **Changes That May Require Code Changes** tab lists all of the deprecated APIs, their replacement APIs, and the earliest that each API could be retired. APIs that are retired are listed on the tab, along with when the API was removed and their replacement.

Open C API changes are listed in the NX release notes, in the *New Open C routines*, *Obsoleted Open C routines*, and *Deleted Open C routines* topics.

Function declarations that are newly retired (obsolete/deprecated) are moved to the **uf_retiring.h** file, which contains a complete list of Open C functions that could be deleted in the next NX release.

Release upgrade process for a functional release

For each functional release of NX, you should perform the following steps to migrate your application to the release:

1. Review the **Changes That May Require Code Changes** tab for the release you are going from and upgrading to. This helps you to understand what changes are being made in the release and what changes are planned for the next release.

2. For any methods or properties that have been deleted and for those which you have not already replaced in your code, implement the replacement code.

3. Recompile your entire source base using the current compile time files and settings for the given release.

4. For any methods or properties that produce deprecation warnings, decide if you are going to replace this code now or in the next release. Implement the replacement code for any deprecated methods you want to replace now and recompile.

5. Link your application with the appropriate NX Open libraries.

6. Perform a full suite of application testing in a stable NX environment.

7. Distribute your applications to your user base.

Release upgrade process for a monthly release

For each monthly release of NX, you should perform the following steps to migrate your application to the release:

1. Review any changes for the monthly release in **Support Center**.

2. Any API changes that may require rework or a recompile of your code will only be made if absolutely necessary. In this case, you will have to rework or recompile the affected applications, then test and redistribute. These changes are provided in **Support Center**.

Product Notes

Certified compilers and environments to use for NX Open programs

This table specifies the compiler (or installation for Python) to use for the applicable platform. These certified compilers are the ones used in testing for a particular release and are the only ones certified for use with NX Open. You can also review this information in **What's changed in NX Configuration and NX Open** in Support Center. On that page, select **NX Open APIs**, then the **Compiler Changes** tab.

NX Open may require you to use a C++ language standard that is higher than the default for your compiler. For example, Visual Studio 2019 defaults to C++ 14. Projects that are created using the NX Open wizard for the current release have the minimum C++ language required for that release set correctly.

	Windows	**Linux**
C/C++	Visual Studio 2019 Professional 16.11.26 Build 19.29.30148.0 ––––––––––––– C++ 20	C: gcc 11.2.1 C++: g++11.2.1 ––––––––––––– C++ 20
.NET	.NET 4.8 Framework	N/A
Java (64–bit)	NX Open **.jar** files built with Eclipse Adoptium (JDK) Temurin Development Kit with Hotspot 11.0.14.1+9 (x64). Certified with Java 11. Any Java provider can be used.	NX Open **.jar** files built with Eclipse Adoptium (JDK) Temurin Development Kit with Hotspot 11.0.14.1+9 (x64). Certified with Java 11. Any Java provider can be used.
Python (64–bit)	Python 3.10.12	Python 3.10.12

Use of a non-certified compiler

- **C++**

  For Windows, any version of a Visual Studio compiler should work within the release family as long as the version is equal to or higher than the certified version.

  For Linux and **gcc** versions, any version of **gcc** should work as long as the version is equal to or higher than the certified version and in the same release family.

  To find the current C++ linker version installed on your system, run **link /?**. For additional information, see Signing Process.

- **.Net**

The .Net framework provided by Microsoft attempts to maximize backward compatibility. The version used should be equal to or greater than the certified version. Additional information is provided by Microsoft at **Version Compatibility in the .NET Framework**.

- **Java**

  The Java Runtime Environment (JRE) has historically attempted to maximize backward compatibility. The version of Java used should be equal to or greater than the certified version.

- **Python**

  The Python version that NX delivers is built at the certified version for the release. In addition, Python is binary compatible in a major release family, so for example, if we build and certify against X.Y.4, our NX Open Python binaries will be binary compatible for all releases in the X.Y series (X.Y.5, X.Y.6, and so on). But they will not be binary compatible with a different major release family, for example, the X.Z series (X.Z.3, X.Z.4, and so on).

Using a non-certified compiler or environment is not without risks, which is assumed by the user. If an issue arises using a non-certified compiler or environment, Siemens Digital Industries Software can investigate with the following caveats:

- We need to be able to reproduce the issue easily so we can investigate the problem.
- If the compiler or environment regresses, it is expected that the user will investigate the issue in their environment to rule out improper usage on their part. For example, it is expected that users will rule out scenarios where the latest version of the compiler or environment fixes a bug that now catches something invalid that wasn't caught before in a previous version.
- There exists the possibility that the solution to the issue encountered cannot be realized in the release reported, but will be deferred to a later release.

Running Java and .NET programs

To run programs created in NX Open that require Java or .NET, the following is required:

	Windows	**Linux**
.NET	.NET 4.8 Framework	N/A
Java (64–bit)	NX Open **.jar** files built with Eclipse Adoptium (JDK) Temurin Development Kit with Hotspot 11.0.14.1+9 (x64). Certified with Java 11. Any Java provider can be used.	NX Open **.jar** files built with Eclipse Adoptium (JDK) Temurin Development Kit with Hotspot 11.0.14.1+9 (x64). Certified with Java 11. Any Java provider can be used.

Note:

- Some standard NX applications, such as translators, require installation of the JRE. For additional information, see **Java** in the NX System Information section of the release notes.
- If you are running a Siemens product other than NX, you may need to run a different version of Java according to the requirements for that product.

.NET Core

Microsoft is transitioning from the .NET Framework to .NET Core. We are working to implement .NET Core into NX Open in a future release, so you can use .NET Core as your design platform for developing NX Open programs.

NX Open for GRIP, UF, UGOpen++

The NX Open for GRIP, UF, and UGOpen++ programming tools are no longer being actively developed or enhanced. These tools are frozen and will not be updated.

Using the SCL compiler on Red Hat Enterprise Linux

To use the SCL compiler g++ 11.2.1 on Red Hat 8.7 or later, add a repository and install the compiler package:

1. Register the system (required for Red Hat Linux). If you don't register, the following commands will fail.

2. Install a package with a repository for your system. Run the following to enable the RHSCL repository:

```
sudo yum-config-manager –enable rhel-server-rhscl-7-rpms
```

3. Run the following to install the collection:

```
sudo yum install devtoolset-7
```

4. The compiler is installed. However, the default compiler recognized is still the one found by paths and environment variables. To use your compiler, either invoke a shell or set environment variables.

   - Invoke a shell using the SCL compiler:

```
scl enable devtoolset-7 bash (or ksh)
```

   - Set environment variables:

- INFOPATH /opt/rh/devtoolset-7/root/usr/share/info:$INFOPATH
- LD_LIBRARY_PATH /opt/rh/devtoolset-7/root/usr/lib64:/opt/rh/devtoolset-7 /root/usr/lib:/opt/rh/devtoolset-7 /root/usr/lib64/dyninst:/opt/rh/devtoolset-7/root/usr/lib/dyninst:/opt/rh/devtoolset-7 /root/usr/lib64:/opt/rh/devtoolset-7/root/usr/lib:$LD_LIBRARY_PATH
- MANPATH /opt/rh/devtoolset-7/root/usr/share/man:$MANPATH
- PATH /opt/rh/devtoolset-7/root/usr/bin:$PATH

> Note:
>
> When invoking a shell to use the compiler, it's possible you could have problems when running Perl scripts. In that case, setting environment variables is the preferred method.

Remote jar files deprecated

Starting in NX 2212, the use of remoting **.jar** files, which includes **NXOpenUIRemote.jar**, **SafetyOpenRemote.jar**, and other **<task>Remote.jar** files in NX, is no longer applicable. All of the remoting **.jar** files are deprecated and will not be available moving forward. It is recommended that you move your program remote dependency to **<task>Open.jar** from **<task>Remote.jar**.

The Java skeleton interface (part of Java RMI), which **Remote.jar** uses, is deprecated and no longer required or recommended. See **Tools and Components Removed and Deprecated in JDK 15**.

If you need to continue to use remoting **.jar** files in the short term, you can enable the **UGII_JAVA_USE_RETIRING_REMOTING_JARS=1** environment variable.

Internet browsers

If you do not see the latest documentation for the following programming tools, it may be a cache issue. Try clearing your cache, or using a different browser.

- *Open C Reference Guide*
- *NX Open C++ Reference Guide*
- *Open for Java*
- *NX Open Python*
- *Open for .NET*

In the *Open C Reference Guide*, if links to the example *.c* files do not work in IE, the *.c* file may not have an associated program to open it. Follow the instructions for your operating system to associate the *.c* file to a program, such as Notepad.

Signing applications

An executable for an application created with NX Open must be signed before it can be executed by anybody who does not have a NX Open Author license. This typically includes the primary users of the executable, such as the mechanical designers at your site. The signing process is performed with the NX Open signing utility before the application is distributed to the user base.

Starting in NX 2007 for Java jar files, you can sign the executable using the NX Open signing utility and also have the executable digitally signed using jarsigner. Previously, you could not do both in Java as the NX Open signing process was not compatible with digital signing. Adding digital signing provides an additional level of security and protection especially if the application is going to other sites.

Starting in NX 2212, legacy signatures for C++, .Net, and Java applications created using the NX Open signing scheme that was used before digital signing are not honored; only NX Open signatures that allow digital signatures will be honored. Both methods of signing were allowed to coexist for an application previously. Compilers are expected to be updated in NX 2212, so it is anticipated that all applications will need to be recompiled and resigned, and will use NX Open signatures that allow the digital signature.

Executing NX Open for .NET batch programs

Starting in NX 2007, use the **run_dotnet_nxopen.exe** standalone executable in the **UGII_BASE_DIR\nxbin** directory instead of **run_managed.exe** to run a batch program.

All of the functionality used for executing NX Open .NET applications is unchanged in the **run_dotnet_nxopen.exe** standalone executable.

Wizards

Starting in NX 2406, the old NX Open wizards are removed from **%UGII_BASE_DIR%\ugopen\vs_files**. The new VSIX Project templates located at **%UGII_BASE_DIR%\ugopen\NXOpenWizard\NXOpen<programming language>Application** should be used instead.

Creating programs with Visual Basic

Microsoft has announced that it does not plan to continue to evolve and advance the Visual Basic programming language. Therefore, we recommend that going forward you create new NX Open programs using C# instead of Visual Basic.