SIEMENS

NX 1899 Series – Release Notes

Welcome to NX

April 2020

Dear Customer:

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

Sincerely,

Siemens Digital Industries Software

CONTENTS

NOTE

These release notes provide information about updates, fixes, and enhancements that are included in NX Releases.

For all other Siemens Digital Industries Software product updates, please refer to each products' respective release notes provided on [Support Center](#).

NX 1899 SERIES SUMMARY

Release Availability

A complete list of NX 1899 Series releases is included below.

Release	Release Date	Automatic Update	Download Server
1915	Apr 2020	✔	✔
1911	Mar 2020	✔	✔
1907	Feb 2020	✔	✔
1904	Feb 2020	✘	✔
1903	Jan 2020	✔	✔

Release Content

The following table represents a release overview of the PR and ER fix counts. More details can be found in the **NX Release Summaries** section in this document.

Release	PR Count	ER Count	Total
1915	59	0	59
1911	41	0	41
1907	48	0	48
1904	1	0	1
1903	88	1	89
Total	**237**	**1**	**238**

Caveat - Siemens Cloud Connector Service Form

Users will see below issues while trying to login using Siemens Cloud Connector Service form:

1. The Siemens Cloud Connector Service login form embedded in NX does not have scroll bar. Users will have to re-size the window to see the "Sign In" button. <u>This issue will be addressed in 1926.</u>
2. Siemens Cloud Connector Service will remember the webkey login credentials entered by user even if user reboots his machine. To work around this, users will need to clear the cache on the web browser.

NX 1915

1915 UPDATES

1915 FIXED PROBLEM REPORTS

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

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

Application	Count of PR Number
CAE	15
KDA	10
SYSENG	9
CAM	5
NXMANAGER	3
PMI	3
SHIP_DESIGN	3
TRANSLATOR	2
ASSEMBLIES	2
DESIGN	2
ARCHITECTURE	2
ROUTING_GENERAL	1
AUTOMATN_DESIGN	1
MECHATRONICS	1
Total	**59**

1915 ENHANCEMENTS

There are no enhancements included in this release.

NX 1911

1911 UPDATES

Caveats

Removed fields feature toggle for Lattice and Rod Thickness

Create a lattice with a variable rod diameter functionality was under a feature toggle in "Fields_AvailableForLatticeRodDiameter" in NX1899. This feature toggle is removed, and the functionality is available with no feature toggle in NX1911.

1911 FIXED PROBLEM REPORTS

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

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

Application	Count of PR Number
CAE	20
KDA	5
ASSEMBLIES	4
AUTOMATN_DESIGN	2
SYSENG	2
NASTRAN	2
CAM	1
GATEWAY	1
ROUTING_GENERAL	1
DESIGN	1
NXMANAGER	1
PMI	1
Total	**41**

1911 ENHANCEMENTS

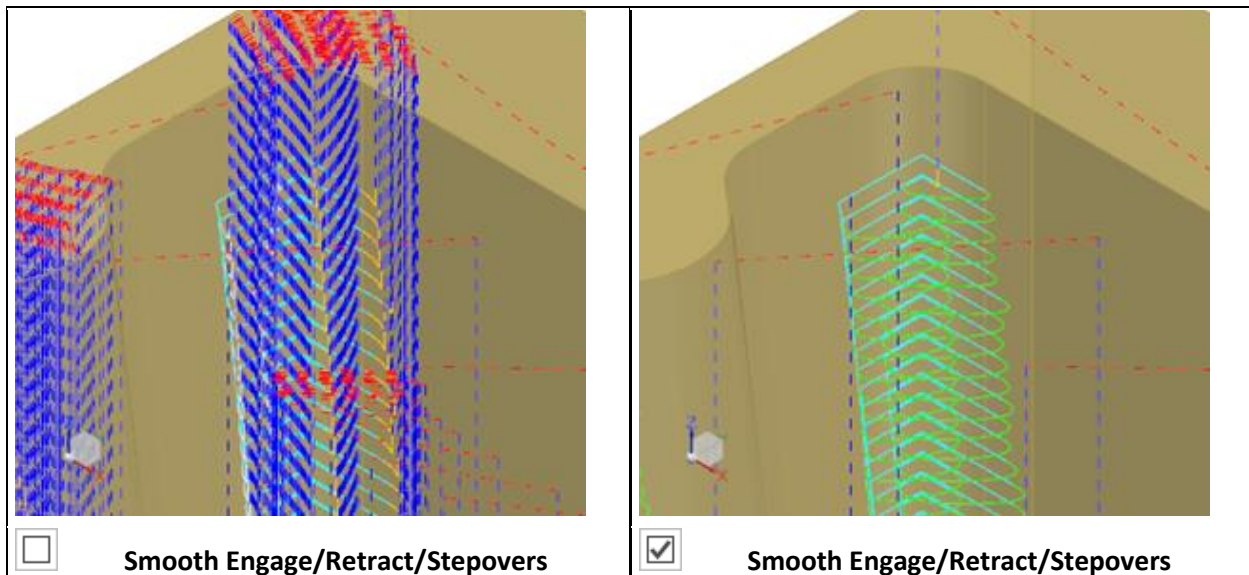Please see below for a list of enhancements included in this release.

CAM : Non-cutting moves in Rest Milling operations

Introduction

What is it?

The smoothing commands for non-cutting moves are now updated in in Rest Milling operations.

☐	**Smooth Engage/Retract/Stepovers**	☑	**Smooth Engage/Retract/Stepovers**

- The default setting for **Max Stepover** is **100 %Tool**.

  The maximum value for **Max Stepover** is **400 %Tool**.
- The **Smooth Engage/Retract/Stepovers** command does not override the **Engage/Retract** settings.
- The **Smooth Engage/Retract/Stepovers** command is available when you define the **IPW/Reference Tool**.

If blank geometry is defined with part offset or additional stock, the **Smooth Engage/Retract/Stepovers** command is not available.

Where do I find it?

Application	Manufacturing
Dialog box	**Non Cutting Moves** node→**Smoothing**→**Transition Smoothing**→ ☑ **Smooth Engage/Retract/Stepovers**

CAM : Report Shortest Tool Enhancements

Introduction

What is it?

The **Report Shortest Tool** command has the following enhancements:

- NX supports the shank as well as the holder when calculating the shortest tool length.
- NX considers the holder and shank clearance when calculating the shortest tool length.
- When both the holder and shank are defined for a tool, NX reports the shortest tool that satisfies both the holder and shank clearance values.
- Changing the **Tool Insertion** offset value does not affect the shortest tool value.

In all cases, NX rounds up the reported value to 1 decimal point for metric parts and 2 decimal points for inch parts. For example:

- 120.206 mm = 120.3 mm
- 30.206 in = 30.21 in

Note:

The **Minimum Tool Length** command is renamed to **Report Shortest Tool**. When you customize a dialog box, the customizable item is also renamed from **Minimum Tool Length** to **Report Shortest Tool**.

Where do I find it?

Application	Manufacturing
Initial Release	NX 1911

Prerequisite	**Cavity Mill**, **Adaptive Milling**, **Plunge Milling**, or **Rest Milling** operation
	The tool used in the operation must have a shank or holder defined.
	The operation must have a generated tool path.
Operation Navigator	Right-click the operation→**Tool Path**→**Report Shortest Tool**

NX 1907

1907 UPDATES

Caveats

Bolt Connection Element Error

An error will be raised upon attempting to create a Bolt Connection Element from the Connection Element Manager dialog under these conditions.

1. A new FEM/AFM file is created and saved in Simcenter 3D 2020.1 version 1907 and it is opened in Simcenter 3D 2020 version 1899
2. A Bolt Connection element is created in Simcenter 3D 2020.1 version 1907 FEM/AFM file, when opening the file in Simcenter 3D 2020.1 version 1899, editing the Bolt Connection Element can trigger the error and obstruct the edit operation

The influence of the errors raised in the above-mentioned cases on further operations in the Simcenter 3D is not known, so for safety reasons it is recommended to close the session and reopen Simcenter 3D after the error is triggered.

1907 FIXED PROBLEM REPORTS

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

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

Application	Count of PR Number
CAE	22
DRAFTING	5
KDA	5
CAM	4
ROUTING_GENERAL	2
SYSENG	2
NASTRAN	2
NX_SHEET_METAL	2
VOLUME	1
TRANSLATOR	1
SHIP_DESIGN	1
PCB_EXCHANGE	1
Total	**48**

1907 ENHANCEMENTS

No enhancements were included in this release.

NX 1904

1904 UPDATES

1904 FIXED PROBLEM REPORTS

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

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

Release	PR Number	Description	Application	Function	Sub_Function
1904	8441216	Internal error when using delete template	AUTOMATN_DESIGN	PAGE	BASIC_CAAPABILIT

1904 ENHANCEMENTS

There are no enhancements included in this release.

NX 1903

1903 UPDATES

1903 FIXED PROBLEM REPORTS

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

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

Application	Count of PR Number
CAE	21
CAM	15
KDA	13
SYSENG	6
NX_SHEET_METAL	5
ASSEMBLIES	3
CMM_INSPECTION	3
ROUTING_GENERAL	3
SHIP_DESIGN	3
TC_FEATURES	3
DESIGN	2
NXMANAGER	2
CORRUPTED_PARTS	1
DRAFTING	1
FLEXIBLE_PIPE	1
MECHATRONICS	1
NASTRAN	1
TRANSLATOR	1
Total	**85**

1903 ENHANCEMENTS

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

- ER8562281 - Enhancement Request for SC3D Motion TWR

Teamcenter Integration: Support for Mentor Graphics Capital Integration

Introduction

Enhancement Scope

Mentor Graphics Capital software can communicate to NX using connected mode. This requires the two software to be running simultaneously on same machine or two different machines.

Earlier, the connected mode setup was a part of Capital build and the user had to manually set the environment in order to get that working. With this enhancement, we will ship the connected mode package as a part of NX kits thereby not requiring any need for manual setup

Setup Improvements

As mentioned above, earlier connected mode was established by setting the following environment variables:

- MGLS_PKGINFO_FILE
- MGLS_DLL
- CAPITAL_NX_PLUGIN_LOCATION
- PATH=%PATH%, <Path_To>/jetinitheap.dll

With this enhancement, these values will be set at the time of NX install and the user will no longer need to set these values manually.

Newly Added Buttons

As part of enhancement, we have added two new buttons under Route List application

1. Connect: On click of this button, an RMI connection is established which enables the exchange of route list and diagram data between NX and Capital.
2. Harness 2D View: On click of this button, the diagram selected in Capital is displayed for cross-probing of routing objects
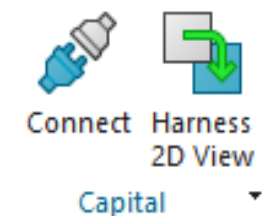
Above buttons will always be visible in NX regardless of whether Customer has Capital software installed or not. They will work only when the Capital software is running.
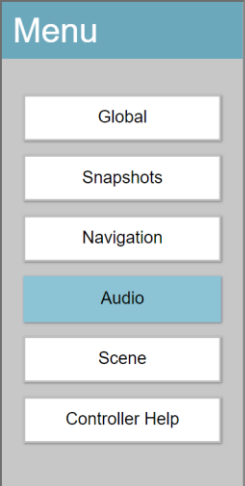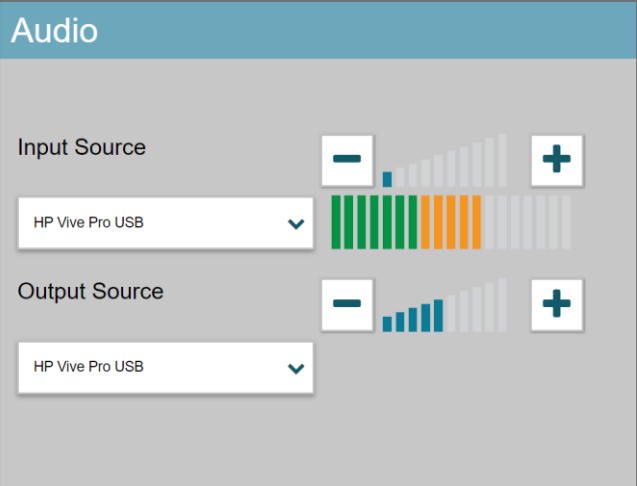
Architecture: NX Virtual Reality enhancements

Introduction

Conversing in a multi-user collaborative VR session

All the participants in a multi-user collaborative VR session can speak and hear simultaneously through the Head Mounted Device (HMD). NX VR integrated audio also supports spatial sound which helps you hear other participants relative to their position in a 3D space.

You can control the NX VR audio settings from the **Audio** tab of the **Global Menu**.

Global Menu

Options	Description
Input Source	Lists all the supported and enabled microphone sources that are available on the system. Point the right controller laser to the list and select the microphone that you want to use. To adjust the volume of the audio recording device, point the right control to ➕ or ➖, and press and hold the laser trigger. You must release the laser trigger to stop the volume adjustment.
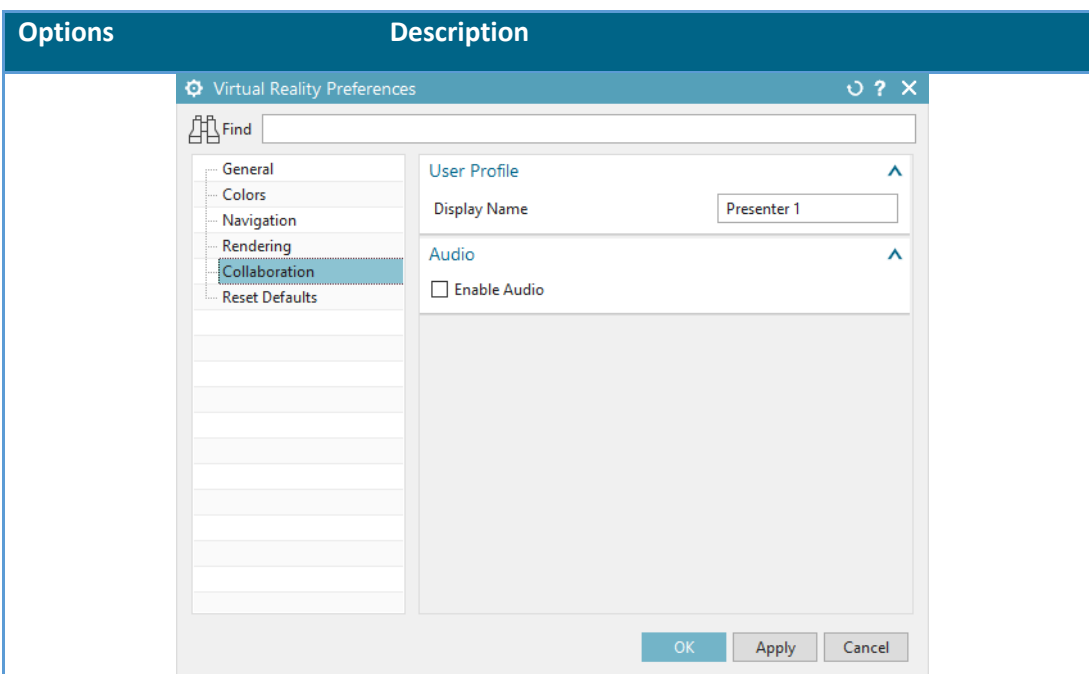
Output Source	Lists all the supported and enabled speakers that are available on the system. Point the right controller laser to the list and select the speaker that you want to use. To adjust the volume of the speaker, point the right control to ➕ or ➖, and press and hold the laser trigger. You must release the laser trigger to stop the volume adjustment.

Note: If you are unable to hear the audio, check the Input and Output settings for the HMD device.

Virtual Reality Preferences

Options	Description
Enable Audio	Enables the audio of the HMD in a multi-user collaborative VR session.

CMM: Multiple Curve Segments for Curve Scan

Introduction

CMM Inspection – Path Dialog – Curve Scans

When programming curve scans the user can define scan parameters such as scan speed at each control point. In NX1903, the user has two new ways to define the control points:

- Midpoint of curve
- Screen select point on curve

Line Designer: Native Mode Support

Introduction

Working with a plant layout in NX managed and native mode

What is it?

In the native Line Designer application, you can now modify the plant layout to add resources. Also, you can create detailed designs. You can:

- Add, remove, and modify the components of a plant layout. For example, use the NX Modeling application to create new components.

- Display the attributes of the component files in the **Plant Navigator**.

- Add, edit, and remove attributes of the components in the **Properties** dialog box and in the **Plant Navigator** for configured columns.

- Save the modified plant structure data in NX native mode, and optionally update the plant layout in the NX Managed mode.

- Work with connections, attachments, mounts, and kinematic objects.
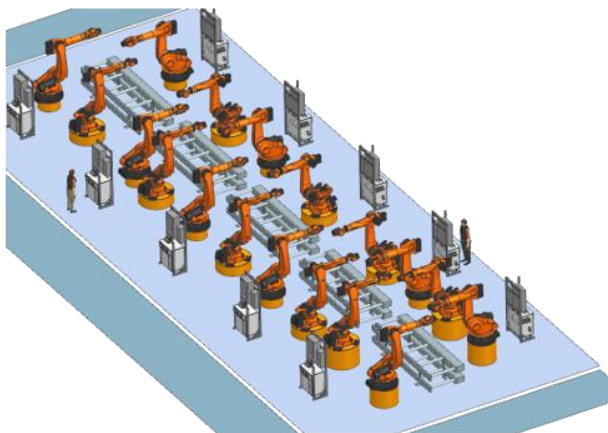
Why should I use it?

You can now author Line Designer plant layouts in NX native.

Where do I find it?

Application	Line Designer
Resource bar	**File** tab→**Export Assembly outside Teamcenter** **File** tab→**Import Assembly into Teamcenter**

Technical documentation

Managing a plant layout in NX managed and native mode

Exporting and importing a plant layout

For various business processes and customer requirements, and for general product development, you need to move the plant layout from one database to another. This helps you to conveniently provide required data to a third party for further modifications. To do this, use the **Export Assembly outside Teamcenter** command.

You can modify and detail the plant layout and add, remove, modify, rename, and revise its components in NX native mode.

Once you modify the plant layout, you can import back this data into NX integrated with the Teamcenter environment. To do this, use the **Import Assembly into Teamcenter** command. You can verify and validate the changes to the plant layoutfixture data.

You can move the plant layout data in the following way:

Export Assembly outside Teamcenter	In NX integrated with Teamcenter, you can launch the **Plant Navigator** and load a plant layout. Before you export the plant layout data, make sure that you do the following:

- Save the plant layout data in Line Designer before you export the data, with or without any changes.

- Confirm that the plant layout data contains part files for all the objects. This is necessary so that Line Designer in the native mode recognizes these objects in the **Plant Navigator**.

- Use the mapping file to map the object properties to the part file. You can use this mapping file in the NX native mode and show more

information to the native user. To do this, use the attribute mapping tool.

For more information, see *Export Assembly outside Teamcenter Overview* in NX help.

Import Assembly into Teamcenter	Import the modified plant layout data to NX integrated with Teamcenter. This helps you to validate the plant layout with a detailed plan and to check for the modified data. For more information, see *Import Assembly into Teamcenter* in NX help.

If the plant layout data contains the JT parts as components, you can export them to NX native in the local directory and import them back in NX integrated with Teamcenter. NX exports the JT parts as components in the plant layout. For more information, see *Importing and exporting Multi-CAD* in NX help.

Where do I find it?

Application	**Line Designer**
Resource bar	**File** tab→**Export Assembly outside Teamcenter** **File** tab→**Import Assembly into Teamcenter**

Managing plant layout data from NX managed to NX native

The following graphic shows a typical workflow for exporting the plant layout data from NX integrated with Teamcenter (Managed) mode to the native NX mode and importing back to managed NX mode.

NX Integrated with Teamcenter

Open the plant layout

Export the plant layout to native NX with the reuse library objects

Import the modified plant layout

Validate the plant layout

Native NX

Load the part libraries in the Reuse Library

Import and load the plant layout

Modify and save the plant layout
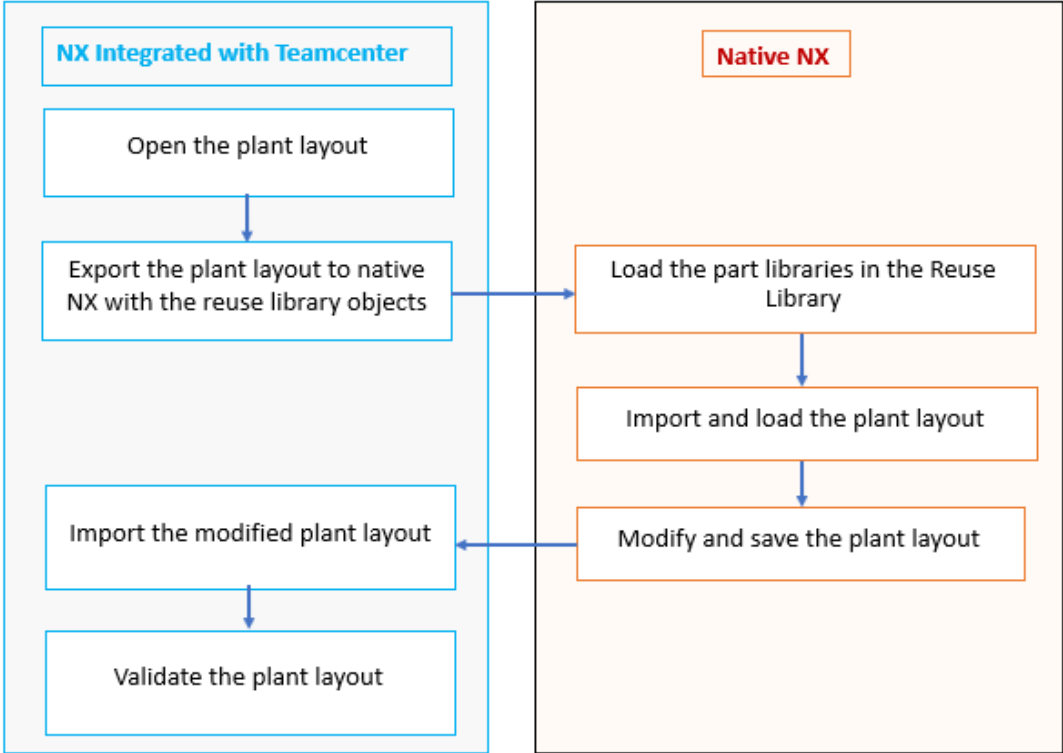
Managing plant layout data in NX native

The following graphic shows a typical workflow for working with a plant layout in NX native mode.
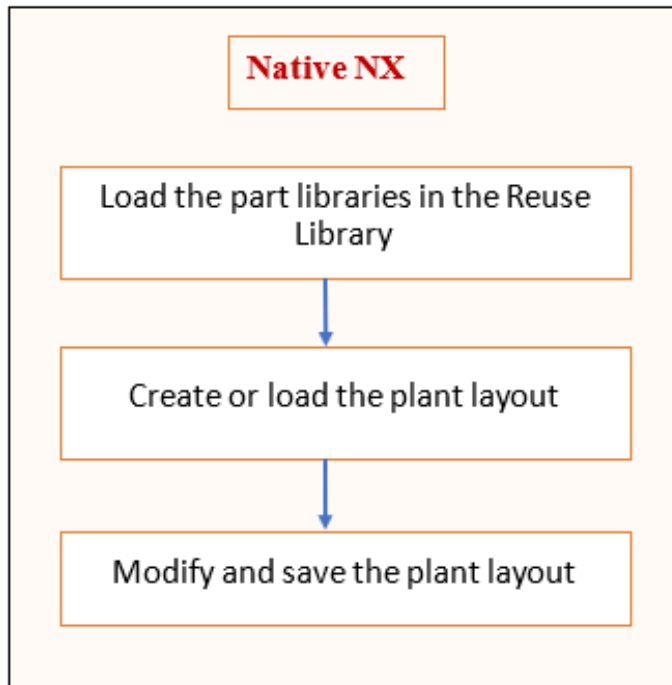
Managing plant layout data from NX native to NX managed

The following graphic shows a typical workflow for working with a plant layout in NX native mode and then importing the data in NX integrated with Teamcenter (Managed) mode.

Native NX

- Load the part libraries in the Reuse Library
- Create or load the plant layout
- Modify and save the plant layout

NX Integrated with Teamcenter

- Import the modified plant layout
- Open the plant layout
- Validate the plant layout
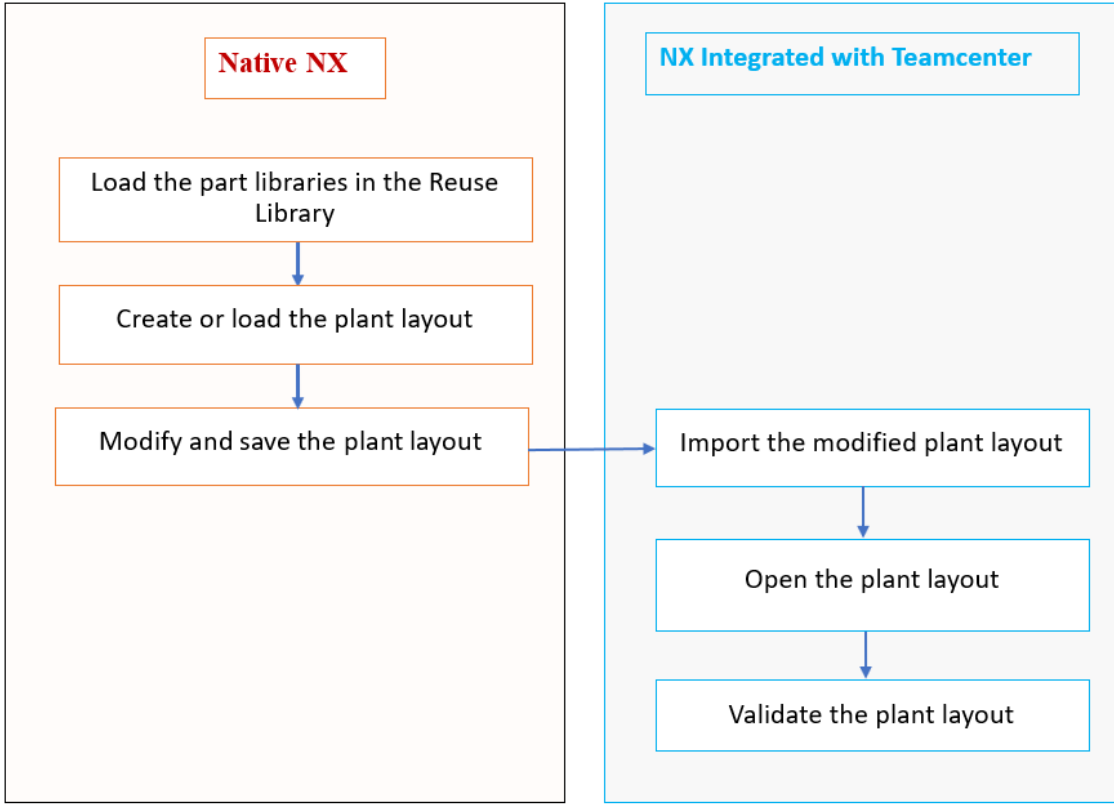
Managing PAX files for NX Native

A PAX file is actually a palette file with a .pax extension. A palette file is an XML file that contains a list of palette entries, each of which contains a specific identification number and a standard set of data that describes the entry.

You can create structure and resource objects that you can load in Teamcenter as a specific **Item Type**. The pax file for a work area template in Line Designer is named *ugs_linedesigner_workarea_templates.pax* and for the resource templates is named *ugs_linedesigner_templates.pax*.

The following is an example of the XML in a .pax file that defines the template for the new work area. The type specified in the object data section is mapped to the **Item Type** in Teamcenter, which you can use while importing layout data to Teamcenter.

```xml
<PaletteEntry id="d2">
    <References/>
    <Presentation name="Workarea" description="NX Line Designer Workarea">
        <PreviewImage type="UGPart" location="linedesigner_zone_template.jpg"/>
    </Presentation>
    <ObjectData class="LineDesignerTemplate">
        <Filename>linedesigner-inch-part-template.prt</Filename>
        <Units>English</Units>
        <Type>MEWorkarea</Type>
    </ObjectData>
</PaletteEntry>
```

You can customize the pax files to achieve the work area and resource types you want, regardless of the listed work areas and resources below.

Supported work areas

By default, these work areas are available to use in NX native:

- Workarea

- Plant

- Site

- Department

- Line

- Zone

- Station

- Blank (Custom)

Supported resource templates

By default, these resource templates are available to use in NX native:

- Equipment

- Resource

- Conveyor

- Robot

- Factory Tool

- Cable Resources

- Sensors

- Fixture Root

- Blank (Custom)

Working with Attribute templates files and icons in native NX

Icons from item types

When you work in a plant layout, you can see the component icons in the **Plant Navigator** based on the **Item Type** definition that you specify in the part file. This helps you identify the different types of work areas and resources in the layout.

You can customize the default icons to use the desired bmp or png files and to align with the **Item Types** and with the icons that are available in Teamcenter.

You can also display customized icons in the **Plant Navigator** for the default **Item Types** included with Line Designer or for custom **Item Types**. To do this, you must store the respective .bmp or .png files in the installation folder.

Here is the installation folder path:
%UGII_BASE_DIR%\MANUFACTURING_PLANNING\line_designer\application\class_bit maps

You can use the following icon name formats:

Icon name format for .bmp file: <internal name>.bmp

Icon name format for .png file: <internal name>.png

For more information, see *Customizing Plant Navigator icons* in NX help.

Attribute templates

When you work in a plant layout in native NX, you have the ability add, edit, and delete attributes from the part files. You can modify the component properties that you can later

import to Teamcenter, if required. You can edit the attributes of a component in the **Properties** dialog box and in the **Plant Navigator** for configured columns.

For NX native mode, you can use the attributes based on the default Teamcenter **Item Types** and the attribute mapping file from the default Line Designer installation. When you create a **New Item**, NX includes the component properties in the part file. You can find the **Attribute Templates** that you use to add component properties in the attribute templates folder.

Here is the attribute templates folder path:
%UGII_BASE_DIR%\MANUFACTURING_PLANNING\line_designer\startup\pdm_attribu te_templates

The **Attribute Templates** are based on the attribute mappings that you define in Teamcenter. You can export these templates from NX and customize them. For more information, see *Attribute templates* in NX help.

For detailed information about the mapping file, see *Mapping Attributes in Teamcenter and Teamcenter Integration* in NX help.

Modifying the plant layout in NX native

To align with changing customer requirements, you might need to frequently change the design of a plant layout. This helps you create new designs, develop new products, and stay current in the market.

You can modify the plant layout to add components. Also, you can create detailed designs. To do this, you can use the Line Designer in NX native mode.

You can export the plant layout from NX managed mode to the NX native mode and load the whole plant layout structure.
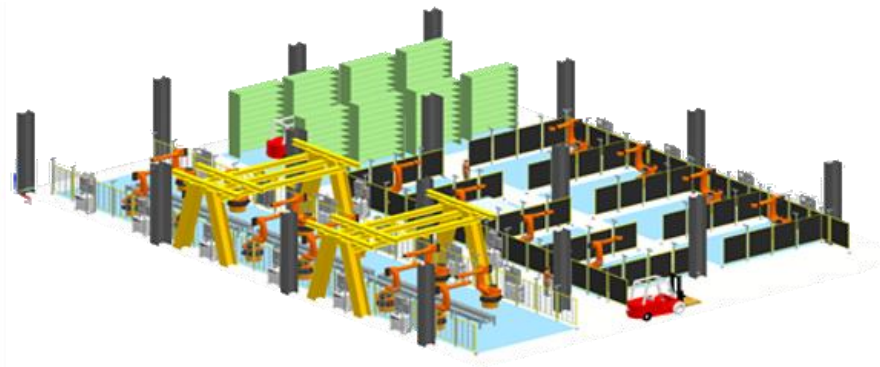
When you load the plant layout data in NX native, you can:

- Add, remove, and modify the components of a plant layout. For example, use the NX Modeling application to create new components.

- Display the attributes of the component files in the **Plant Navigator**.

- Add, edit, and remove attributes of the components using the **Properties** dialog box and in the **Plant Navigator** for configured columns.

- Save the modified plant structure data in NX native mode, and optionally update the plant layout in the NX Managed mode.

Work with connections, attachments, mounts, and kinematic objects.

CAE: Pre/Post for Ansys

Introduction

Importing All KEYOPTs as User Defined KEYOPTs

Many ANSYS element types have key options (KEYOPTs) that you can specify. Each KEYOPT is identified by its number and may have different values. For example in ANSYS version 19.1, SHELL181 has 7 KEYOPTs: KEYOPT(1), (3), (5), (8), (9), (10) and (11). The value of KEYOPT(3) could be 0 or 2. The KEYOPT numbers and its values are defined in the XML. They can be changed, removed or added in ANSYS from release to release.

When a particular KEYOPT is supported by Pre/Post, but one of its values is unsupported, this value is lost on import, and Pre/Post uses the default value instead. You can now select a new Customer Default, "Import all KEYOPTS as user defined KEYOPTS", to ensure that unsupported KEYOPT values are imported.

Accessibility

You can select this new option from the Customer Defaults dialog box as follows: Customer Defaults → Simulation → Pre/Post → ANSYS → Import →General Options (see the image).

Image: Toggle of "Import all KEYOPTS as user defined KEYOPTS" in Customer Defaults
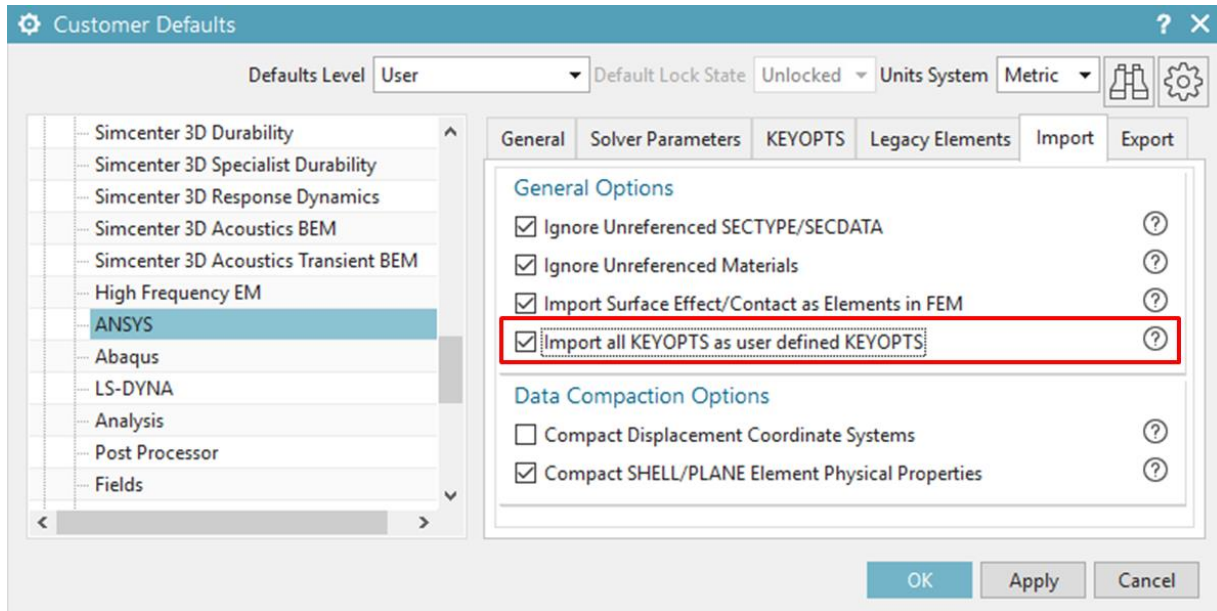
Import behavior

How Pre/Post imports KEYOPTs in an ANSYS input file depends on whether the new customer default is selected:

- When the new default is turned off, Pre/Post imports KEYOPTs into Pre/Post as in previous releases.

- When the new default is selected, Pre/Post imports all KEYOPTs as User Defined KEYOPTs.

The following images illustrate these two behaviors:

Image: KEYOPTs numbers and values example for SOLID186 with an unsupported value in the input file

Image: SOLID186 ET modeling object with the loss of unsupported value when the customer default is turned off
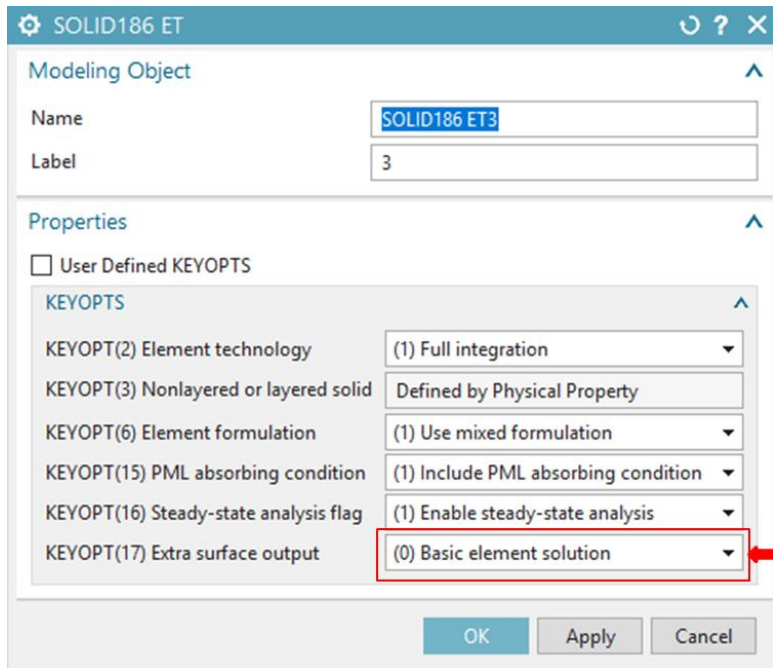
Image: SOLID186 ET modeling object when the customer default is selected
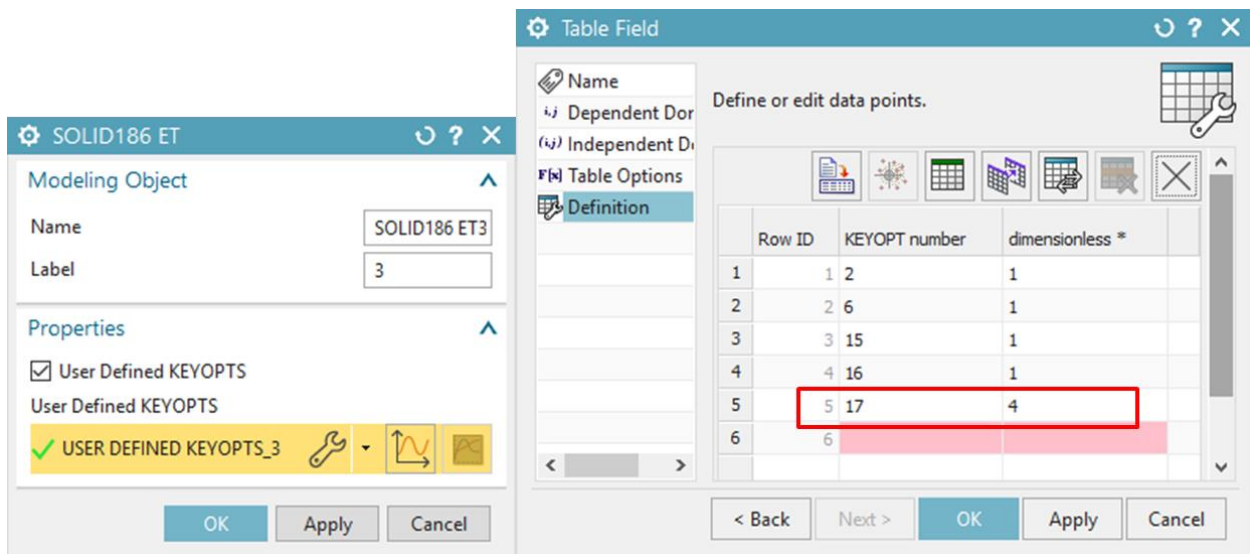
Note: This support applies to all element types accept the Legacy Elements that are not supported as user defined KEYOPTS. The Legacy Elements are BEAM4, BEAM44, LINK8, LINK10, CONTAC12, CONTAC52, PLANE42, PLANE82, SHELL63, SHELL57, SHELL91, SHELL93, SHELL99, SOLID45, SOLID92, SOLID95,

SOLID191. Pre/Post always imports the KEYOPTs for these Legacy Elements as standard KEYOPTs regadless the status of the new default.

Image: SOLID95 ET modeling object (Legacy Element)
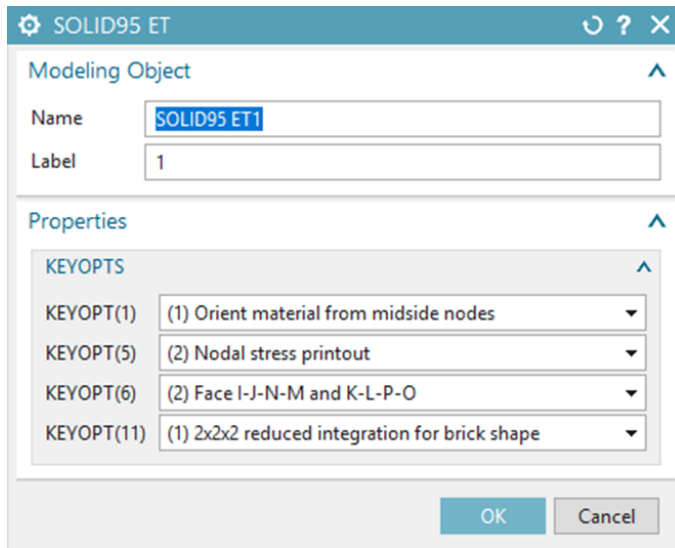
NX RELEASE SUMMARIES

The following tables show high level area overviews of the PR fixes in each NX release.

1915

Application	Function	Sub-function	PR Count	ER Count
ARCHITECTURE	OM	SAVE	1	0
ARCHITECTURE	UNITS	ALL	1	0
ASSEMBLIES	ASSEMBLY_MODEL	COMP_PATTERN	2	0
AUTOMATN_DESIGN	LAYOUT	ALL	1	0
CAE	ADV_SIMULATION	CHK_LOAD_SUM	1	0
CAE	ADV_SIMULATION	CONNECTIONS_UNI	2	0
CAE	ADV_SIMULATION	DUR_SPEC_SOLV	1	0
CAE	ADV_SIMULATION	DUR_SPEC_UI	1	0
CAE	ADV_SIMULATION	POST_PROBE	1	0
CAE	ADV_SIMULATION	POST_RES_MANIP	1	0
CAE	ADV_SIMULATION	SYSNOISE_SOLVER	1	0
CAE	ADV_SIMULATION	UNIF_DATA_VIE	1	0
CAE	MOTION	CONTACT	2	0
CAE	MOTION	JOINT	1	0
CAE	MOTION	VL2NXMIGRATION	1	0
CAE	RESULTS_VIEWER	RESULTS_VIEWER	2	0
CAM	BASIC_FUNCTIONS	UNDETERMINED	1	0
CAM	TURNING	GEOMETRY	3	0
CAM	ZLEVEL_MILL	OTHER	1	0
DESIGN	SKETCHER	UNDETERMINED	1	0
DESIGN	WAVE	COPY_THREADS	1	0
KDA	FEATURE2COST	MOLDING_COST	1	0
KDA	KNOW_FUSION	KF_FEATURE	1	0
KDA	KNOW_FUSION	LANGUAGE	1	0
KDA	REUSE_LIBRARY	2D_SECTION	1	0
KDA	REUSE_LIBRARY	INDEX_SEARCH	2	0
KDA	VALIDATION	HD3D_RESULTS	1	0
KDA	WELD_ASSISTANT	CONN_FACE_FIND	1	0
KDA	WELD_ASSISTANT	FEATURE_PUBLISH	1	0
KDA	WELD_ASSISTANT	WELD_PT_WIZARD	1	0
MECHATRONICS	BASIC_PHYSICS	RIGID_BODY	1	0
NXMANAGER	ATTRIBUTES	MAPPING	1	0
NXMANAGER	FILE_NEW	ALL	2	0
PMI	DIMENSION	LINEAR	2	0
PMI	WAVE	CORE	1	0
ROUTING_GENERAL	LEGACY_STOCK	EDIT_STOCK	1	0
SHIP_DESIGN	DRAFTING	SHIP_SECTN_DRWG	1	0

SHIP_DESIGN	STEEL_FEATURES	SPLIT_PRFL_PLAT	2	0
SYSENG	NXOPEN	JAVA	1	0
SYSENG	UI_TOOLS	UI_TOOLS	1	0
SYSENG	UI_TOOLS	WEB_BROWSER	7	0
TRANSLATOR	NX_JT	GEOMETRY	1	0
TRANSLATOR	STEP_AP214	ALL	1	0
Total			**59**	**0**

1911

Application	Function	Sub-function	PR Count	ER Count
ASSEMBLIES	ARCHITECTURE	ALL	2	-
ASSEMBLIES	ASSEMBLY_MODEL	DESIGN_FEATURE	1	-
ASSEMBLIES	PART_FAMILIES	GENERAL	1	-
AUTOMATN_DESIGN	LAYOUT	ALL	1	-
AUTOMATN_DESIGN	PAGE	BASIC_CAPABILIT	1	-
CAE	ADV_SIMULATION	CONNECTIONS_UNI	6	-
CAE	ADV_SIMULATION	DUR_SPEC_SOLV	2	-
CAE	ADV_SIMULATION	DUR_SPEC_UI	1	-
CAE	ADV_SIMULATION	FEMALTREP_SET	1	-
CAE	ADV_SIMULATION	MESH_3D_TET	1	-
CAE	ADV_SIMULATION	POST_RESULT	1	-
CAE	ADV_SIMULATION	SYSNOISE_EXPORT	1	-
CAE	ADV_SIMULATION	THERM_FLOW	1	-
CAE	ADV_SIMULATION	UNIF_DATA_VIE	1	-
CAE	ADV_SIMULATION	VL2NX_ACOUSTIC	2	-
CAE	ADV_SIMULATION	XYPLOT	1	-
CAE	DES_SIMULATION	POST_RESULT	1	-
CAE	MOTION	FLEX_BODY_SC	1	-
CAM	TURNING	IPW	1	-
DESIGN	FREE_FORM_MODEL	EXTENSION	1	-
GATEWAY	OPEN_JT	UNDETERMINED	1	-
KDA	GENERAL_PACKAGE	PEDPRO	1	-
KDA	KNOW_FUSION	LANGUAGE	1	-
KDA	MOLDWIZARD	STANDARD_PARTS	1	-
KDA	REUSE_LIBRARY	INDEX_SEARCH	2	-
NASTRAN	STRUCTURES	SOL_101_SESTAT	2	-
NXMANAGER	FILE_OPEN	PERFORMANCE	1	-
PMI	PLMXML	ALL	1	-
ROUTING_GENERAL	ADMINISTRATION	PART_LIBRARY	1	-
SYSENG	NXOPEN	PYTHON	1	-
SYSENG	UI_TOOLS	TOOLBAR	1	-
Total			**41**	**0**

1907

Application	Function	Sub-function	PR Count	ER Count
CAE	ADV_SIMULATION	CONNECTIONS_UNI	6	-
CAE	ADV_SIMULATION	DUR_SPEC_SOLV	1	-
CAE	ADV_SIMULATION	FEM_CORRELATION	1	-
CAE	ADV_SIMULATION	FIELDS	2	-
CAE	ADV_SIMULATION	MATERIALS	1	-
CAE	ADV_SIMULATION	MESH_MATING	1	-
CAE	ADV_SIMULATION	SAMCEF_EXPORT	1	-
CAE	ADV_SIMULATION	SELECTION_RECIP	1	-
CAE	ADV_SIMULATION	THERM_FLOW	1	-
CAE	ADV_SIMULATION	UNIF_DATA_VIE	1	-
CAE	ADV_SIMULATION	VL2NX_ACOUSTIC	2	-
CAE	MOTION	FUNCTION_EDITOR	2	-
CAE	MOTION	LINK	1	-
CAE	MOTION	SOLVER_LMS	1	-
CAM	FLOOR_WALL_MILL	CONTAINMENT	1	-
CAM	FLOOR_WALL_MILL	NON_CUT_MOVES	1	-
CAM	TEMPLATES	DEFAULTS	1	-
CAM	TOOL_PATH	LIST	1	-
DRAFTING	SYMBOL	CUSTOM_SYMBOL	4	-
DRAFTING	SYMBOL	REUSE_LIBRARY	1	-
KDA	MOLDED_PART_VAL	FLOW_ANALYSIS	1	-
KDA	MOLDWIZARD	CREATE_POCKETS	1	-
KDA	MOLDWIZARD	LOAD_PRODUCT	1	-
KDA	VALIDATION	TC_INTEG	1	-
KDA	WELD_ASSISTANT	TRANSFORM	1	-
NASTRAN	STRUCTURES	SOL_101_SESTAT	2	-
NX_SHEET_METAL	3_BEND_CORNER	ALL	1	-
NX_SHEET_METAL	CLOSED_CORNER	ALL	1	-
PCB_EXCHANGE	IDX_IMPORT	ALL	1	-
ROUTING_GENERAL	CONSTRAINTS	GENERAL	1	-
ROUTING_GENERAL	LEGACY_STOCK	INSERT_STOCK	1	-
SHIP_DESIGN	STEEL_FEATURES	MIRROR_COPY	1	-
SYSENG	NXOPEN	LICENSING	1	-
SYSENG	UI_TOOLS	COMMAND_FINDER	1	-
TRANSLATOR	STEP_AP214	STEP_TO_UG	1	-
VOLUME	VOLUME_ACCESS	ALL	1	-
Total			**48**	**0**

1904

Application	Function	Sub-function	PR Count	ER Count
AUTOMATN_DESIGN	PAGE	BASIC_CAPABILIT	1	-
Total			**1**	**0**

1903

Application	Function	Sub-function	PR Count	ER Count
ASSEMBLIES	SEQUENCING	MOTION	2	-
ASSEMBLIES	WEIGHT_MANAGE	ALL	1	-
CAE	ADV_SIMULATION	CHK_ELEMENTS	1	-
CAE	ADV_SIMULATION	DURABILITY	1	-
CAE	ADV_SIMULATION	DUR_SPEC_SOLV	1	-
CAE	ADV_SIMULATION	FILE_OPERATIONS	2	-
CAE	ADV_SIMULATION	IPRT_MODEL_OPS	1	-
CAE	ADV_SIMULATION	MATERIALS	1	-
CAE	ADV_SIMULATION	POST_OTHER	2	-
CAE	ADV_SIMULATION	POST_RESULT	4	-
CAE	ADV_SIMULATION	THERM_FLOW	1	-
CAE	ADV_SIMULATION	UNIVERSAL_FILES	1	-
CAE	MOTION	CONTACT	1	-
CAE	MOTION	FORCES	1	-
CAE	MOTION	FTK	1	-
CAE	MOTION	TWR	-	1
CAE	MOTION	VL2NXMIGRATION	3	-
CAM	POSTPROCESS	ALL	2	-
CAM	TC_MFG	MRL_CONNECT	1	-
CAM	TC_MFG	OTHER	8	-
CAM	TURNING	OTHER	2	-
CAM	WORK_INSTRUCTN	ALL	2	-
CMM_INSPECTION	PATH	ALL	1	-
CMM_INSPECTION	POSTPROCESSING	DMIS_30	1	-
CMM_INSPECTION	SIMULATION	ALL	1	-
CORRUPTED_PARTS	KDA	ALL	1	-
DESIGN	EXPRESSION	MEASUREMENTS	1	-
DESIGN	UFUNC	MODELING	1	-
DRAFTING	SHIPBUILDING	VIEW_CREATION	1	-
FLEXIBLE_PIPE	GENERAL	ALL	1	-
KDA	PROGRESSIVE_DIE	FACE_COLOR_MGT	1	-
KDA	PROGRESSIVE_DIE	MOTION_SIMULATE	1	-
KDA	REUSE_LIBRARY	ADD_REUSE_COMP	1	-

KDA	WELD_ASSISTANT	EASY_SPOT	2	-
KDA	WELD_ASSISTANT	GROOVE	5	-
KDA	WELD_ASSISTANT	IMPACT_ANALYSIS	1	-
KDA	WELD_ASSISTANT	WELD_PT_WIZARD	2	-
MECHATRONICS	OPC	UA_SERVER	1	-
NASTRAN	STRUCTURES	SOL_402	1	-
NX_SHEET_METAL	FLANGE	ALL	1	-
NX_SHEET_METAL	FLAT_PATTERN	ALL	2	-
NX_SHEET_METAL	FLAT_SOLID	ALL	1	-
NX_SHEET_METAL	PREFERENCES	ALL	1	-
NXMANAGER	ASSEMBLY	UPDATE_STRUCT	1	-
NXMANAGER	FILE_OPEN	NX_MASTER_PART	1	-
ROUTING_GENERAL	CUST_DEFAULTS	GENERAL	1	-
ROUTING_GENERAL	REUSE_LIBRARY	PLACE_PART	1	-
SHIP_DESIGN	DRAFTING	SHIP_SECTN_DRWG	1	-
SHIP_DESIGN	DRAWING_AUTO	INVRS_BEND_LINE	2	-
SYSENG	CONFIG_UPGRADE	ROLES	1	-
SYSENG	NXOPEN	UDO	1	-
SYSENG	SELECTION	SNAP_POINT	1	-
SYSENG	UG/OPEN	UDO	1	-
SYSENG	UI_TOOLS	ROLES	2	-
SYSTEM	WNT_INSTALL	ALL	4	-
TC_FEATURES	RESOURCE_MGR	AUTO_ASSEMBLY	3	-
TRANSLATOR	STEP_AP214	ALL	1	-
Total			**88**	**1**

GLOBAL TECHNICAL ACCESS CENTER

Installation Assistance

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

Website:

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

Phone:

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

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